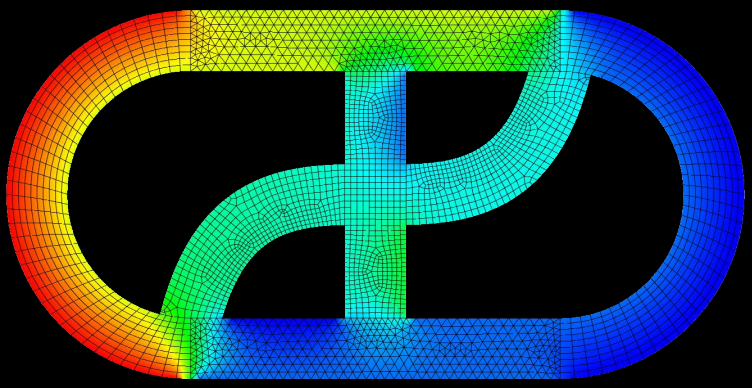


CFD EXPERTS

Simulate the Future

WWW.CFDEXPERTS.NET

Ansys 2021 R2
Engineering What's Ahead.

©2021 ANSYS, Inc.
All Rights Reserved.
Unauthorized use, distribution or duplication is prohibited.

Ansys CFX Tutorials

Ansys
ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
http://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 2021 R2
July 2021

ANSYS, Inc. and Ansys Europe, Ltd. are UL registered ISO 9001: 2015 companies.

Copyright and Trademark Information

© 2021 ANSYS, Inc. Unauthorized use, distribution or duplication is prohibited.

Ansys, Ansys Workbench, AUTODYN, CFX, FLUENT and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries located in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. CFX is a trademark of Sony Corporation in Japan. All other brand, product, service and feature names or trademarks are the property of their respective owners. FLEXlm and FLEXnet are trademarks of Flexera Software LLC.

Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.

ANSYS, Inc. and Ansys Europe, Ltd. are UL registered ISO 9001: 2015 companies.

U.S. Government Rights

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

Third-Party Software

See the legal information in the product help files for the complete Legal Notice for Ansys proprietary software and third-party software. If you are unable to access the Legal Notice, contact ANSYS, Inc.

Published in the U.S.A.

Table of Contents

1. Introduction to the Ansys CFX Tutorials 31
1.1. Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode 33
1.2. Running Ansys CFX Tutorials Using Ansys Workbench 34
1.2.1. Setting Up the Project 34
1.2.2. Writing the CFX-Solver Input (.def) File 35
1.2.3. Obtaining the Solution Using CFX-Solver Manager 35
1.2.4. Viewing the Results Using CFD-Post 36
1.2.5. Creating CFX Component Systems for Multiple Simulations 36
1.2.6. Closing the Applications 36
1.3. Changing the Display Colors 37
1.4. Editor Buttons 37
1.5. Using Help 37
2. Simulating Flow in a Static Mixer Using CFX in Stand-alone Mode 39
2.1. Tutorial Features 39
2.2. Overview of the Problem to Solve 40
2.3. Preparing the Working Directory 41
2.4. Defining the Case Using CFX-Pre 41
2.4.1. Starting Quick Setup Mode 43
2.4.2. Setting the Physics Definition 43
2.4.3. Importing a Mesh 43
2.4.4. Using the Viewer 44
2.4.4.1. Using the Zoom Tools 44
2.4.4.2. Rotating the Geometry 44
2.4.5. Defining Model Data 45
2.4.6. Defining Boundaries 46
2.4.7. Setting Boundary Data 46
2.4.8. Setting Flow Specification 46
2.4.9. Setting Temperature Specification 46
2.4.10. Reviewing the Boundary Condition Definitions 47
2.4.11. Creating the Second Inlet Boundary Definition 47
2.4.12. Creating the Outlet Boundary Definition 48
2.4.13. Moving to General Mode 48
2.4.14. Setting Solver Control 48
2.4.15. Writing the CFX-Solver Input (.def) File 49
2.5. Obtaining the Solution Using CFX-Solver Manager 49
2.5.1. Starting the Run 50
2.5.2. Moving from CFX-Solver Manager to CFD-Post 51
2.6. Viewing the Results Using CFD-Post 51
2.6.1. Setting the Edge Angle for a Wireframe Object 52
2.6.2. Creating a Point for the Origin of the Streamline 54
2.6.3. Creating a Streamline Originating from a Point 55
2.6.4. Rearranging the Point 56
2.6.5. Configuring a Default Legend 57
2.6.6. Creating a Slice Plane 58
2.6.7. Defining Slice Plane Geometry 59
2.6.8. Configuring Slice Plane Views 59
2.6.9. Rendering Slice Planes 60
2.6.10. Coloring the Slice Plane 61
2.6.11. Moving the Slice Plane 62

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.6.12. Adding Contours 62
2.6.13. Working with Animations 64
2.6.13.1. Showing the Animation Dialog Box 64
2.6.13.2. Creating the First Keyframe 65
2.6.13.3. Creating the Second Keyframe 66
2.6.13.4. Viewing the Animation 67
2.6.13.5. Modifying the Animation 67
2.6.13.6. Saving a Movie 69
2.6.14. Quitting CFD-Post 69
3. Simulating Flow in a Static Mixer Using Workbench 71
3.1. Tutorial Features 71
3.2. Overview of the Problem to Solve 72
3.3. Preparing the Working Directory 73
3.4. Setting Up the Project 73
3.5. Defining the Case Using CFX-Pre 74
3.5.1. Creating the Simulation Definition 76
3.5.2. Setting the Physics Definition 76
3.5.3. Defining Boundaries 77
3.5.4. Setting Boundary Data 77
3.5.5. Creating the Second Inlet Boundary Definition 77
3.5.6. Creating the Outlet Boundary Definition 78
3.5.7. Moving to General Mode 78
3.5.8. Using the Viewer 78
3.5.8.1. Using the Zoom Tools 79
3.5.8.2. Rotating the Geometry 79
3.5.9. Setting Solver Control 80
3.6. Obtaining the Solution Using CFX-Solver Manager 81
3.7. Viewing the Results Using CFD-Post 82
3.7.1. Setting the Edge Angle for a Wireframe Object 84
3.7.2. Creating a Point for the Origin of the Streamline 85
3.7.3. Creating a Streamline Originating from a Point 86
3.7.4. Rearranging the Point 87
3.7.5. Configuring a Default Legend 88
3.7.6. Creating a Slice Plane 89
3.7.7. Defining Slice Plane Geometry 90
3.7.8. Configuring Slice Plane Views 90
3.7.9. Rendering Slice Planes 91
3.7.10. Coloring the Slice Plane 92
3.7.11. Moving the Slice Plane 93
3.7.12. Adding Contours 93
3.7.13. Working with Animations 95
3.7.13.1. Showing the Animation Dialog Box 95
3.7.13.2. Creating the First Keyframe 96
3.7.13.3. Creating the Second Keyframe 97
3.7.13.4. Viewing the Animation 98
3.7.13.5. Modifying the Animation 98
3.7.13.6. Saving a Movie 100
3.7.14. Closing the Applications 100
4. Flow in a Static Mixer (Refined Mesh) 103
4.1. Tutorial Features 103
4.2. Overview of the Problem to Solve 104

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.3. Preparing the Working Directory 105
4.4. Defining the Case Using CFX-Pre 105
4.4.1. Importing a Mesh 105
4.4.2. Importing Settings from Tutorial 1 106
4.4.3. Viewing Domain Settings 107
4.4.4. Viewing the Boundary Condition Setting 108
4.4.5. Defining Solver Parameters 108
4.4.6. Writing the CFX-Solver Input (.def) File 109
4.5. Obtaining the Solution Using CFX-Solver Manager 110
4.5.1. Starting the Run with an Initial Values File 110
4.5.2. Confirming Results 110
4.5.3. Moving from CFX-Solver Manager to CFD-Post 111
4.6. Viewing the Results Using CFD-Post 111
4.6.1. Creating a Slice Plane 112
4.6.2. Coloring the Slice Plane 113
4.6.3. Loading Results from Tutorial 1 for Comparison 113
4.6.4. Comparing Slice Planes Using Multiple Views 114
4.6.5. Viewing the Surface Mesh on the Outlet 115
4.6.6. Looking at the Inflated Elements in Three Dimensions 116
4.6.7. Viewing the Surface Mesh on the Mixer Body 117
4.6.8. Viewing the Layers of Inflated Elements on a Plane 118
4.6.9. Viewing the Mesh Statistics 118
4.6.10. Viewing the Mesh Elements with Largest Face Angle 119
4.6.11. Viewing the Mesh Elements with Largest Face Angle Using a Point 119
4.6.12. Quitting CFD-Post 120
5. Optimizing Flow in a Static Mixer 121
5.1. Tutorial Features 121
5.2. Overview of the Problem to Solve 122
5.3. Setting Up Ansys Workbench 123
5.4. Creating the Project 123
5.5. Creating the Geometry in DesignModeler 124
5.5.1. Creating the Solid 124
5.5.1.1. Setting Up the Grid 124
5.5.1.2. Creating the Basic Geometry 125
5.5.1.3. Revolving the Sketch 126
5.5.1.4. Create the First Inlet Pipe 127
5.5.1.4.1. Extrude the First Side Pipe 128
5.5.1.4.2. Make the Solid Visible 128
5.5.1.5. Create the Second Inlet Pipe 129
5.5.1.6. Create Named Selections 131
5.6. Creating the Mesh 132
5.7. Setting up the Case with CFX-Pre 133
5.8. Setting the Output Parameter in CFD-Post 136
5.9. Investigating the Impact of Changing Design Parameters Manually 138
5.10. Using Design of Experiments 140
5.11. Viewing the Response Surface 141
5.12. Optimization based on the Response Surface 142
6. Flow in a Process Injection Mixing Pipe 145
6.1. Tutorial Features 145
6.2. Overview of the Problem to Solve 146
6.3. Preparing the Working Directory 147

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.4. Defining the Case Using CFX-Pre 148
6.4.1. Importing a Mesh 148
6.4.2. Setting Temperature-Dependent Material Properties 149
6.4.3. Plotting an Expression 150
6.4.4. Evaluating an Expression 150
6.4.5. Modify Material Properties 150
6.4.6. Creating the Domain 151
6.4.7. Creating the Side Inlet Boundary 152
6.4.8. Creating the Main Inlet Boundary 153
6.4.9. Creating the Main Outlet Boundary 154
6.4.10. Setting Initial Values 155
6.4.11. Setting Solver Control 155
6.4.12. Writing the CFX-Solver Input (.def) File 156
6.5. Obtaining the Solution Using CFX-Solver Manager 157
6.5.1. Starting the Run 157
6.5.2. Moving from CFX-Solver Manager to CFD-Post 157
6.6. Viewing the Results Using CFD-Post 157
6.6.1. Modifying the Outline of the Geometry 157
6.6.2. Creating and Modifying Streamlines Originating from the Main Inlet 158
6.6.3. Modifying Streamline Color Ranges 158
6.6.4. Coloring Streamlines with a Constant Color 159
6.6.5. Creating Streamlines Originating from the Side Inlet 159
6.6.6. Examining Turbulence Kinetic Energy 160
6.6.7. Quitting CFD-Post 161
7. Flow from a Circular Vent 163
7.1. Tutorial Features 163
7.2. Overview of the Problem to Solve 164
7.3. Preparing the Working Directory 165
7.4. Defining the Case Using CFX-Pre 165
7.4.1. Importing the Mesh 165
7.4.2. Creating an Additional Variable 166
7.4.3. Defining the Steady-State Analysis 166
7.4.3.1. Renaming the Analysis 166
7.4.3.2. Creating the Domain 166
7.4.3.3. Creating the Boundaries 168
7.4.3.3.1. Inlet Boundary 168
7.4.3.3.2. Opening Boundary 169
7.4.3.3.3. Inlet for the Vent 170
7.4.3.4. Setting Initial Values 171
7.4.3.5. Setting Solver Control 171
7.4.4. Defining the Transient Analysis 172
7.4.4.1. Creating the Analysis 172
7.4.4.2. Modifying the Analysis Type 172
7.4.4.3. Modifying the Boundary Conditions 173
7.4.4.3.1. To Modify the Vent Inlet Boundary Condition 173
7.4.4.3.2. Plotting Smoke Concentration 174
7.4.4.4. Initialization Values 176
7.4.4.5. Modifying the Solver Control 176
7.4.4.6. Setting Output Control 176
7.4.5. Configuring Simulation Control 178
7.4.5.1. Configuration Control for the Steady State Analysis 178

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.5.2. Configuration Control for the Transient Analysis 178
7.4.6. Writing the CFX-Solver Input (.mdef) File 179
7.5. Obtaining the Solution Using CFX-Solver Manager 179
7.6. Viewing the Results Using CFD-Post 180
7.6.1. Displaying Smoke Density Using an Isosurface 181
7.6.2. Viewing the Results at Different Time Steps 181
7.6.3. Generating Titled Image Files 182
7.6.3.1. Adding a Title 182
7.6.3.2. JPEG output 183
7.6.4. Generating a Movie 183
7.6.5. Viewing the Dispersion of Smoke at the Final Time Step 184
8. Flow Around a Blunt Body 187
8.1. Tutorial Features 187
8.2. Overview of the Problem to Solve 188
8.3. Preparing the Working Directory 189
8.4. Defining the Case Using CFX-Pre 189
8.4.1. Importing the Mesh 189
8.4.2. Creating the Domain 190
8.4.3. Creating Composite Regions 191
8.4.4. Creating the Boundaries 191
8.4.4.1. Inlet Boundary 191
8.4.4.2. Outlet Boundary 192
8.4.4.3. Free-Slip Wall Boundary 192
8.4.4.4. Symmetry Plane Boundary 193
8.4.4.5. Wall Boundary on the Blunt Body Surface 193
8.4.5. Setting Initial Values 194
8.4.6. Setting Solver Control 194
8.4.7. Writing the CFX-Solver Input (.def) File 195
8.5. Obtaining the Solution Using CFX-Solver Manager 195
8.5.1. Obtaining a Solution in Serial 196
8.5.2. Obtaining a Solution in Parallel 196
8.5.2.1. Background to Parallel Running in CFX 196
8.5.2.2. Obtaining a Solution with Local Parallel 197
8.5.2.3. Obtaining a Solution with Distributed Parallel 197
8.5.2.4. Text Output when Running in Parallel 199
8.6. Viewing the Results Using CFD-Post 200
8.6.1. Using Symmetry Plane to Display the Full Geometry 201
8.6.1.1. Manipulating the Geometry 201
8.6.1.2. Creating an Instance Transform 201
8.6.1.3. Using the Reflection Transform 202
8.6.2. Creating Velocity Vectors 202
8.6.2.1. Creating the Sampling Plane 202
8.6.2.2. Creating a Vector Plot Using Different Sampling Methods 203
8.6.3. Displaying Pressure Distribution on Body and Symmetry Plane 204
8.6.4. Creating Surface Streamlines to Display the Path of Air along the Surface of the Body 205
8.6.5. Moving Objects 206
8.6.6. Creating a Surface Plot of y+ 206
8.6.7. Viewing the Mesh Partitions (Parallel Only) 207
9. Free Surface Flow Over a Bump 209
9.1. Tutorial Features 209
9.2. Overview of the Problem to Solve 210

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.3. Preparing the Working Directory 211
9.4. Defining the Case Using CFX-Pre 211
9.4.1. Importing the Mesh 211
9.4.2. Viewing the Region Labels 212
9.4.3. Creating Expressions for Initial and Boundary Conditions 212
9.4.3.1. Creating Expressions in CEL 213
9.4.3.2. Reading Expressions From a File 214
9.4.4. Creating the Domain 214
9.4.5. Creating the Boundaries 216
9.4.5.1. Inlet Boundary 216
9.4.5.2. Outlet Boundary 217
9.4.5.3. Symmetry Boundaries 217
9.4.5.4. Opening and Wall Boundaries 218
9.4.6. Setting Initial Values 219
9.4.7. Setting Mesh Adaption Parameters 221
9.4.8. Setting the Solver Controls 221
9.4.9. Writing the CFX-Solver Input (.def) File 222
9.5. Obtaining the Solution Using CFX-Solver Manager 223
9.6. Viewing the Results Using CFD-Post 224
9.6.1. Creating Velocity Vector Plots 224
9.6.2. Viewing Mesh Refinement 226
9.6.3. Creating an Isosurface to Show the Free Surface 229
9.6.4. Creating a Polyline that Follows the Free Surface 230
9.6.5. Creating a Chart to Show the Height of the Surface 230
9.6.6. Further Postprocessing 231
9.7. Further Discussion 231
10. Supersonic Flow Over a Wing 233
10.1. Tutorial Features 233
10.2. Overview of the Problem to Solve 234
10.3. Preparing the Working Directory 234
10.4. Defining the Case Using CFX-Pre 235
10.4.1. Importing the Mesh 235
10.4.2. Creating the Domain 236
10.4.3. Creating the Boundaries 236
10.4.3.1. Creating an Inlet Boundary 236
10.4.3.2. Creating an Outlet Boundary 237
10.4.3.3. Creating the Symmetry Plane Boundaries 238
10.4.3.4. Creating a Free Slip Boundary 238
10.4.3.5. Creating a Wall Boundary 239
10.4.4. Creating Domain Interfaces 239
10.4.5. Setting Initial Values 240
10.4.6. Setting the Solver Controls 241
10.4.7. Writing the CFX-Solver Input (.def) File 241
10.5. Obtaining the Solution Using CFX-Solver Manager 242
10.6. Viewing the Results Using CFD-Post 242
10.6.1. Displaying Mach Information 242
10.6.2. Displaying Pressure Information 243
10.6.3. Displaying Temperature Information 243
10.6.4. Displaying Pressure With User Vectors 243
11. Flow Through a Butterfly Valve 245
11.1. Tutorial Features 245

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11.2. Overview of the Problem to Solve 246
11.3. Preparing the Working Directory 247
11.4. Defining the Case Using CFX-Pre 247
11.4.1. Importing the Mesh 248
11.4.2. Defining the Properties of the Sand 248
11.4.3. Creating the Domain 249
11.4.4. Creating the Inlet Velocity Profile 254
11.4.5. Creating the Boundary Conditions 255
11.4.5.1. Inlet Boundary 255
11.4.5.2. Outlet Boundary 259
11.4.5.3. Symmetry Plane Boundary 259
11.4.5.4. Pipe Wall Boundary 260
11.4.5.5. Editing the Default Boundary 261
11.4.6. Setting Initial Values 261
11.4.7. Setting the Solver Controls 262
11.4.8. Writing the CFX-Solver Input (.def) File 263
11.5. Obtaining the Solution Using CFX-Solver Manager 264
11.6. Viewing the Results Using CFD-Post 264
11.6.1. Erosion Due to Sand Particles 264
11.6.2. Displaying Erosion on the Pipe Wall 265
11.6.3. Creating Particle Tracks 265
11.6.4. Creating a Particle Track Animation 266
11.6.5. Determining Minimum, Maximum, and Average Pressure Values 268
11.6.6. Other Features 268
12. Flow in a Catalytic Converter 269
12.1. Tutorial Features 269
12.2. Overview of the Problem to Solve 270
12.3. Preparing the Working Directory 271
12.4. Defining the Case Using CFX-Pre 272
12.4.1. Importing the Meshes and CCL File 272
12.4.1.1. Importing the Required Expressions From a CCL File 272
12.4.1.2. Importing the Housing Mesh 273
12.4.1.3. Importing the Pipe and Flange Mesh 274
12.4.1.4. Creating a Second Pipe and Flange Mesh 274
12.4.1.5. Creating a Single Region for Both Pipe and Flange Meshes 275
12.4.2. Creating the Fluid Domain 275
12.4.3. Creating the Porous Domain 276
12.4.4. Creating and Editing the Boundaries 279
12.4.4.1. Creating the Inlet Boundary 279
12.4.4.2. Creating the Outlet Boundary 280
12.4.4.3. Editing the Housing Default Boundary 280
12.4.5. Creating the Domain Interfaces 281
12.4.6. Setting Initial Values 282
12.4.7. Setting Solver Control 283
12.4.8. Writing the CFX-Solver Input (.def) File 284
12.5. Obtaining the Solution Using CFX-Solver Manager 284
12.6. Viewing the Results Using CFD-Post 284
12.6.1. Viewing the Mesh on a GGI Interface 285
12.6.2. Creating User Locations 286
12.6.2.1. Creating a Slice Plane 286
12.6.2.2. Creating a User Surface 287

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12.6.2.3. Creating a Polyline 288
12.6.3. Creating Plots 289
12.6.3.1. Creating a Contour Plot of Pressure 289
12.6.3.2. Creating a Vector Plot on the Slice Plane 289
12.6.3.3. Creating a Chart of Pressure versus the Z Coordinate 290
12.6.4. Exporting Polyline Data 291
13. Non-Newtonian Fluid Flow in an Annulus 293
13.1. Tutorial Features 293
13.2. Overview of the Problem to Solve 293
13.3. Background Theory 294
13.4. Preparing the Working Directory 296
13.5. Defining the Case Using CFX-Pre 296
13.5.1. Importing the Mesh 296
13.5.2. Creating the Fluid 296
13.5.3. Creating the Domain 298
13.5.4. Creating the Boundaries 298
13.5.4.1. Wall Boundary for the Inner Pipe 299
13.5.4.2. Symmetry Plane Boundary 299
13.5.5. Setting Initial Values 300
13.5.6. Setting Solver Control 301
13.5.7. Writing the CFX-Solver Input (.def) File 302
13.6. Obtaining the Solution Using CFX-Solver Manager 302
13.7. Viewing the Results Using CFD-Post 302
14. Flow in an Axial Turbine Stage 307
14.1. Tutorial Features 307
14.2. Overview of the Problem to Solve 308
14.3. Preparing the Working Directory 311
14.4. Simulating the Stage with the Steady-State Frozen Rotor Model 311
14.4.1. Defining the Case Using CFX-Pre 311
14.4.1.1. Basic Settings 311
14.4.1.2. Component Definition 312
14.4.1.3. Physics Definition 313
14.4.1.4. Interface Definition 315
14.4.1.5. Boundary Definition 315
14.4.1.6. Final Operations 315
14.4.1.7. Writing the CFX-Solver Input (.def) File 315
14.4.2. Obtaining the Solution Using CFX-Solver Manager 316
14.4.2.1. Obtaining a Solution in Serial 316
14.4.2.2. Obtaining a Solution With Local Parallel 316
14.4.2.3. Obtaining a Solution with Distributed Parallel 317
14.4.2.4. Viewing the Results Using CFD-Post 317
14.4.2.4.1. Initializing Turbo-Post 317
14.4.2.4.2. Viewing Three Domain Passages 318
14.4.2.4.3. Blade Loading Turbo Chart 319
14.4.3. Simulating the Stage with the Exit Corrected Mass Flow Rate Boundary Condition 319
14.4.3.1. Writing the CFX-Solver Input (.def) File 321
14.4.3.2. Obtaining the Solution Using CFX-Solver Manager 322
14.4.4. Comparing the Two Cases Using CFD-Post 322
14.5. Simulating the Stage with the Transient Rotor-Stator Model 324
14.5.1. Defining the Case Using CFX-Pre 324
14.5.1.1. Modifying the Physics Definition 325

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.5.1.2. Setting Output Control 326
14.5.1.3. Modifying Execution Control 327
14.5.1.4. Writing the CFX-Solver Input (.def) File 327
14.5.2. Obtaining the Solution Using CFX-Solver Manager 328
14.5.2.1. Serial Solution 328
14.5.2.2. Parallel Solution 328
14.5.2.3. Monitoring the Run 328
14.5.3. Viewing the Results Using CFD-Post 328
14.5.3.1. Initializing Turbo-Post 329
14.5.3.2. Displaying a Surface of Constant Span 329
14.5.3.3. Using Multiple Turbo Viewports 329
14.5.3.4. Creating a Turbo Surface at Mid-Span 330
14.5.3.5. Setting up Instancing Transformations 330
14.5.3.6. Animating the Movement of the Rotor Relative to the Stator 330
14.5.3.7. Further Postprocessing 332
14.6. Live Monitoring of the Solution in CFD-Post 332
14.6.1. Creating and Exporting Surfaces in CFD-Post 332
14.6.1.1. Creating a Turbo Surface at Mid-Span of the Rotor and Stator 333
14.6.1.2. Exporting the Surfaces 333
14.6.2. Defining Monitor Surfaces in CFX-Pre 334
14.6.2.1. Importing the User Surfaces 334
14.6.2.2. Creating a Rotating Coordinate Frame 335
14.6.2.3. Defining the Monitor Surfaces 336
14.6.2.4. Reducing Solution Data 337
14.6.2.5. Defining the Number of Timesteps 337
14.6.2.6. Modifying Execution Control 337
14.6.2.7. Writing the CFX-Solver Input (.def) File 337
14.6.3. Starting the Solution Using CFX-Solver Manager 337
14.6.3.1. Serial Solution 338
14.6.4. Monitoring the Solution in CFD-Post 338
14.6.4.1. Creating a Text Label to View the Current Timestep 338
14.6.4.2. Creating a Contour 338
14.6.4.3. Live Monitoring of the Solution 339
14.6.4.4. Creating a Vector 339
14.6.4.5. Resuming Live Monitoring of the Solution 340
15. Reacting Flow in a Mixing Tube 341
15.1. Tutorial Features 341
15.2. Overview of the Problem to Solve 342
15.3. Preparing the Working Directory 344
15.4. Defining the Case Using CFX-Pre 345
15.4.1. Importing the Mesh 345
15.4.2. Creating a Multicomponent Fluid 345
15.4.2.1. Acid Properties 345
15.4.2.2. Alkali Properties 347
15.4.2.3. Reaction Product Properties 349
15.4.2.4. Fluid Properties 350
15.4.3. Creating an Additional Variable to Model pH 351
15.4.4. Formulating the Reaction and pH as Expressions 351
15.4.4.1. Stoichiometric Ratio 352
15.4.4.2. Reaction Source Terms 354
15.4.4.3. Calculating pH 355

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15.4.4.4. Loading the Expressions to Model the Reaction and pH 357
15.4.5. Creating the Domain 357
15.4.6. Creating a Subdomain to Model the Chemical Reactions 360
15.4.7. Creating the Boundary Conditions 363
15.4.7.1. Water Inlet Boundary 363
15.4.7.2. Acid Inlet Boundary 364
15.4.7.3. Alkali Inlet Boundary 365
15.4.7.4. Outlet Boundary 366
15.4.7.5. Symmetry Boundary 366
15.4.7.6. Default Wall Boundary 367
15.4.8. Setting Initial Values 367
15.4.9. Setting Solver Control 369
15.4.10. Writing the CFX-Solver Input (.def) File 369
15.5. Obtaining the Solution Using CFX-Solver Manager 370
15.6. Viewing the Results Using CFD-Post 370
16. Heat Transfer from a Heating Coil 373
16.1. Tutorial Features 373
16.2. Overview of the Problem to Solve 374
16.3. Preparing the Working Directory 375
16.4. Defining the Case Using CFX-Pre 375
16.4.1. Importing the Mesh 376
16.4.2. Editing the Material Properties 376
16.4.3. Defining the Calcium Carbonate Deposit Material 377
16.4.4. Creating the Domains 378
16.4.4.1. Creating a Fluid Domain 378
16.4.4.2. Creating a Solid Domain 379
16.4.5. Creating the Boundaries 380
16.4.5.1. Heating Coil Boundaries 380
16.4.5.2. Inlet Boundary 381
16.4.5.3. Opening Boundary 381
16.4.6. Creating the Domain Interface 382
16.4.7. Setting Solver Control 383
16.4.8. Writing the CFX-Solver Input (.def) File 384
16.5. Obtaining the Solution using CFX-Solver Manager 384
16.6. Viewing the Results Using CFD-Post 384
16.6.1. Heating Coil Temperature Range 385
16.6.2. Creating a Cylindrical Locator 385
16.6.2.1. Expression 385
16.6.2.2. Variable 386
16.6.2.3. Isosurface of the variable 386
16.6.2.4. Creating a Temperature Profile Chart 387
16.6.3. Specular Lighting 388
16.6.4. Moving the Light Source 388
17. Multiphase Flow in a Mixing Vessel 389
17.1. Tutorial Features 389
17.2. Overview of the Problem to Solve 390
17.3. Preparing the Working Directory 391
17.4. Defining the Case Using CFX-Pre 392
17.4.1. Importing the Meshes 392
17.4.1.1. Importing the Mixer Tank Mesh 393
17.4.1.2. Importing the Impeller Mesh 393

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

17.4.1.3. Relocating the Impeller Mesh 394
17.4.1.4. Viewing the Mesh at the Tank Periodic Boundary 394
17.4.2. Creating the Domains 395
17.4.2.1. Rotating Domain for the Impeller 395
17.4.2.2. Stationary Domain for the Main Tank 399
17.4.3. Creating the Boundaries 399
17.4.3.1. Air Inlet Boundary 400
17.4.3.2. Degassing Outlet Boundary 401
17.4.3.3. Thin Surface for the Baffle 402
17.4.3.4. Wall Boundary for the Shaft 403
17.4.3.5. Required Boundary in the Impeller Domain 405
17.4.3.6. Modifying the Default Wall Boundary 405
17.4.4. Creating the Domain Interfaces 406
17.4.4.1. Modeling the Blade Using a Domain Interface 406
17.4.4.2. Rotational Periodic Interfaces 408
17.4.4.3. Frozen Rotor Interfaces 410
17.4.5. Setting Initial Values 412
17.4.6. Setting Solver Control 413
17.4.7. Adding Monitor Points 414
17.4.8. Writing the CFX-Solver Input (.def) File 415
17.5. Obtaining the Solution Using CFX-Solver Manager 415
17.6. Viewing the Results Using CFD-Post 416
17.6.1. Creating a Plane Locator 416
17.6.2. Plotting Velocity 417
17.6.3. Plotting Pressure Distribution 417
17.6.4. Plotting Volume Fractions 418
17.6.5. Plotting Shear Strain Rate and Shear Stress 418
17.6.6. Calculating Torque and Power Requirements 419
18. Gas-Liquid Flow in an Airlift Reactor 421
18.1. Tutorial Features 421
18.2. Overview of the Problem to Solve 422
18.3. Preparing the Working Directory 423
18.4. Defining the Case Using CFX-Pre 423
18.4.1. Importing the Mesh 424
18.4.2. Creating the Domain 424
18.4.3. Creating the Boundary Conditions 428
18.4.3.1. Inlet Boundary 428
18.4.3.2. Outlet Boundary 430
18.4.3.3. Draft Tube Boundaries 430
18.4.3.4. Symmetry Plane Boundary 431
18.4.3.5. Modifying the Default Boundary 432
18.4.4. Setting Initial Values 432
18.4.5. Setting Solver Control 435
18.4.6. Writing the CFX-Solver Input (.def) File 436
18.5. Obtaining the Solution Using CFX-Solver Manager 437
18.6. Viewing the Results Using CFD-Post 437
18.6.1. Creating Water Velocity Vector Plots 438
18.6.2. Creating Volume Fraction Plots 438
18.6.3. Displaying the Entire Airlift Reactor Geometry 440
18.7. Further Discussion 441
19. Air Conditioning Simulation 443

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

19.1. Tutorial Features 443
19.2. Overview of the Problem to Solve 444
19.3. Preparing the Working Directory 446
19.4. Defining the Case Using CFX-Pre 446
19.4.1. Importing the Mesh 446
19.4.2. Importing CEL Expressions 447
19.4.3. Compiling the Fortran Subroutine for the Thermostat 448
19.4.4. Creating a User CEL Function for the Thermostat 449
19.4.5. Setting the Analysis Type 451
19.4.6. Creating the Domain 452
19.4.7. Creating the Boundaries 453
19.4.7.1. Inlet Boundary 453
19.4.7.2. Outlet Boundary 454
19.4.7.3. Window Boundary 455
19.4.7.4. Default Wall Boundary 456
19.4.8. Closet Wall Interface 457
19.4.9. Creating a Logical Expression for the Conditional GGI Interface 458
19.4.10. Creating a Conditional GGI Interface for the Closet Door 458
19.4.11. Creating Space Under the Closet Door 460
19.4.12. Setting Initial Values 460
19.4.13. Setting Solver Control 463
19.4.14. Setting Output Control 463
19.4.15. Writing the CFX-Solver Input (.def) File 465
19.5. Obtaining the Solution Using CFX-Solver Manager 465
19.6. Viewing the Results Using CFD-Post 466
19.6.1. Creating Graphics Objects 466
19.6.1.1. Creating Planes 466
19.6.1.2. Creating an Isosurface 467
19.6.1.3. Adjusting the Legend 467
19.6.1.4. Creating a Point for the Thermometer 468
19.6.1.5. Creating a Text Label 468
19.6.2. Creating an Animation 469
19.7. Further Discussion 470
20. Combustion and Radiation in a Can Combustor 471
20.1. Tutorial Features 471
20.2. Overview of the Problem to Solve 472
20.3. Preparing the Working Directory 474
20.4. Defining the Case Using CFX-Pre 474
20.4.1. Importing the Mesh 474
20.4.2. Creating a Reacting Mixture 475
20.4.2.1. To create the variable composition mixture 475
20.4.3. Creating the Domain 476
20.4.4. Creating the Boundaries 477
20.4.4.1. Fuel Inlet Boundary 477
20.4.4.2. Bottom Air Inlet Boundary 478
20.4.4.3. Side Air Inlet Boundary 478
20.4.4.4. Outlet Boundary 479
20.4.4.5. Vanes Boundary 479
20.4.4.6. Default Wall Boundary 481
20.4.5. Setting Initial Values 481
20.4.6. Setting Solver Control 483

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

20.4.7. Writing the CFX-Solver Input (.def) File 483
20.5. Obtaining the Solution Using CFX-Solver Manager 484
20.6. Viewing the Results Using CFD-Post 484
20.6.1. Temperature Within the Domain 484
20.6.2. The NO Concentration in the Combustor 485
20.6.3. Printing a Greyscale Graphic 485
20.6.4. Calculating NO Mass Fraction at the Outlet 485
20.6.5. Viewing Flow Field 486
20.6.6. Viewing Radiation 487
21. Cavitation Around a Hydrofoil 489
21.1. Tutorial Features 489
21.2. Overview of the Problem to Solve 490
21.3. Preparing the Working Directory 490
21.4. Simulating the Hydrofoil without Cavitation 491
21.4.1. Defining the Case Using CFX-Pre 491
21.4.1.1. Importing the Mesh 491
21.4.1.2. Loading Materials 491
21.4.1.3. Creating the Domain 492
21.4.1.4. Creating the Boundaries 493
21.4.1.4.1. Inlet Boundary 493
21.4.1.4.2. Outlet Boundary 494
21.4.1.4.3. Free Slip Wall Boundary 494
21.4.1.4.4. Symmetry Plane Boundaries 495
21.4.1.5. Setting Initial Values 495
21.4.1.6. Setting Solver Control 496
21.4.1.7. Writing the CFX-Solver Input (.def) File 497
21.4.2. Obtaining the Solution using CFX-Solver Manager 497
21.4.3. Viewing the Results Using CFD-Post 498
21.4.3.1. Plotting Pressure Distribution Data 498
21.4.3.2. Exporting Pressure Distribution Data 501
21.4.3.3. Saving the Postprocessing State 502
21.5. Simulating the Hydrofoil with Cavitation 502
21.5.1. Defining the Case Using CFX-Pre 502
21.5.1.1. Adding Cavitation 502
21.5.1.2. Modifying Solver Control 503
21.5.1.3. Modifying Execution Control 503
21.5.1.4. Writing the CFX-Solver Input (.def) File 504
21.5.2. Obtaining the Solution using CFX-Solver Manager 504
21.5.3. Viewing the Results Using CFD-Post 505
22. Modeling a Ball Check Valve using Mesh Deformation and the CFX Rigid Body Solver 509
22.1. Tutorial Features 509
22.2. Overview of the Problem to Solve 510
22.3. Preparing the Working Directory 511
22.4. Defining the Case Using CFX-Pre 511
22.4.1. Importing the Mesh 512
22.4.2. Defining a Transient Simulation 512
22.4.3. Editing the Domain 513
22.4.4. Creating a Coordinate Frame 516
22.4.5. Creating a Rigid Body 516
22.4.6. Creating the Subdomain 518
22.4.7. Creating the Boundaries 519

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

22.4.7.1. Ball Boundary 519
22.4.7.2. Symmetry Boundary 520
22.4.7.3. Vertical Valve Wall Boundary 520
22.4.7.4. Tank Opening Boundary 521
22.4.7.5. Valve Opening Boundary 521
22.4.8. Setting Initial Values 522
22.4.9. Setting Solver Control 523
22.4.10. Setting Output Control 524
22.4.11. Writing the CFX-Solver Input (.def) File 525
22.5. Obtaining the Solution Using CFX-Solver Manager 525
22.6. Viewing the Results Using CFD-Post 526
22.6.1. Creating a Slice Plane 526
22.6.2. Creating Points and a Vector Plot 527
22.6.3. Creating an Animation 528
23. Aerodynamic and Structural Performance of a Centrifugal Compressor 531
23.1. Tutorial Features 531
23.2. Overview of the Problem to Solve 532
23.3. Preparing the Working Directory 534
23.4. Setting Up the Project 534
23.5. Defining the Geometry Using Ansys BladeGen 534
23.5.1. Changing the Blade Design Properties 535
23.5.2. Reviewing the Geometry 536
23.6. Defining the Mesh 537
23.6.1. Defining the CFD Mesh Using Ansys TurboGrid 537
23.6.1.1. Defining the Shroud Tip 538
23.6.1.2. Creating the Topology and Mesh 538
23.6.1.3. Changing the Mesh Data Settings 538
23.6.2. Defining the Structural Mesh Using Mechanical Model 539
23.6.2.1. Specifying the Global Mesh Controls 540
23.6.2.2. Defining the Virtual Topology 540
23.6.2.3. Specifying the Sizing Controls 543
23.6.2.4. Specifying the Face Meshing Controls 547
23.6.2.5. Specifying the Method Controls 548
23.6.2.6. Generating the Mesh 549
23.7. Defining the Case Using CFX-Pre 549
23.7.1. Defining the Fluid Region Using Turbo Mode 550
23.7.1.1. Configuring the Basic Settings 550
23.7.1.2. Defining the Components 551
23.7.1.3. Defining the Physics 552
23.7.1.4. Specifying the Domain Interfaces 553
23.7.1.5. Specifying the Boundaries 553
23.7.1.6. Setting the Final Operations 554
23.7.2. Defining the Solid Region Using General Mode 554
23.7.2.1. Specifying the Domains 554
23.7.2.2. Specifying the Boundaries 555
23.7.2.3. Specifying the Domain Interfaces 556
23.8. Obtaining the Solution Using CFX-Solver Manager 557
23.9. Viewing the Results Using CFD-Post 557
23.10. Simulating the Structural Performance Using Static Structural 558
23.10.1. Simulating the Structural Performance without Rotational Velocity 559
23.10.1.1. Importing the Loads 559

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

23.10.1.2. Specifying the Supports 560
23.10.1.3. Obtaining the Solution 561
23.10.2. Simulating the Structural Performance with Rotational Velocity 561
23.10.2.1. Specifying the Loads 562
23.10.2.2. Obtaining the Solution 562
24. Axial Turbine Equilibrium and Non-Equilibrium Steam Predictions 565
24.1. Tutorial Features 565
24.2. Overview of the Problem to Solve 567
24.3. Preparing the Working Directory 568
24.4. Simulating the Equilibrium Phase Change Case 569
24.4.1. Defining the Case Using CFX-Pre 569
24.4.1.1. Basic Settings 569
24.4.1.2. Component Definition 570
24.4.1.3. Physics Definition 571
24.4.1.4. Interface Definition 572
24.4.1.5. Boundary Definition 573
24.4.1.6. Final Operations 573
24.4.1.7. Defining the Properties of Water 573
24.4.1.8. Modifications to Domain and Boundary Conditions 577
24.4.1.9. Setting Initial Values 577
24.4.1.10. Writing the CFX-Solver Input (.def) File 579
24.4.2. Obtaining the Solution Using CFX-Solver Manager 579
24.4.3. Viewing the Results Using CFD-Post 579
24.4.3.1. Specifying Locators for Plots 580
24.4.3.2. Static Pressure and Mass Fraction Contour Plots 580
24.4.3.3. Static Temperature Contour Plots 581
24.5. Simulating the Non-equilibrium Phase Change Case 581
24.5.1. Defining the Case Using CFX-Pre 581
24.5.1.1. Modifying the Domains 582
24.5.1.2. Writing the CFX-Solver Input (.def) File 587
24.5.2. Obtaining the Solution Using CFX-Solver Manager 587
24.5.3. Viewing the Results Using CFD-Post 588
24.5.3.1. Specifying Locators for Plots 588
24.5.3.2. Supercooling Contour Plot 588
24.5.3.3. Nucleation Rate and Droplet Number Contour Plots 589
24.5.3.4. Mass Fraction and Particle Diameter Contour Plots 590
24.5.3.5. Gas and Condensed Phase Static Temperature Contour Plots 591
25. Modeling a Gear Pump using an Immersed Solid 593
25.1. Tutorial Features 593
25.2. Overview of the Problem to Solve 594
25.3. Preparing the Working Directory 595
25.4. Defining the Case Using CFX-Pre 595
25.4.1. Importing the Mesh 595
25.4.2. Creating Expressions for Time Step and Total Time 596
25.4.3. Setting the Analysis Type 596
25.4.4. Creating the Domains 597
25.4.4.1. Creating an Immersed Solid Domain 597
25.4.4.2. Creating the Stationary Fluid Domain 598
25.4.4.3. Creating the Rotating Fluid Domain 601
25.4.5. Creating the Domain Interface 602
25.4.6. Creating Boundary Conditions 605

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

25.4.6.1. Inlet Boundary 605
25.4.6.2. Outlet Boundary 606
25.4.7. Setting Solver Control 607
25.4.8. Setting Output Control 607
25.4.9. Writing the CFX-Solver Input (.def) File 608
25.5. Obtaining the Solution Using CFX-Solver Manager 609
25.6. Viewing the Results Using CFD-Post 610
25.6.1. Creating a Chart of Mass Flow versus Time 611
25.6.2. Creating a Velocity Vector Plot 612
25.6.3. Changing the Appearance in Preparation for an Animation 613
25.6.4. Creating a Keyframe Animation 615
26. Drop Curve for Cavitating Flow in a Pump 617
26.1. Tutorial Features 617
26.2. Overview of the Problem to Solve 618
26.3. Preparing the Working Directory 619
26.4. Simulating the Pump with High Inlet Pressure 619
26.4.1. Defining the Case Using CFX-Pre 619
26.4.1.1. Importing the Mesh 620
26.4.1.2. Loading Materials 620
26.4.1.3. Creating the Domain 620
26.4.1.4. Creating the Boundaries 621
26.4.1.4.1. Inlet Boundary 621
26.4.1.4.2. Outlet Boundary 622
26.4.1.4.3. Wall Boundaries 623
26.4.1.5. Creating Domain Interfaces 623
26.4.1.5.1. Inblock to Passage Interface 624
26.4.1.5.2. Passage to Outblock Interface 624
26.4.1.6. Setting Initial Values 625
26.4.1.7. Setting Solver Controls 626
26.4.1.8. Writing the CFX-Solver Input (.def) File 627
26.4.2. Obtaining the Solution Using CFX-Solver Manager 627
26.4.3. Viewing the Results Using CFD-Post 628
26.5. Simulating the Pump with Cavitation and High Inlet Pressure 628
26.5.1. Defining the Case Using CFX-Pre 628
26.5.1.1. Modifying the Domain and Boundary Conditions 629
26.5.1.2. Creating Expressions 631
26.5.1.3. Adding Monitor Points 632
26.5.1.4. Writing the CFX-Solver Input (.def) File 632
26.5.2. Obtaining the Solution using CFX-Solver Manager 633
26.5.3. Viewing the Results Using CFD-Post 633
26.6. Simulating the Pump with Cavitation and a Range of Inlet Pressures 634
26.6.1. Writing CFX-Solver Input (.def) Files for Lower Inlet Pressures 635
26.6.2. Obtaining the Solutions using CFX-Solver Manager 635
26.6.3. Viewing the Results Using CFD-Post 636
26.6.3.1. Generating a Drop Curve 636
26.6.3.1.1. Creating a Table of the Head and NPSH Values 636
26.6.3.1.2. Creating a Head-versus-NPSH Chart 637
26.6.3.1.3. Viewing the Drop Curve 638
26.6.3.1.4. Creating a Head-versus-NPSH Chart (Optional Exercise) 639
26.6.3.1.5. Viewing the Drop Curve 640
26.6.3.2. Visualizing the Cavitation Regions (Optional Exercise) 641

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.6.3.3. Restoring CFX run history and multi-configuration options 642
27. Spray Dryer 643
27.1. Tutorial Features 643
27.2. Overview of the Problem to Solve 644
27.3. Preparing the Working Directory 645
27.4. Defining the Case Using CFX-Pre 645
27.4.1. Importing the Mesh 646
27.4.2. Importing the Evaporating CCL Drops Model Template 646
27.4.3. Editing the Domain 647
27.4.4. Creating and Editing the Boundary Conditions 649
27.4.4.1. Water Nozzle Boundary 649
27.4.4.2. Air Inlet Boundary 651
27.4.4.3. Outlet Boundary 652
27.4.4.4. Domain 1 Default 652
27.4.5. Creating a Domain Interface 653
27.4.6. Setting Solver Control 653
27.4.7. Setting Output Control 654
27.4.8. Writing the CFX-Solver Input (.def) File 655
27.5. Obtaining the Solution Using CFX-Solver Manager 655
27.6. Viewing the Results Using CFD-Post 655
27.6.1. Displaying the Temperature Using a Contour Plot 656
27.6.2. Displaying the Water Mass Fraction Using a Contour Plot 656
27.6.3. Displaying the Liquid Water Averaged Mean Particle Diameter Using a Contour Plot 656
27.6.4. Displaying the Liquid Water Averaged Temperature Using a Contour Plot 657
27.6.5. Displaying the Liquid Water Temperature Using Particle Tracking 657
27.6.6. Displaying the Diameter of a Water Drop Using Particle Tracking 657
28. Coal Combustion 659
28.1. Tutorial Features 659
28.2. Overview of the Problem to Solve 660
28.3. Preparing the Working Directory 661
28.4. Simulating the Coal Combustion without Swirl and without Nitrogen Oxide 662
28.4.1. Defining the Case Using CFX-Pre 662
28.4.1.1. Importing the Mesh 662
28.4.1.2. Importing the Coal Combustion Materials CCL File 663
28.4.1.3. Creating the Domain 664
28.4.1.4. Creating the Boundary Conditions 668
28.4.1.4.1. Coal Inlet Boundary 668
28.4.1.4.2. Air Inlet Boundary 671
28.4.1.4.3. Outlet Boundary 672
28.4.1.4.4. Coal Gun No-Slip Wall Boundary 673
28.4.1.4.5. Coal Inlet No-Slip Wall Boundary 674
28.4.1.4.6. Air Inlet No-Slip Wall Boundary 674
28.4.1.4.7. Furnace No-Slip Wall Boundary 675
28.4.1.4.8. Quarl No-Slip Wall Boundary 676
28.4.1.4.9. Symmetry Plane Boundaries 676
28.4.1.5. Setting Solver Control 677
28.4.1.6. Writing the CFX-Solver Input (.def) File 679
28.4.2. Obtaining the Solution using CFX-Solver Manager 679
28.4.3. Viewing the Results Using CFD-Post 680
28.4.3.1. Displaying the Temperature on a Symmetry Plane 680
28.4.3.2. Displaying the Water Mass Fraction 680

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.4.3.3. Displaying the Radiation Intensity 680
28.4.3.4. Displaying the Temperature of the Fuel Particles 681
28.4.3.5. Displaying the Ash Mass Fraction using Particle Tracking 681
28.5. Simulating the Coal Combustion with Swirl and without Nitrogen Oxide 681
28.5.1. Defining the Case Using CFX-Pre 682
28.5.1.1. Editing the Boundary Conditions 682
28.5.1.1.1. Air Inlet Boundary 682
28.5.1.1.2. Outlet Boundary 683
28.5.1.1.3. Deleting the Symmetry Plane Boundaries 683
28.5.1.2. Creating a Domain Interface 683
28.5.1.3. Writing the CFX-Solver Input (.def) File 684
28.5.2. Obtaining the Solution Using CFX-Solver Manager 684
28.5.3. Viewing the Results Using CFD-Post 685
28.5.3.1. Displaying the Temperature on a Periodic Interface 685
28.5.3.2. Displaying the Water Mass Fraction 685
28.5.3.3. Displaying the Radiation Intensity 685
28.5.3.4. Displaying the Temperature using Particle Tracking 685
28.5.3.5. Displaying the Ash Mass Fraction using Particle Tracking 686
28.6. Simulating the Coal Combustion with Swirl and with Nitrogen Oxide 686
28.6.1. Defining the Case Using CFX-Pre 686
28.6.1.1. Editing the Domain 686
28.6.1.2. Writing the CFX-Solver Input (.def) File 688
28.6.2. Obtaining the Solution Using CFX-Solver Manager 688
28.6.3. Viewing the Results Using CFD-Post 689
29. Steam Jet 691
29.1. Tutorial Features 691
29.2. Overview of the Problem to Solve 692
29.3. Preparing the Working Directory 693
29.4. Defining the Case Using CFX-Pre 693
29.4.1. Importing the Mesh 693
29.4.2. Importing the Steam Jet CCL 694
29.4.3. Creating a Steady-State Analysis 695
29.4.4. Creating and Loading Materials 696
29.4.4.1. Loading the Steam3v, Steam3l, and Steam3vl Materials 696
29.4.4.2. Creating the Gas Mixture Material 697
29.4.4.3. Creating the Liquid Mixture Material 697
29.4.5. Creating the Domain 697
29.4.6. Creating Subdomains 702
29.4.6.1. Gas-to-Liquid Source Subdomain 702
29.4.6.2. Liquid-to-Gas Source Subdomain 708
29.4.7. Creating Boundaries 713
29.4.7.1. Inlet Boundary 713
29.4.7.2. Opening Boundary for the Outside Edges 715
29.4.7.3. Creating the Symmetry Plane Boundaries 717
29.4.8. Creating a Time Step Expression 718
29.4.9. Setting Solver Control 718
29.4.10. Writing the CFX-Solver Input (.def) File 720
29.5. Obtaining the Solution Using CFX-Solver Manager 720
29.6. Viewing the Results Using CFD-Post 720
29.6.1. Displaying the Steam Molar Fraction 720
29.6.2. Displaying the Gas-to-Liquid Mass Transfer Rate 721

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

29.6.3. Displaying the Liquid-to-Gas Mass Transfer Rate 721
29.6.4. Displaying the Gas-to-Liquid and Liquid-to-Gas Phase Transfer Rates in Synchronous Views 722
29.6.5. Creating a Chart to Plot the False Time Step Along a Line 722
30. Modeling a Buoy using the CFX Rigid Body Solver 725
30.1. Tutorial Features 725
30.2. Overview of the Problem to Solve 726
30.3. Preparing the Working Directory 728
30.4. Simulating the Buoy with Fully Coupled Mesh Motion 728
30.4.1. Defining the Case Using CFX-Pre 728
30.4.1.1. Importing the Mesh 728
30.4.1.2. Importing the Required Expressions From a CCL File 729
30.4.1.3. Defining a Transient Simulation 729
30.4.1.4. Editing the Domain 731
30.4.1.5. Creating a Rigid Body 735
30.4.1.6. Creating the Boundary Conditions 737
30.4.1.6.1. Symmetry Boundaries 737
30.4.1.6.2. Wall Boundaries 738
30.4.1.6.3. Opening Boundary 740
30.4.1.7. Setting Initial Values 742
30.4.1.8. Setting the Solver Control 744
30.4.1.9. Setting the Output Control 746
30.4.1.10. Writing the CFX-Solver Input (.def) File 748
30.4.2. Obtaining the Solution Using CFX-Solver Manager 748
30.4.3. Viewing the Results Using CFD-Post 749
30.4.3.1. Creating a Contour Plot 749
30.4.3.2. Creating a Keyframe Animation 750
30.4.3.3. Calculating the Minimum Mesh Face Angle 752
30.5. Simulating the Buoy with Decoupled Mesh Motion 752
30.5.1. Defining the Case Using CFX-Pre 752
30.5.1.1. Creating a Subdomain 753
30.5.1.2. Editing the Domain Interfaces 754
30.5.1.3. Writing the CFX-Solver Input (.def) File 755
30.5.2. Obtaining the Solution Using CFX-Solver Manager 755
30.5.3. Viewing the Results Using CFD-Post 756
30.5.3.1. Loading a Contour Plot from the State File 756
30.5.3.2. Creating a Keyframe Animation 756
30.5.3.3. Calculating the Minimum Mesh Face Angle 757
30.6. Comparing the Two Cases Using CFD-Post 758
31. Time Transformation Method for an Inlet Disturbance Case 761
31.1. Tutorial Features 761
31.2. Overview of the Problem to Solve 761
31.3. Preparing the Working Directory 763
31.4. Defining a Steady-state Case in CFX-Pre 764
31.4.1. Basic Settings 764
31.4.2. Components Definition 764
31.4.3. Physics Definition 765
31.4.4. Modifying the Fluid Model Settings 766
31.4.5. Initializing Profile Boundary Conditions 766
31.4.6. Modifying Inlet and Outlet Boundary Conditions 767
31.4.7. Writing the CFX-Solver Input (.def) File 768

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.5. Obtaining a Solution to the Steady-state Case 768
31.6. Defining a Transient Blade Row Case in CFX-Pre 769
31.6.1. Opening the Existing Case 769
31.6.2. Modifying the Analysis Type 769
31.6.3. Creating the Local Rotating Coordinate Frame 769
31.6.4. Setting up a Transient Blade Row Model 770
31.6.5. Applying the Local Rotating Frame to the Inlet Boundary 772
31.6.6. Setting the Output Control and Creating Monitor Points 773
31.6.7. Writing the CFX-Solver Input (.def) File 774
31.7. Obtaining a Solution to the Transient Blade Row Case 775
31.8. Viewing the Time Transformation Results in CFD-Post 776
31.8.1. Creating a Turbo Surface 776
31.8.2. Creating a Contour Plot 776
31.8.3. Animating Temperature 777
32. Fourier Transformation Method for an Inlet Disturbance Case 779
32.1. Tutorial Features 779
32.2. Overview of the Problem to Solve 780
32.3. Preparing the Working Directory 781
32.4. Defining a Transient Blade Row Case in CFX-Pre 782
32.4.1. Basic Settings 782
32.4.2. Components Definition 782
32.4.3. Physics Definition 783
32.4.4. Disturbance Definition 784
32.4.5. Modifying the Fluid Model Settings 785
32.4.6. Initializing Profile Boundary Conditions 785
32.4.7. Creating the Local Rotating Coordinate Frame 787
32.4.8. Modifying Inlet and Outlet Boundary Conditions 788
32.4.9. Setting up a Transient Blade Row Model 789
32.4.10. Setting the Output Control and Creating Monitor Points 790
32.4.11. Setting the Execution Control 792
32.4.12. Writing the CFX-Solver Input (.def) File 794
32.5. Obtaining a Solution to the Transient Blade Row Case 795
32.6. Viewing the Fourier Transformation Results in CFD-Post 796
32.6.1. Creating a Turbo Surface 796
32.6.2. Creating a Contour Plot 796
32.6.3. Animating Temperature 796
33. Time Transformation Method for a Transient Rotor-stator Case 799
33.1. Tutorial Features 799
33.2. Overview of the Problem to Solve 800
33.3. Preparing the Working Directory 801
33.4. Defining a Steady-state Case in CFX-Pre 801
33.4.1. Basic Settings 802
33.4.2. Components Definition 802
33.4.3. Physics Definition 803
33.4.4. Additional Fluid Model Settings 804
33.4.5. Initializing Profile Boundary Conditions 805
33.4.6. Modifying Inlet and Outlet Boundary Conditions 805
33.4.7. Visualizing the Profile Boundary Value 807
33.4.8. Writing the CFX-Solver Input (.def) File 807
33.5. Obtaining a Solution to the Steady-state Case 808
33.6. Defining a Transient Blade Row Case in CFX-Pre 808

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

33.6.1. Opening the Existing Case 808
33.6.2. Modifying the Analysis Type 809
33.6.3. Modifying the Rotor–stator Interface 809
33.6.4. Setting up a Transient Blade Row Model 809
33.6.5. Setting Output Control and Creating Monitor Points 811
33.6.6. Writing the CFX-Solver Input (.def) File 812
33.7. Obtaining a Solution to the Transient Blade Row Case 813
33.8. Viewing the Time Transformation Results in CFD-Post 814
33.8.1. Creating a Turbo Surface 814
33.8.2. Creating a Vector Plot 815
33.8.3. Creating a Contour Plot 815
33.8.4. Creating a Variable Time Chart 816
33.8.5. Setting up Data Instancing Transformations 817
33.8.6. Setting up Graphical Instancing Transformations 818
33.8.7. Animating the Movement of the Rotor Relative to the Stator 819
34. Time Transformation Method for a Transient Rotor-stator Case with Conjugate Heat Transfer 821
34.1. Tutorial Features 821
34.2. Overview of the Problem to Solve 822
34.3. Preparing the Working Directory 823
34.4. Defining a Transient Blade Row Case with Conjugate Heat Transfer in CFX-Pre 824
34.4.1. Basic Settings 824
34.4.2. Components Definition 825
34.4.3. Physics Definition 826
34.4.4. Additional Fluid Model Settings 827
34.4.5. Importing the Solid Blade Mesh File 827
34.4.6. Creating the Solid Blade Domain 828
34.4.7. Modifying the Boundary Conditions 829
34.4.7.1. Initializing Profile Boundary Conditions 829
34.4.7.2. Modifying Inlet and Outlet Boundary Conditions 830
34.4.7.3. Visualizing the Profile Boundary Value 832
34.4.7.4. Creating the Blade Base Boundary 832
34.4.8. Creating the Conjugate Heat Transfer Interface 833
34.4.9. Setting up a Transient Blade Row Model 833
34.4.10. Setting Output Control and Creating Monitor Points 834
34.4.11. Writing the CFX-Solver Input (.def) File 837
34.5. Obtaining a Solution to the Transient Blade Row Case 837
34.6. Viewing the Time Transformation Results in CFD-Post 839
34.6.1. Creating a Slice Plane 840
34.6.2. Creating a Contour Plot 840
34.6.3. Creating a Variable Time Chart 841
34.6.4. Setting up Data Instancing Transformations 842
35. Fourier Transformation Method for a Transient Rotor-stator Case 843
35.1. Tutorial Features 843
35.2. Overview of the Problem to Solve 844
35.3. Preparing the Working Directory 846
35.4. Defining and Obtaining a Solution for the Time Integration Solution Method Case 846
35.4.1. Basic Settings 847
35.4.2. Components Definition 847
35.4.3. Physics Definition 848
35.4.4. Disturbance Definition 850
35.4.5. Additional Fluid Model Settings 850

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.4.6. Initializing Profile Boundary Conditions 851
35.4.7. Modifying Inlet and Outlet Boundary Conditions 851
35.4.8. Setting up a Transient Blade Row Model 853
35.4.9. Setting Output Control and Creating Monitor Points 854
35.4.10. Setting the Execution Control 856
35.4.11. Writing the CFX-Solver Input (.def) File 858
35.4.12. Obtaining a Solution for the Time Integration Solution Method Case 858
35.4.12.1. Confirming Convergence Using Derived Variables 859
35.5. Defining and Obtaining a Solution for the Harmonic Balance Solution Method Case 861
35.5.1. Opening the Existing Case 861
35.5.2. Modifying the Transient Blade Row Model 862
35.5.3. Modifying the Solver Control 862
35.5.4. Modifying the Output Control 863
35.5.5. Modifying Execution Control 865
35.5.6. Writing the CFX-Solver Input (.def) File 865
35.5.7. Obtaining a Solution for the Harmonic Balance Solution Method Case 866
35.6. Postprocessing the Transient Rotor-stator Solution 866
35.6.1. Creating a Turbo Surface 866
35.6.2. Creating a Vector Plot 867
35.6.3. Creating a Contour Plot 867
35.6.4. Creating a Variable Time Chart 868
35.6.5. Viewing the Harmonic Forced Response Results 870
35.6.6. Setting up Data Instancing Transformations 870
35.6.7. Setting up Graphical Instancing Transformations 871
35.6.8. Animating the Movement of the Rotor Relative to the Stator 871
36. Fourier Transformation Method for a Blade Flutter Case 875
36.1. Tutorial Features 875
36.2. Overview of the Problem to Solve 876
36.3. Preparing the Working Directory 879
36.4. Defining and Obtaining a Solution for the Steady-state Case 879
36.4.1. Importing the Mesh 879
36.4.2. Expanding Profile Data 880
36.4.3. Initializing Profile Data 882
36.4.4. Creating the Domain 882
36.4.5. Creating the Boundaries 884
36.4.5.1. Inlet Boundary 884
36.4.5.2. Outlet Boundary 886
36.4.5.3. Wall Boundaries 887
36.4.6. Creating Domain Interfaces 889
36.4.7. Writing the CFX-Solver Input (.def) File 893
36.4.8. Obtaining a Solution for the Steady-state Case 893
36.5. Defining and Obtaining a Solution for the Time Integration Solution Method Case 893
36.5.1. Opening the Existing Case 893
36.5.2. Modifying the Analysis Type 894
36.5.3. Modifying the Domain 894
36.5.4. Creating Expressions for Frequency and Scaling Factor 895
36.5.5. Modifying the R1 Blade Boundary 895
36.5.6. Setting up a Transient Blade Row Model 897
36.5.7. Setting Output Control and Creating Monitor Points 899
36.5.8. Writing the CFX-Solver Input (.def) File 905
36.5.9. Obtaining a Solution for the Time Integration Solution Method Case 905

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.6. Defining and Obtaining a Solution for the Harmonic Balance Solution Method Case 906
36.6.1. Opening the Existing Case 906
36.6.2. Modifying the Transient Blade Row Model 907
36.6.3. Modifying the Solver Control 907
36.6.4. Modifying the Output Control 908
36.6.5. Writing the CFX-Solver Input (.def) File 911
36.6.6. Obtaining a Solution for the Harmonic Balance Solution Method Case 911
36.7. Postprocessing the Blade Flutter Solution 911
36.7.1. Viewing Results in CFX-Solver Manager 912
36.7.2. Viewing Results in CFD-Post 912
36.7.2.1. Displaying Total Wall Work on the Blade 913
36.7.2.2. Creating a Contour Plot for Total Wall Work on the Blade 913
36.7.2.3. Creating an Animation for Total Wall Work on the Blade 914
37. Time Transformation Method for a 1.5-stage Transient Rotor-stator Case 917
37.1. Tutorial Features 917
37.2. Overview of the Problem to Solve 918
37.3. Preparing the Working Directory 919
37.4. Defining a Steady-state Case in CFX-Pre 920
37.4.1. Basic Settings 920
37.4.2. Components Definition 920
37.4.3. Physics Definition 921
37.4.4. Additional Fluid Model Settings 922
37.4.5. Initializing Profile Boundary Conditions 923
37.4.6. Modifying Inlet and Outlet Boundary Conditions 924
37.4.7. Visualizing the Profile Boundary Value 925
37.4.8. Writing the CFX-Solver Input (.def) File 926
37.5. Obtaining a Solution to the Steady-state Case 926
37.6. Defining a Transient Blade Row Case in CFX-Pre 926
37.6.1. Opening the Existing Case 926
37.6.2. Modifying the Analysis Type 927
37.6.3. Modifying the Rotor–stator Interfaces 927
37.6.4. Setting up a Transient Blade Row Model 928
37.6.5. Setting Output Control and Creating Monitor Points 930
37.6.6. Writing the CFX-Solver Input (.def) File 932
37.7. Obtaining a Solution to the Transient Blade Row Case 933
37.8. Viewing the Time Transformation Results in CFD-Post 934
37.8.1. Creating a Turbo Surface 934
37.8.2. Creating a Vector Plot 935
37.8.3. Creating a Blade-to-Blade Plot 935
37.8.4. Creating a Chart of Force on a Rotor Blade versus Time 936
37.8.5. Creating a Chart of a Solution Monitor and Fourier Coefficient Data for Pressure versus Time 937
37.8.6. Setting up Data Instancing Transformations 939
37.8.7. Animating the Movement of the Rotor Relative to the Stator 940
38. Film Cooling in a Turbine 943
38.1. Tutorial Features 943
38.2. Overview of the Problem to Solve 944
38.3. Preparing the Working Directory 944
38.4. Defining the Case Using CFX-Pre 945
38.4.1. Changing the Inlet Temperature 945
38.4.2. Initializing Profile Data 945

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

38.4.3. Preparing an Expanded Profile for Later Use 947
38.4.4. Mapping Data to Injection Region Settings 949
38.4.5. Adding Cooling Holes to the Rotor and Hub 950
38.4.6. Using an Injection Region as a Locator in a CEL Expression 953
38.4.7. Modifying Execution Control 954
38.4.8. Writing the CFX-Solver Input (.def) File 955
38.5. Obtaining the Solution Using CFX-Solver Manager 955
38.6. Viewing the Results Using CFD-Post 959
38.6.1. Showing the Mesh Lines 960
38.6.2. Creating a Contour Plot of Temperature 960
38.6.3. Showing the Cooling Hole Locations using Point Clouds 962
38.6.4. Creating Streamlines from the Point Clouds 963
39. Operating Map for a Centrifugal Compressor 967
39.1. Tutorial Features 967
39.2. Overview of the Problem to Solve 968
39.3. Preparing the Working Directory 968
39.4. Defining the Case Using CFX-Pre 969
39.4.1. Creating a Table Function for Operating Point Input Parameters 969
39.4.2. Defining the Operating Point Output Parameters 972
39.4.3. Creating the Operating Points Object 973
39.4.4. Applying the Operating Point Input Parameters to the Case 974
39.4.4.1. Setting Solver Control 975
39.4.5. Writing the CFX-Solver Input (.def) File 975
39.5. Obtaining the Solution Using CFX-Solver Manager 976
39.6. Viewing the Results Using CFD-Post 977
39.6.1. Viewing and Modifying the Pre-defined Operating Map Chart 977
39.6.2. Visually Checking the Quality of a Response Surface by Creating a New Operating Map Chart 988
40. Flow in an Intake Manifold using GT-SUITE Coupling 991
40.1. Tutorial Features 991
40.2. Overview of the Problem to Solve 991
40.3. Preparing the Working Directory 992
40.4. Defining the Case Using CFX-Pre 993
40.4.1. Importing a Mesh 993
40.4.2. Importing the Intake Manifold CCL File 993
40.4.3. Initializing GT-SUITE Coupling 994
40.4.4. Editing the Boundaries 995
40.4.5. Writing the CFX-Solver Input (.def) File 996
40.5. Obtaining the Solution Using CFX-Solver Manager 996
40.5.1. Starting the Run 996

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

List of Figures

2.1. Static Mixer with 2 Inlet Pipes and 1 Outlet Pipe 40
3.1. Static Mixer with 2 Inlet Pipes and 1 Outlet Pipe 73
5.1. Static Mixer with 2 Inlet Pipes and 1 Outlet Pipe 123
6.1. Injection Mixing Pipe 147
8.1. External Air Flow Over a Generic Vehicle Body 188
9.1. Mesh around the bump 227
13.1. Shear Stress of a Shear-thickening Fluid 295
13.2. Apparent Viscosity of a Shear-thickening Fluid 295
14.1. Geometry subsection 310
17.1. Cut-away Diagram of the Mixer 391
17.2. Impeller Mesh Region 392
18.1. Cut-away Diagram of the Airlift Reactor 422
23.1. Small Face at the Leading Edge of the Blade 541
23.2. Back View of the Blade 542
23.3. Small Face at the Trailing Edge of the Blade 542
23.4. Bottom View of the Blade 543
23.5. Three Edges at the Leading Edge of the Blade 543
23.6. Two Edges on the Back of the Blade 544
23.7. Edge on the Trailing Edge of the Blade 545
23.8. Top View of the Blade 546
23.9. Leading Edge of the Blade 547
25.1. Velocity Vectors on the Outlet 610
36.1. Single-row Reference Case Containing 36 Blades 877
38.1. Injection Positions 947
38.2. Mass Flow through each Injection Region 956
38.3. User Points versus Accumulated Time Step 958
38.4. Conservation-related User Points versus Accumulated Time Step 959
38.5. Temperature on Blade and Hub 961
38.6. Hole Locations shown using Point Cloud 963
38.7. Streamlines starting from Cooling Holes 965

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

List of Tables

15.1. Properties of the Dilute Sulfuric Acid Solution 342
15.2. Properties of the Dilute Sodium Hydroxide Solution 343
15.3. Properties of the Salt Water Product 343
26.1. Pump Performance Data 634

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 1: Introduction to the Ansys CFX Tutorials

The CFX tutorials are designed to introduce general techniques used in CFX and to provide tips on advanced modeling.

The initial tutorials introduce general principles used in CFX, including setting up the physical models, running CFX-Solver and visualizing the results in CFD-Post; the later tutorials highlight specialized features of CFX.

This manual contains the following tutorials:

- Simulating Flow in a Static Mixer Using CFX in Stand-alone Mode (p. 39) simulates a static mixer consisting of two inlet pipes delivering water into a mixing vessel; the water exits through an outlet pipe. A general workflow is established for analyzing the flow of fluid into and out of a mixer.
- Simulating Flow in a Static Mixer Using Workbench (p. 71) simulates the previous tutorial using Ansys Workbench.
- Flow in a Static Mixer (Refined Mesh) (p. 103) uses a refined mesh to obtain a better solution to the Static Mixer problem created in the first tutorial.
- Optimizing Flow in a Static Mixer (p. 121) shows how to use Design Points and DesignXplorer to optimize the static mixer first shown in Simulating Flow in a Static Mixer Using CFX in Stand-alone Mode (p. 39).
- Flow in a Process Injection Mixing Pipe (p. 145) describes the general approach taken when working with an existing mesh.
- Flow from a Circular Vent (p. 163) simulates a chimney stack releasing smoke that is dispersed into the atmosphere with an oncoming side wind.
- Flow Around a Blunt Body (p. 187) models the behavior of fluid flow around a generic vehicle body.
- Free Surface Flow Over a Bump (p. 209) demonstrates the simulation of a free surface flow in which the bottom of the channel is interrupted by a semicircular bump.
- Supersonic Flow Over a Wing (p. 233) simulates supersonic flow over a symmetric NACA0012 airfoil at an angle of attack of 0°.
- Flow Through a Butterfly Valve (p. 245) investigates the detailed flow pattern around a valve to better understand why flow losses occur.
- Flow in a Catalytic Converter (p. 269) models a catalytic converter in order to determine the pressure drop and heat transfer through it.
- Non-Newtonian Fluid Flow in an Annulus (p. 293) simulates a shear-thickening liquid rotating in a 2D eccentric annular pipe gap.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Flow in an Axial Turbine Stage (p. 307) sets up a transient calculation of an axial turbine stage.
- Reacting Flow in a Mixing Tube (p. 341) models basic reacting flows using a multicomponent fluid and CEL expressions.
- Heat Transfer from a Heating Coil (p. 373) models the transfer of thermal energy from an electrically-heated solid copper coil to the water flowing around it.
- Multiphase Flow in a Mixing Vessel (p. 389) simulates the mixing of water and air in a mixing vessel.
- Gas-Liquid Flow in an Airlift Reactor (p. 421)demonstrates the Eulerian-Eulerian multiphase model by simulating an airlift reactor (a tall, gas-liquid contacting vessel used in processes where gas absorption is important).
- Air Conditioning Simulation (p. 443) simulates a room with a thermostat-controlled air conditioner.
- Combustion and Radiation in a Can Combustor (p. 471) gives a qualitative impression of the flow and temperature distributions inside a can combustor that burns methane in air.
- Cavitation Around a Hydrofoil (p. 489) demonstrates cavitation in the flow of water around a hydrofoil by modeling a thin slice of the hydrofoil and using two symmetry boundary conditions.
- Modeling a Ball Check Valve using Mesh Deformation and the CFX Rigid Body Solver (p. 509) uses an example of a ball check valve to demonstrate both two-way Fluid-Structure Interaction (FSI) between a ball and a fluid, as well as the mesh-deformation capabilities of Ansys CFX.
- Aerodynamic and Structural Performance of a Centrifugal Compressor (p. 531) simulates the aerodynamic and structural performance of a centrifugal compressor.
- Axial Turbine Equilibrium and Non-Equilibrium Steam Predictions (p. 565) uses an axial turbine to demonstrate setting up and executing equilibrium and non-equilibrium steam calculations using the IAPWS water database for properties.
- Modeling a Gear Pump using an Immersed Solid (p. 593) simulates a gear pump that drives a flow of water.
- Drop Curve for Cavitating Flow in a Pump (p. 617) uses a simple pump to illustrate the basic concepts of setting up, running, and postprocessing a cavitation problem.
- Spray Dryer (p. 643) models the way in which water drops are evaporated by a hot air flow.
- Coal Combustion (p. 659) models coal combustion and radiation in a furnace.
- Steam Jet (p. 691) simulates a high-speed wet steam jet into air.
- Modeling a Buoy using the CFX Rigid Body Solver (p. 725) models the interaction between a rigid body (represented by a buoy) and two fluids (air and water).
- Time Transformation Method for an Inlet Disturbance Case (p. 761) sets up a transient blade row calculation to model an inlet disturbance (frozen gust) in an axial turbine using the Time Transformation model.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Fourier Transformation Method for an Inlet Disturbance Case (p. 779) sets up a transient blade row calculation to model an inlet disturbance (frozen gust) in an axial turbine using the Fourier Transformation model.
- Time Transformation Method for a Transient Rotor-stator Case (p. 799) sets up a transient blade row calculation using the Time Transformation model.
- Time Transformation Method for a Transient Rotor-stator Case with Conjugate Heat Transfer (p. 821) sets up a transient blade row calculation with conjugate heat transfer using the Time Transformation model.
- Fourier Transformation Method for a Transient Rotor-stator Case (p. 843) sets up a transient blade row calculation using the Fourier Transformation model.
- Fourier Transformation Method for a Blade Flutter Case (p. 875) sets up a transient blade row simulation using the Fourier Transformation model as part of blade flutter modeling. The tutorial demonstrates two approaches: Time Integration Solution Method and Harmonic Balance Solution Method.
- Time Transformation Method for a 1.5-stage Transient Rotor-stator Case (p. 917) sets up a transient blade row calculation with a multistage machine using the Time Transformation model.
- Film Cooling in a Turbine (p. 943) defines injection regions on a turbine and shows the injection positions in CFX-Pre and CFD-Post.
- Operating Map for a Centrifugal Compressor (p. 967) sets up an operating point case and displays the results using operating maps.
- Flow in an Intake Manifold using GT-SUITE Coupling (p. 991) sets up a case that is coupled with GT-SUITE.

You should review the following topics before attempting to start a tutorial for the first time:

1.1. Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode
1.2. Running Ansys CFX Tutorials Using Ansys Workbench
1.3. Changing the Display Colors
1.4. Editor Buttons
1.5. Using Help

1.1. Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode

Before you start CFX-Pre, CFX-Solver Manager, or CFD-Post, set the working directory. The procedure for setting the working directory and starting Ansys CFX in stand-alone is listed below:

1. Start the Ansys CFX Launcher.

   You can start the launcher in any of the following ways:

   - On Windows:

     – From the **Start** menu, select **All Programs > ANSYS 2021 R2 > Fluid Dynamics > CFX 2021 R2**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - In a Command Prompt that has its path set up correctly to run CFX, enter `cfx5` (otherwise, you will need to type the full pathname of the `cfx5` command).

- On Linux, enter `cfx5` in a terminal window that has its path set up to run CFX.

2. Specify the **Working Directory** in the launcher window.
3. Click the **CFX-Pre 2021 R2** button.
4. If you were directed here at some point during a tutorial, return to that location.

Note:

All tutorials assume that the **CFX run history and multi-configuration options**, under the **Load Results File** dialog box in CFD-Post, is set to **Load only the last results**.

1.2. Running Ansys CFX Tutorials Using Ansys Workbench

Most Ansys CFX tutorials are written to work in stand-alone mode. This section includes the steps required to run these tutorials in Ansys Workbench:

1.2.1. Setting Up the Project
1.2.2. Writing the CFX-Solver Input (.def) File
1.2.3. Obtaining the Solution Using CFX-Solver Manager
1.2.4. Viewing the Results Using CFD-Post
1.2.5. Creating CFX Component Systems for Multiple Simulations
1.2.6. Closing the Applications

Tip:

You may find it useful to open the Ansys CFX help from the Ansys CFX Launcher (which does not take up a license).

1.2.1. Setting Up the Project

1. Start Ansys Workbench.

   - To launch Ansys Workbench on Windows, click the **Start** menu, then select **All Programs > ANSYS 2021 R2 > Workbench 2021 R2**.

   - To launch Ansys Workbench on Linux, open a command line interface, type the path to `runwb2` (for example, `~/ansys_inc/v212/Framework/bin/Linux64/runwb2`), then press **Enter**.

2. From the main menu, select **File** > **Save As**.
3. In the **Save As** dialog box, browse to a working directory and specify a filename for the project.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Click **Save**.

5. In the **Toolbox** pane, open **Component Systems** and double-click **CFX**. A **CFX** system opens in the **Project Schematic**.

   Note:

   You use a **CFX** component system because you are starting with a mesh. If you want to create the geometry and mesh, you will start with a **Fluid Flow (CFX)** system.

6. Type in the new name, such as `System 1`, to replace the highlighted text below the system.

   Alternatively, you can right-click the first cell in the system and select **Rename**. The name will be highlighted. Now you can change the highlighted text by typing in the new name.

7. In the **Project Schematic**, right-click the **Setup** cell and select **Edit** to launch CFX-Pre.

8. Continue from the section of the tutorial that provides instructions for defining the case using CFX-Pre. The section is typically named *Defining the Case Using CFX-Pre*.

1.2.2. Writing the CFX-Solver Input (.def) File

When running Ansys CFX within Ansys Workbench, no action is required for this section of the tutorial. The required files are automatically transferred between the cells within the **CFX** component system. Continue from Obtaining the Solution Using CFX-Solver Manager (p. 35).

1.2.3. Obtaining the Solution Using CFX-Solver Manager

Once the simulation setup is complete, the **Solution** cell prompts you to refresh it. To refresh that cell:

- Right-click the **Solution** cell and select **Refresh**.

  Note:

  If the **Solution** cell displays a prompt to perform an update, ignore it and proceed to the next step.

To obtain a solution, you need to launch the CFX-Solver Manager and subsequently use it to start the solver:

1. Right-click the **Solution** cell and select **Edit**.

   The CFX-Solver Manager appears with the **Define Run** dialog box displayed.

2. Continue from the *Obtaining a Solution Using CFX-Solver Manager* section of the tutorial.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.2.4. Viewing the Results Using CFD-Post

When CFX-Solver has finished, a completion message appears in a dialog box. Click **OK**.

Alternatively, a message saying `This run of the Ansys CFX-Solver has finished` is displayed in the final line of the CFX-Solver Output file in the CFX-Solver Manager.

Once CFX-Solver has finished, you can use CFD-Post to review the finished results. At this point, the **Results** cell in Ansys Workbench prompts you to refresh:

1. Right-click the **Results** cell and select **Refresh**.
2. When the refresh is complete, right-click the **Results** cell and select **Edit** to open CFD-Post.
3. Continue from the *Viewing the Results in CFD-Post* section of the tutorial.

If this is the final section of your tutorial, continue from Closing the Applications (p. 36). If you are running Ansys CFX tutorials with a sequence of multiple simulations, continue from Creating CFX Component Systems for Multiple Simulations (p. 36).

1.2.5. Creating CFX Component Systems for Multiple Simulations

Now that you have set the physics in the initial state, you will duplicate the **CFX** component system created earlier and edit the physics in the new system. To duplicate the existing **CFX** component system:

1. In the Ansys Workbench **Project Schematic**, right-click the first cell in `System 1` and select **Duplicate**.

   A new system named `Copy of System 1` will appear in the **Project Schematic**.
2. Type in the new name `System 2` to replace the highlighted text below the system.
3. Click the **Solution** cell of `System 1` and drag it to the **Solution** cell of `System 2`.

   You will now see a line, indicating a transfer connection, going from **Solution** cell of `System 1` to the **Solution** cell of `System 2`.
4. Once you have set up the new **CFX** component system, continue from Step 5 of Setting Up the Project (p. 34).

> **Note:**
>
> In the tutorial, ignore the steps that tell you to set the initial values file in the **Define Run** dialog box for CFX-Solver Manager. Dragging the solution cell between systems automatically sets the initialization options in CFX-Solver Manager.

1.2.6. Closing the Applications

Close Ansys Workbench (and the applications it launched) by selecting **File** > **Exit** from Ansys Workbench. Ansys Workbench prompts you to save all your project files.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.3. Changing the Display Colors

If viewing objects in Ansys CFX becomes difficult due to contrast with the background, you can change the colors for improved viewing. The color options are set in different places, depending on how you run CFX:

1. Select **Edit** > **Options**.

   The **Options** dialog box appears.

2. Adjust the color settings under `CFX-Pre` > `Graphics Style` (for CFX-Pre) or `CFD-Post` > `Viewer` (for CFD-Post).

3. Click **OK**.

4. If you were directed here at some point during a tutorial, return to that location.

1.4. Editor Buttons

The Ansys CFX interface uses editors to enter the data required to set up and post-process a simulation. The editors have standard buttons, which are described next:

- **Apply** applies the information contained within all the tabs of an editor.
- **OK** is the same as **Apply**, except that the editor automatically closes.
- **Cancel** and **Close** both close the editor without applying or saving any changes.
- **Reset** returns the settings for the object to those stored in the database for all the tabs. The settings are stored in the database each time the **Apply** button is clicked.
- **Defaults** restores the system default settings for all the tabs of the edited object.

1.5. Using Help

To open the Ansys Help, select **Help** > **Contents**.

Context-sensitive help is provided for many of the details views and other parts of the interface. To invoke the context-sensitive help for a particular details view or other feature, ensure that the window is active, place the mouse pointer over the feature, and press **F1**. Not every area of the interface supports context-sensitive help.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 2: Simulating Flow in a Static Mixer Using CFX in Stand-alone Mode

This tutorial includes:

2.1. Tutorial Features

2.2. Overview of the Problem to Solve

2.3. Preparing the Working Directory

2.4. Defining the Case Using CFX-Pre

2.5. Obtaining the Solution Using CFX-Solver Manager

2.6. Viewing the Results Using CFD-Post

This tutorial simulates a static mixer consisting of two inlet pipes delivering water into a mixing vessel; the water exits through an outlet pipe. A general workflow is established for analyzing the flow of fluid into and out of a mixer.

2.1. Tutorial Features

In this tutorial you will learn about:

- Using Quick Setup mode in CFX-Pre to set up a problem.
- Using CFX-Solver Manager to obtain a solution.
- Modifying the outline plot in CFD-Post.
- Using streamlines in CFD-Post to trace the flow field from a point.
- Viewing temperature using colored planes and contours in CFD-Post.
- Creating an animation and saving it as a movie file.

Component	Feature	Details
CFX-Pre	User Mode	Quick Setup Wizard
	Analysis Type	Steady State
	Fluid Type	General Fluid
	Domain Type	Single Domain
	Turbulence Model	k-Epsilon
	Heat Transfer	Thermal Energy
	Boundary Conditions	Inlet (Subsonic)
		Outlet (Subsonic)

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Component	Feature	Details
		Wall: No-Slip
		Wall: Adiabatic
	Timestep	Physical Time Scale
CFD-Post	Animation	Keyframe
	Plots	Contour
		Outline Plot (Wireframe)
		Point
		Slice Plane
		Streamline

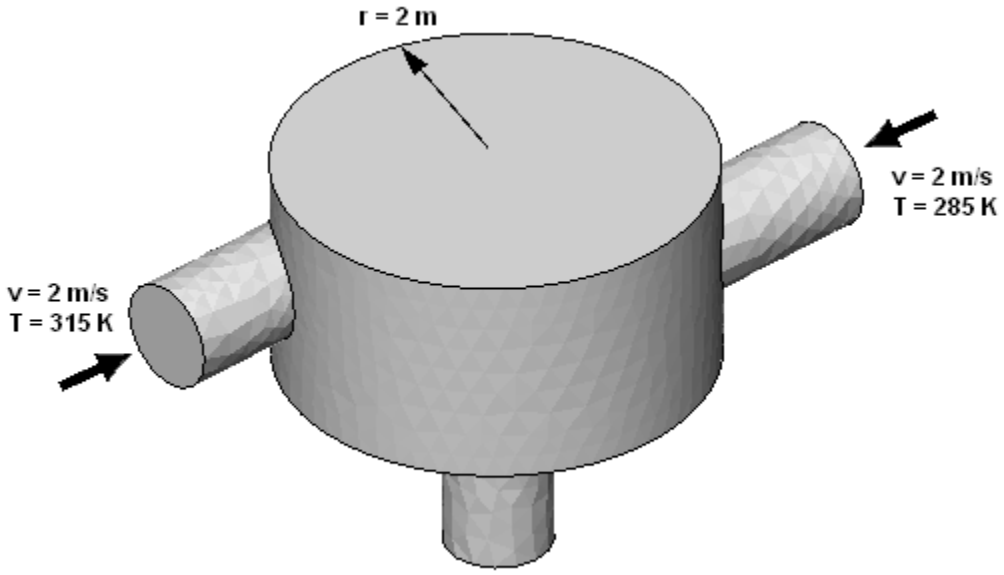
2.2. Overview of the Problem to Solve

This tutorial simulates a static mixer consisting of two inlet pipes delivering water into a mixing vessel; the water exits through an outlet pipe. A general workflow is established for analyzing the flow of fluid into and out of a mixer.

Water enters through both pipes at the same rate but at different temperatures. The first entry is at a rate of 2 m/s and a temperature of 315 K and the second entry is at a rate of 2 m/s at a temperature of 285 K. The radius of the mixer is 2 m.

Your goal in this tutorial is to understand how to use CFX to determine the speed and temperature of the water when it exits the static mixer.

Figure 2.1: Static Mixer with 2 Inlet Pipes and 1 Outlet Pipe



If this is the first tutorial you are working with, it is important to review the following topics before beginning:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Running Ansys CFX Tutorials Using Ansys Workbench (p. 34)
- Changing the Display Colors (p. 37)

2.3. Preparing the Working Directory

1. Create a working directory.

   Ansys CFX uses a working directory as the default location for loading and saving files for a particular session or project.

2. Download the `static_mixer_standalone.zip` file here.

3. Unzip `static_mixer_standalone.zip` to your working directory.

   Ensure that the following tutorial input file is in your working directory:

   - `StaticMixerMesh.gtm`

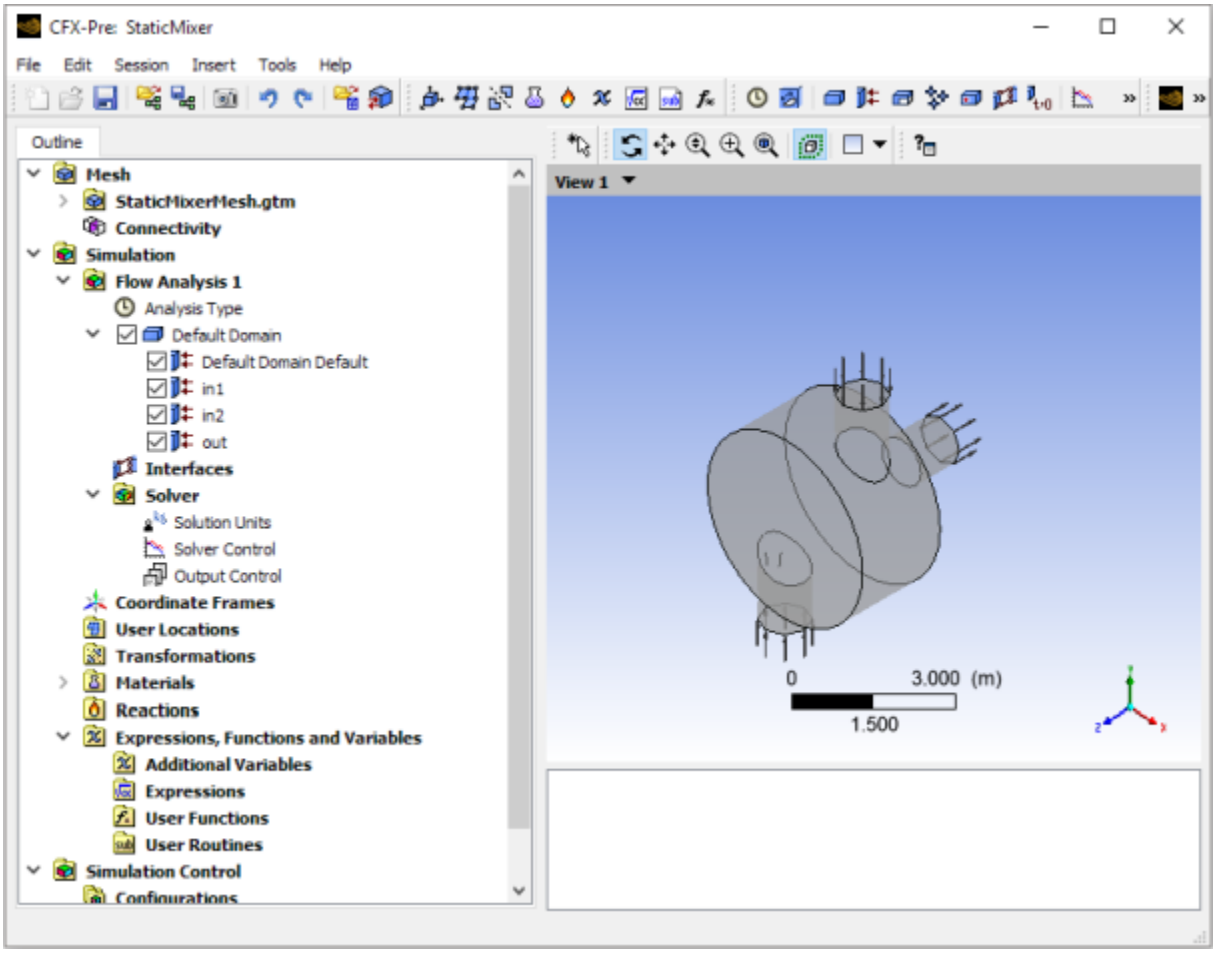
4. Set the working directory and start CFX-Pre.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

2.4. Defining the Case Using CFX-Pre

Because you are starting with an existing mesh, you can immediately use CFX-Pre to define the simulation. This is how CFX-Pre will look with the imported mesh:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The left pane of CFX-Pre displays the **Outline** workspace.

The tutorial follows this general workflow for Quick Setup mode in CFX-Pre:

2.4.1. Starting Quick Setup Mode
2.4.2. Setting the Physics Definition
2.4.3. Importing a Mesh
2.4.4. Using the Viewer
2.4.5. Defining Model Data
2.4.6. Defining Boundaries
2.4.7. Setting Boundary Data
2.4.8. Setting Flow Specification
2.4.9. Setting Temperature Specification
2.4.10. Reviewing the Boundary Condition Definitions
2.4.11. Creating the Second Inlet Boundary Definition
2.4.12. Creating the Outlet Boundary Definition
2.4.13. Moving to General Mode
2.4.14. Setting Solver Control
2.4.15. Writing the CFX-Solver Input (.def) File

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.4.1. Starting Quick Setup Mode

Quick Setup mode provides a simple wizard-like interface for setting up simple cases. This is useful for getting familiar with the basic elements of a CFD problem setup. This section describes using Quick Setup mode to develop a simulation in CFX-Pre.

1. In CFX-Pre, select **File** > **New Case**.

   The **New Case File** dialog box is displayed.

2. Select **Quick Setup** and click **OK**.

   Note:

   If this is the first time you are running this software, a message box will appear notifying you that automatic generation of the default domain is active. To avoid seeing this message again clear **Show This Message Again**.

3. Select **File** > **Save Case As**.

4. Under **File name**, type: `StaticMixer`

5. Click **Save**.

2.4.2. Setting the Physics Definition

You need to specify the fluids used in a simulation. A variety of fluids are already defined as library materials. For this tutorial you will use a prepared fluid, Water, which is defined to be water at 25°C.

1. Ensure that the **Simulation Definition** panel is displayed at the top of the details view.

2. Under **Working Fluid** > **Fluid** select `Water`.

2.4.3. Importing a Mesh

At least one mesh must be imported before physics are applied.

1. In the **Simulation Definition** panel, under **Mesh Data** > **Mesh File**, click *Browse* .

   The **Import Mesh** dialog box appears.

2. Under **Files of type**, select `CFX Mesh (*gtm *cfx)`.

3. From your working directory, select `StaticMixerMesh.gtm`.

4. Click **Open**.

   The mesh loads.

5. Click **Next**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.4.4. Using the Viewer

Now that the mesh is loaded, take a moment to explore how you can use the viewer toolbar to zoom in or out and to rotate the object in the viewer.

2.4.4.1. Using the Zoom Tools

There are several icons available for controlling the level of zoom in the viewer.

1. Click *Zoom Box*
2. Click and drag a rectangular box over the geometry.
3. Release the mouse button to zoom in on the selection.

   The geometry zoom changes to display the selection at a greater resolution.
4. Click *Fit View* to re-center and re-scale the geometry.

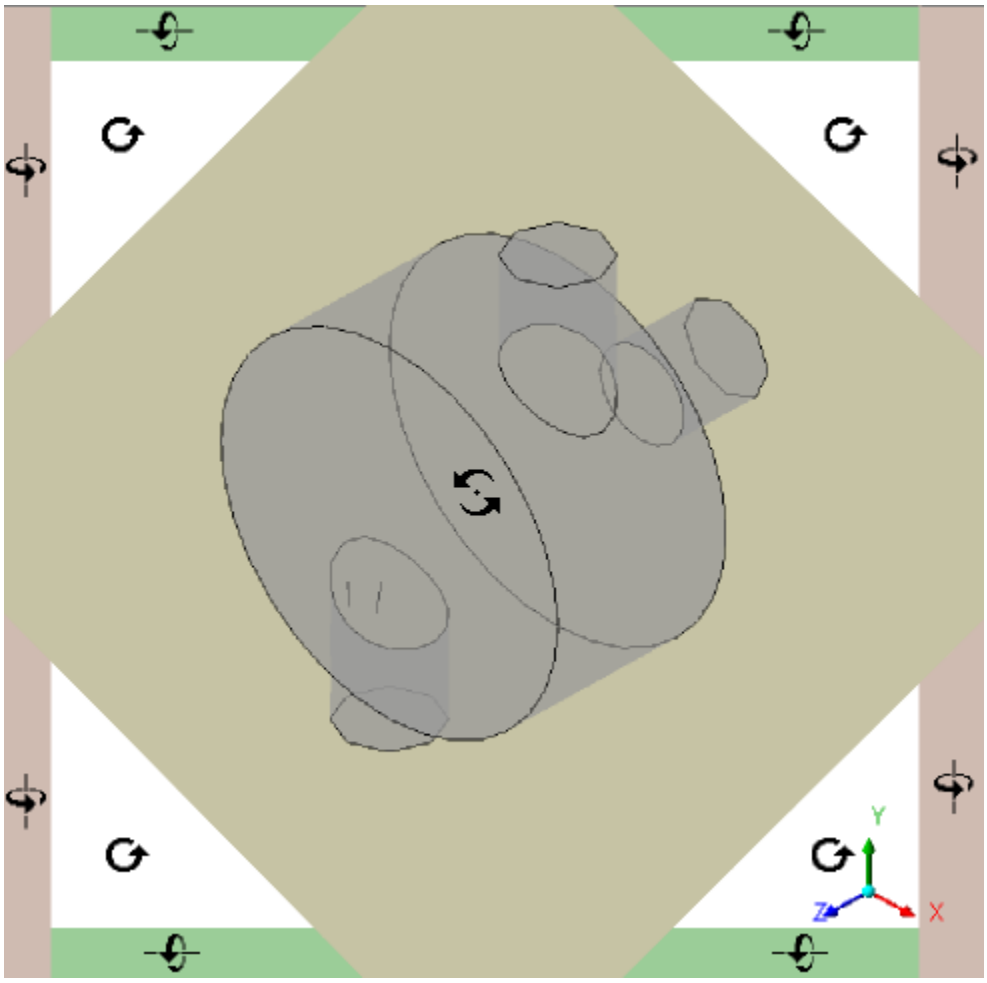
2.4.4.2. Rotating the Geometry

If you need to rotate an object or to view it from a new angle, you can use the viewer toolbar.

1. Click *Rotate* on the viewer toolbar.
2. Click and drag within the geometry repeatedly to test the rotation of the geometry.

   The geometry rotates based on the direction of mouse movement and based on the initial mouse cursor shape, which changes depending on where the mouse cursor is in the viewer. If the mouse drag starts near a corner of the viewer window, the motion of the geometry will be constrained to rotation about a single axis, as indicated by the mouse cursor shape.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Right-click a blank area in the viewer and select **Predefined Camera** > **View From -X**.
4. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z Up)**.

   A clearer view of the mesh is displayed.

2.4.5. Defining Model Data

You need to define the type of flow and the physical models to use in the fluid domain.

You will specify the flow as steady-state with turbulence and heat transfer. Turbulence is modeled using the k-ε turbulence model and heat transfer using the thermal energy model. The k-ε turbulence model is a commonly used model and is suitable for a wide range of applications. The thermal energy model neglects high speed energy effects and is therefore suitable for low speed flow applications.

1. Ensure that the **Physics Definition** panel is displayed.
2. Under **Model Data**, set **Reference Pressure** to `1 [atm]`.

   All other pressure settings are relative to this reference pressure.
3. Set **Heat Transfer** to `Thermal Energy`.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Set **Turbulence** to `k-Epsilon`.
5. Click **Next**.

2.4.6. Defining Boundaries

The CFD model requires the definition of conditions on the boundaries of the domain.

1. Ensure that the **Boundary Definition** panel is displayed.
2. Delete `Inlet` and `Outlet` from the list by right-clicking each and selecting **Delete Boundary**.
3. Right-click in the blank area where `Inlet` and `Outlet` were listed, then select **Add Boundary**.
4. Set **Name** to `in1`.
5. Click **OK**.

   The boundary is created and, when selected, properties related to the boundary are displayed.

2.4.7. Setting Boundary Data

Once boundaries are created, you need to create associated data. Based on Figure 2.1: Static Mixer with 2 Inlet Pipes and 1 Outlet Pipe (p. 40), you will define the velocity and temperature for the first inlet.

1. Ensure that **in1** is displayed on the **Boundary Definition** panel.
2. Set **Boundary Type** to `Inlet`.
3. Set **Location** to `in1`.

2.4.8. Setting Flow Specification

Once boundary data is defined, the boundary needs to have the flow specification assigned.

1. Ensure that **Flow Specification** is displayed on the **Boundary Definition** panel.
2. Set **Option** to `Normal Speed`.
3. Set **Normal Speed** to `2 [m s^-1]`.

2.4.9. Setting Temperature Specification

Once flow specification is defined, the boundary needs to have temperature assigned.

1. Ensure that **Temperature Specification** is displayed on the **Boundary Definition** panel.
2. Set **Static Temperature** to `315 [K]`.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.4.10. Reviewing the Boundary Condition Definitions

Defining the boundary condition for `in1` required several steps. Here the settings are reviewed for accuracy.

Based on Figure 2.1: Static Mixer with 2 Inlet Pipes and 1 Outlet Pipe (p. 40), the first inlet boundary condition consists of a velocity of 2 m/s and a temperature of 315 K at one of the side inlets.

- Review the boundary `in1` settings on the **Boundary Definition** panel for accuracy. They should be as follows:

Setting	Value
in1 > **Boundary Type**	 Inlet
in1 > **Location**	 in1
Flow Specification > **Option**	 Normal Speed
Flow Specification > **Normal Speed**	 2 [m s^-1]
Temperature Specification > **Static Temperature**	 315 [K]

2.4.11. Creating the Second Inlet Boundary Definition

Based on Figure 2.1: Static Mixer with 2 Inlet Pipes and 1 Outlet Pipe (p. 40), you know the second inlet boundary condition consists of a velocity of 2 m/s and a temperature of 285 K at one of the side inlets. You will define that now.

1. Under the **Boundary Definition** panel, right-click in the selector area and select **Add Boundary**.
2. Create a new boundary named `in2` with these settings:

Setting	Value
in2 > **Boundary Type**	 Inlet
in2 > **Location**	 in2
Flow Specification > **Option**	 Normal Speed
Flow Specification	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
> **Normal Speed**	2 [m s^-1]
Temperature Specification > **Static Temperature**	 285 [K]

2.4.12. Creating the Outlet Boundary Definition

Now that the second inlet boundary has been created, the same concepts can be applied to building the outlet boundary.

1. Create a new boundary named `out` with these settings:

Setting	Value
out > **Boundary Type**	 Outlet
out > **Location**	 out
Flow Specification > **Option**	 Average Static Pressure
Flow Specification > **Relative Pressure**	 0 [Pa]

2. Click **Next**.

2.4.13. Moving to General Mode

There are no further boundary conditions that need to be set. All 2D exterior regions that have not been assigned to a boundary condition are automatically assigned to the default boundary condition.

- Set **Operation** to `Enter General Mode` and click **Finish**.

  The three boundary conditions are displayed in the viewer as sets of arrows at the boundary surfaces. Inlet boundary arrows are directed into the domain. Outlet boundary arrows are directed out of the domain.

2.4.14. Setting Solver Control

Solver Control parameters control aspects of the numerical solution generation process.

While an upwind advection scheme is less accurate than other advection schemes, it is also more robust. This advection scheme is suitable for obtaining an initial set of results, but in general should not be used to obtain final accurate results.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The time scale can be calculated automatically by the solver or set manually. The `Automatic` option tends to be conservative, leading to reliable, but often slow, convergence. It is often possible to accelerate convergence by applying a time scale factor or by choosing a manual value that is more aggressive than the `Automatic` option. In this tutorial, you will select a physical time scale, leading to convergence that is twice as fast as the `Automatic` option.

1. Click *Solver Control* .
2. On the **Basic Settings** tab, set **Advection Scheme** > **Option** to `Upwind`.
3. Set **Convergence Control** > **Fluid Timescale Control** > **Timescale Control** to `Physical Timescale` and set the physical timescale value to `2 [s]`.
4. Click **OK**.

2.4.15. Writing the CFX-Solver Input (.def) File

The simulation file, `StaticMixer.cfx`, contains the simulation definition in a format that can be loaded by CFX-Pre, enabling you to complete (if applicable), restore, and modify the simulation definition. The simulation file differs from the CFX-Solver input file in that it can be saved at any time while defining the simulation.

1. Click *Define Run* .
2. Set **File name** to `StaticMixer.def`.
3. Click **Save**.

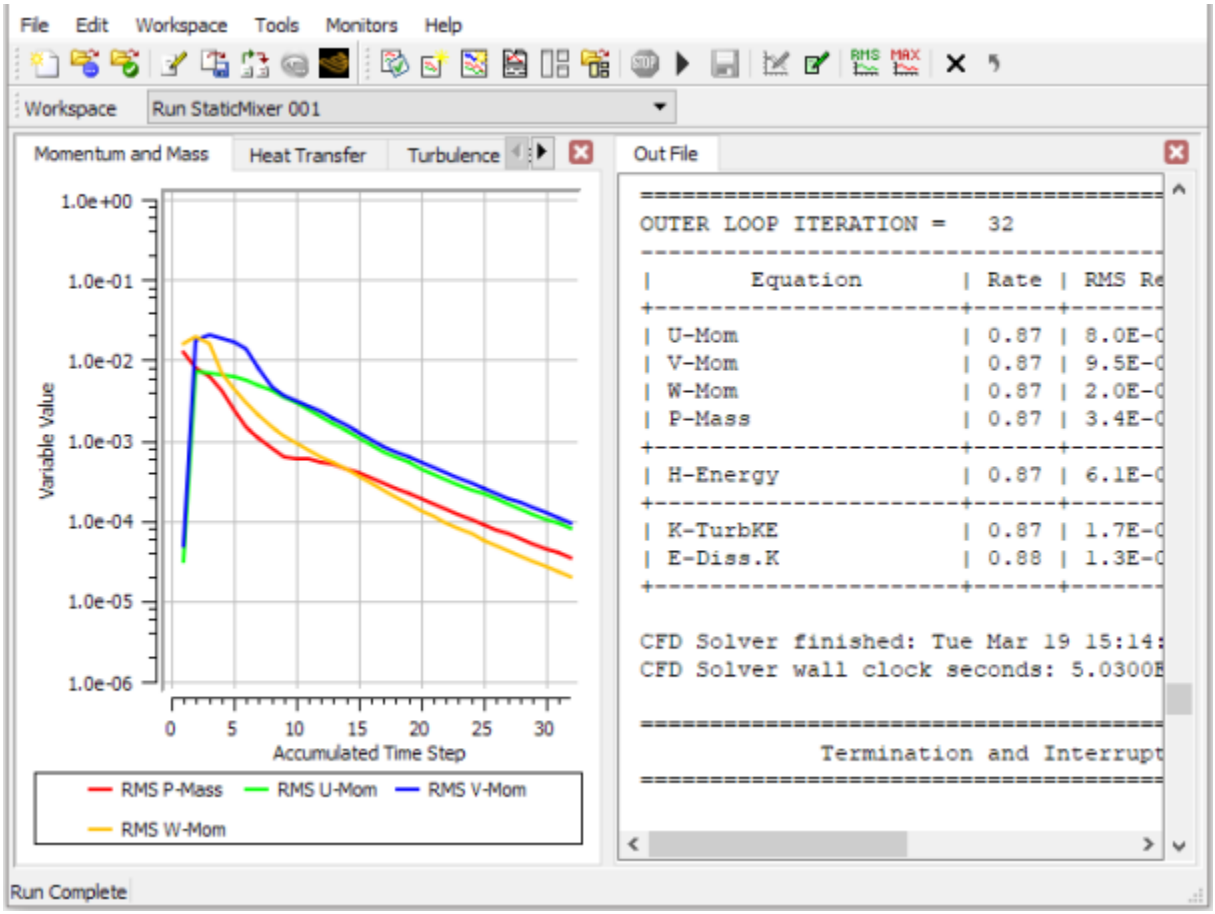
   The CFX-Solver input file (`StaticMixer.def`) is created. CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, **Solver Input File** is set.
4. If you are notified the file already exists, click **Overwrite**.
5. When you are finished, select **File** > **Quit** in CFX-Pre.
6. If prompted, click **Yes** or **Save & Quit** to save `StaticMixer.cfx`.

2.5. Obtaining the Solution Using CFX-Solver Manager

CFX-Solver Manager has a visual interface that displays a variety of results and should be used when plotted data must be viewed during problem solving.

Two windows are displayed when CFX-Solver Manager runs. There is an adjustable split between the windows, which is oriented either horizontally or vertically depending on the aspect ratio of the entire CFX-Solver Manager window (also adjustable).

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



One window shows the convergence history plots and the other displays text output from CFX-Solver.

The text lists physical properties, boundary conditions and various other parameters used or calculated in creating the model. All the text is written to the CFX-Solver Output file automatically (in this case, `StaticMixer_001.out`).

2.5.1. Starting the Run

The **Define Run** dialog box allows configuration of a run for processing by CFX-Solver.

When CFX-Solver Manager is launched automatically from CFX-Pre, all of the information required to perform a new serial run (on a single processor) is entered automatically. You do not need to alter the information in the **Define Run** dialog box. This is a very quick way to launch into CFX-Solver without having to define settings and values.

1. Ensure that the **Define Run** dialog box is displayed.
2. Optionally specify a local parallel run:
   a. Set **Run Mode** to a parallel mode suitable for your configuration; for example, `Intel MPI Local Parallel`.

      This is the recommended method for most applications.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. If required, click *Add Partition* to add more partitions.

Ideally, the number of partitions should not exceed the number of available processor cores.

More detailed information about setting up CFX to run in parallel is available in Flow Around a Blunt Body (p. 187).

3. Click **Start Run**.

CFX-Solver launches and a split screen appears and displays the results of the run graphically and as text. The panes continue to build as CFX-Solver Manager operates.

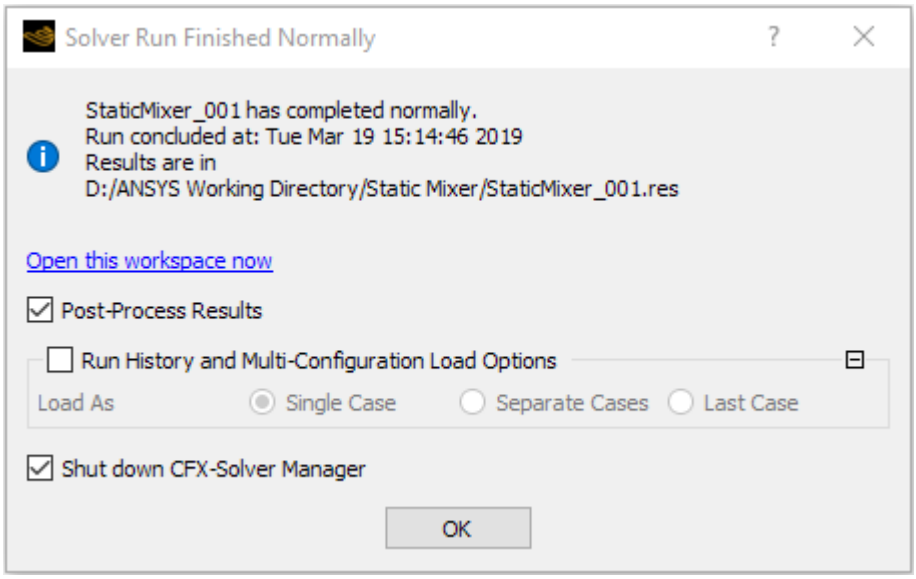
Note:

Once the second iteration appears, data begins to plot. Plotting may take a long time depending on the amount of data to process. Let the process run.

2.5.2. Moving from CFX-Solver Manager to CFD-Post

Once CFX-Solver has finished, you can use CFD-Post to review the finished results.

1. When CFX-Solver is finished, select the check box next to **Post-Process Results**.



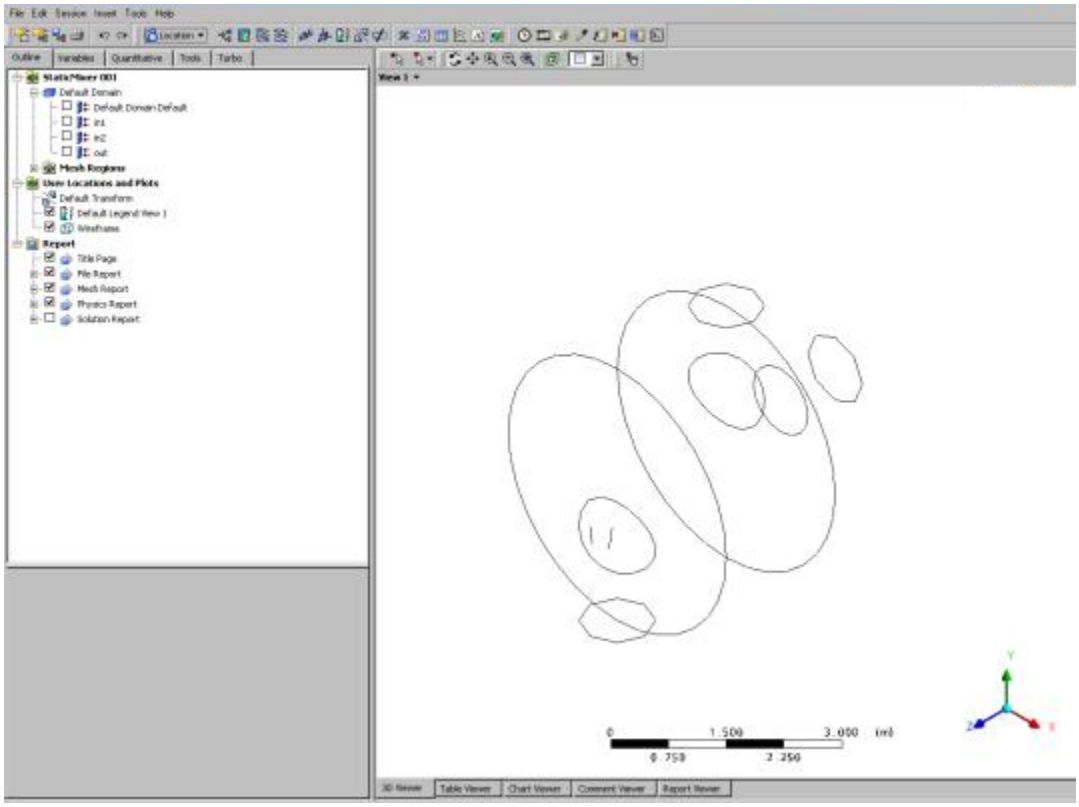
2. If using stand-alone mode, select the check box next to **Shut down CFX-Solver Manager**.

3. Click **OK**. After a short pause, CFX-Solver Manager closes and CFD-Post opens.

2.6. Viewing the Results Using CFD-Post

When CFD-Post starts, the viewer and **Outline** workspace are displayed.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The viewer displays an outline of the geometry and other graphic objects. You can use the mouse or the toolbar icons to manipulate the view, exactly as in CFX-Pre.

The tutorial follows this general workflow for viewing results in CFD-Post:

2.6.1. Setting the Edge Angle for a Wireframe Object

2.6.2. Creating a Point for the Origin of the Streamline

2.6.3. Creating a Streamline Originating from a Point

2.6.4. Rearranging the Point

2.6.5. Configuring a Default Legend

2.6.6. Creating a Slice Plane

2.6.7. Defining Slice Plane Geometry

2.6.8. Configuring Slice Plane Views

2.6.9. Rendering Slice Planes

2.6.10. Coloring the Slice Plane

2.6.11. Moving the Slice Plane

2.6.12. Adding Contours

2.6.13. Working with Animations

2.6.14. Quitting CFD-Post

2.6.1. Setting the Edge Angle for a Wireframe Object

The outline of the geometry is called the *wireframe* or *outline plot*.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

By default, CFD-Post displays only some of the surface mesh. This sometimes means that when you first load your results file, the geometry outline is not displayed clearly. You can control the amount of the surface mesh shown by editing the `Wireframe` object listed in the **Outline** tree view.

The check boxes next to each object name in the **Outline** control the visibility of each object. Currently only the `Wireframe` and `Default Legend` objects have visibility turned on.

The edge angle determines how much of the surface mesh is visible. If the angle between two adjacent faces is greater than the edge angle, then that edge is drawn. If the edge angle is set to 0°, the entire surface mesh is drawn. If the edge angle is large, then only the most significant corner edges of the geometry are drawn.

For this geometry, a setting of approximately 15° lets you view the model location without displaying an excessive amount of the surface mesh.

In this module you can also modify the zoom settings and view of the wireframe.

1. In the **Outline**, under `User Locations and Plots`, double-click `Wireframe`.

   Tip:

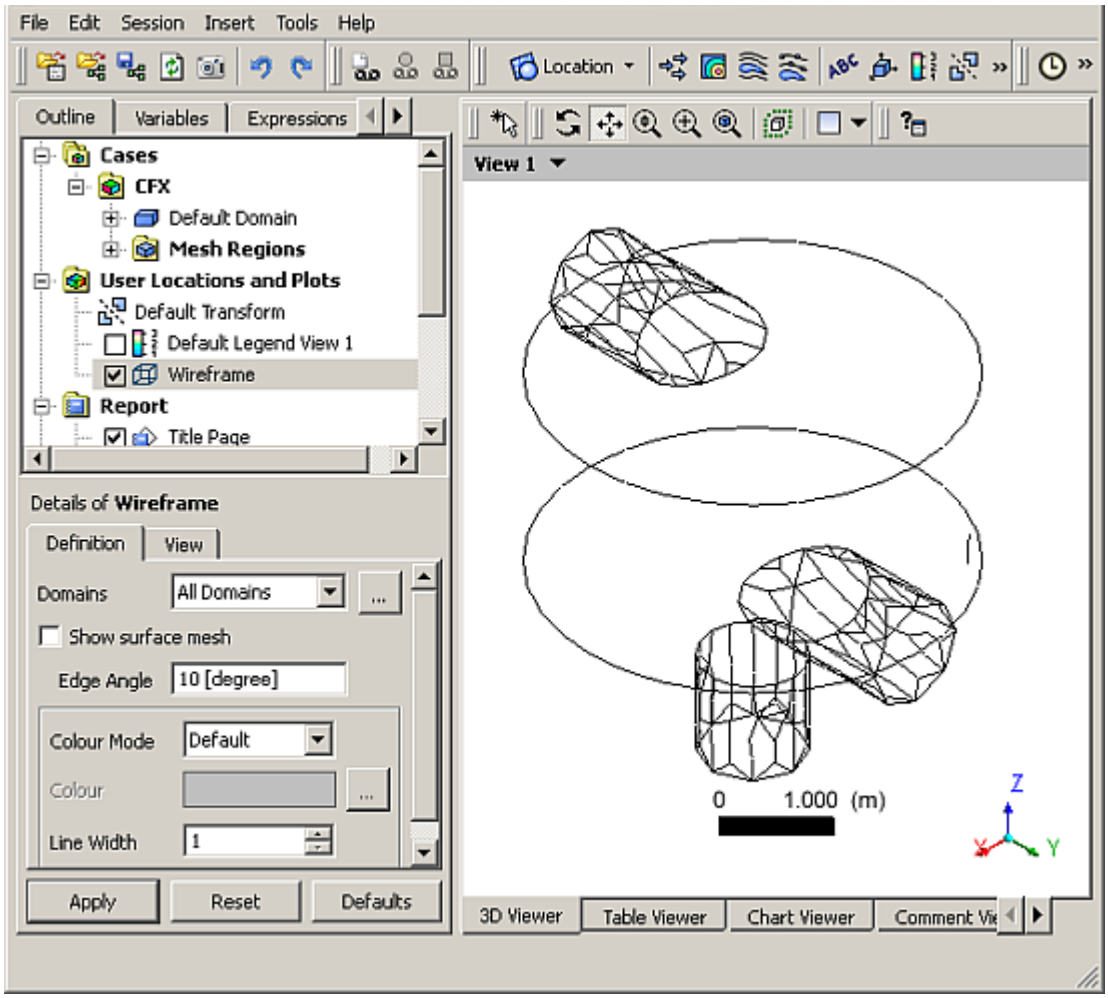
   While it is not necessary to change the view to set the angle, do so to explore the practical uses of this feature.

2. Right-click a blank area anywhere in the viewer, select **Predefined Camera** from the shortcut menu, and select **Isometric View (Z up)**.
3. In the `Wireframe` details view, under **Definition**, click in the **Edge Angle** box.

   An embedded slider is displayed.
4. Type a value of `10 [degree]`.
5. Click **Apply** to update the object with the new setting.

   Notice that more surface mesh is displayed.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



6. Drag the embedded slider to set the **Edge Angle** value to approximately `45 [degree]`.
7. Click **Apply** to update the object with the new setting.

   Less of the outline of the geometry is displayed.
8. Type a value of `15 [degree]`.
9. Click **Apply** to update the object with the new setting.

2.6.2. Creating a Point for the Origin of the Streamline

A *streamline* is the path that a particle of zero mass would follow through the domain.

1. Select **Insert** > **Location** > **Point** from the main menu.

   You can also use the toolbars to create a variety of objects. Later modules and tutorials explore this further.
2. Click **OK**.

   This accepts the default name.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Under **Definition**, ensure that **Method** is set to XYZ.
4. Under **Point**, enter the following coordinates: `-1, -1, 1`.

   This is a point near the first inlet.
5. Click **Apply**.

   The point appears as a symbol in the viewer as a crosshair symbol.

2.6.3. Creating a Streamline Originating from a Point

Where applicable, streamlines can trace the flow direction forwards (downstream) and/or backwards (upstream).

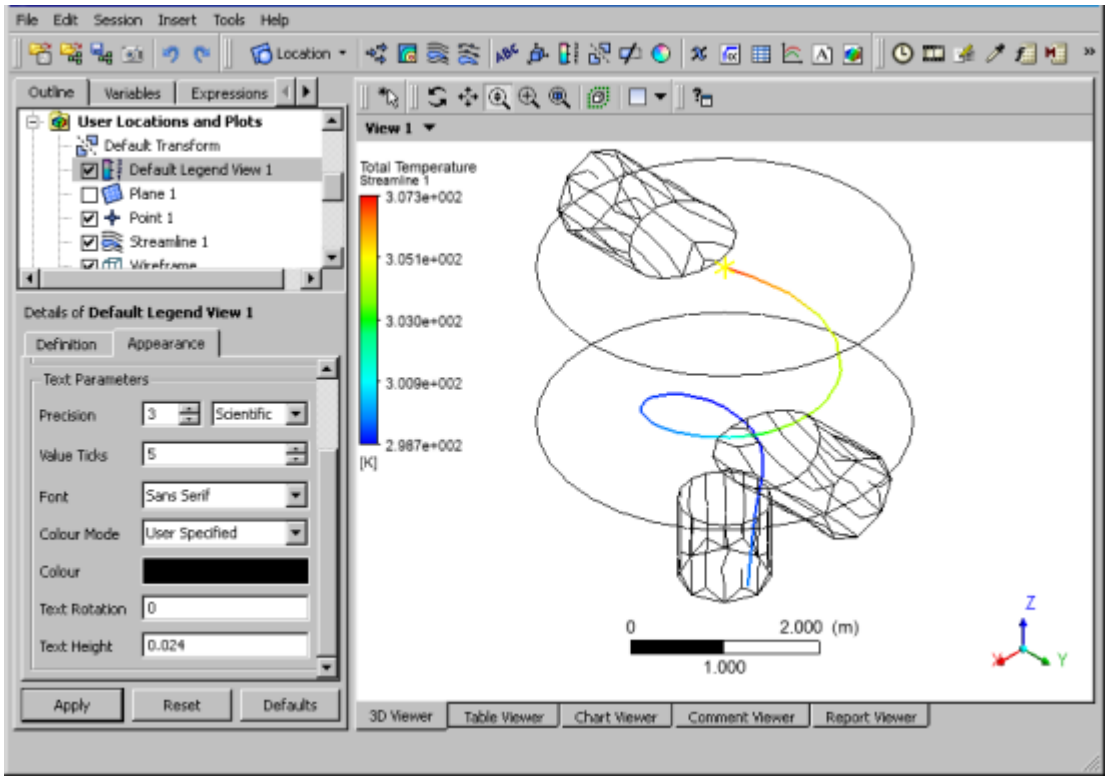
1. From the main menu, select **Insert** > **Streamline**.
2. Click **OK**.
3. Set **Definition** > **Start From** to `Point 1`.

   Tip:

   To create streamlines originating from more than one location, click the *Ellipsis* icon to the right of the **Start From** box. This displays the **Location Selector** dialog box, where you can use the **Ctrl** and **Shift** keys to pick multiple locators.
4. Click the **Color** tab.
5. Set **Mode** to `Variable`.
6. Set **Variable** to `Total Temperature`.
7. Set **Range** to `Local`.
8. Click **Apply**.

   The streamline shows the path of a zero mass particle from `Point 1`. The temperature is initially high near the hot inlet, but as the fluid mixes the temperature drops.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2.6.4. Rearranging the Point

Once created, a point can be rearranged manually or by setting specific coordinates.

Tip:

In this module, you may choose to display various views and zooms from the **Predefined Camera** option in the shortcut menu (such as **Isometric View (Z up)** or **View From -X**) and by using *Zoom Box* if you prefer to change the display.

---

1. In **Outline**, under `User Locations and Plots` double-click `Point 1`.

   Properties for the selected user location are displayed.

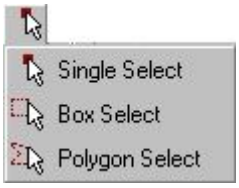
2. Under **Point**, set these coordinates: `-1, -2.9, 1`.

3. Click **Apply**.

   The point is moved and the streamline redrawn.

4. In the viewer toolbar, click *Select* and ensure that the adjacent toolbar icon is set to **Single Select** .

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



While in select mode, you cannot use the left mouse button to re-orient the object in the viewer.

5. In the viewer, drag `Point 1` (appears as a yellow addition sign) to a new location within the mixer.

   The point position is updated in the details view and the streamline is redrawn at the new location. The point moves normal in relation to the viewing direction.

6. Click *Rotate* .

   Tip:

   You can also click in the viewer area, and press the space bar to toggle between Select and Viewing Mode. A way to pick objects from Viewing Mode is to hold down **Ctrl** + **Shift** while clicking on an object with the left mouse button.

7. Under **Point**, reset these coordinates: `-1, -1, 1`.

8. Click **Apply**.

   The point appears at its original location.

9. Right-click a blank area in the viewer and select **Predefined Camera** > **View From -X**.

2.6.5. Configuring a Default Legend

You can modify the appearance of the default legend.

The default legend appears whenever a plot is created that is colored by a variable. The streamline color is based on temperature; therefore, the legend shows the temperature range. The color pattern on the legend's color bar is banded in accordance with the bands in the plot.

Note:

If a user-specified range is used for the legend, one or more bands may represent values beyond the legend's range. In this case, these band colors are extrapolated slightly past the range of colors shown in the legend.

The default legend displays values for the last eligible plot that was opened in the details view. To maintain a legend definition during a CFD-Post session, you can create a new legend by clicking *Legend* .

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Because there are many settings that can be customized for the legend, this module allows you the freedom to experiment with them. In the last steps you will set up a legend, based on the default legend, with a minor modification to the position.

Tip:

When editing values, you can restore the values that were present when you began editing by clicking **Reset**. To restore the factory-default values, click **Default**.

1. Double-click `Default Legend View 1`.

   The **Definition** tab of the default legend is displayed.

2. Configure the following setting(s):

Tab	Setting	Value
Definition	**Title Mode**	User Specified
	Title	Streamline Temp.
	Horizontal	(Selected)
	Location > **Y Justification**	 Bottom

3. Click **Apply**.

   The appearance and position of the legend changes based on the settings specified.

4. Modify various settings in **Definition** and click **Apply** after each change.

5. Select the **Appearance** tab.

6. Modify a variety of settings in the **Appearance** and click **Apply** after each change.

7. Click **Defaults**.

8. Click **Apply**.

9. Under **Outline**, in `User Locations and Plots`, clear the check boxes for `Point 1` and `Streamline 1`.

   Since both are no longer visible, the associated legend no longer appears.

2.6.6. Creating a Slice Plane

Defining a slice plane allows you to obtain a cross-section of the geometry.

In CFD-Post you often view results by coloring a graphic object. The graphic object could be an isosurface, a vector plot, or in this case, a plane. The object can be a fixed color or it can vary based on the value of a variable.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You already have some objects defined by default (listed in the **Outline**). You can view results on the boundaries of the static mixer by coloring each boundary object by a variable. To view results within the geometry (that is, on non-default locators), you will create new objects.

You can use the following methods to define a plane:

- `Three Points`: creates a plane from three specified points.
- `Point and Normal`: defines a plane from one point on the plane and a normal vector to the plane.
- `YZ Plane`, `ZX Plane`, and `XY Plane`: similar to `Point and Normal`, except that the normal is defined to be normal to the indicated plane.

1. From the main menu, select **Insert** > **Location** > **Plane** or click **Location** > **Plane**.
2. In the **Insert Plane** window, type: `Slice`
3. Click **OK**.

   The **Geometry**, **Color**, **Render**, and **View** tabs enable you to switch between settings.
4. Click the **Geometry** tab.

2.6.7. Defining Slice Plane Geometry

You need to choose the vector normal to the plane. You want the plane to lie in the x-y plane, hence its normal vector points along the z-axis. You can specify any vector that points in the Z direction, but you will choose the most obvious (0,0,1).

1. If required, under **Geometry**, expand **Definition**.
2. Under **Method** select `Point and Normal`.
3. Under **Point** enter `0,0,1`.
4. Under **Normal** enter `0,0,1`.
5. Ensure that the **Plane Type** > **Slice** is selected.
6. Click **Apply**.

   `Slice` appears under **User Locations and Plots**. Rotate the view to see the plane.

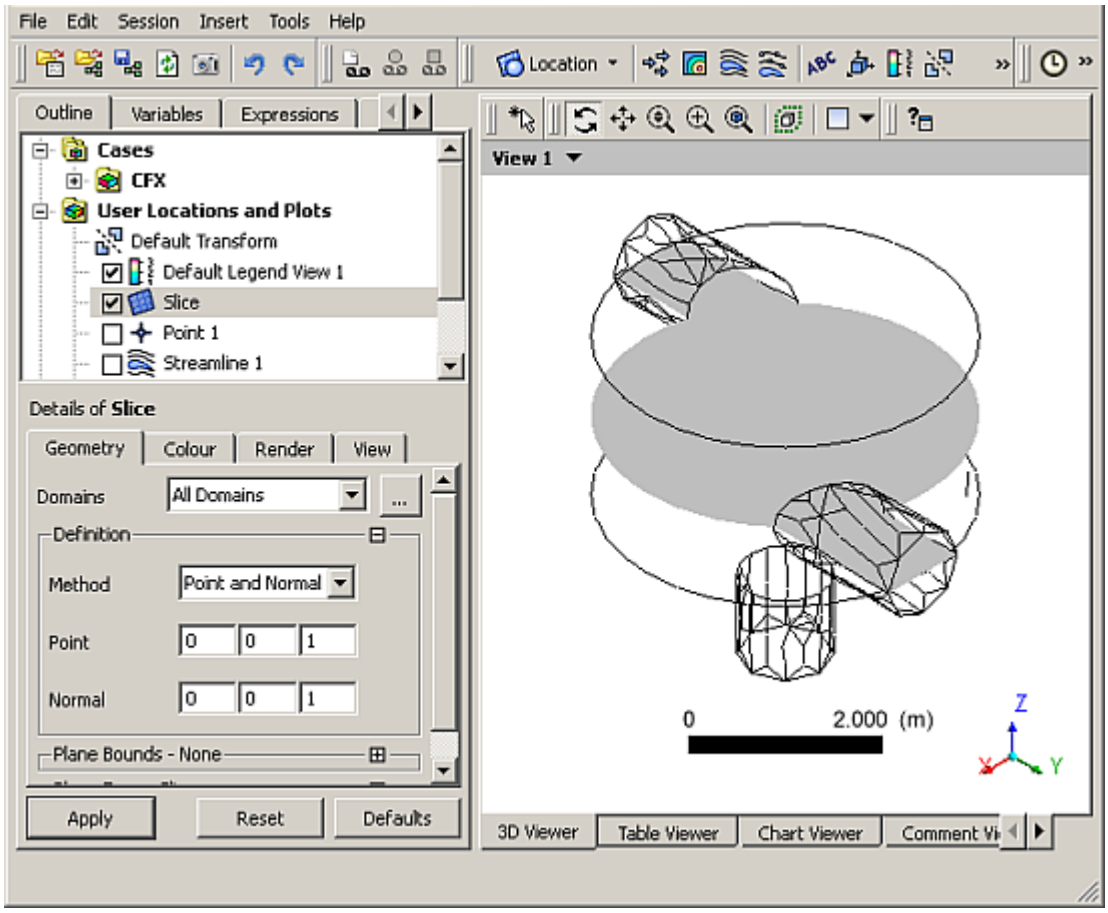
2.6.8. Configuring Slice Plane Views

Depending on the view of the geometry, various objects may not appear because they fall in a 2D space that cannot be seen.

1. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)**.

   The slice is now visible in the viewer.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Click *Zoom Box* .
3. Click and drag a rectangular selection over the geometry.
4. Release the mouse button to zoom in on the selection.
5. Click *Rotate* .
6. Click and drag the mouse pointer down slightly to rotate the geometry towards you.
7. Select **Isometric View (Z up)** as described earlier.

2.6.9. Rendering Slice Planes

Render settings determine how the plane is drawn.

1. In the details view for **Slice**, select the **Render** tab.
2. Clear **Show Faces**.
3. Select **Show Mesh Lines**.
4. Under **Show Mesh Lines** change **Color Mode** to `User Specified`.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Click the current color in **Line Color** to change to a different color.

   For a greater selection of colors, click the *Ellipsis* icon to use the **Select color** dialog box.

6. Click **Apply**.

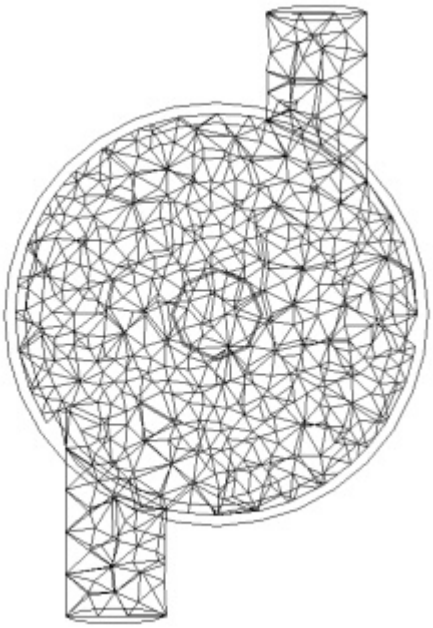
7. Click *Zoom Box* .

8. Zoom in on the geometry to view it in greater detail.

   The line segments show where the slice plane intersects with mesh element faces. The end points of each line segment are located where the plane intersects mesh element edges.

9. Right-click a blank area in the viewer and select **Predefined Camera** > **View From +Z**.

   The image shown below can be used for comparison with Flow in a Static Mixer (Refined Mesh) (p. 103) (in the section Creating a Slice Plane (p. 112)), where a refined mesh is used.



2.6.10. Coloring the Slice Plane

The **Color** panel is used to determine how the object faces are colored.

1. Configure the following setting(s) of `Slice`:

Tab	Setting	Value
Color	**Mode**	Variable [a]
	Variable	Temperature
Render	**Show Faces**	(Selected)

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Show Mesh Lines	(Cleared)

a. You can specify the variable (in this case, temperature) used to color the graphic element. The `Constant` mode allows you to color the plane with a fixed color.

2. Click **Apply**.

   Hot water (red) enters from one inlet and cold water (blue) from the other.

2.6.11. Moving the Slice Plane

The plane can be moved to different locations.

1. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)** from the shortcut menu.
2. Click the **Geometry** tab.

   Review the settings in **Definition** under **Point** and under **Normal**.

3. Click *Single Select* .
4. Click and drag the plane to a new location that intersects the domain.

   As you drag the mouse, the viewer updates automatically. Note that **Point** updates with new settings.

5. Set **Point** settings to `0,0,1`.
6. Click **Apply**.
7. Click *Rotate* .
8. Turn off visibility of `Slice` by clearing the check box next to `Slice` in the **Outline** tree view.

2.6.12. Adding Contours

Contours connect all points of equal value for a scalar variable (for example, `Temperature`) and help to visualize variable values and gradients. Colored bands fill the spaces between contour lines. Each band is colored by the average color of its two bounding contour lines (even if the latter are not displayed).

1. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)** from the shortcut menu.
2. Select **Insert** > **Contour** from the main menu or click *Contour* .

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Insert Contour** dialog box is displayed.

3. Set **Name** to `Slice Contour`.
4. Click **OK**.
5. Configure the following setting(s):

Tab	Setting	Value
Geometry	**Locations**	Slice
	Variable	Temperature
Render	**Show Contour Bands**	(Selected)

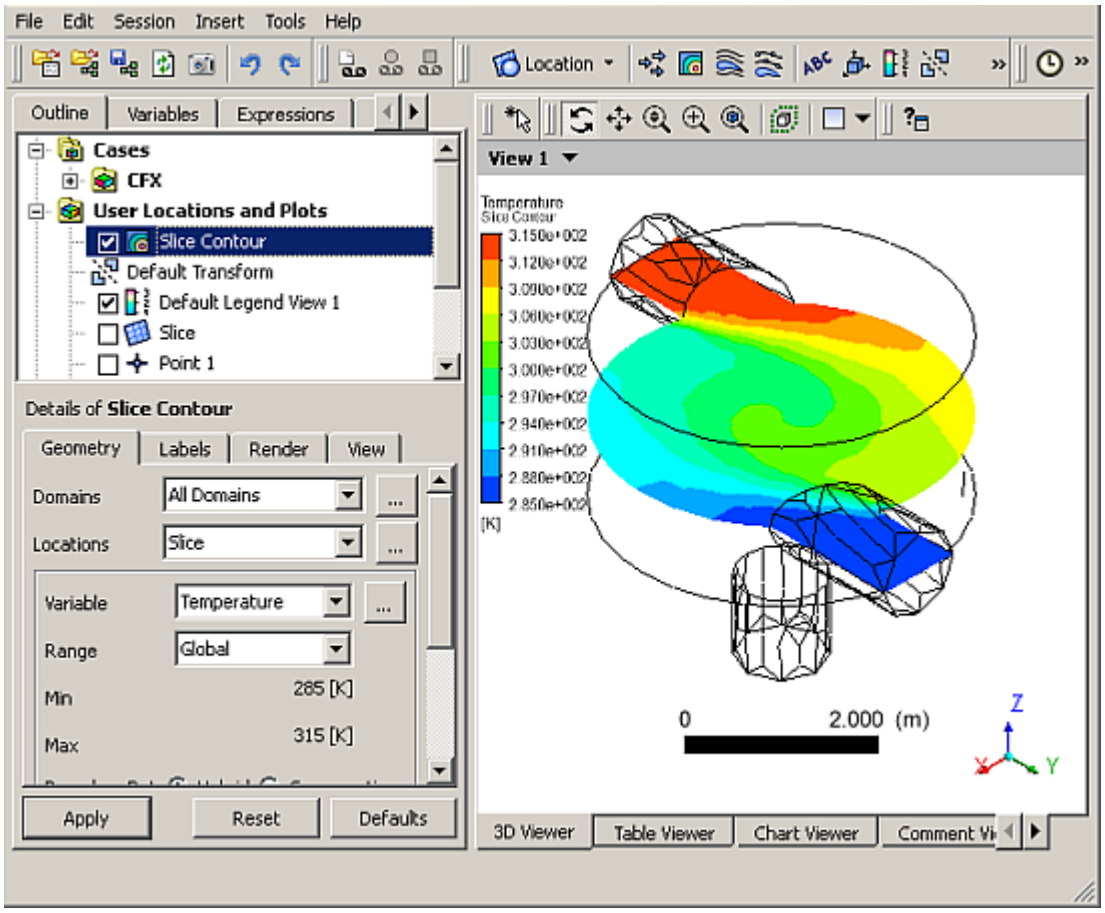
6. Click **Apply**.

Important:

The colors of 3D graphics object faces are slightly altered when lighting is on. To view colors with highest accuracy, go to the **Render** tab and, under **Show Contour Bands**, clear **Lighting** and click **Apply**.

The graphic element faces are visible, producing a contour plot as shown.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

Make sure that the visibility for `Slice` (in the **Outline** tree view) is turned off.

2.6.13. Working with Animations

Animations build transitions between views for development of video files.

The tutorial follows this general workflow for creating a keyframe animation:

2.6.13.1. Showing the Animation Dialog Box
2.6.13.2. Creating the First Keyframe
2.6.13.3. Creating the Second Keyframe
2.6.13.4. Viewing the Animation
2.6.13.5. Modifying the Animation
2.6.13.6. Saving a Movie

2.6.13.1. Showing the Animation Dialog Box

The **Animation** dialog box is used to define keyframes and to export to a video file.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Select **Tools** > **Animation** or click *Animation* .

  The **Animation** dialog box appears.

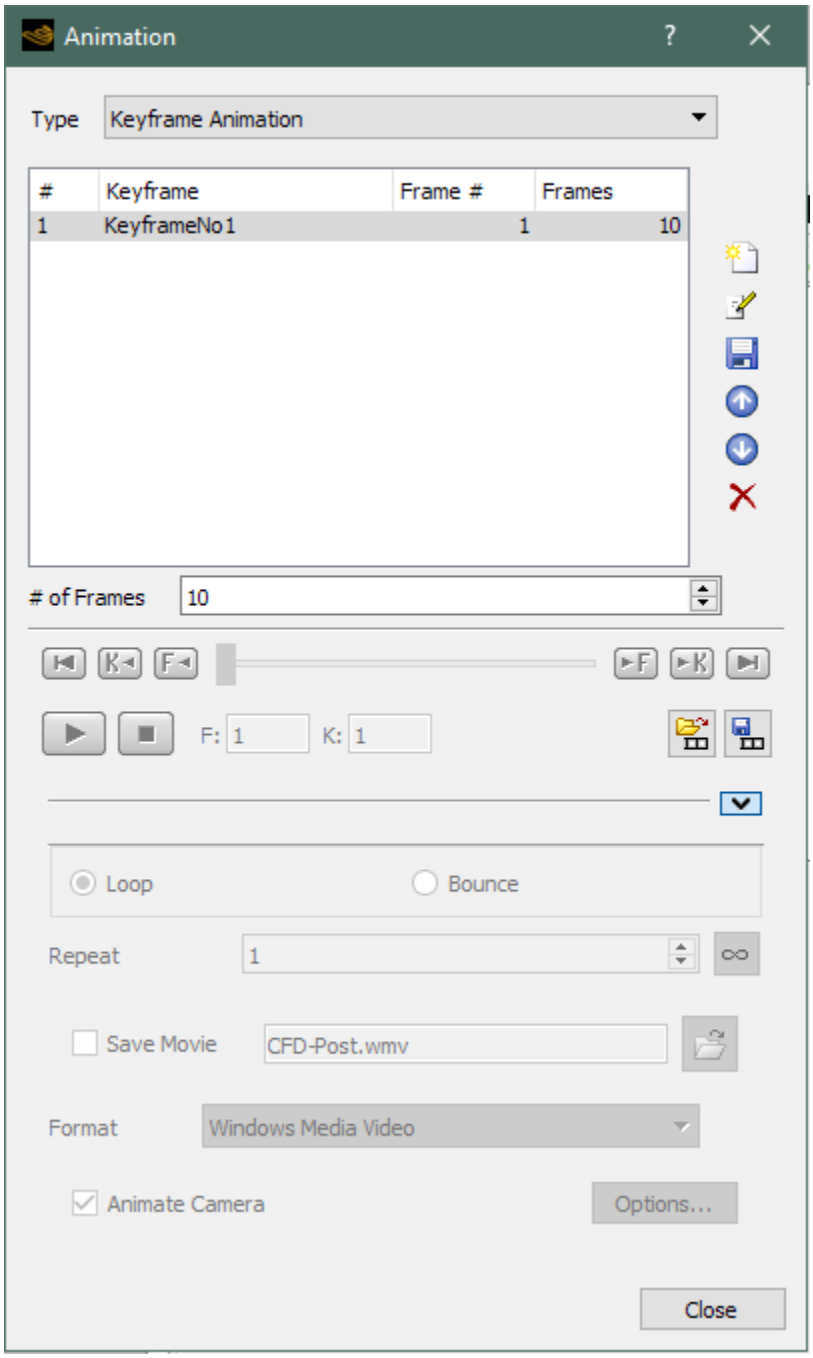
2.6.13.2. Creating the First Keyframe

Keyframes are required in order to produce an animation. You need to define the first viewer state, a second (and final) viewer state, and set the number of interpolated intermediate frames.

1. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)**.
2. In the **Outline**, under `User Locations and Plots`, turn off the visibility of `Slice Contour` and turn on the visibility of `Slice`.
3. Set **Type** to **Keyframe Animation**.
4. In the **Animation** dialog box, click *New* .

   A new keyframe named `KeyframeNo1` is created. This represents the current image displayed in the viewer.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2.6.13.3. Creating the Second Keyframe

Define the second keyframe and the number of intermediate frames:

1. In the **Outline**, under `User Locations and Plots`, double-click `Slice`.
2. On the **Geometry** tab, set **Point** coordinate values to `(0,0,-1.99)`.
3. Click **Apply**.

   The slice plane moves to the bottom of the mixer.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. In the **Animation** dialog box, click *New* .

   KeyframeNo2 is created and represents the image displayed in the viewer.

5. Select KeyframeNo1.

6. Set **# of Frames** (located below the list of keyframes) to 20.

   This is the number of intermediate frames used when going from KeyframeNo1 to KeyframeNo2. This number is displayed in the **Frames** column for KeyframeNo1.

7. Press **Enter**.

   The **Frame #** column shows the frame in which each keyframe appears. KeyframeNo1 appears at frame 1 since it defines the start of the animation. KeyframeNo2 is at frame 22 since you have 20 intermediate frames (frames 2 to 21) in between KeyframeNo1 and KeyframeNo2.

2.6.13.4. Viewing the Animation

More keyframes could be added, but this animation has only two keyframes (which is the minimum possible).

The controls previously grayed-out in the **Animation** dialog box are now available. The number of intermediate frames between keyframes is listed beside the keyframe having the lowest number of the pair. The number of frames listed beside the last keyframe is ignored.

1. Click *To Beginning* .

   This ensures that the animation will begin at the first keyframe.

2. Click *Play the animation* .

   The animation plays from frame 1 to frame 22. It plays relatively slowly because the slice plane must be updated for each frame.

2.6.13.5. Modifying the Animation

To make the plane sweep through the whole geometry, you will set the starting position of the plane to be at the top of the mixer. You will also modify the **Range** properties of the plane so that it shows the temperature variation better. As the animation is played, you can see the hot and cold water entering the mixer. Near the bottom of the mixer (where the water flows out) you can see that the temperature is quite uniform. The new temperature range lets you view the mixing process more accurately than the global range used in the first animation.

1. Configure the following setting(s) of Slice:

Tab	Setting	Value
Geometry	**Point**	0, 0, 1.99
Color	**Mode**	Variable
	Variable	Temperature

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Range	User Specified
	Min	295 [K]
	Max	305 [K]

2. Click **Apply**.

   The slice plane moves to the top of the static mixer.

   Note:

   Do not double-click in the next step.

3. In the **Animation** dialog box, single click (*do not double-click*) `KeyframeNo1` to select it.

   If you had double-clicked `KeyFrameNo1`, the plane and viewer states would have been re-defined according to the stored settings for `KeyFrameNo1`. If this happens, click *Undo* and try again to select the keyframe.

4. Click *Set Keyframe* .

   The image in the viewer replaces the one previously associated with `KeyframeNo1`.

5. Double-click `KeyframeNo2.`

   The object properties for the slice plane are updated according to the settings in `KeyFrameNo2`.

6. Configure the following setting(s) of `Slice`:

Tab	Setting	Value
Color	**Mode**	Variable
	Variable	Temperature
	Range	User Specified
	Min	295 [K]
	Max	305 [K]

7. Click **Apply**.

8. In the **Animation** dialog box, single-click `KeyframeNo2`.

9. Click *Set Keyframe* to save the new settings to `KeyframeNo2`.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.6.13.6. Saving a Movie

1. Click *More Animation Options* to view the additional options.

   The **Loop** and **Bounce** option buttons determine what happens when the animation reaches the last keyframe. When **Loop** is selected, the animation repeats itself the number of times defined by **Repeat**. When **Bounce** is selected, every other cycle is played in reverse order, starting with the second.

2. Select the check box next to **Save Movie**.

3. Set **Format** to `MPEG1`.

4. Click *Browse* next to **Save Movie**.

5. Under **File name** type: `StaticMixer.mpg`

6. If required, set the path location to a different directory.

7. Click **Save**.

   The movie filename (including path) has been set, but the animation has not yet been produced.

8. Click *To Beginning* .

9. Click *Play the animation* .

10. If prompted to overwrite an existing movie click **Overwrite**.

    The animation plays and builds an MPEG file.

11. Click the **Options** button at the bottom of the **Animation** dialog box.

    In **Advanced**, you can see that a **Frame Rate** of 24 frames per second was used to create the animation. The animation you produced contains a total of 22 frames, so it takes just under 1 second to play in a media player.

12. Click **Cancel** to close the dialog box.

13. Close the **Animation** dialog box.

14. Review the animation in third-party software as required.

2.6.14. Quitting CFD-Post

When finished with CFD-Post, exit the current window:

1. When you are finished, select **File** > **Quit** to exit CFD-Post.

2. Click **Quit** if prompted to save.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 3: Simulating Flow in a Static Mixer Using Workbench

This tutorial simulates a static mixer consisting of two inlet pipes delivering water into a mixing vessel; the water exits through an outlet pipe. A general workflow is established for analyzing the flow of fluid into and out of a mixer using Ansys Workbench.

This tutorial includes:

3.1. Tutorial Features

3.2. Overview of the Problem to Solve

3.3. Preparing the Working Directory

3.4. Setting Up the Project

3.5. Defining the Case Using CFX-Pre

3.6. Obtaining the Solution Using CFX-Solver Manager

3.7. Viewing the Results Using CFD-Post

3.1. Tutorial Features

In this tutorial you will learn about:

- Using Ansys Workbench to set up a project.
- Using Quick Setup mode in CFX-Pre to set up a problem.
- Using Ansys CFX-Solver Manager to obtain a solution.
- Modifying the outline plot in CFD-Post.
- Using streamlines in CFD-Post to trace the flow field from a point.
- Viewing temperature using colored planes and contours in CFD-Post.
- Creating an animation and saving it as a movie file.

Component	Feature	Details
CFX-Pre	User Mode	Quick Setup mode
	Analysis Type	Steady State
	Fluid Type	General Fluid
	Domain Type	Single Domain
	Turbulence Model	k-Epsilon

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Component	Feature	Details
	Heat Transfer	Thermal Energy
	Boundary Conditions	Inlet (Subsonic)
		Outlet (Subsonic)
		Wall: No-Slip
		Wall: Adiabatic
	Timestep	Physical Time Scale
CFD-Post	Animation	Keyframe
	Plots	Contour
		Outline Plot (Wireframe)
		Point
		Slice Plane
		Streamline

3.2. Overview of the Problem to Solve

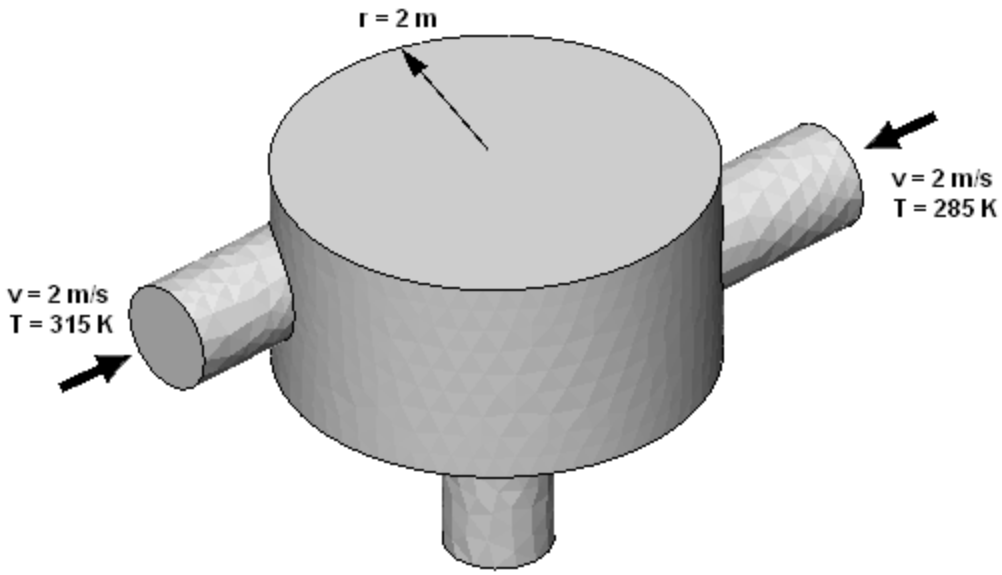
This tutorial simulates a static mixer consisting of two inlet pipes delivering water into a mixing vessel; the water exits through an outlet pipe. A general workflow is established for analyzing the flow of fluid into and out of a mixer.

Water enters through both pipes at the same rate but at different temperatures. The first entry is at a rate of 2 m/s and a temperature of 315 K and the second entry is at a rate of 2 m/s at a temperature of 285 K. The radius of the mixer is 2 m.

Your goal in this tutorial is to understand how to use CFX in Workbench to determine the speed and temperature of the water when it exits the static mixer.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 3.1: Static Mixer with 2 Inlet Pipes and 1 Outlet Pipe



If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- Running Ansys CFX Tutorials Using Ansys Workbench (p. 34)
- Changing the Display Colors (p. 37)

3.3. Preparing the Working Directory

1. Create a working directory.

   Ansys CFX uses a working directory as the default location for loading and saving files for a particular session or project.

2. Download the `static_mixer_workbench.zip` file here.

3. Unzip `static_mixer_workbench.zip` to your working directory.

   Ensure that the following tutorial input file is in your working directory:

   - `StaticMixerMesh.gtm`

3.4. Setting Up the Project

1. Start Ansys Workbench.

   - To launch Ansys Workbench on Windows, click the **Start** menu, then select **All Programs > ANSYS 2021 R2 > Workbench 2021 R2**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To launch Ansys Workbench on Linux, open a command line interface, type the path to `runwb2` (for example, `~/ansys_inc/v212/Framework/bin/Linux64/runwb2`), then press **Enter**.

2. From the main menu, select **File** > **Save As**.
3. In the **Save As** dialog box, browse to the working directory and set **File name** to `StaticMixer`.
4. Click **Save**.

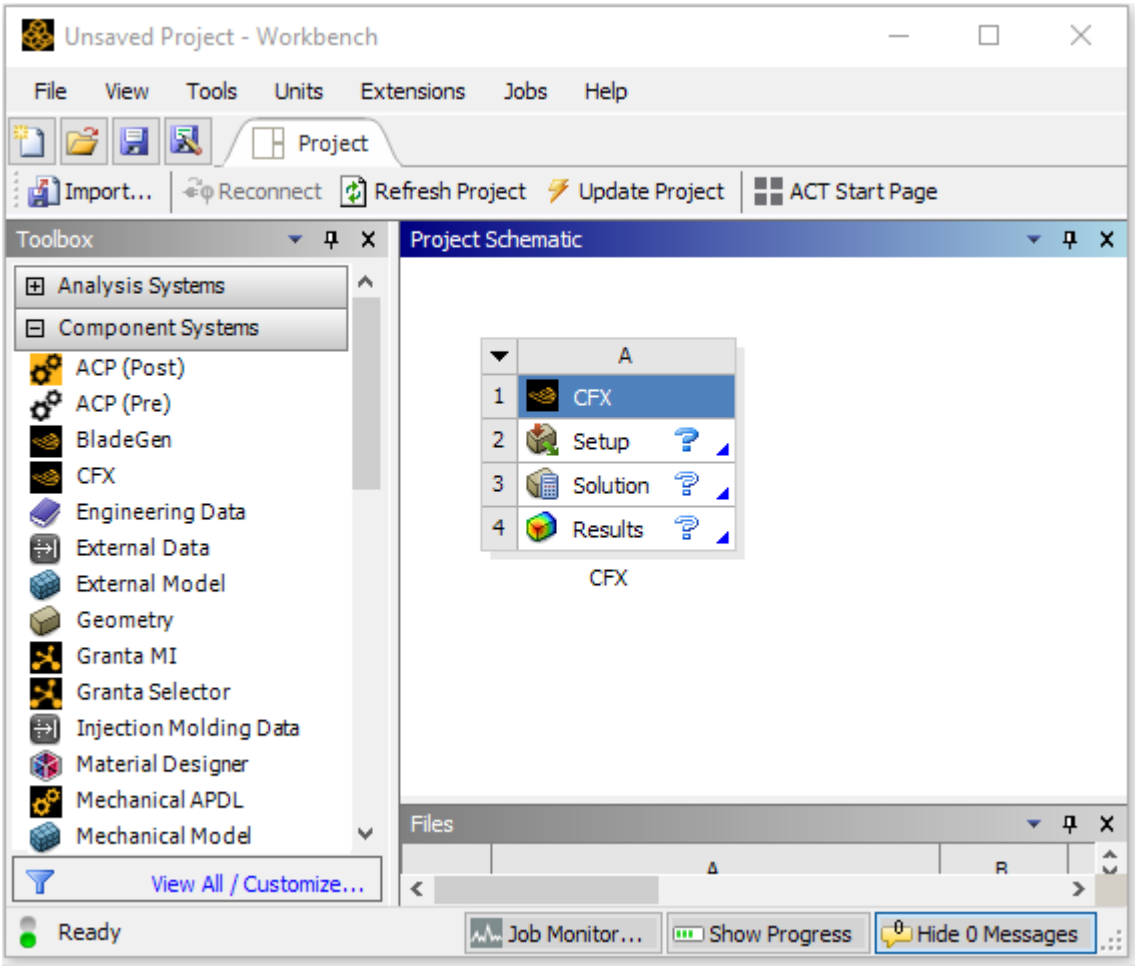
3.5. Defining the Case Using CFX-Pre

Because you are starting with an existing mesh, you can immediately use CFX-Pre to define the simulation. To launch CFX-Pre:

1. In the **Toolbox** pane, open **Component Systems** and double-click **CFX**.

   A **CFX** system opens in the **Project Schematic**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

You use a **CFX** component system because you are starting with a mesh. If you wanted to create the geometry and mesh, you would start with a **Fluid Flow (CFX)** analysis system.

2. Right-click the blue **CFX** cell (A1) and select **Rename**.
3. Change the name of the system to `Static Mixer`. Finish by pressing **Enter** or by clicking outside the cell.
4. In Ansys Workbench, enable **View** > **Files** and **View** > **Progress** so that you can see the files that are written and the time remaining to complete operations.
5. In the Workbench **Project Schematic**, double-click the **Setup** cell of the CFX component system.

   CFX-Pre opens.
6. Optionally, change the background color of the viewer in CFX-Pre for improved viewing:

   a. Select **Edit** > **Options**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Options** dialog box appears.

b. Adjust the color settings under **CFX-Pre** > **Graphics Style**.

For example, you could set the **Background** > **Color Type** to **Solid** and the **Color** to white.

c. Click **OK**.

3.5.1. Creating the Simulation Definition

Before importing and working with the mesh, you need to create a simulation; in this example, you will use Quick Setup mode. Quick Setup mode provides a simple wizard-like interface for setting up simple cases. This is useful for getting familiar with the basic elements of a CFD problem setup.

1. In CFX-Pre, select **Tools** > **Quick Setup Mode**.

   The Quick Setup Wizard opens, enabling you to define this single-phase simulation.

2. Under **Working Fluid** > **Fluid** select `Water`.

   This is a fluid already defined in the library of materials as water at 25°C.

3. Under **Mesh Data** > **Mesh File**, click *Browse* .

   The **Import Mesh** dialog box appears.

4. Under **Files of type**, select `CFX Mesh (*gtm *cfx)`.

5. From your working directory, select `StaticMixerMesh.gtm`.

6. Click **Open**.

   The mesh loads, which enables you to apply physics.

7. Click **Next**.

3.5.2. Setting the Physics Definition

You need to define the type of flow and the physical models to use in the fluid domain.

The flow is steady-state and you will specify the turbulence and heat transfer. Turbulence is modeled using the k-ε turbulence model and heat transfer using the thermal energy model. The k-ε turbulence model is a commonly used model and is suitable for a wide range of applications. The thermal energy model neglects high speed energy effects and is therefore suitable for low speed flow applications.

1. Under **Model Data**, note that the **Reference Pressure** is set to `1 [atm]`.

   All other pressure settings are relative to this reference pressure.

2. Set **Heat Transfer** to `Thermal Energy`.

3. Leave **Turbulence** at its default setting, `k-Epsilon`.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Click **Next**.

3.5.3. Defining Boundaries

The CFD model requires the definition of conditions on the boundaries of the domain.

1. Delete `Inlet` and `Outlet` from the list by right-clicking each and selecting **Delete Boundary**.
2. Right-click in the blank area where `Inlet` and `Outlet` were listed, then select **Add Boundary**.
3. Set **Name** to `in1`.
4. Click **OK**.

   The boundary is created and, when selected, properties related to the boundary are displayed.

3.5.4. Setting Boundary Data

Once boundaries are created, you need to create associated data. Based on Figure 3.1: Static Mixer with 2 Inlet Pipes and 1 Outlet Pipe (p. 73), you will define the velocity and temperature for the first inlet.

1. Set **in1** > **Boundary Type** to `Inlet`.
2. Set **Location** to `in1`.
3. Set the **Flow Specification** > **Option** to `Normal Speed` and set **Normal Speed** to: `2 [m s^-1]`
4. Set the **Temperature Specification** > **Static Temperature** to `315 [K]` (note the units).

3.5.5. Creating the Second Inlet Boundary Definition

Based on Figure 3.1: Static Mixer with 2 Inlet Pipes and 1 Outlet Pipe (p. 73), you know the second inlet boundary condition consists of a velocity of 2 m/s and a temperature of 285 K at one of the side inlets. You will define that now.

1. Under the **Boundary Definition** panel, right-click in the selector area and select **Add Boundary**.
2. Create a new boundary named `in2` with these settings:

Setting	Value
in2 > **Boundary Type**	Inlet
in2 > **Location**	in2
Flow Specification > **Option**	Normal Speed

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
Flow Specification > **Normal Speed**	 2 [m s^-1]
Temperature Specification > **Static Temperature**	 285 [K]

3.5.6. Creating the Outlet Boundary Definition

Now that the second inlet boundary has been created, the same concepts can be applied to building the outlet boundary.

1. Create a new boundary named `out` with these settings:

Setting	Value
out > **Boundary Type**	 Outlet
out > **Location**	 out
Flow Specification > **Option**	 Average Static Pressure
Flow Specification > **Relative Pressure**	 0 [Pa]

2. Click **Next**.

3.5.7. Moving to General Mode

There are no further boundary conditions that need to be set. All 2D exterior regions that have not been assigned to a boundary condition are automatically assigned to the default boundary condition.

1. Set **Operation** to `Enter General Mode`.
2. Click **Finish**.

   The three boundary conditions are displayed in the viewer as sets of arrows at the boundary surfaces. Inlet boundary arrows are directed into the domain. Outlet boundary arrows are directed out of the domain.

3.5.8. Using the Viewer

Now that the simulation is loaded, take a moment to explore how you can use the viewer toolbar to zoom in or out and to rotate the object in the viewer.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.5.8.1. Using the Zoom Tools

There are several icons available for controlling the level of zoom in the viewer.

1. Click *Zoom Box*
2. Click and drag a rectangular box over the geometry.
3. Release the mouse button to zoom in on the selection.

   The geometry zoom changes to display the selection at a greater resolution.
4. Click *Fit View* to re-center and re-scale the geometry.

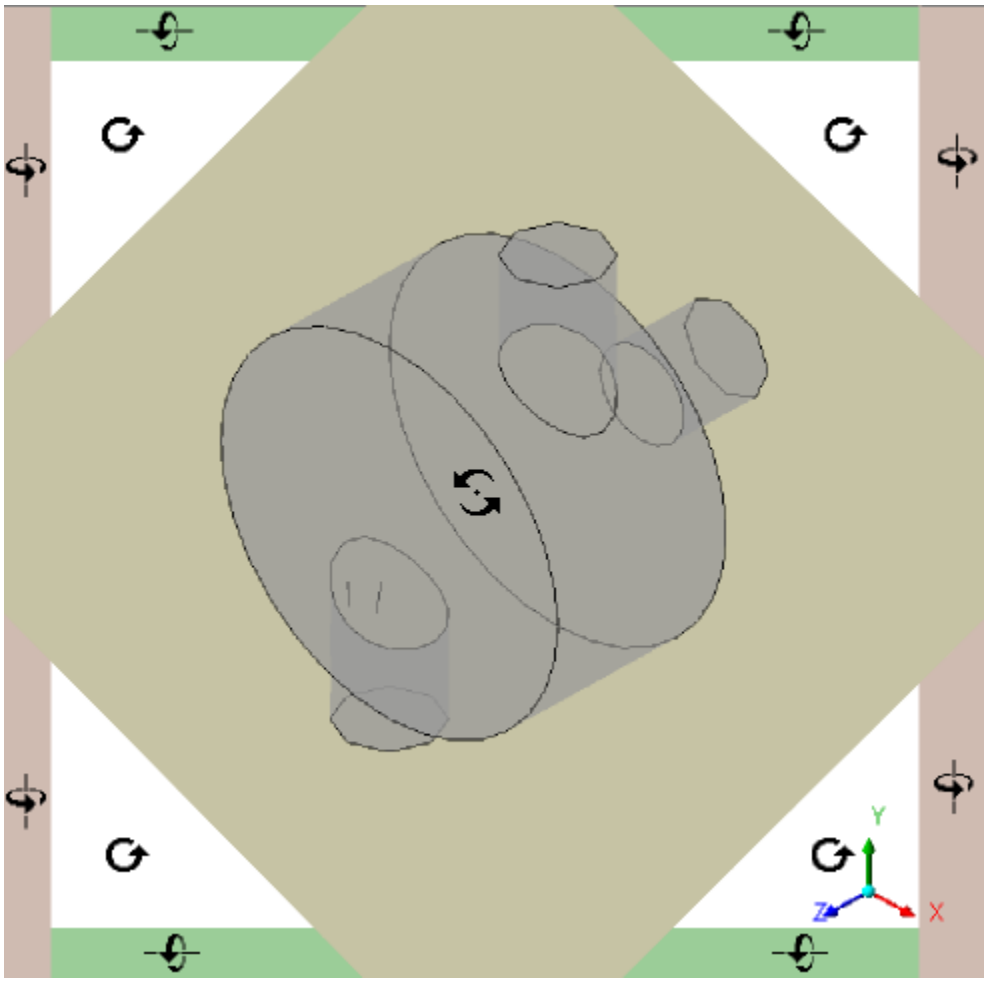
3.5.8.2. Rotating the Geometry

If you need to rotate an object or to view it from a new angle, you can use the viewer toolbar.

1. Click *Rotate* on the viewer toolbar.
2. Click and drag within the geometry repeatedly to test the rotation of the geometry.

   The geometry rotates based on the direction of mouse movement and based on the initial mouse cursor shape, which changes depending on where the mouse cursor is in the viewer. If the mouse drag starts near a corner of the viewer window, the motion of the geometry will be constrained to rotation about a single axis, as indicated by the mouse cursor shape.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Right-click a blank area in the viewer and select **Predefined Camera** > **View From -X**.
4. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z Up)**.

   A clearer view of the mesh is displayed.

3.5.9. Setting Solver Control

Solver Control parameters control aspects of the numerical solution generation process.

While an upwind advection scheme is less accurate than other advection schemes, it is also more robust. This advection scheme is suitable for obtaining an initial set of results, but in general should not be used to obtain final accurate results.

The time scale can be calculated automatically by the solver or set manually. The `Automatic` option tends to be conservative, leading to reliable, but often slow, convergence. It is often possible to accelerate convergence by applying a time scale factor or by choosing a manual value that is more aggressive than the `Automatic` option. In this tutorial, you will select a physical time scale, leading to convergence that is twice as fast as the `Automatic` option.

1. In the CFX-Pre toolbar, click *Solver Control* .

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. On the **Basic Settings** tab, set **Advection Scheme** > **Option** to `Upwind`.
3. Set **Convergence Control** > **Fluid Timescale Control** > **Timescale Control** to `Physical Timescale` and set the physical timescale value to `2 [s]`.
4. Click **OK**.

3.6. Obtaining the Solution Using CFX-Solver Manager

To obtain a solution, you need to launch the CFX-Solver Manager and subsequently use it to start the solver:

1. Double-click the Ansys Workbench **Solution** cell. The CFX-Solver Manager appears with the **Define Run** dialog box displayed.

   The **Define Run** dialog box enables configuration of a run for processing by CFX-Solver. In this case, all of the information required to perform a new serial run (on a single processor) is entered automatically. You do not need to alter the information in the **Define Run** dialog box.

2. Optionally specify a local parallel run:

   a. Set **Run Mode** to a parallel mode suitable for your configuration; for example, `Intel MPI Local Parallel`.

      This is the recommended method for most applications.

   b. If required, click *Add Partition* to add more partitions.

      Ideally, the number of partitions should not exceed the number of available processor cores.

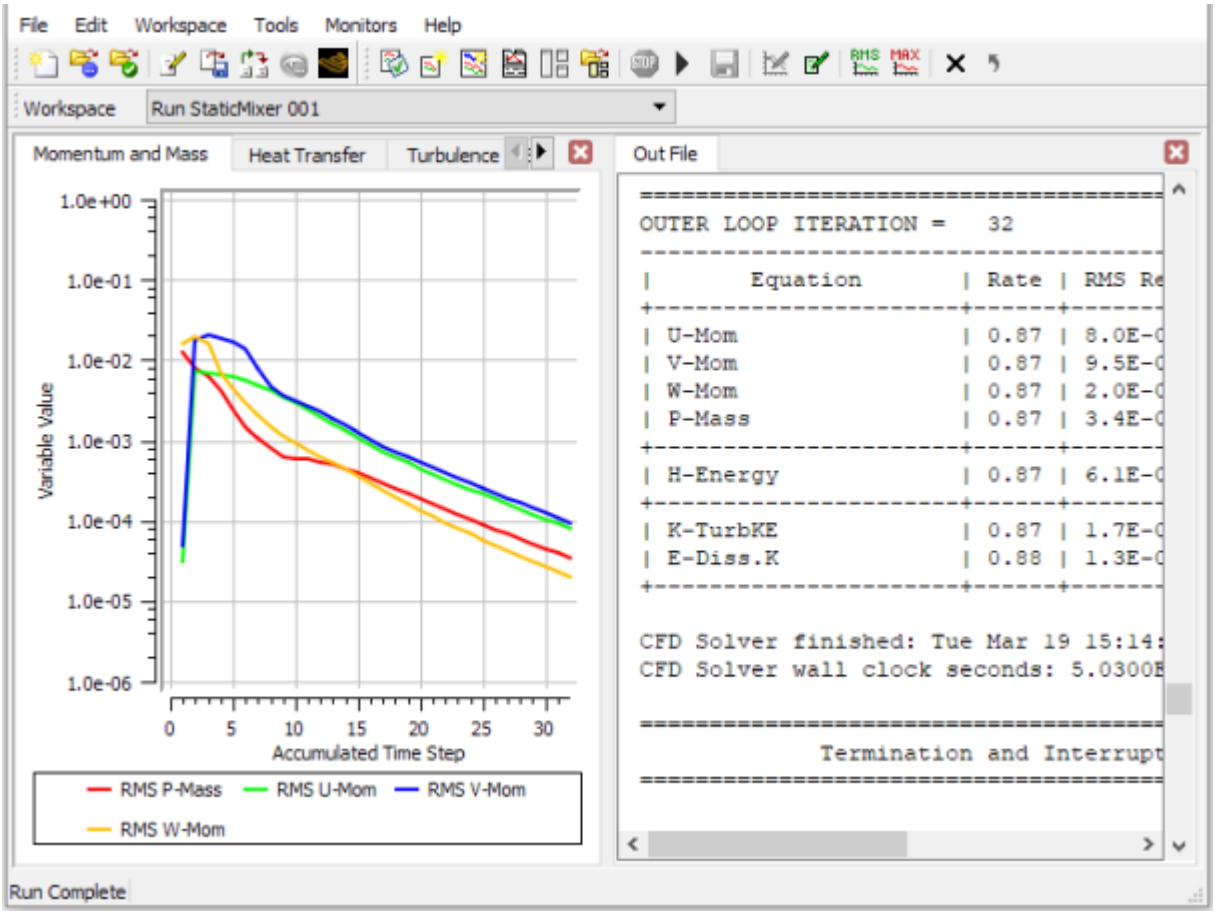
   More detailed information about setting up CFX to run in parallel is available in Flow Around a Blunt Body (p. 187).

3. Click **Start Run**.

   CFX-Solver launches and a split screen appears and displays the results of the run graphically and as text. The panes continue to build as CFX-Solver Manager operates.

   One window shows the convergence history plots and the other displays text output from CFX-Solver. The text lists physical properties, boundary conditions, and various other parameters used or calculated in creating the model. All the text is written to the CFX-Solver Output file automatically (in this case, `StaticMixer_001.out`).

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

Once the second iteration appears, data begins to plot. Plotting may take a long time depending on the amount of data to process. Let the process run.

When CFX-Solver is finished, a message is displayed and the final line in the CFX-Solver Output file (which you can see in the CFX-Solver Manager) is:

```
This run of the Ansys CFX Solver has finished.
```

3.7. Viewing the Results Using CFD-Post

Once CFX-Solver has finished, you can use CFD-Post to review the finished results:

1. In Ansys Workbench, right-click the **Results** cell and select **Refresh**.
2. When the refresh is complete, double-click the **Results** cell. CFD-Post appears.

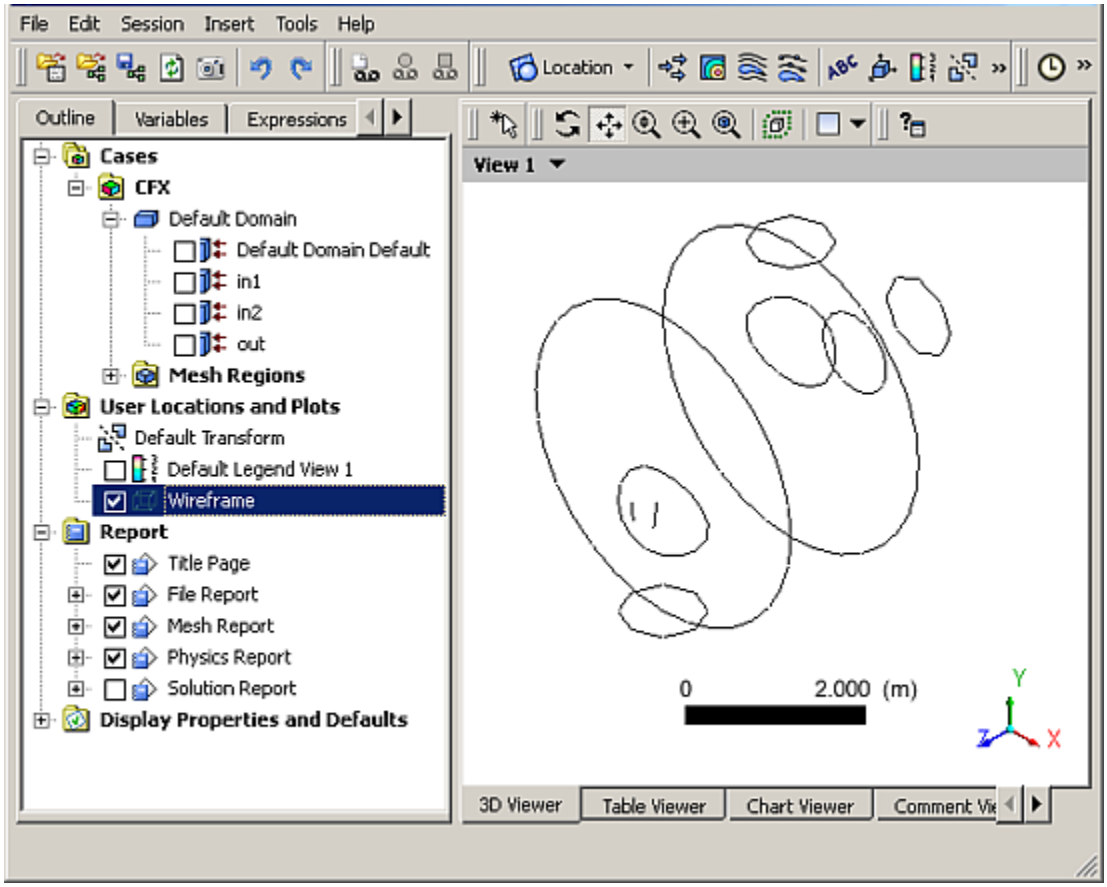
When CFD-Post starts, the viewer and **Outline** workspace are displayed. Optionally, change the background color of the viewer for improved viewing:

1. In CFD-Post, select **Edit** > **Options**. The **Options** dialog box appears.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Adjust the color settings under **CFD-Post** > **Viewer**. For example, you could set the **Background** > **Color Type** to **Solid** and the **Color** to white.

3. Click **OK**.



The viewer displays an outline of the geometry and other graphic objects. You can use the mouse or the toolbar icons to manipulate the view, exactly as in CFX-Pre.

The tutorial follows this general workflow for viewing results in CFD-Post:

3.7.1. Setting the Edge Angle for a Wireframe Object
3.7.2. Creating a Point for the Origin of the Streamline
3.7.3. Creating a Streamline Originating from a Point
3.7.4. Rearranging the Point
3.7.5. Configuring a Default Legend
3.7.6. Creating a Slice Plane
3.7.7. Defining Slice Plane Geometry
3.7.8. Configuring Slice Plane Views
3.7.9. Rendering Slice Planes
3.7.10. Coloring the Slice Plane
3.7.11. Moving the Slice Plane
3.7.12. Adding Contours

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.7.13. Working with Animations
3.7.14. Closing the Applications

3.7.1. Setting the Edge Angle for a Wireframe Object

The outline of the geometry is called the *wireframe* or *outline plot*.

By default, CFD-Post displays only some of the surface mesh. This sometimes means that when you first load your results file, the geometry outline is not displayed clearly. You can control the amount of the surface mesh shown by editing the `Wireframe` object listed in the **Outline**.

The check boxes next to each object name in the **Outline** tree view control the visibility of each object. Currently only the `Wireframe` and `Default Legend` objects have visibility turned on.

The edge angle determines how much of the surface mesh is visible. If the angle between two adjacent faces is greater than the edge angle, then that edge is drawn. If the edge angle is set to 0°, the entire surface mesh is drawn. If the edge angle is large, then only the most significant corner edges of the geometry are drawn.

For this geometry, a setting of approximately 15° lets you view the model location without displaying an excessive amount of the surface mesh.

In this module you can also modify the zoom settings and view of the wireframe.

1. In the **Outline**, under `User Locations and Plots`, double-click `Wireframe`.
2. Right-click a blank area anywhere in the viewer, select **Predefined Camera** from the shortcut menu, and select **Isometric View (Z up)**.

   Tip:

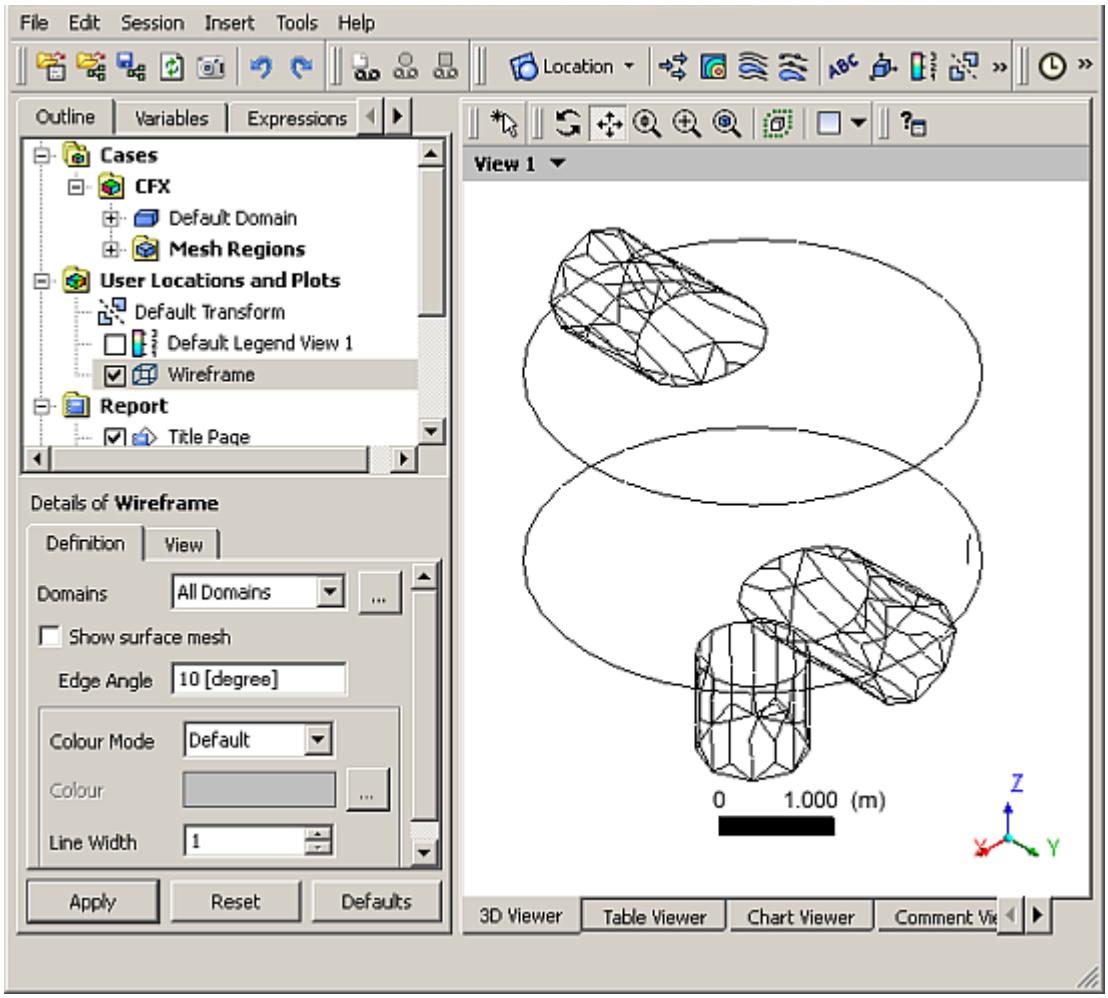
   While it is not necessary to change the view to set the edge angle for the wireframe, doing so enables you to explore the practical uses of this feature.

3. In the `Wireframe` details view, under **Definition**, click in the **Edge Angle** box.

   An embedded slider is displayed.
4. Type a value of `10 [degree]`.
5. Click **Apply** to update the object with the new setting.

   Notice that more surface mesh is displayed.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



6. Drag the embedded slider to set the **Edge Angle** value to approximately `45 [degree]`.
7. Click **Apply** to update the object with the new setting.

   Less of the outline of the geometry is displayed.
8. Type a value of `15 [degree]`.
9. Click **Apply** to update the object with the new setting.

3.7.2. Creating a Point for the Origin of the Streamline

A *streamline* is the path that a particle of zero mass would follow through the domain.

1. Select **Insert** > **Location** > **Point** from the main menu.

   You can also use the toolbars to create a variety of objects. Later modules and tutorials will explore this further.
2. Click **OK**.

   This accepts the default name.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Set **Definition** > **Method** to XYZ.
4. Under **Point**, enter the following coordinates: `-1, -1, 1`.

   This is a point near the first inlet.
5. Click **Apply**.

   The point appears as a symbol in the viewer as a crosshair symbol.

3.7.3. Creating a Streamline Originating from a Point

Where applicable, streamlines can trace the flow direction forwards (downstream) and/or backwards (upstream).

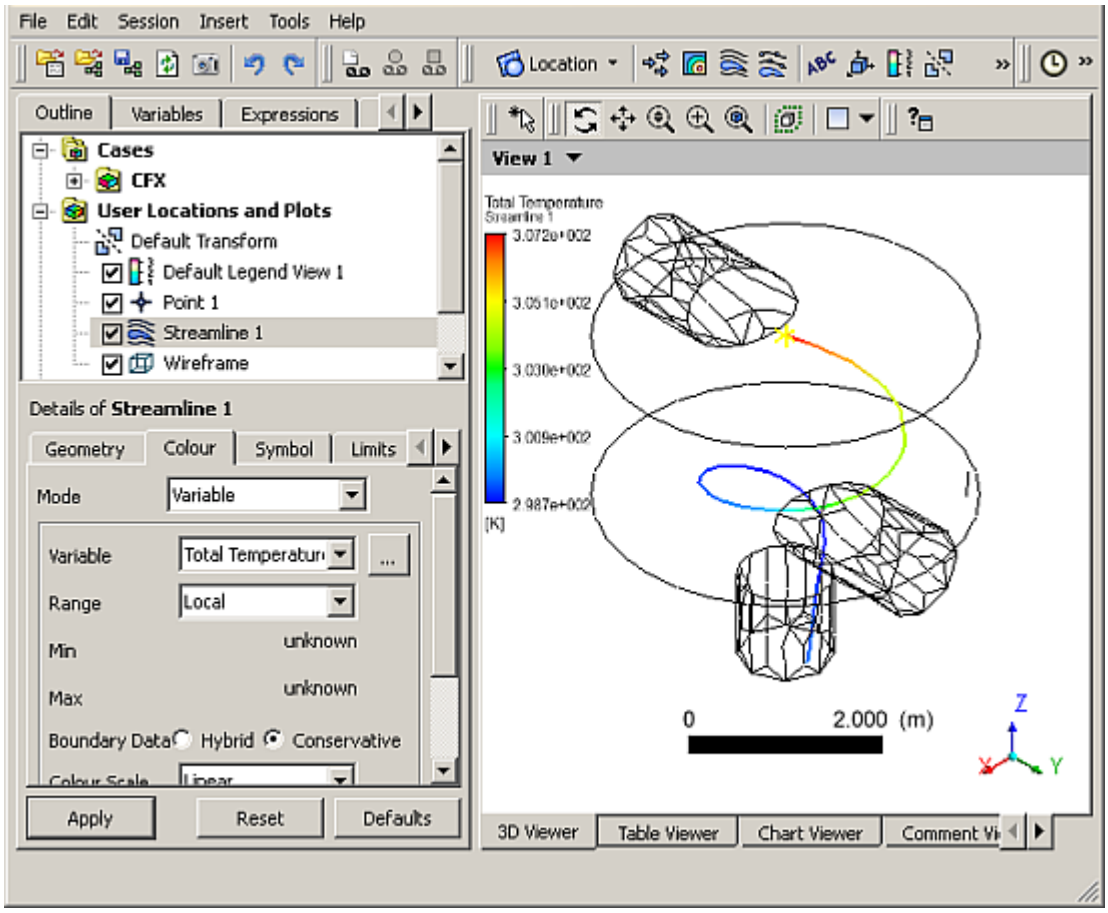
1. From the main menu, select **Insert** > **Streamline**.
2. Click **OK**.
3. Set **Definition** > **Start From** to `Point 1`.

   Tip:

   To create streamlines originating from more than one location, click the *Ellipsis* icon to the right of the **Start From** box. This displays the **Location Selector** dialog box, where you can use the **Ctrl** and **Shift** keys to pick multiple locators.
4. Click the **Color** tab.
5. Set **Mode** to `Variable`.
6. Set **Variable** to `Total Temperature`.
7. Set **Range** to `Local`.
8. Click **Apply**.

   The streamline shows the path of a zero mass particle from `Point 1`. The temperature is initially high near the hot inlet, but as the fluid mixes the temperature drops.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3.7.4. Rearranging the Point

Once created, a point can be rearranged manually or by setting specific coordinates.

Tip:

In this module, you may choose to display various views and zooms from the **Predefined Camera** option in the shortcut menu (such as **Isometric View (Z up)** or **View From -X**) and by using *Zoom Box* if you prefer to change the display.

---

1. In **Outline**, under `User Locations and Plots` double-click `Point 1`.

   Properties for the selected user location are displayed.

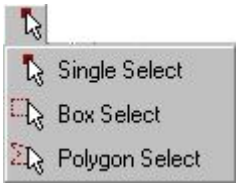
2. Under **Point**, set these coordinates: `-1, -2.9, 1`.

3. Click **Apply**.

   The point is moved and the streamline redrawn.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. In the viewer toolbar, click *Select* and ensure that the adjacent toolbar icon is set to **Single Select**.



While in select mode, you cannot use the left mouse button to re-orient the object in the viewer.

5. In the viewer, drag `Point 1` (appears as a yellow addition sign) to a new location within the mixer.

The point position is updated in the details view and the streamline is redrawn at the new location. The point moves normal in relation to the viewing direction.

6. Click *Rotate*.

> **Tip:**
>
> You can also click in the viewer area, and press the space bar to toggle between Select and Viewing Mode. A way to pick objects from Viewing Mode is to hold down **Ctrl** + **Shift** while clicking on an object with the left mouse button.

7. Under **Point**, reset these coordinates: `-1, -1, 1`.

8. Click **Apply**.

The point appears at its original location.

9. Right-click a blank area in the viewer and select **Predefined Camera** > **View From -X**.

3.7.5. Configuring a Default Legend

You can modify the appearance of the default legend.

The default legend appears whenever a plot is created that is colored by a variable. The streamline color is based on temperature; therefore, the legend shows the temperature range. The color pattern on the legend's color bar is banded in accordance with the bands in the plot.

> **Note:**
>
> If a user-specified range is used for the legend, one or more bands may represent values beyond the legend's range. In this case, these band colors are extrapolated slightly past the range of colors shown in the legend.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The default legend displays values for the last eligible plot that was opened in the details view. To maintain a legend definition during a CFD-Post session, you can create a new legend by clicking *Legend* .

Because there are many settings that can be customized for the legend, this module allows you the freedom to experiment with them. In the last steps you will set up a legend, based on the default legend, with a minor modification to the position.

Tip:

When editing values, you can restore the values that were present when you began editing by clicking **Reset**. To restore the factory-default values, click **Default**.

1. Double-click `Default Legend View 1`.

   The **Definition** tab of the default legend is displayed.

2. Configure the following setting(s):

Tab	Setting	Value
Definition	**Title Mode**	User Specified
	Title	Streamline Temp.
	Horizontal	(Selected)
	Location > **Y Justification**	 Bottom

3. Click **Apply**.

   The appearance and position of the legend changes based on the settings specified.

4. Modify various settings in **Definition** and click **Apply** after each change.

5. Select **Appearance**.

6. Modify a variety of settings in the **Appearance** and click **Apply** after each change.

7. Click **Defaults**.

8. Click **Apply**.

9. Under **Outline**, in `User Locations and Plots`, clear the check boxes for `Point 1` and `Streamline 1`.

   Since both are no longer visible, the associated legend no longer appears.

3.7.6. Creating a Slice Plane

Defining a slice plane allows you to obtain a cross-section of the geometry.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In CFD-Post you often view results by coloring a graphic object. The graphic object could be an isosurface, a vector plot, or in this case, a plane. The object can be a fixed color or it can vary based on the value of a variable.

You already have some objects defined by default (listed in the **Outline**). You can view results on the boundaries of the static mixer by coloring each boundary object by a variable. To view results within the geometry (that is, on non-default locators), you will create new objects.

You can use the following methods to define a plane:

- `Three Points`: creates a plane from three specified points.
- `Point and Normal`: defines a plane from one point on the plane and a normal vector to the plane.
- `YZ Plane`, `ZX Plane`, and `XY Plane`: similar to `Point and Normal`, except that the normal is defined to be normal to the indicated plane.

1. From the main menu, select **Insert** > **Location** > **Plane** or click **Location** > **Plane**.
2. In the **Insert Plane** window, type: `Slice`
3. Click **OK**.

   The details view for the plane appears; the **Geometry**, **Color**, **Render**, and **View** tabs enable you to configure the characteristics of the plane.

3.7.7. Defining Slice Plane Geometry

You need to choose the vector normal to the plane. You want the plane to lie in the x-y plane, hence its normal vector points along the Z axis. You can specify any vector that points in the Z direction, but you will choose the most obvious (0,0,1).

1. On the **Geometry** tab, expand **Definition**.
2. Under **Method** select `Point and Normal`.
3. Under **Point** enter `0,0,1`.
4. Under **Normal** enter `0,0,1`.
5. Click **Apply**.

   `Slice` appears under **User Locations and Plots**. Rotate the view to see the plane.

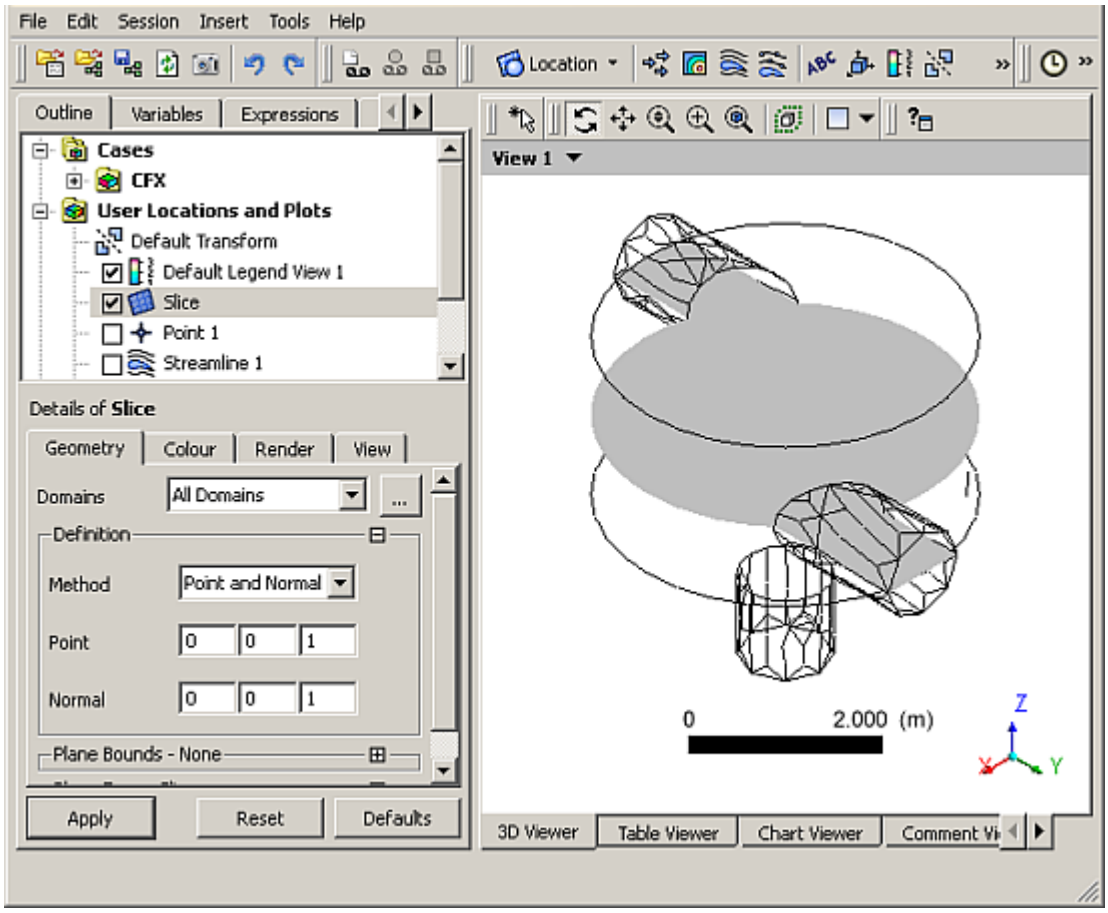
3.7.8. Configuring Slice Plane Views

Depending on the view of the geometry, various objects may not appear because they fall in a 2D space that cannot be seen.

1. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)**.

   The slice is now visible in the viewer.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Click *Zoom Box* .
3. Click and drag a rectangular selection over the geometry.
4. Release the mouse button to zoom in on the selection.
5. Click *Rotate* .
6. Click and drag the mouse pointer down slightly to rotate the geometry towards you.
7. Select **Isometric View (Z up)** as described earlier.

3.7.9. Rendering Slice Planes

Render settings determine how the plane is drawn.

1. In the details view for **Slice**, select the **Render** tab.
2. Clear **Show Faces**.
3. Select **Show Mesh Lines**.
4. Under **Show Mesh Lines** change **Color Mode** to `User Specified`.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Click the current color in **Line Color** to change to a different color.

   For a greater selection of colors, click the *Ellipsis* icon to use the **Color selector** dialog box.

6. Click **Apply**.

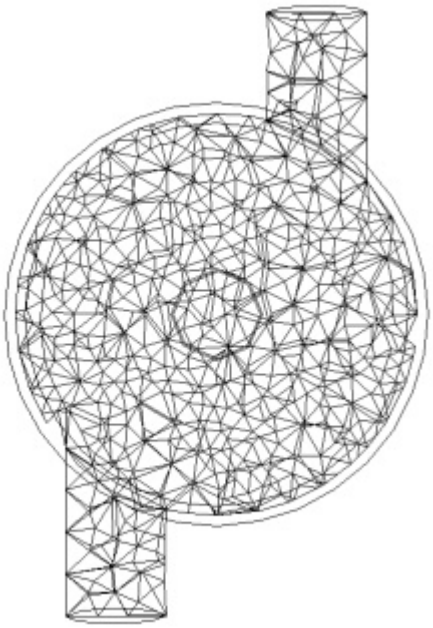
7. Click *Zoom Box*.

8. Zoom in on the geometry to view it in greater detail.

   The line segments show where the slice plane intersects with mesh element faces. The end points of each line segment are located where the plane intersects mesh element edges.

9. Right-click a blank area in the viewer and select **Predefined Camera** > **View From +Z**.

   The image shown below can be used for comparison with Flow in a Static Mixer (Refined Mesh) (p. 103) (in the section Creating a Slice Plane (p. 112)), where a refined mesh is used.



3.7.10. Coloring the Slice Plane

The **Color** panel is used to determine how the object faces are colored.

1. Configure the following setting(s) of `Slice`:

Tab	Setting	Value
Color	**Mode**	Variable [a]
	Variable	Temperature
Render	**Show Faces**	(Selected)

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Show Mesh Lines	(Cleared)

a. You can specify the variable (in this case, temperature) used to color the graphic element. The `Constant` mode allows you to color the plane with a fixed color.

2. Click **Apply**.

   Hot water (red) enters from one inlet and cold water (blue) from the other.

3.7.11. Moving the Slice Plane

You can move the plane to different locations:

1. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)** from the shortcut menu.
2. Click the **Geometry** tab.

   Review the settings in **Definition** under **Point** and under **Normal**.

3. Click *Single Select* .
4. Click and drag the plane to a new location that intersects the domain.

   As you drag the mouse, the viewer updates automatically. Note that **Point** updates with new settings.

5. Type in **Point** settings of `0,0,1`.
6. Click **Apply**.
7. Click *Rotate* .
8. Turn off the visibility for `Slice` by clearing the check box next to `Slice` in the **Outline** tree view.

3.7.12. Adding Contours

Contours connect all points of equal value for a scalar variable (for example, `Temperature`) and help to visualize variable values and gradients. Colored bands fill the spaces between contour lines. Each band is colored by the average color of its two bounding contour lines (even if the latter are not displayed).

1. Right-click a blank area in the viewer and select **Predefined Camera > Isometric View (Z up)** from the shortcut menu.
2. Select **Insert** > **Contour** from the main menu or click *Contour* .

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Insert Contour** dialog box is displayed.

3. Set **Name** to `Slice Contour`.
4. Click **OK**.
5. Configure the following setting(s):

Tab	Setting	Value
Geometry	**Locations**	Slice
	Variable	Temperature
Render	**Show Contour Lines**	(Selected)

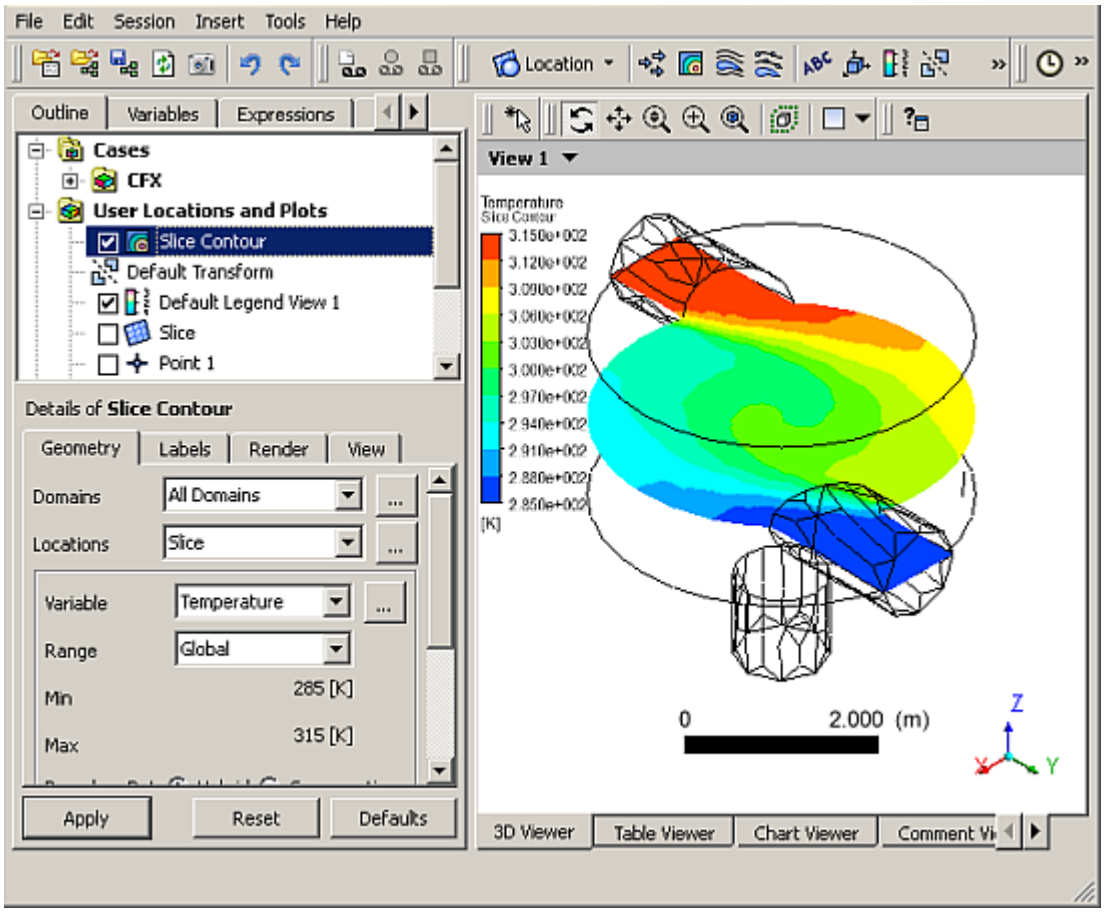
6. Click **Apply**.

Important:

The colors of 3D graphics object faces are slightly altered when lighting is on. To view colors with highest accuracy, go to the **Render** tab and, under **Show Faces**, clear **Lighting** and click **Apply**.

The graphic element faces are visible, producing a contour plot as shown.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

Make sure that the visibility of `Slice` (in the **Outline** tree view) is turned off.

3.7.13. Working with Animations

Animations build transitions between views for development of video files.

The tutorial follows this general workflow for creating a keyframe animation:

3.7.13.1. Showing the Animation Dialog Box
3.7.13.2. Creating the First Keyframe
3.7.13.3. Creating the Second Keyframe
3.7.13.4. Viewing the Animation
3.7.13.5. Modifying the Animation
3.7.13.6. Saving a Movie

3.7.13.1. Showing the Animation Dialog Box

The **Animation** dialog box is used to define keyframes and to export to a video file.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select **Tools** > **Animation** or click *Animation* .

   The **Animation** dialog box appears.

2. Set **Type** to **Keyframe Animation**.

3.7.13.2. Creating the First Keyframe

Keyframes are required in order to produce a keyframe animation. You need to define the first viewer state, a second (and final) viewer state, and set the number of interpolated intermediate frames.

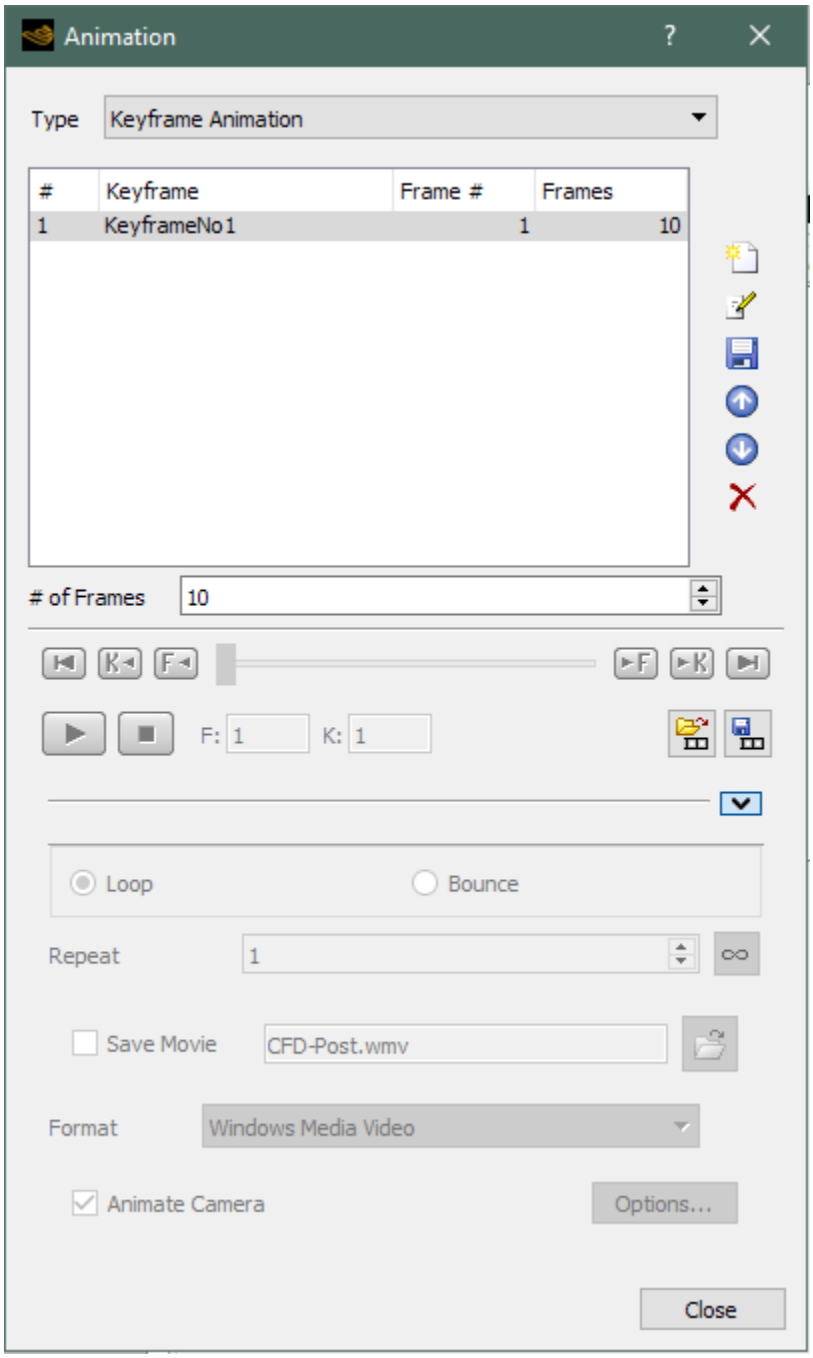
1. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)**.

2. In the **Outline**, under `User Locations and Plots`, turn off the visibility of `Slice Contour` and turn on the visibility of `Slice`.

3. In the **Animation** dialog box, click *New* .

   A new keyframe named `KeyframeNo1` is created. This represents the current image displayed in the viewer.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3.7.13.3. Creating the Second Keyframe

Define the second keyframe and the number of intermediate frames:

1. In the **Outline**, under `User Locations and Plots`, double-click `Slice`.
2. On the **Geometry** tab, set **Point** coordinate values to `(0,0,-1.99)`.
3. Click **Apply**.

   The slice plane moves to the bottom of the mixer.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. In the **Animation** dialog box, click *New* .

   `KeyframeNo2` is created and represents the image displayed in the viewer.

5. Select `KeyframeNo1` so that you can set the number of frames to be interpolated between the two keyframes.

6. Set **# of Frames** (located below the list of keyframes) to `20`.

   This is the number of intermediate frames used when going from `KeyframeNo1` to `KeyframeNo2`. This number is displayed in the **Frames** column for `KeyframeNo1`.

7. Press **Enter**.

   The **Frame #** column shows the frame in which each keyframe appears. `KeyframeNo1` appears at frame 1 since it defines the start of the animation. `KeyframeNo2` is at frame 22 since you have 20 intermediate frames (frames 2 to 21) in between `KeyframeNo1` and `KeyframeNo2`.

3.7.13.4. Viewing the Animation

More keyframes could be added, but this animation has only two keyframes (which is the minimum possible).

The controls previously grayed-out in the **Animation** dialog box are now available. The number of intermediate frames between keyframes is listed beside the keyframe having the lowest number of the pair. The number of frames listed beside the last keyframe is ignored.

1. Click *To Beginning* .

   This ensures that the animation will begin at the first keyframe.

2. Click *Play the animation* .

   The animation plays from frame 1 to frame 22. It plays relatively slowly because the slice plane must be updated for each frame.

3.7.13.5. Modifying the Animation

To make the plane sweep through the whole geometry, you will set the starting position of the plane to be at the top of the mixer. You will also modify the **Range** properties of the plane so that it shows the temperature variation better. As the animation is played, you can see the hot and cold water entering the mixer. Near the bottom of the mixer (where the water flows out) you can see that the temperature is quite uniform. The new temperature range lets you view the mixing process more accurately than the global range used in the first animation.

1. Configure the following setting(s) of `Slice`:

Tab	Setting	Value
Geometry	**Point**	0, 0, 1.99
Color	**Mode**	Variable

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Variable	Temperature
	Range	User Specified
	Min	295 [K]
	Max	305 [K]

2. Click **Apply**.

   The slice plane moves to the top of the static mixer.

   Note:

   Do not double-click in the next step.

3. In the **Animation** dialog box, single click (*do not double-click*) `KeyframeNo1` to select it.

   If you had double-clicked `KeyFrameNo1`, the plane and viewer states would have been re-defined according to the stored settings for `KeyFrameNo1`. If this happens, click *Undo* and try again to select the keyframe.

4. Click *Set Keyframe* .

   The image in the viewer replaces the one previously associated with `KeyframeNo1`.

5. Double-click `KeyframeNo2`.

   The object properties for the slice plane are updated according to the settings in `KeyFrameNo2`.

6. Configure the following setting(s) of `Slice`:

Tab	Setting	Value
Color	**Mode**	Variable
	Variable	Temperature
	Range	User Specified
	Min	295 [K]
	Max	305 [K]

7. Click **Apply**.

8. In the **Animation** dialog box, single-click `KeyframeNo2`.

9. Click *Set Keyframe* to save the new settings to `KeyframeNo2`.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.7.13.6. Saving a Movie

1. Click *More Animation Options* to view the additional options.

   The **Loop** and **Bounce** option buttons determine what happens when the animation reaches the last keyframe. When **Loop** is selected, the animation repeats itself the number of times defined by **Repeat**. When **Bounce** is selected, every other cycle is played in reverse order, starting with the second.

2. Select **Save Movie**.

3. Set **Format** to `MPEG1`.

4. Click *Browse* next to **Save Movie**.

5. Under **File name** type: `StaticMixer.mpg`

6. If required, set the path location to a different directory. You may want to set the directory to your working directory so that the animation will be in the same location as the project files.

7. Click **Save**.

   The movie filename (including path) has been set, but the animation has not yet been produced.

8. Click *To Beginning*.

9. Click *Play the animation*.

10. If prompted to overwrite an existing movie click **Overwrite**.

    The animation plays and builds an MPEG file.

11. Click the **Options** button at the bottom of the **Animation** dialog box.

    In **Advanced**, you can see that a **Frame Rate** of 24 frames per second was used to create the animation. The animation you produced contains a total of 22 frames, so it takes just under 1 second to play in a media player.

12. Click **Cancel** to close the dialog box.

13. Close the **Animation** dialog box.

14. View the animation using a media player.

3.7.14. Closing the Applications

Before you close the project, take a moment to look at the files listed in the **Files** view. You will see the project file, `StaticMixer.wbpj`, and the files that Ansys Workbench created (such as CFX-Solver Input, CFX-Solver Output, CFX-Solver Results, CFX-Pre Case, CFD-Post State, and Design Point files).

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Close Ansys Workbench (and the applications it launched) by selecting **File** > **Exit** from Ansys Workbench. Ansys Workbench prompts you to save all of your project files.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 4: Flow in a Static Mixer (Refined Mesh)

This tutorial includes:

4.1. Tutorial Features

4.2. Overview of the Problem to Solve

4.3. Preparing the Working Directory

4.4. Defining the Case Using CFX-Pre

4.5. Obtaining the Solution Using CFX-Solver Manager

4.6. Viewing the Results Using CFD-Post

Important:

This tutorial requires files `StaticMixer.def` and `StaticMixer_001.res`, which are produced by following tutorial Simulating Flow in a Static Mixer Using CFX in Stand-alone Mode (p. 39).

4.1. Tutorial Features

In this tutorial you will learn about:

- Using the General mode of CFX-Pre (this mode is used for more complex cases).
- Rerunning a problem with a refined mesh.
- Importing a CCL (CFX Command Language) file to copy the definition of a different simulation into the current simulation.
- Viewing the mesh with a Sphere Volume locator and a Surface Plot.
- Using the **Report Viewer** to analyze mesh quality.

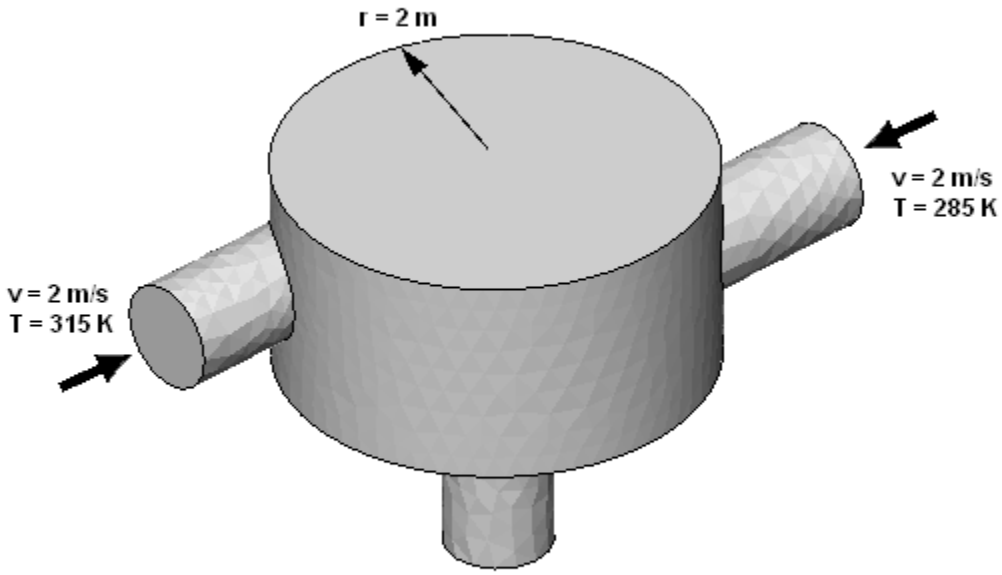
Component	Feature	Details
CFX-Pre	User Mode	General mode
	Analysis Type	Steady State
	Fluid Type	General Fluid
	Domain Type	Single Domain
	Turbulence Model	k-Epsilon
	Heat Transfer	Thermal Energy
	Boundary Conditions	Inlet (Subsonic)
		Outlet (Subsonic)

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Component	Feature	Details
		Wall: No-Slip
		Wall: Adiabatic
	Timestep	Physical Time Scale
CFD-Post	Plots	Slice Plane
		Sphere Volume
	Other	Viewing the Mesh

4.2. Overview of the Problem to Solve

In this tutorial, you use a refined mesh to obtain a better solution to the Static Mixer problem created in Simulating Flow in a Static Mixer Using CFX in Stand-alone Mode (p. 39). You establish a general workflow for analyzing the flow of fluid into and out of a mixer. This tutorial uses a specific problem to teach the general approach taken when working with an existing mesh.



You start a new simulation in CFX-Pre and import the refined mesh. This tutorial introduces General mode (the mode used for most tutorials) in CFX-Pre. The physics for this tutorial are the same as for Simulating Flow in a Static Mixer Using CFX in Stand-alone Mode (p. 39); therefore, you can import the physics settings used in that tutorial to save time.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.3. Preparing the Working Directory

1. Create a working directory.

   Ansys CFX uses a working directory as the default location for loading and saving files for a particular session or project.

2. Download the `static_mixer_refined_mesh.zip` file here.

3. Unzip `static_mixer_refined_mesh.zip` to your working directory.

   Ensure that the following tutorial input files are in your working directory:

   - `StaticMixerRefMesh.gtm`
   - `StaticMixer.def` (produced by following tutorial Simulating Flow in a Static Mixer Using CFX in Stand-alone Mode (p. 39))
   - `StaticMixer_001.res` (produced by following tutorial Simulating Flow in a Static Mixer Using CFX in Stand-alone Mode (p. 39))

4. Set the working directory and start CFX-Pre.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

4.4. Defining the Case Using CFX-Pre

The tutorial follows this general workflow for setting up the case in CFX-Pre:

4.4.1. Importing a Mesh
4.4.2. Importing Settings from Tutorial 1
4.4.3. Viewing Domain Settings
4.4.4. Viewing the Boundary Condition Setting
4.4.5. Defining Solver Parameters
4.4.6. Writing the CFX-Solver Input (.def) File

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** in the **New Case** dialog box and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `StaticMixerRef` and click **Save**.

4.4.1. Importing a Mesh

At least one mesh must be imported before physics are applied.

1. Select **File** > **Import** > **Mesh**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Import Mesh** dialog box appears.

2. Configure the following setting(s):

Setting	Value
Files of type	CFX Mesh (*gtm *cfx)
File name	StaticMixerRefMesh.gtm [a]

a. This mesh is more refined than the one used in the other "Static Mixer" tutorials (`StaticMixerMesh.gtm`).

3. Click **Open**.

The **Mesh** tree shows the regions in the `StaticMixerRefMesh.gtm` assembly in a tree structure. The first tree branch displays the 3D regions and the level below each 3D region shows the 2D regions associated with it. The check box next to each item in the `Mesh` tree indicates the visibility status of the object in the viewer; you can click these to toggle visibility.

Note:

An *assembly* is a group of mesh regions that are topologically connected. Each assembly can contain only one mesh, but multiple assemblies are permitted.

4. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)** from the shortcut menu.

4.4.2. Importing Settings from Tutorial 1

Because the physics and region names for this simulation are very similar to that for Tutorial 1, you can save time by importing the settings used there. You will be importing CCL from Tutorial 1 that contains settings that reference mesh regions. For example, the outlet boundary condition references the mesh region named `out`. In this tutorial, the name of the mesh regions are the same as in Tutorial 1, so you can import the CCL without error.

1. Select **File** > **Import** > **CCL**.

The **Import CCL** dialog box appears.

2. Under **Import Method**, select **Replace**.

Replace is useful if you have defined physics and want to update or replace them with newly imported physics.

3. Under **Files of type**, select `CFX-Solver input files (*def *res)`.

4. Select `StaticMixer.def`.

5. Click **Open**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. View the **Outline** tab.

   The tree view displays a summary of the current simulation in a tree structure. Some items may be recognized from Tutorial 1; for example the boundary condition objects `in1`, `in2`, and `out`.

Note:

- If you import CCL that references nonexistent mesh regions, you will get errors.
- A number of file types can be used as sources to import CCL, including:
  - Simulation files (`*.cfx`)
  - Results files (`*.res`)
  - CFX-Solver input files (`*.def`)
- The physics for a simulation can be saved to a CCL file at any time by selecting **File** > **Export** > **CCL**.

4.4.3. Viewing Domain Settings

It is useful to review the options available in General mode.

Various domain settings can be set. These include:

- **Basic Settings**

  Specifies the location of the domain, coordinate frame settings and the fluids/solids that are present in the domain. You also reference pressure, buoyancy and whether the domain is stationary or rotating. Mesh motion can also be set.

- **Fluid Models**

  Sets models that apply to the fluid(s) in the domain, such as heat transfer, turbulence, combustion, and radiation models. An option absent in Tutorial 1 is **Turbulent Wall Functions**, which is set to **Scalable**. Wall functions model the flow in the near-wall region. For the k-epsilon turbulence model, you should always use scalable wall functions.

- **Initialization**

  Sets the initial conditions for the current domain only. This is generally used when multiple domains exist to enable setting different initial conditions in each domain, but can also be used to initialize single-domain simulations. Global initialization allows the specification of initial conditions for all domains that do not have domain-specific initialization.

1. In the **Outline** tree view, under `Simulation` > `Flow Analysis 1`, double-click `Default Domain`.

   The domain `Default Domain` is opened for editing.

2. Click the **Basic Settings** tab and review, but do not change, the current settings.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click **Fluid Models** and review, but do not change, the current settings.
4. Click **Initialization** and review, but do not change, the current settings.
5. Click **Close**.

4.4.4. Viewing the Boundary Condition Setting

For the k-epsilon turbulence model, you must specify the turbulent nature of the flow entering through the inlet boundary. For this simulation, the default setting of `Medium (Intensity = 5%)` is used. This is a sensible setting if you do not know the turbulence properties of the incoming flow.

1. Under `Default Domain`, double-click `in1`.
2. Click the **Boundary Details** tab and review the settings for **Flow Regime**, **Mass and Momentum**, **Turbulence** and **Heat Transfer**.
3. Click **Close**.

4.4.5. Defining Solver Parameters

Solver Control parameters control aspects of the numerical-solution generation process.

In Tutorial 1 you set some solver control parameters, such as **Advection Scheme** and **Timescale Control**, while other parameters were set automatically by CFX-Pre.

In this tutorial, **High Resolution** is used for the advection scheme. This is more accurate than the **Upwind Scheme** used in Tutorial 1. You usually require a smaller time step when using this model. You can also expect the solution to take a higher number of iterations to converge when using this model.

1. Select **Insert** > **Solver** > **Solver Control** from the menu bar or click *Solver Control* .
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Advection Scheme > **Option**	High Resolution
	Convergence Control > **Max. Iterations** [a]	150
	Convergence Control > Fluid Timescale Control > **Timescale Control**	Physical Timescale
	Convergence Control > Fluid Timescale Control	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Physical Timescale**	0.5 [s]
a. If your solution does not meet the convergence criteria after this number of time steps, the CFX-Solver will stop.		

3. Click **Apply**.
4. Click the **Advanced Options** tab.

Tip:

To select **Advanced Options** you may need to click the navigation icons next to the tabs to move 'forward' or 'backward' through the various tabs.

5. Ensure that **Global Dynamic Model Control** is selected.
6. Click **OK**.

4.4.6. Writing the CFX-Solver Input (.def) File

Once all boundaries are created you move from CFX-Pre into CFX-Solver.

The simulation file, `StaticMixerRef.cfx`, contains the simulation definition in a format that can be loaded by CFX-Pre, enabling you to complete (if applicable), restore, and modify the simulation definition. The simulation file differs from the CFX-Solver input file in two important ways:

- The simulation file can be saved at any time while defining the simulation.
- The CFX-Solver input file is an encapsulated set of meshes and CCL defining a solver run, and is a subset of the data in the simulation file.

1. Click *Define Run* .

   The **Write Solver Input File** dialog box is displayed.

2. If required, set the path to your working directory.
3. Configure the following setting(s):

Setting	Value
File name	StaticMixerRef.def

4. Click **Save**.

   The CFX-Solver input file (`StaticMixerRef.def`) is created. CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, **Solver Input File** is set.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. If you are notified in CFX-Pre that the file already exists, click **Overwrite**.

6. Quit CFX-Pre, saving the simulation (`.cfx`) file.

4.5. Obtaining the Solution Using CFX-Solver Manager

Two windows are displayed when CFX-Solver Manager runs. There is an adjustable split between the windows that is oriented either horizontally or vertically, depending on the aspect ratio of the entire CFX-Solver Manager window (also adjustable).

The tutorial follows this general workflow for generating the solution in CFX-Solver Manager:

4.5.1. Starting the Run with an Initial Values File

4.5.2. Confirming Results

4.5.3. Moving from CFX-Solver Manager to CFD-Post

4.5.1. Starting the Run with an Initial Values File

In CFX-Solver Manager, the **Define Run** dialog box is visible and **Solver Input File** has automatically been set to the CFX-Solver input file from CFX-Pre: `StaticMixerRef.def`.

Configure the settings so that the results from Tutorial 1 (contained in `StaticMixer_001.res`) will be used to initialize the solution:

1. Under the **Initial Values** tab, select **Initial Values Specification**.

2. Set **Initial Values Specification** > **Initial Values** > **Initial Values 1** > **Initial Values 1 Settings** > **File Name** to `StaticMixer_001.res`.

   If you did not complete the first tutorial, you can use `StaticMixer_001.res` from your working directory.

3. Set **Use Mesh From** to `Solver Input File`.

4. Clear **Initial Values Specification** > **Continue History From**.

   This will cause the CFX-Solver to use the results in the `StaticMixer_001.res` file as a basic initial guess for the solution, and will cause the iteration count to start from 1 instead of from the last iteration number in the initial values file.

5. Click **Start Run**.

   Note:

   Convergence information is plotted once the second outer loop iteration is complete.

4.5.2. Confirming Results

When the run is finished, specific information appears in the text window of CFX-Solver Manager.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To confirm that results interpolation was successful, look in the text window in CFX-Solver Manager. The following text appears before the convergence history begins:

```
+--------------------------------------------------------+
| Initial Conditions Supplied by Fields in the Input Files |
+--------------------------------------------------------+
```

This lists the variables that were interpolated from the results file.

After the final iteration, a message similar to the following content appears in a message window:

```
StaticMixerRef_001 has completed normally.
Run concluded at: Fri Nov 27 11:57:57 2009
```

This indicates that CFX-Solver has successfully calculated the solution for the problem to the specified accuracy or has run out of coefficient loops.

1. In the **Solver Run Finished Normally** window, ensure that the check box next to **Post-Process Results** is cleared to prevent CFD-Post from launching at this time.
2. Click **OK**.
3. Review the CFX-Solver Manager's **Out File** tab for details on the run results.

4.5.3. Moving from CFX-Solver Manager to CFD-Post

Once CFX-Solver has finished, you can use CFD-Post to review the finished results.

1. On the CFX-Solver Manager, select **Tools** > **Post-Process Results** or click *Post-Process Results* in the toolbar.
2. In the **Start CFD-Post** dialog box, next to **Results File**, ensure that `StaticMixerRef_001.res` is set. If it is not, click *Browse* and select `StaticMixerRef_001.res` (located in the working directory).
3. If using CFX-Solver in stand-alone mode, select **Shut down CFX-Solver Manager**.

   This forces stand-alone CFX-Solver to close. This option is not required in Workbench.
4. Click **OK**.

   After a short pause, CFD-Post starts.

4.6. Viewing the Results Using CFD-Post

In the following sections, you will explore the differences between the mesh and the results from this tutorial and Tutorial 1.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.6.1. Creating a Slice Plane

More information exists for use by CFD-Post in this tutorial than in Tutorial 1 because the slice plane is more detailed.

Once a new slice plane is created it can be compared with Tutorial 1. There are three noticeable differences between the two slice planes.

- Around the edges of the mixer geometry there are several layers of narrow rectangles. This is the region where the mesh contains prismatic elements (which are created as inflation layers). The bulk of the geometry contains tetrahedral elements.
- There are more lines on the plane than there were in Tutorial 1. This is because the slice plane intersects with more mesh elements.
- The curves of the mixer are smoother than in Tutorial 1 because the finer mesh better represents the true geometry.

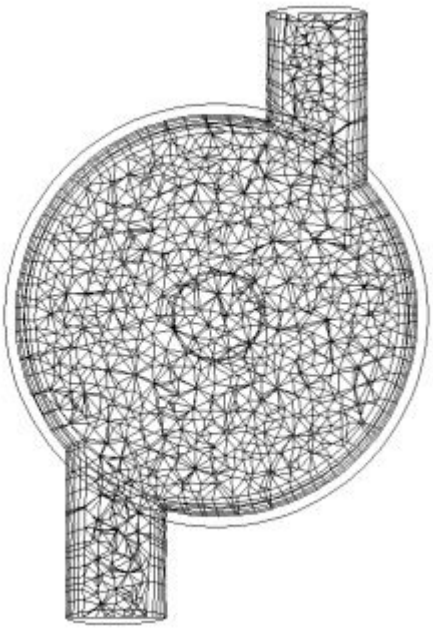
1. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)**.
2. From the menu bar, select **Insert** > **Location** > **Plane** or under **Location**, click **Plane**.
3. In the **Insert Plane** dialog box, type `Slice` and click **OK**.

   The **Geometry**, **Color**, **Render** and **View** tabs enable you to switch between settings.
4. Configure the following setting(s):

Tab	Setting	Value
Geometry	**Domains**	Default Domain
	Definition > **Method**	XY Plane
	Definition > **Z**	1 [m]
	Plane Type	Slice
Render	**Show Faces**	(Cleared)
	Show Mesh Lines	(Selected)

5. Click **Apply**.
6. Right-click a blank area in the viewer and select **Predefined Camera** > **View From +Z**.
7. If necessary, click *Zoom Box* and zoom in on the geometry to view it in greater detail.
8. Compare the on-screen image with the equivalent picture from Simulating Flow in a Static Mixer Using CFX in Stand-alone Mode (p. 39) (in the section Rendering Slice Planes (p. 60)).

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4.6.2. Coloring the Slice Plane

Here, you will color the plane by temperature.

1. Configure the following setting(s):

Tab	Setting	Value
Color	**Mode** [a]	Variable
	Variable	Temperature
	Range	Global
Render	**Show Faces**	(Selected)
	Show Mesh Lines	(Cleared)

a. A mode setting of `Constant` would enable you to color the plane with a fixed color.

2. Click **Apply**.

4.6.3. Loading Results from Tutorial 1 for Comparison

In CFD-Post, you may load multiple results files into the same instance for comparison.

1. To load the results file from Tutorial 1, select **File** > **Load Results** or click *Load Results* .

2. (Do not click **Open** until instructed to do so.) In the **Load Results File** dialog box, select `StaticMixer_001.res` from your working directory.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. On the right side of the dialog box, there are three frames:

   - **Case options**
   - **Additional actions**
   - **CFX run history and multi-configuration options**.

   Under **Case options**, select **Keep current cases loaded** and ensure that **Open in new view** is selected.

4. Under **Additional actions**, ensure that the **Clear user state before loading** check box is cleared.

5. Under **CFX run history and multi-configuration options**, ensure that **Load only the last results** is selected.

6. Click **Open** to load the results.

   In the tree view, there is now a second group of domains, meshes and boundary conditions with the heading `StaticMixer_001`.

   In the **3D Viewer**, there are two viewports named **View 1** and **View 2**; the former shows Static-Mixer_001 and the latter shows StaticMixerRef_001.

7. Double-click the `Wireframe` object under `User Locations and Plots`.

8. In the **Definition** tab, set **Edge Angle** to `5 [degree]`.

9. Click **Apply**.

10. Click *Synchronize camera in displayed views* so that all viewports maintain the same camera position.

11. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)**.

    Both meshes are now displayed in a line along the Y axis. Notice that one mesh is of a higher resolution than the other.

12. Set **Edge Angle** to `30 [degree]`.

13. Click **Apply**.

4.6.4. Comparing Slice Planes Using Multiple Views

The visibility status of each object is maintained separately for each viewport. This allows some planes to be shown while others are hidden. However, you can also use the *Synchronize visibility in displayed views* to synchronize the visibility of objects that you add.

1. Under **User Locations and Plots**, select the check box beside `Slice`.

2. Right-click in the viewer and select **Predefined Camera** > **View From -Z**.

   Note the difference in temperature distribution.

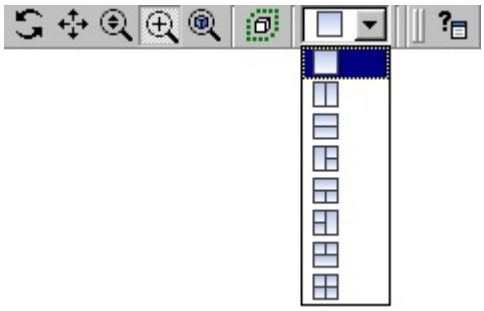
Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. In the viewer toolbar, click *Synchronize visibility in displayed views* to deselect the option.

4. Click in **View 1** in the **3D Viewer**, then clear the visibility check box for `Slice` in the **Outline** tree view.

5. Click in **View 2**. Note that the visibility check box for `Slice` has been re-selected as it describes the state of the plane for **View 2**. Clear the visibility check box for `Slice` in this view.

6. To return to a single viewport, select the option with a single rectangle.



7. Ensure that the visibility check box for `Slice` is cleared.

8. Right-click `StaticMixer_001` in the tree view and select **Unload**.

4.6.5. Viewing the Surface Mesh on the Outlet

In this part of the tutorial, you will view the mesh on the outlet. You will see five layers of inflated elements against the wall. You will also see the triangular faces of the tetrahedral elements closer to the center of the outlet.

1. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)**.

2. In the tree view, ensure that the visibility check box for `StaticMixerRef_001` > `Default Domain` > `out` is selected, then double-click `out` to open it for editing.

   Because the boundary location geometry was defined in CFX-Pre, the details view does not display a **Geometry** tab as it did for the planes.

3. Configure the following setting(s):

Tab	Setting	Value
Render	**Show Faces**	(Cleared)
	Show Mesh Lines	(Selected)
	Color Mode	User Specified
	Line Color	(Select any light color)

4. Click **Apply**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Click *Zoom Box* .
6. Zoom in on the geometry to view `out` in greater detail.
7. Click *Rotate* on the **Viewing Tools** toolbar.
8. Rotate the image as required to clearly see the mesh.

4.6.6. Looking at the Inflated Elements in Three Dimensions

To show more clearly what effect inflation has on the shape of the elements, you will use volume objects to show two individual elements. The first element that will be shown is a normal tetrahedral element; the second is a prismatic element from an inflation layer of the mesh.

Leave the surface mesh on the outlet visible to help see how surface and volume meshes are related.

1. From the menu bar, select **Insert** > **Location** > **Volume** or, under **Location** click **Volume**.
2. In the **Insert Volume** dialog box, type `Tet Volume` and click **OK**.
3. Configure the following setting(s):

Tab	Setting	Value
Geometry	Definition > **Method**	Sphere
	Definition > **Point** [a]	0.08, 0, -2
	Definition > **Radius**	0.14 [m]
	Definition > **Mode**	Below Intersection
	Inclusive [b]	(Cleared)
Color	**Color**	Red
Render	Show Faces > **Transparency**	0.3
	Show Mesh Lines	(Selected)
	Show Mesh Lines > **Line Width**	1
	Show Mesh Lines > **Color Mode**	User Specified

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Show Mesh Lines > **Line Color**	 Grey

a. The z slider's minimum value corresponds to the minimum z value of the entire geometry, which, in this case, occurs at the outlet.

b. Only elements that are entirely contained within the sphere volume will be included.

4. Click **Apply** to create the volume object.
5. Right-click `Tet Volume` and choose **Duplicate**.
6. In the **Duplicate Tet Volume** dialog box, type `Prism Volume` and click **OK**.
7. Double-click `Prism Volume`.
8. Configure the following setting(s):

Tab	Setting	Value
Geometry	Definition > **Point**	 -0.22, 0.4, -1.85
	Definition > **Radius**	 0.206 [m]
Color	**Color**	Orange

9. Click **Apply**.

4.6.7. Viewing the Surface Mesh on the Mixer Body

1. Double-click the `Default Domain Default` object.
2. Configure the following setting(s):

Tab	Setting	Value
Render	**Show Faces**	(Selected)
	Show Mesh Lines	(Selected)
	Line Width	2

3. Click **Apply**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.6.8. Viewing the Layers of Inflated Elements on a Plane

You will see the layers of inflated elements on the wall of the main body of the mixer. Within the body of the mixer, there will be many lines that are drawn wherever the face of a mesh element intersects the slice plane.

1. From the menu bar, select **Insert** > **Location** > **Plane** or under **Location**, click **Plane**.
2. In the **Insert Plane** dialog box, type `Slice 2` and click **OK**.
3. Configure the following setting(s):

Tab	Setting	Value
Geometry	Definition > **Method**	YZ Plane
	Definition > **X**	0 [m]
Render	**Show Faces**	(Cleared)
	Show Mesh Lines	(Selected)

4. Click **Apply**.
5. Turn off the visibility of all objects except `Slice 2`.
6. To see the plane clearly, right-click in the viewer and select **Predefined Camera** > **View From -X**.

4.6.9. Viewing the Mesh Statistics

You can use the **Report Viewer** to check the quality of your mesh. For example, you can load a `.def` file into CFD-Post and check the mesh quality before running the `.def` file in the solver.

1. Click the **Report Viewer** tab (located below the viewer window).

   A report appears. Look at the table shown in the "Mesh Report" section.
2. Double-click `Report` > `Mesh Report` in the **Outline** tree view.
3. In the **Mesh Report** details view, select **Statistics** > **Maximum Face Angle**.
4. Click **Refresh Preview**.

   Note that a new table, showing the maximum face angle for all elements in the mesh, has been added to the "Mesh Report" section of the report. The maximum face angle is reported as 148.95°.

As a result of generating this mesh statistic for the report, a new variable, `Maximum Face Angle`, has been created and stored at every node. This variable will be used in the next section.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.6.10. Viewing the Mesh Elements with Largest Face Angle

In this section, you will visualize the mesh elements that have a `Maximum Face Angle` value greater than 140°.

1. Click the **3D Viewer** tab (located below the viewer window).
2. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)**.
3. In the **Outline** tree view, select the visibility check box of `Wireframe`.
4. From the menu bar, select **Insert** > **Location** > **Volume** or under **Location**, click **Volume**.
5. In the **Insert Volume** dialog box, type `Max Face Angle Volume` and click **OK**.
6. Configure the following setting(s):

Tab	Setting	Value
Geometry	Definition > **Method**	Isovolume
	Definition > **Variable**	Maximum Face Angle [a]
	Definition > **Mode**	Above Value
	Definition > **Value**	140 [degree]
	Inclusive [b]	(Selected)

a. Select `Maximum Face Angle` from the larger list of variables available by clicking ... to the right of the **Variable** box.

b. This includes any elements that have at least one node with a variable value greater than or equal to the given value.

7. Click **Apply**.

   The volume object appears in the viewer.

4.6.11. Viewing the Mesh Elements with Largest Face Angle Using a Point

Next, you will create a point object to show a node that has the maximum value of `Maximum Face Angle`. The point object will be represented by a 3D yellow crosshair symbol. In order to avoid obscuring the point object with the volume object, you may want to turn off the visibility of the latter.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. From the menu bar, select **Insert** > **Location** > **Point** or under **Location**, click **Point**.
2. Click **OK** to use the default name.
3. Configure the following setting(s):

Tab	Setting	Value
Geometry	Definition > **Method**	Variable Maximum
	Definition > **Location**	Default Domain
	Definition > **Variable**	Maximum Face Angle
Symbol	**Symbol Size**	2

4. Click **Apply**.

4.6.12. Quitting CFD-Post

1. When you are finished, select **File** > **Quit** to exit CFD-Post.
2. If prompted by a dialog box, save the state at your discretion.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 5: Optimizing Flow in a Static Mixer

DesignXplorer is a Workbench component that you can use to examine the effect of changing parameters in a system. In this tutorial, two streams of water at different temperatures are mixed in a static mixer. You will see how changing the geometry and inlet flow rate affects the mixing effectiveness, as measured by the range of temperature over the outlet.

This tutorial includes:

5.1. Tutorial Features

5.2. Overview of the Problem to Solve

5.3. Setting Up Ansys Workbench

5.4. Creating the Project

5.5. Creating the Geometry in DesignModeler

5.6. Creating the Mesh

5.7. Setting up the Case with CFX-Pre

5.8. Setting the Output Parameter in CFD-Post

5.9. Investigating the Impact of Changing Design Parameters Manually

5.10. Using Design of Experiments

5.11. Viewing the Response Surface

5.12. Optimization based on the Response Surface

Note:

Some of the instructions in this tutorial assume that you have sufficient licensing to have multiple applications open. If you do not have sufficient licensing, you may not be able to keep as many of the applications open as this tutorial suggests. In this case, simply close the applications as you finish with them.

5.1. Tutorial Features

In this tutorial you will learn about:

- Creating a geometry in DesignModeler and creating a mesh.
- Using General mode in CFX-Pre to set up a problem.
- Using design points to manually vary characteristics of the problem to see how you can improve the mixing.
- Using DesignXplorer to vary characteristics of the problem programmatically to find an optimal design.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Component	Feature	Details
DesignModeler		Geometry Creation
		Named Selections
Meshing Application		Mesh Creation
CFX-Pre	User Mode	General mode
	Analysis Type	Steady State
	Fluid Type	General Fluid
	Domain Type	Single Domain
	Turbulence Model	k-Epsilon
	Heat Transfer	Thermal Energy
	Boundary Conditions	Inlet (Subsonic)
		Outlet (Subsonic)
		Wall: No-Slip
		Wall: Adiabatic
	Timescale	Physical Timescale
	Expressions	Workbench input parameter
CFD-Post	Expressions	Workbench output parameter
Parameters	Design Points	Manual changes
DesignXplorer	Response Surface Optimization	Design of Experiments
		Response Surface
		Optimization

5.2. Overview of the Problem to Solve

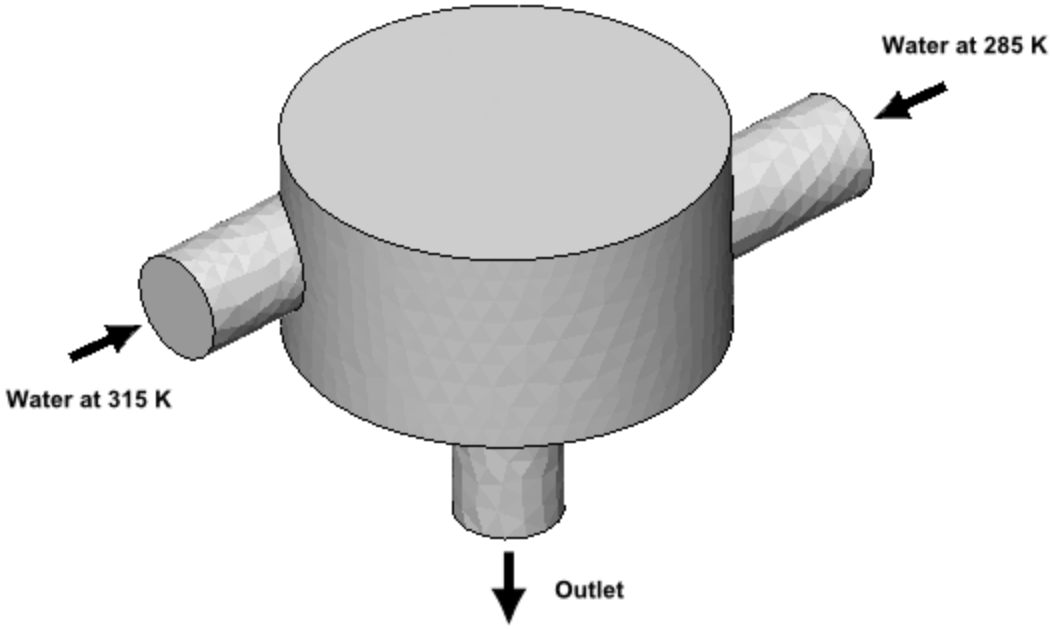
This tutorial simulates a static mixer consisting of two inlet pipes delivering water into a mixing vessel; the water exits through an outlet pipe. A general workflow is established for analyzing the flow of fluid into and out of a mixer.

Initially, water enters through both pipes at the same rate but at different temperatures. The first inlet has an initial mass flow rate of 1500 kg/s and a temperature of 315 K. The second inlet has an initial mass flow rate of 1500 kg/s and a temperature of 285 K. The radius of the mixer is 2 m.

Your goal in this tutorial is to understand how to use Design Points and DesignXplorer to maximize the amount of mixing, as measured by the range of water temperature at the outlet.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.1: Static Mixer with 2 Inlet Pipes and 1 Outlet Pipe



5.3. Setting Up Ansys Workbench

Before you begin using Ansys Workbench, you have to configure the **Geometry Import** option settings for use with this tutorial:

1. Launch Ansys Workbench.
2. From the Ansys Workbench menu bar, select **Tools** > **Options**.

   The **Options** dialog box appears.
3. In the **Options** dialog box, select **Geometry Import**.
4. Ensure that **Parameters** is set to **Independent**.
5. Remove "DS" (and the unneeded delimiter, ";") from **Filtering Prefixes and Suffixes**.
6. Select **Named Selections**.
7. Remove "NS" from **Filtering Prefixes**.
8. Click **OK**.

5.4. Creating the Project

To create the project, you save an empty project:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. From the Ansys Workbench menu bar, select **File** > **Save As** and save the project as `StaticMixerDX.wbpj` in the directory of your choice.

2. From **Toolbox** > **Analysis Systems**, drag the **Fluid Flow (CFX)** system onto the **Project Schematic**.

5.5. Creating the Geometry in DesignModeler

Now you can create a geometry by using DesignModeler:

1. In the **Fluid Flow (CFX)** system, right-click **Geometry** and select **New DesignModeler Geometry**.

   DesignModeler starts.

2. If DesignModeler displays a dialog box for selecting the required length unit, select **Meter** as the required length unit and click **OK**.

   Note that this dialog box will not appear if you have previously set a default unit of measurement.

5.5.1. Creating the Solid

You create geometry in DesignModeler by creating two-dimensional sketches and extruding, revolving, sweeping, or lofting these to add or remove material. To create the main body of the static mixer, you will draw a sketch of a cross-section and revolve it.

1. In the **Tree Outline**, click **ZXPlane**.

   Each sketch is created in a plane. By selecting **ZXPlane** before creating a sketch, you ensure that the sketch you are about to create is based on the ZX plane.

2. Click *New Sketch* on the Active Plane/Sketch toolbar, which is located above the Graphics window.

3. In the **Tree Outline**, click **Sketch1**.

4. Select the **Sketching** tab (below the **Tree Outline**) to view the available sketching toolboxes.

5.5.1.1. Setting Up the Grid

Before starting your sketch, set up a grid on the plane in which you will draw the sketch. The grid facilitates the precise positioning of points (when **Snap** is selected).

1. Click **Settings** (in the **Sketching** tab) to open the **Settings** toolbox.

2. Click **Grid** and select **Show in 2D** and **Snap**.

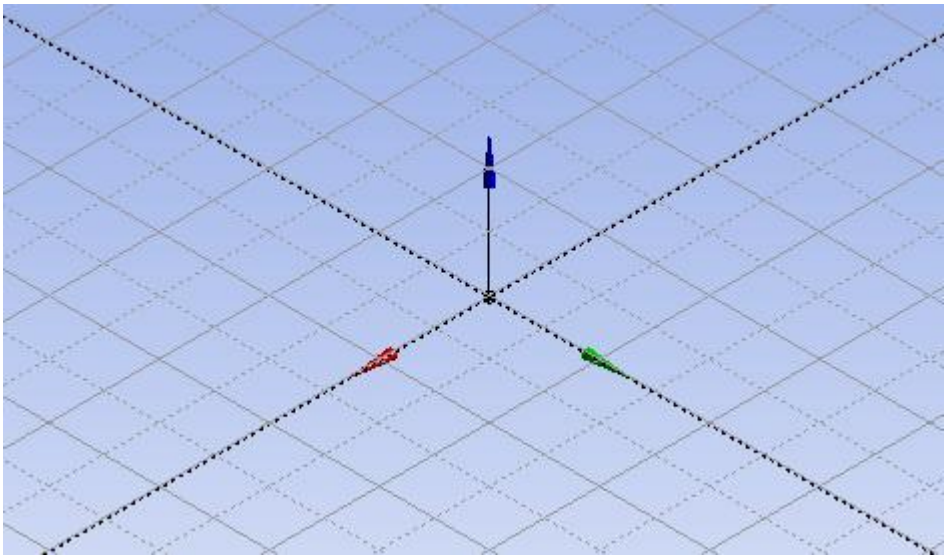
3. Click **Major Grid Spacing** and set it to 1.

4. Click **Minor-Steps per Major** and set it to 2.

5. To see the effect of changing **Minor-Steps per Major**, press and hold the right mouse button above and to the left of the plane center in the Graphics window, then drag the mouse down and right to make a box around the plane center.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you release the mouse button, the model is magnified to show the selected area.



You now have a grid of squares with the smallest squares being 50 cm across. Because snap is selected, you can select only points that are on this grid to build your geometry. Using snap can help you to position objects correctly.

The triad at the center of the grid indicates the local coordinate frame. The color of the arrow indicates the local axis: red for X, green for Y, and blue for Z.

5.5.1.2. Creating the Basic Geometry

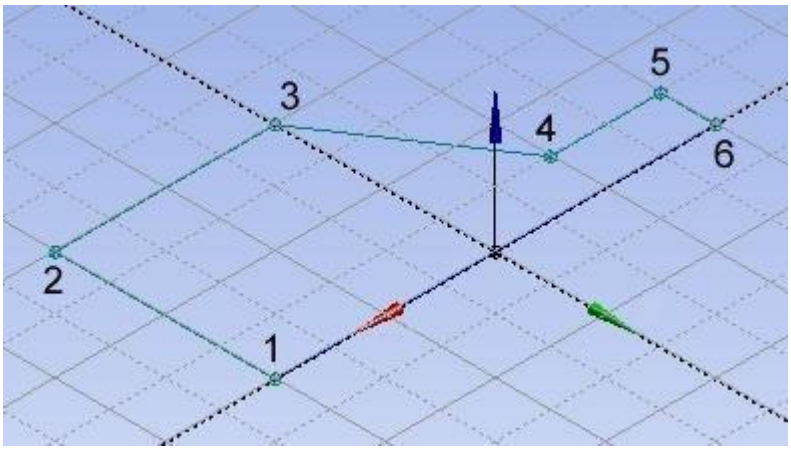
Start by creating the main body of the mixer:

1. From the **Sketching** tab, select the **Draw** toolbox.
2. Click **Polyline** and then create the shape shown below as follows:
   a. Click the grid in the position where one of the points from the shape must be placed (it does not matter which point, but a suggested order is given in the graphic below).
   b. Click each successive point to make the shape.

      If at any time you click the wrong place, right-click over the Graphics window and select **Back** from the shortcut menu to undo the last point selection.
   c. To close the polyline after selecting the last point, right-click and select **Closed End** from the shortcut menu.

Information about the new sketch, **Sketch1**, appears in the details view. Note that the longest straight line (4 m long) in the diagram below is along the local X axis (located at Y = 0 m). The numbers and letters in the image below are added here for your convenience but do not appear in the software.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5.5.1.3. Revolving the Sketch

You will now create the main body of the mixer by revolving the new sketch around the local X axis.

1. Click *Revolve* from the toolbar above the Graphics window.

   Details of the Revolve feature are shown in the details view at the bottom left of the window.

2. Leave the name of the Revolve feature at its default value: `Revolve1`.

3. Click **Apply** in the details view under **Details of Revolve1** > **Geometry** to select `Sketch1` as the geometry to be revolved.

   The **Geometry** specifies which sketch is to be revolved.

4. In the **Details View** you should see **Apply** and **Cancel** buttons next to the **Axis** property; if those buttons are not displayed, double-click the word **Axis**.

5. In the Graphics window, click the grid line that is aligned with the local X axis (the local X axis, represented by a red arrow, is parallel to the global Z axis in this case), then click **Apply** in the **Details View**.

6. Leave **Operation** set to **Add Material** because you need to create a solid (which will eventually represent a fluid region).

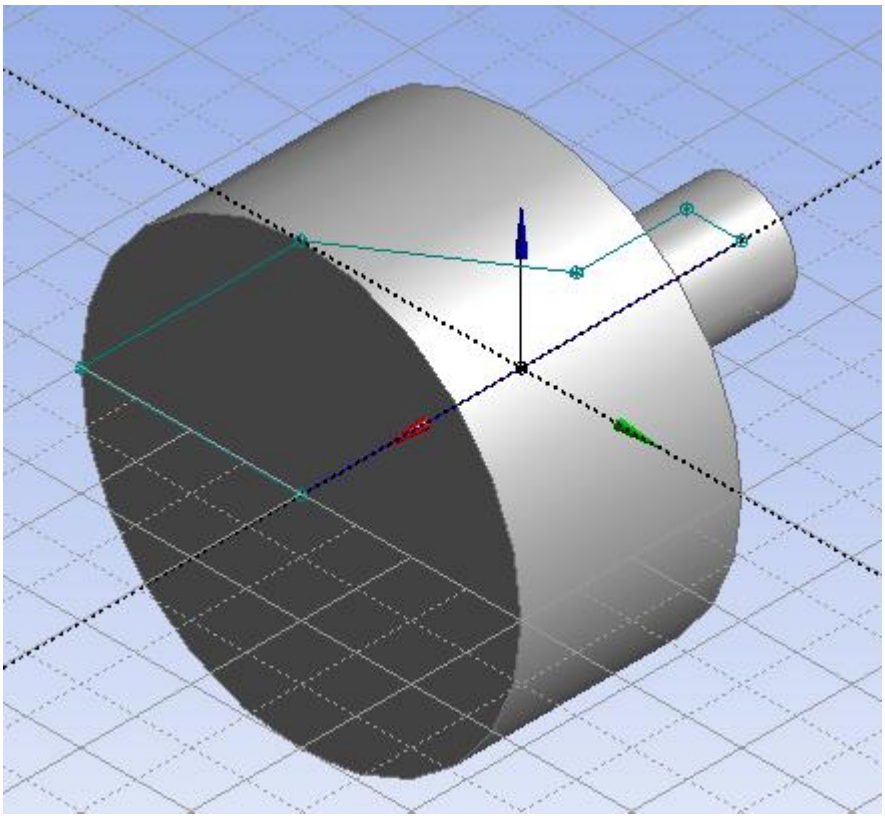
7. Ensure that **Angle** is set to **360°** and leave the other settings at their defaults.

8. Click *Generate* to activate the Revolve operation.

   You can select this from the 3D Features Toolbar, from the shortcut menu by right-clicking in the Graphics window, or from the shortcut menu by right-clicking the **Revolve1** object in the **Tree Outline**.

After generation, you should find that you have a solid as shown below.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

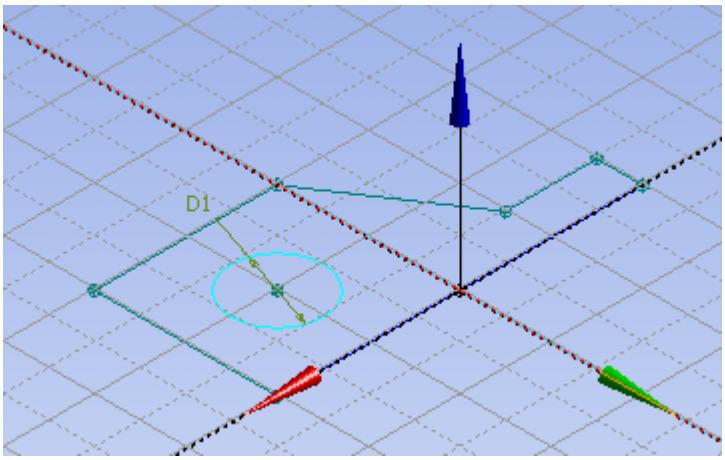


5.5.1.4. Create the First Inlet Pipe

To create the inlet pipes, you will create two sketches and extrude them. For clear viewing of the grid during sketching, you will hide the previously created geometry.

1. In the **Tree Outline**, click the plus sign next to **1 Part, 1 Body** to expand the tree structure.
2. Right-click **Solid** and select **Hide Body**.
3. Select **ZXPlane** in the **Tree Outline**.
4. Click *New Sketch* .
5. From the **Sketching** tab, select the **Draw** toolbox.
6. Click **Circle** and then create the circle shown below as follows:
   a. Click and hold the left mouse button at the center of the circle.
   b. While still holding the mouse button, drag the mouse to set the radius.
   c. Release the mouse button.
7. Select the **Dimensions** toolbox, select **General**, click the circle in the sketch, then click near the circle to set a dimension. In the Details View, select the check box beside **D1**. When prompted, rename the parameter to `inDia` and click **OK**. This dimension will be a parameter that is modified in DesignXplorer.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5.5.1.4.1. Extrude the First Side Pipe

To create the first side pipe, extrude the sketch:

1. Click *Extrude* from the 3D Features toolbar, located above the Graphics window.
2. In the **Details View**, change **Direction** to **Reversed** to reverse the direction of the extrusion (that is, click the word **Normal**, then from the drop-down menu select **Reversed**).
3. Change **Depth** to **3** (meters) and press **Enter** to set this value.

   All other settings should remain at their default values. Note that the **Operation** property is set to **Add Material**, which indicates that material is to be added to the existing solid.
4. Click *Generate* to perform the extrusion.

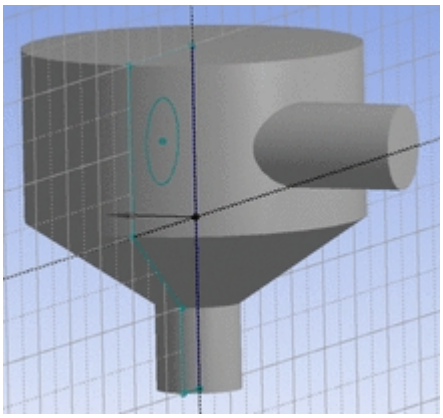
   Initially, you will not see the geometry.

5.5.1.4.2. Make the Solid Visible

To see the result of the previous operation, make the solid visible:

1. In the **Tree Outline**, right-click **Solid** and select **Show Body**.
2. Click and hold the middle mouse button over the middle of the Graphics window and drag the mouse to rotate the model. The solid should be similar to the one shown below.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Right-click **Solid** and select **Hide Body**.

5.5.1.5. Create the Second Inlet Pipe

You will create the second inlet so that the relative angle between the two inlets is controlled parametrically, enabling you to evaluate the effects of different relative inlet angles:

1. In the **Tree Outline**, select **ZXPlane**.
2. In the toolbar, click *New Plane* .

   The new plane (**Plane4**) appears in the **Tree Outline**.
3. In the **Details View**, click beside **Transform 1 (RMB)** and choose the axis about which you want to rotate the inlet: **Rotate about X**.
4. Select the check box for the **FD1, Value 1** property then, when prompted, set the name to `in2Angle` and click **OK**.

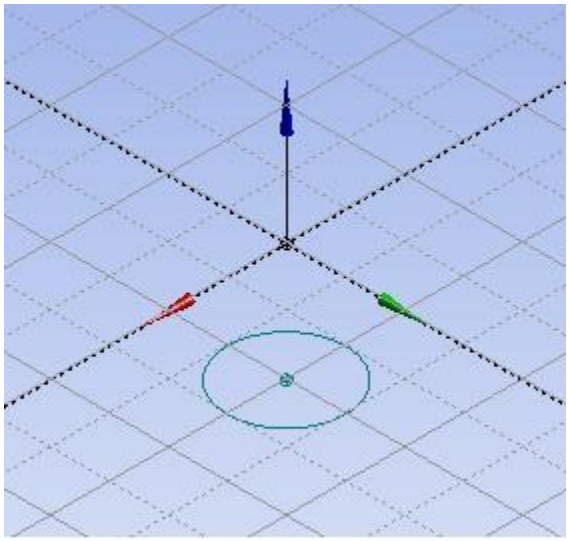
   This makes the angle of rotation of this plane a design parameter.
5. Click *Generate* .
6. In the **Tree Outline** click **Plane4**.
7. Create a new sketch (**Sketch3**) based on **Plane4** by clicking *New Sketch* .
8. Select the **Sketching** tab.
9. Click **Settings** to open the **Settings** toolbox.
10. Click **Grid** and select **Show in 2D** and **Snap**.
11. Click **Major Grid Spacing** and set it to **1**.
12. Click **Minor-Steps per Major** and set it to **2**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13. Right-click over the Graphics window and select **Isometric View** to put the sketch into a sensible viewing position.

14. Zoom in, if required, to see the level of detail in the image below.

15. From the **Draw** Toolbox, select **Circle** and create a circle as shown below:



16. Select the **Dimensions** toolbox, click **General**, click the circle in the sketch, then click near the circle to set a dimension.

17. In the **Details View**, select the check box for the **D1** property then, when prompted, set the name to `inDia` and click **OK**.

18. Click *Extrude* .

19. In the **Details View**, ensure that **Direction** is set to **Normal** in order to extrude in the same direction as the plane normal.

20. Ensure that **Depth** is set to **3** (meters).

    All other settings should remain at their default values.

21. Click *Generate* to perform the extrusion.

22. Right-click **Solid** in the **Tree Outline** and select **Show Body**.

    The geometry is now complete.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.5.1.6. Create Named Selections

Named selections enable you to specify and control like-grouped items. Here, you will create named selections so that you can specify boundary conditions in CFX-Pre for these specific regions.

Note:

The Graphics window must be in "viewing mode" for you to be able to orient the geometry, and must be in "select mode" for you to be able to select a boundary in the geometry. You set viewing mode or select mode by clicking the icons in the toolbar:

Create named selections as follows:

1. In viewing mode, orient the static mixer so that you can see the inlet that has the lowest value of (global) Y-coordinate.

   You can rotate the mixer by holding down the middle-mouse button (or the mouse scroll wheel) while moving the mouse.

2. In select mode, with *Selection Filter: Faces* active (in the toolbar), click the inlet face to select it, then right-click the inlet and select **Named Selection**.

3. In the **Details View**, click **Apply**.

4. Set **Named Selection** to `in1`.

5. Click *Generate* .

6. In viewing mode, orient the static mixer so that you can see the inlet that has the highest value of (global) Y-coordinate.

7. In select mode, click the inlet face to select it, then right-click the inlet and select **Named Selection**.

8. In the **Details View**, click **Apply**.

9. Set **Named Selection** to `in2`.

10. Click *Generate* .

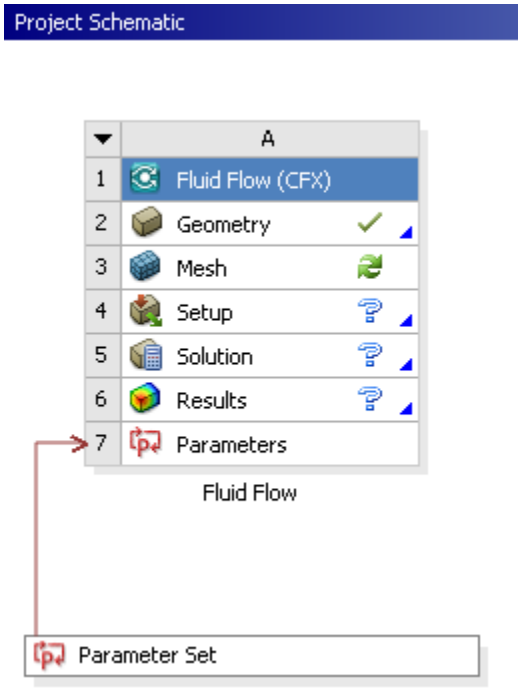
11. In viewing mode, orient the static mixer so that you can see the outlet (the face with the lowest value of (global) Z-coordinate).

12. In select mode, click the outlet face to select it, then right-click the outlet and select **Named Selection**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13. In the **Details View**, click **Apply**.
14. Set **Named Selection** to `out`.
15. Click *Generate* .
16. Click **Save** on the Ansys Workbench toolbar.

    This enables you to recover the work that you have performed to this point if needed (until the next time you save the tutorial).



5.6. Creating the Mesh

To create the mesh:

1. In the **Project Schematic** view, right-click the **Mesh** cell and select **Edit**.

   The Meshing application appears.
2. Right-click **Project** > **Model (A3)** > **Mesh** and select **Generate Mesh**.
3. After the mesh has been produced, return to the **Project Schematic**, right-click the **Mesh** cell, and select **Update**.
4. In the Meshing application, select **File** > **Close Meshing**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.7. Setting up the Case with CFX-Pre

Now that the mesh has been created, you can use CFX-Pre to define the simulation. To set up the case with CFX-Pre:

1. Double-click the **Setup** cell.

   CFX-Pre appears with the mesh file loaded.

2. In CFX-Pre, create an expression named `inMassFlow` as follows:

   a. In the **Outline** tree view, expand **Expressions, Functions and Variables**, right-click **Expressions**, and select **Insert** > **Expression**.

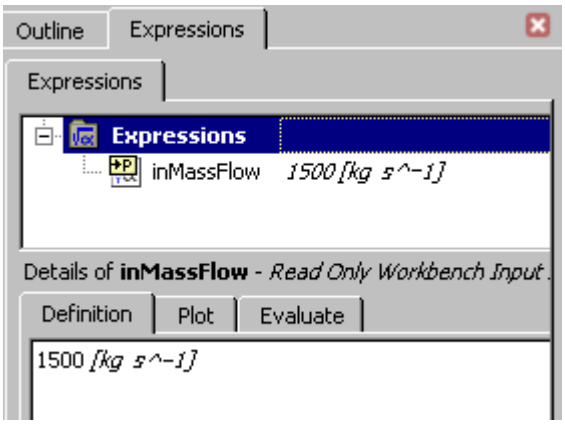
   b. Give the new expression the name: `inMassFlow`

   c. In the **Definition** area, type: `1500 [kg s^-1]`

   d. Click **Apply**.

3. Right-click **inMassFlow** in the **Expressions** area and select **Use as Workbench Input Parameter**.

   A small "P" with a right-pointing arrow appears on the expression's icon.



4. Define the characteristics of the domain:

   a. Click the **Outline** tab.

   b. Double-click **Simulation** > **Flow Analysis 1** > **Default Domain** to open it for editing.

   c. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Fluid and Particle Definitions**	Fluid 1
	Fluid and Particle Definitions > Fluid 1	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Material**	Water
	Domain Models > Pressure > **Reference Pressure**	 1 [atm]
Fluids Models	Heat Transfer > **Option**	 Thermal Energy

d. Click **OK**.

5. Create the first inlet boundary:

   a. From the CFX-Pre menu bar, select **Insert** > **Boundary**.

   b. In the **Insert Boundary** dialog box, name the new boundary `in1` and click **OK**.

   c. Configure the following setting(s) of `in1`:

Tab	Setting	Value
Basic Settings	**Boundary Type**	Inlet
	Location	in1
Boundary Details	Mass and Momentum > **Option**	 Mass Flow Rate
	Mass Flow Rate	inMassFlow [a]
	Heat Transfer > **Static Temperature**	 315 [K]

a. To enter this expression name into the Mass Flow Rate field, click in the blank field, click the *Enter Expression* icon that appears, right-click in the blank field, then select the inMassFlow expression that appears.

   d. Click **OK**.

6. Create the second inlet boundary:

   a. From the CFX-Pre menu bar, select **Insert** > **Boundary**.

   b. In the **Insert Boundary** dialog box, name the new boundary `in2` and click **OK**.

   c. Configure the following setting(s) of `in2`:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Basic Settings	**Boundary Type**	Inlet
	Location	in2
Boundary Details	Mass and Momentum > **Option**	Mass Flow Rate
	Mass Flow Rate	inMassFlow [a]
	Heat Transfer > **Static Temperature**	285 [K]

a. To enter this expression name into the Mass Flow Rate field, click in the blank field, click the *Enter Expression* icon that appears, right-click in the blank field, then select the inMassFlow expression that appears.

d. Click **OK**.

7. Create the outlet boundary:

   a. From the CFX-Pre menu bar, select **Insert** > **Boundary**.

   b. In the **Insert Boundary** dialog box, name the new boundary `out` and click **OK**.

   c. Configure the following setting(s) of `out`:

Tab	Setting	Value
Basic Settings	**Boundary Type**	Outlet
	Location	out
Boundary Details	Mass and Momentum > **Option**	Static Pressure
	Mass and Momentum > **Relative Pressure**	0 [Pa]

   d. Click **OK**.

   CFX-Pre and Ansys Workbench both update automatically. The three boundary conditions are displayed in the viewer as sets of arrows at the boundary surfaces. Inlet boundary arrows are directed into the domain; outlet boundary arrows are directed out of the domain.

8. Solver Control parameters control aspects of the numerical solution generation process. Set the solver controls as follows:

   a. Click *Solver Control* .

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. On the **Basic Settings** tab, set **Advection Scheme** > **Option** to `Upwind`.

While an upwind advection scheme is less accurate than other advection schemes, it is also more robust. This advection scheme is suitable for obtaining an initial set of results, but in general should not be used to obtain final results.

c. Set **Convergence Control** > **Min. Iterations** to 5.

This change is required because when the solver is restarted from a previous "converged" solution for each design point, the solver may "think" the solution is converged after one or two iterations and halt the solution prematurely if the default setting (1) is maintained.

d. Set **Convergence Control** > **Fluid Timescale Control** > **Timescale Control** to `Physical Timescale` and set the physical timescale value to 2 [s].

The time scale can be calculated automatically by the solver or set manually. The `Automatic` option tends to be conservative, leading to reliable, but often slow, convergence. It is often possible to accelerate convergence by applying a time scale factor or by choosing a manual value that is more aggressive than the `Automatic` option. By selecting a physical time scale, you obtain convergence that is at least twice as fast as with the `Automatic` option.

e. Click **OK**.

CFX-Pre and Ansys Workbench both update automatically.

9. In the **Project Schematic** view, right-click the **Solution** cell and select **Update**.

CFX-Solver obtains a solution.

10. When the **Solution** cell shows an up-to-date state, right-click the **Results** cell and select **Refresh**.

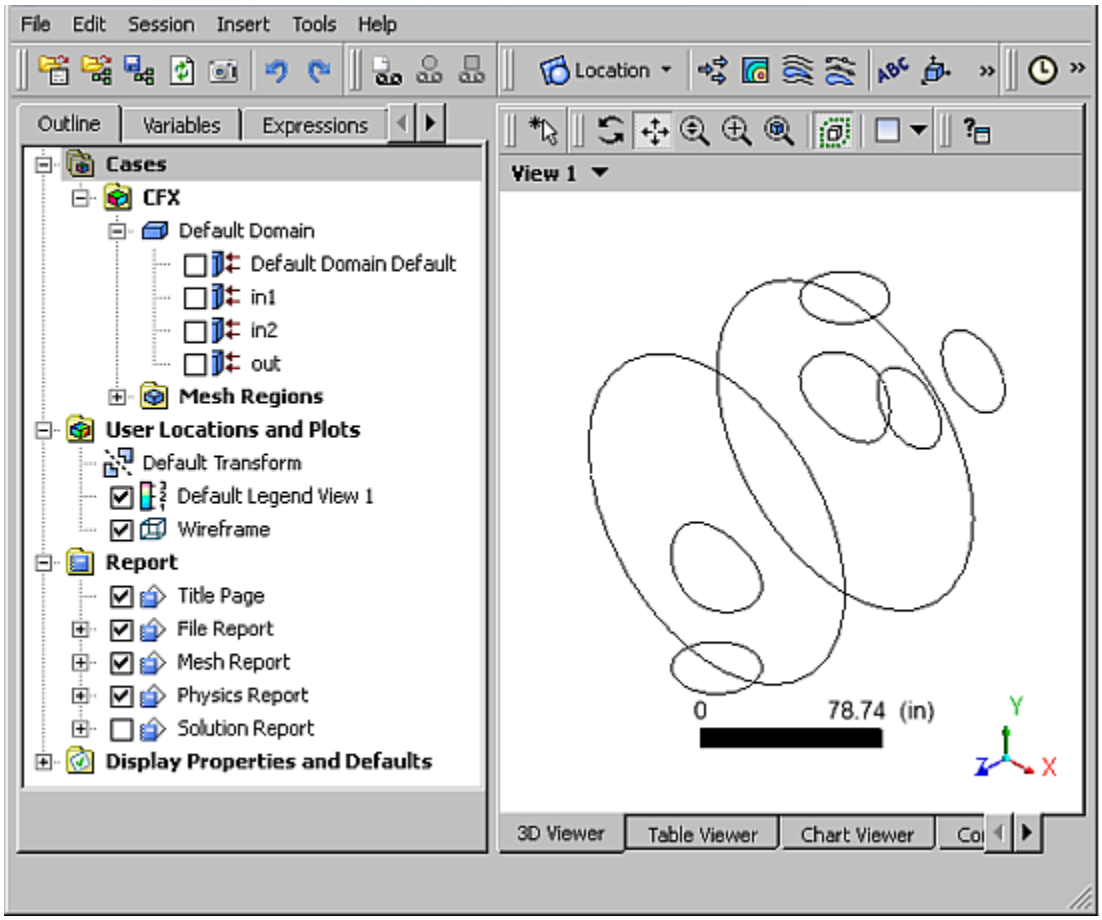
11. When the refresh is complete, right-click the **Results** cell again and select **Edit**.

CFD-Post starts.

5.8. Setting the Output Parameter in CFD-Post

When CFD-Post starts, it displays the **Outline** workspace and the **3D Viewer**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You need to create an expression for the response parameter to be examined (Outlet Temperature) called `OutTempRange`, which will be the maximum output temperature minus the minimum output temperature:

1. On the **Expressions** tab, right-click in the **Expressions** tree view and select **New**.
2. Type `OutTempRange` and click **OK**.
3. In the **Definition** area:
   a. Right-click and select **Functions** > **CFD-Post** > **maxVal**.
   b. With the cursor between the parentheses, right-click and select **Variables** > **Temperature**.
   c. Click after the @, then right-click and select **Locations** > **out**.
   d. Continue to edit the expression until it appears as follows:

```
maxVal(Temperature)@out - minVal(Temperature)@out
```

   e. Click **Apply**.

   The new expression appears in the **Expressions** list. Note the value of the expression.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. In the **Expressions** list, right-click `OutTempRange` and select **Use as Workbench Output Parameter**.

   A small "P" with a right-pointing arrow appears on the expression's icon.

5. Repeat the steps above for a second expression called `OutTempAve`, defined by:

   ```
   massFlowAve(Temperature)@out
   ```

   Be sure to also set this expression to **Use as Workbench Output Parameter**. When you click **Apply**, note the value of the expression.

   This expression will be used to monitor the output temperature. The overall output temperature is expected to be the average of the two input temperatures, given that the incoming mass flows are equal.

6. Click **Save** on the Ansys Workbench toolbar to save the project.

7. In CFD-Post, select **File** > **Close CFD-Post**.

5.9. Investigating the Impact of Changing Design Parameters Manually

Now you will manually change the values of some design parameters to see what effect each has on the rate of mixing. These combinations of parameter values where you perform calculations are called *design points*. As you make changes to parameters in Ansys Workbench, Ansys CFX-Pre and Ansys DesignModeler will reflect the updated values automatically. If you leave CFX-Pre and DesignModeler open, you can see the changes taking place. In particular, you can see changes in the **Expressions** view of CFX-Pre.

1. In the **Project Schematic** view, right-click **Parameters** (cell A7) and select **Edit**.

   A **Parameters** tab appears.

2. Resize the Ansys Workbench window to be larger.

   Tip:

   If necessary, you can close the **Toolbox** view to gain more space. To restore it, select **View** > **Toolbox**.

When you edit the **Parameters** cell (cell A7), the new views that appear on the **Parameters** tab are:

- **Outline of Schematic A7: Parameters**, which lists the input and output parameters and their values (which match the values observed in previous steps).
- **Table of Design Points**, which lists one design point named **Current**. Ensure that this view is wide enough to display the **Retain** column.

Now you will change the design parameter values from the **Outline of Schematic A7: Parameters** view:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. On the **Parameters** tab, in the **Outline of Schematic A7: Parameters** view, change the **in2Angle** value from `0` to `-45` and press **Enter**.

2. Switch to the **Project** tab.

3. In the **Project Schematic** view, right-click **Geometry** and select **Update**.

   Notice how the geometry changes in Ansys DesignModeler.

4. In Ansys Workbench, switch to the **Parameters** tab, then in the **Outline of Schematic A7: Parameters** view, change the **in2Angle** value from `-45` back to `0` and press **Enter**.

5. In the **Project Schematic** view, update **Geometry**.

   Again, notice how the geometry changes in Ansys DesignModeler.

6. Switch to the **Parameters** tab, then in the **Outline of Schematic A7: Parameters** view, change the **inMassFlow** value from `1500` to `1600` and press **Enter**.

   Notice how the value of the expression has changed in the **Expressions** tree view in CFX-Pre.

7. On the **Parameters** tab, in the **Outline of Schematic A7: Parameters** view, change the **inMassFlow** value from `1600` back to `1500` and press **Enter**.

You have modified design parameter values and returned each to its original value. In doing this, the **Outline of Schematic A7: Parameters** values and the **Table of Design Points** values have become out-of date.

Update the current design point as follows:

- Right-click any cell in the **Current** row of **Table of Design Points** and select **Update Selected Design Points**.

  This process updates the project and all of the Ansys Workbench views. Ansys Workbench may also close any open Ansys CFX applications and run them in the background. When the update is complete, all of the results cells show current values and all of the cells that display their status are marked as being up-to-date.

Now, you will create three additional design points, **DP 1**, **DP 2**, and **DP 3**:

1. On the **Parameters** tab, in the **Table of Design Points** view in the line under **Current**, make the following entries to create design point **DP 1**:

   - **P1 - inDia**: `1`
   - **P2 - in2Angle**: `-45`
   - **P3 - inMassFlow**: `1500`
   - **Retain**: `(Selected)`

   Note:

   You should save the project once before you retain a design point.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Right-click in the row for **DP 1** and select **Update Selected Design Points**.

Ansys Workbench recalculates all of the values for the input and output parameters. All of the views are updated.

2. Create design point **DP 2** using these values:

   - **P1 - inDia**: `1.5`
   - **P2 - in2Angle**: `0`
   - **P3 - inMassFlow**: `1500`
   - **Retain**: `(Selected)`

   Right-click in the row for **DP 2** and select **Update Selected Design Points**.

3. Create design point **DP 3** using these values:

   - **P1 - inDia**: `1`
   - **P2 - in2Angle**: `0`
   - **P3 - inMassFlow**: `1600`
   - **Retain**: `(Selected)`

   In the Ansys Workbench toolbar, click **Update All Design Points**.

   This command updates any out-of-date design points in a sequential fashion.

4. On the Ansys Workbench toolbar, click **Save** to save the project.

Recall that the goal of this design study is to maximize the amount of mixing by finding the minimum value of `OutTempRange`. Note that `OutTempRange` varies significantly due to changes in the input parameters. Also note that, as expected, `OutTempAve` stays very near `300 K`.

In the next section you will automate the process of changing input parameters by using **Design Exploration**.

5.10. Using Design of Experiments

In this section you will use **Design Exploration**'s Response Surface Optimization feature to minimize the value of `OutTempRange`.

1. Return to the **Project** tab.

2. If you need to restore the **Toolbox**, select **View** > **Toolbox**. If no systems appear in the **Toolbox**, select a cell in the **Project Schematic** view to refresh Ansys Workbench.

3. From the **Design Exploration** toolbox, drag a **Response Surface Optimization** system to the **Project Schematic** (under the **Parameter Set** bar).

4. **Edit** the **Design of Experiments** cell.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. On the **B2:Design of Experiments** tab, in the **Outline of Schematic B2: Design of Experiments** view:

   a. Select the **Enabled** check boxes next to **P2 - in2Angle** (cell A6) and **P3 - inMassFlow** (cell A7), and clear the check box next to **P1 - inDia** (cell A5).

   b. Select cell **P3 - inMassFlow** (cell A7) then, in the **Properties of Outline A7: P3 - inMassFlow** view, set:

      - **Lower Bound:** 1000
      - **Upper Bound:** 2000

   c. Select cell **P2 - in2Angle** (cell A6) then, in the **Properties of Outline A6: P2 - in2Angle** view, set:

      - **Lower Bound:** -45
      - **Upper Bound:** 0

6. In the Ansys Workbench menu bar, ensure that **View** > **Table** is selected.

7. On the **B2:Design of Experiments** tab, on the toolbar, click **Preview**.

8. The table of design points has nine entries in the **Name** column, each of which represents a solver run to be performed. Beside the **Name** column are columns that have the values for the two input parameters, and a column to hold the value the solver will obtain for the output parameter.

9. In the **Project Schematic** view, update the **Design of Experiments** cell. You can monitor the progress of the solver runs by clicking **Show Progress** in the lower-right corner of the Ansys Workbench window.

   When the processing is complete, the table of design points (on the **B2:Design of Experiments** tab) displays the results. Click the down-arrow on the **P4 - OutTempRange (K)** cell to sort the design points according to the value of `OutTempRange`. Note that, in this case and for the range of inputs considered, the best results are returned from the lowest mass flow rate and an input angle of zero.

5.11. Viewing the Response Surface

To view the response surface for this experiment:

1. In the Ansys Workbench menu bar, ensure that **View** > **Chart** is selected.

2. In the **Project Schematic** view, update the **Response Surface**.

3. Right-click **Response Surface** and select **Edit** to open the **B3:Response Surface** tab.

4. In the **Outline of Schematic B3: Response Surface** view, select **Response Surface** > **Response Points** > **Response Point** > **Response**. The results appear in various views:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The **Toolbox** shows the types of charts that are available.

  If the Toolbox is not visible, you can show it by selecting **View** > **Toolbox**.

- **Response Chart for P4 - OutTempRange** shows a 2D graph comparing `OutTempRange` to `in2Angle`.

- The **Properties of Outline A20: Response** shows the values that are being used to display the 2D graph. Note that the other variable that was considered in the Design of Experiments, `in MassFlow`, is held at `1500`.

5. In the **Properties of Outline A20: Response** view, select **Chart** > **Mode** > **3D**. A 3D chart appears, showing the full range of **inMassFlow** results. The chart shows that, in this case and for the range of inputs considered, the best results are returned from the lowest mass flow rate and an input angle of zero.

5.12. Optimization based on the Response Surface

To view the optimization for this experiment:

1. In the **Project Schematic** view, edit the **Optimization** cell.
2. On the **B4:Optimization** tab, in the **Outline of Schematic B4: Optimization** view, select **Objectives and Constraints**.
3. In the **Table of Schematic B4: Optimization** view, add the following objective:

   - **Parameter**: `P4 - OutTempRange`
   - **Type**: `Minimize`

   The **Objective Name** is automatically set to `Minimize P4`.
4. In the **Project Schematic** view, right-click **Optimization** and select **Update**.
5. On the **B4:Optimization** tab, in **Outline of Schematic B4: Optimization**, select **Optimization** > **Results** > **Candidate Points**.

   In the **Table of Schematic B4: Optimization, Candidate Points** view, three design point candidates appear with their interpolated values.

   Recall that design points are combinations of parameter values where you perform real calculations (rather than relying on the interpolated values that appear in the response charts). All of the candidates are represented graphically in the Candidate Points chart.
6. In **Table of Schematic B4: Optimization, Candidate Points**, right-click **Candidate Point 1** and select **Insert as Design Point**.
7. Switch to the **Parameters** tab.

   The various views update and a new design point has appeared in **Table of Design Points**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. In **Table of Design Points**, the new design point, **DP 4** has no values and requests an update. Right-click the lightning icon and select **Update Selected Design Points**.

   You can follow the progress of the update in the **Project Schematic** view while Ansys Workbench reruns its calculations for the design point parameters.

   Note that the resulting value of `OutTempRange` for **DP 4** in **Table of Design Points** is the result of a solver calculation, and differs slightly from the value of `OutTempRange` interpolated from the response surface for **Candidate Point 1**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 6: Flow in a Process Injection Mixing Pipe

This tutorial includes:

6.1. Tutorial Features

6.2. Overview of the Problem to Solve

6.3. Preparing the Working Directory

6.4. Defining the Case Using CFX-Pre

6.5. Obtaining the Solution Using CFX-Solver Manager

6.6. Viewing the Results Using CFD-Post

6.1. Tutorial Features

In this tutorial you will learn about:

- Applying a profile boundary using data stored in a file.
- Visualizing the velocity on a boundary in CFX-Pre.
- Using the CFX Expression Language (CEL) to describe temperature-dependent fluid properties in CFX-Pre.
- Using the k-epsilon turbulence model.
- Using streamlines in CFD-Post to track flow through the domain.

Component	Feature	Details
CFX-Pre	User Mode	General mode
	Analysis Type	Steady State
	Fluid Type	General Fluid
	Domain Type	Single Domain
	Turbulence Model	k-Epsilon
	Heat Transfer	Thermal Energy
	Boundary Conditions	Boundary Profile Visualization
		Inlet (Profile)
		Inlet (Subsonic)
		Outlet (Subsonic)
		Wall: No-Slip
		Wall: Adiabatic

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Component	Feature	Details
	CEL (CFX Expression Language)	
	Timestep	Physical Time Scale
CFD-Post	Plots	Default Locators
		Outline Plot (Wireframe)
		Slice Plane
		Streamline
	Other	Changing the Color Range
		Expression Details View
		Legend
		Viewing the Mesh

6.2. Overview of the Problem to Solve

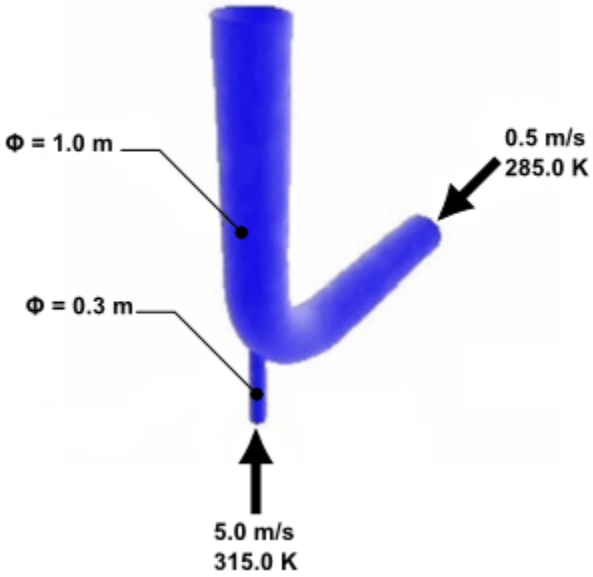
The goal of this tutorial is to understand the general approach taken when working with an existing mesh. In this tutorial, you will go through the three main steps when solving a problem, which are defining a simulation using General mode in CFX-Pre, obtaining a solution using CFX-Solver Manager and viewing the results in CFD-Post.

The injection mixing pipe, common in the process industry, is composed of two pipes: one with a larger diameter than the other. Analyzing and optimizing the mixing process is often critical for many chemical processes. CFD is useful not only in identifying problem areas (where mixing is poor), but also in testing new designs before they are implemented.

The geometry for this example consists of a circular pipe of diameter 1.0 m with a 90° bend, and a smaller pipe of diameter 0.3 m which joins with the main pipe at an oblique angle. Water at 315.0 K enters in the 0.3 m diameter pipe at a rate of 5.0 m/s while water at 285.0 K enters in the 1.0 m diameter pipe at a rate of 0.5 m/s.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.1: Injection Mixing Pipe



In this tutorial, you will establish a general workflow for analyzing the flow of the water fluid into and out of an injection pipe. First, a simulation will be created and an existing mesh will be imported in CFX-Pre. A viscosity expression will also be created, and will be used to modify the water properties later on in this tutorial to increase the solution accuracy. Finally, initial values will be set and a solution will be found using CFX-Solver Manager. The results will then be viewed in CFD-Post. Streamlines originating from the main inlet will be generated to show the flow of the water into and out of the injection pipe.

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- Running Ansys CFX Tutorials Using Ansys Workbench (p. 34)
- Changing the Display Colors (p. 37)

6.3. Preparing the Working Directory

1. Create a working directory.

   Ansys CFX uses a working directory as the default location for loading and saving files for a particular session or project.

2. Download the `inject_mixer.zip` file here.

3. Unzip `inject_mixer.zip` to your working directory.

   Ensure that the following tutorial input files are in your working directory:

   - InjectMixerMesh.gtm
   - InjectMixer_velocity_profile.csv

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Set the working directory and start CFX-Pre.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

6.4. Defining the Case Using CFX-Pre

The tutorial follows this general workflow for setting up the case in CFX-Pre:

6.4.1. Importing a Mesh
6.4.2. Setting Temperature-Dependent Material Properties
6.4.3. Plotting an Expression
6.4.4. Evaluating an Expression
6.4.5. Modify Material Properties
6.4.6. Creating the Domain
6.4.7. Creating the Side Inlet Boundary
6.4.8. Creating the Main Inlet Boundary
6.4.9. Creating the Main Outlet Boundary
6.4.10. Setting Initial Values
6.4.11. Setting Solver Control
6.4.12. Writing the CFX-Solver Input (.def) File

1. In CFX-Pre, select **File** > **New Case**.
2. Ensure that **General** is selected and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `InjectMixer`.
5. Click **Save**.

6.4.1. Importing a Mesh

The following steps will demonstrate how to import a mesh.

1. Right-click `Mesh` and select **Import Mesh** > **CFX Mesh**.

   The **Import Mesh** dialog box appears.

2. Configure the following setting(s):

Setting	Value
File name	InjectMixerMesh.gtm

3. Click **Open**.
4. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Y up)** from the shortcut menu.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.4.2. Setting Temperature-Dependent Material Properties

Viscosity varies with temperature, which implies that the water will behave differently when coming through the 1.0 m and the 0.3 m diameter pipes. In the following steps, you will create an expression for viscosity as a function of temperature. This expression will then be used to modify the properties of the library material: `Water` to increase the accuracy of the solution. By setting temperature-dependent material properties, Viscosity will be made to vary linearly with temperature between the following conditions:

- μ=1.8E-03 N s m^{-2} at T=275.0 K
- μ=5.45E-04 N s m^{-2} at T=325.0 K

The variable T (Temperature) is a CFX System Variable recognized by CFX-Pre, denoting static temperature. All variables, expressions, locators, functions, and constants can be viewed by double-clicking the appropriate entry (such as `Additional Variables` or `Expressions`) in the tree view.

All expressions must have consistent units. You should be careful if using temperature in an expression with units other than `[K]`.

The **Expressions** tab lets you define, modify, evaluate, plot, copy, delete and browse through expressions used within CFX-Pre.

1. From the main menu, select **Insert** > **Expressions, Functions and Variables** > **Expression**.
2. In the **New Expression** dialog box, type `Tupper`.
3. Click **OK**.

   The details view for the `Tupper` equation is displayed.
4. Under **Definition**, type `325 [K]`.
5. Click **Apply** to create the expression.

   The expression is added to the list of existing expressions.
6. Ensure that no expression is highlighted, then right-click in the **Expressions** workspace and select **Insert** > **Expression**.
7. In the **New Expression** dialog box, type `Tlower`.
8. Click **OK**.
9. Under **Definition**, type `275 [K]`.
10. Click **Apply** to create the expression.

    The expression is added to the list of existing expressions.
11. Create expressions for `Visupper`, `Vislower` and `VisT` using the following values.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition
Visupper	5.45E-04 [N s m^-2]
Vislower	1.8E-03 [N s m^-2]
VisT	Vislower+(Visupper-Vislower)*(T-Tlower)/(Tupper-Tlower)

6.4.3. Plotting an Expression

1. Right-click `VisT` in the **Expressions** tree view, and then select **Edit**.

   The **Expressions** details view for `VisT` appears.

   Tip:

   Alternatively, double-clicking the expression also opens the **Expressions** details view.

2. Click the **Plot** tab and configure the following setting(s):

Tab	Setting	Value
Plot	**Number of Points**	10
	T	(Selected)
	Start of Range	275 [K]
	End of Range	325 [K]

3. Click **Plot Expression**.

   A plot showing the variation of the expression `VisT` with the variable `T` is displayed.

6.4.4. Evaluating an Expression

1. Click the **Evaluate** tab.
2. In `T`, type `300 [K]`.

   This is between the start and end range defined in the last module.

3. Click **Evaluate Expression**.

   A value of around 0.0011[kg m^-1 s^-1] for `VisT` at the given value of `T` appears in the **Value** field.

6.4.5. Modify Material Properties

As mentioned earlier in this tutorial, the default material properties of `Water` will be modified using the Viscosity expression to increase the accuracy of the solution.

1. Click the **Outline** tab.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Double-click `Water` under `Materials` to display the **Basic Settings** tab.
3. Click the **Material Properties** tab.
4. Expand **Transport Properties**.
5. Select **Dynamic Viscosity**.
6. Under **Dynamic Viscosity**, click in **Dynamic Viscosity**.
7. Click *Enter Expression* .
8. Enter the expression `VisT` into the data box.
9. Click **OK**.

6.4.6. Creating the Domain

The domain will be set to use the thermal energy heat transfer model, and the $k-\varepsilon$ (k-epsilon) turbulence model.

Both **Basic Settings** and **Fluid Models** are changed in this module. The **Initialization** tab is for setting domain-specific initial conditions, which are not used in this tutorial. Instead, global initialization is used to set the starting conditions.

1. Select **Insert** > **Domain** from the main menu or click *Domain* .
2. In the **Insert Domain** dialog box, type `InjectMixer`.
3. Click **OK**.
4. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Location and Type > **Location**	 B1.P3
	Fluid and Particle Definitions	Fluid 1
	Fluid and Particle Definitions > Fluid 1 > **Material**	 Water
	Domain Models > Pressure > **Reference Pressure**	 0 [atm]

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Click **Fluid Models**.
6. Configure the following setting(s):

Setting	Value
Heat Transfer > **Option**	Thermal Energy

7. Click **OK**.

6.4.7. Creating the Side Inlet Boundary

The side inlet boundary must be defined.

1. Select **Insert** > **Boundary** from the main menu or click *Boundary* .
2. Set **Name** to `side inlet`.

> **Note:**
>
> A boundary named after a region will use that region as its location by default.

3. Click **OK**.
4. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Inlet
	Location	side inlet
Boundary Details	Mass and Momentum > **Option**	Normal Speed
	Mass and Momentum > **Normal Speed**	5 [m s^-1]
	Heat Transfer > **Option**	Static Temperature
	Heat Transfer > **Static Temperature**	315 [K]

5. Click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.4.8. Creating the Main Inlet Boundary

The main inlet boundary for the large pipe must be defined. This inlet is defined using a velocity profile. Profile data must be initialized before the boundary can be created.

You will create a plot showing the velocity profile data, marked by higher velocities near the center of the inlet, and lower velocities near the inlet walls.

1. Select **Tools** > **Initialize Profile Data**.

   The **Initialize Profile Data** dialog box appears.

2. Beside **Profile Data File**, click *Browse* .

   The **Select Profile Data File** dialog box appears.

3. From your working directory, select `InjectMixer_velocity_profile.csv`.

4. Click **Open**.

5. Click **OK**.

   The profile data is read into memory.

   Note:

   After profile data has been initialized from a file, the profile data file should not be deleted or otherwise removed from its directory. By default, the full file path to the profile data file is stored in CFX-Pre, and the profile data file is read directly by CFX-Solver each time the solver is started or restarted.

6. Select **Insert** > **Boundary** from the main menu or click *Boundary* .

7. Set name **Name** to `main inlet`.

8. Click **OK**.

9. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Inlet
	Location	main inlet
	Profile Boundary Conditions > **Use Profile Data**	 (Selected)
	Profile Boundary Setup	 main inlet

Tab	Setting	Value
	> **Profile Name**	

10. Click **Generate Values**.

    This causes the profile values of U, V, W to be applied at the nodes on the main inlet boundary, and U, V, W entries to be made in **Boundary Details**. In order to later reset the velocity values at the main inlet to match those that were originally read from the boundary condition profile file, revisit **Basic Settings** for this boundary and click **Generate Values**.

11. Configure the following setting(s):

Tab	Setting	Value
Boundary Details	Flow Regime > **Option**	Subsonic
	Mass And Momentum > **Option**	Cart. Vel. Components
	Turbulence > **Option**	Medium (Intensity = 5%)
	Heat Transfer > **Option**	Static Temperature
	Heat Transfer > **Static Temperature**	285 [K]
Plot Options	**Boundary Contour**	(Selected)
	Boundary Contour > **Profile Variable**	W

12. Click **OK**.

13. Zoom into the main inlet to view the inlet velocity contour.

6.4.9. Creating the Main Outlet Boundary

In this module you create the outlet boundary. All other surfaces that have not been explicitly assigned a boundary will remain in the `InjectMixer Default` object, which is shown in the tree view. This boundary uses a `No-Slip Adiabatic Wall` by default.

1. Select **Insert** > **Boundary** from the main menu or click *Boundary* .

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Set **Name** to `outlet`.
3. Click **OK**.
4. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Outlet
	Location	outlet
Boundary Details	Flow Regime > **Option**	 Subsonic
	Mass and Momentum > **Option**	 Average Static Pressure
	Relative Pressure	0 [Pa]

5. Click **OK**.

6.4.10. Setting Initial Values

For this tutorial, the initial values will be set automatically. An automatic guess is sufficient for this particular problem.

1. Click *Global Initialization* and review, but do not change, the current settings.
2. Click **Close**.

6.4.11. Setting Solver Control

1. Click *Solver Control*.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Advection Scheme > **Option**	 High Resolution
	Convergence Control > **Max. Iterations**	 50
	Convergence Control	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Fluid Timescale Control > **Timescale Control**	Physical Timescale
	Convergence Control > Fluid Timescale Control > **Physical Timescale**	2 [s] [a]
	Convergence Criteria > **Residual Type**	RMS
	Convergence Criteria > **Residual Target**	1.E-4 [b]

a. The physical timescale that will be set up is derived from the pipe diameter (1 m) and the rate at which the water flows in the pipe (0.5 m/s).

b. An RMS value of at least 1.E-5 is usually required for adequate convergence, but the default value is sufficient for demonstration purposes.

3. Click **OK**.

6.4.12. Writing the CFX-Solver Input (.def) File

Once the problem has been defined you move from General mode into CFX-Solver.

1. Click *Define Run* .

2. Configure the following setting(s):

Setting	Value
File name	InjectMixer.def

3. Click **Save**.

   The CFX-Solver input file (`InjectMixer.def`) is created. CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, **Solver Input File** is set.

4. When you are finished, select **File** > **Quit** in CFX-Pre.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Click **Save & Quit** if prompted, to save `InjectMixer.cfx`

6.5. Obtaining the Solution Using CFX-Solver Manager

You will now generate a solution for the CFD simulation that you just prepared.

6.5.1. Starting the Run

At this point, CFX-Solver Manager is running, and the **Define Run** dialog box is displayed, with the CFX-Solver input file set.

1. Click **Start Run**.
2. When the run ends, ensure that the check box next to **Post-Process Results** is cleared and click **OK** to close the dialog box.

6.5.2. Moving from CFX-Solver Manager to CFD-Post

1. Select **Tools** > **Post-Process Results** or click *Post-Process Results* .
2. If using CFX-Solver Manager in stand-alone mode, optionally select **Shut down CFX-Solver Manager**.
3. Click **OK**.

6.6. Viewing the Results Using CFD-Post

When CFD-Post starts, the viewer and **Outline** workspace display by default.

The tutorial follows this general workflow for viewing results in CFD-Post:

6.6.1. Modifying the Outline of the Geometry
6.6.2. Creating and Modifying Streamlines Originating from the Main Inlet
6.6.3. Modifying Streamline Color Ranges
6.6.4. Coloring Streamlines with a Constant Color
6.6.5. Creating Streamlines Originating from the Side Inlet
6.6.6. Examining Turbulence Kinetic Energy
6.6.7. Quitting CFD-Post

6.6.1. Modifying the Outline of the Geometry

Throughout this and the following examples, use your mouse and the **Viewing Tools** toolbar to manipulate the geometry as required at any time.

1. In the tree view, double-click `Wireframe`.
2. Set the **Edge Angle** to `15 [degree]`.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click **Apply**.

6.6.2. Creating and Modifying Streamlines Originating from the Main Inlet

When you complete this module you will see streamlines (mainly blue and green) starting at the main inlet of the geometry and proceeding to the outlet. Above where the side pipe meets the main pipe, there is an area where the flow re-circulates rather than flowing roughly tangent to the direction of the pipe walls.

1. Select **Insert** > **Streamline** from the main menu or click *Streamline* .
2. Under **Name**, type `MainStream`.
3. Click **OK**.
4. Configure the following setting(s):

Tab	Setting	Value
Geometry	**Type**	3D Streamline
	Definition > **Start From**	 main inlet

5. Click **Apply**.
6. Right-click a blank area in the viewer, select **Predefined Camera** from the shortcut menu, then select **Isometric View (Y up)**.

   The pipe is displayed with the main inlet in the bottom right of the viewer.

6.6.3. Modifying Streamline Color Ranges

You can change the appearance of the streamlines using the **Range** setting on the **Color** tab.

1. Under **User Locations and Plots**, modify the streamline object **MainStream** by applying the following settings

Tab	Setting	Value
Color	**Range**	Local

2. Click **Apply**.

   The color map is fitted to the range of velocities found along the streamlines. The streamlines therefore collectively contain every color in the color map.

3. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Color	**Range**	User Specified
	Min	0.2 [m s^-1]
	Max	2.2 [m s^-1]

Note:

Portions of streamlines that have values outside the range shown in the legend are colored according to the color at the nearest end of the legend. When using tubes or symbols (which contain faces), more accurate colors are obtained with lighting turned off.

4. Click **Apply**.

   The streamlines are colored using the specified range of velocity values.

6.6.4. Coloring Streamlines with a Constant Color

1. Configure the following setting(s):

Tab	Setting	Value
Color	**Mode**	Constant
	Color	(Green)

   Color can be set to green by selecting it from the color pallet, or by repeatedly clicking on the color box until it cycles through to the default green color.

2. Click **Apply**.

6.6.5. Creating Streamlines Originating from the Side Inlet

The following steps illustrate using this feature to add a streamline object that originates at the side inlet.

1. Right-click `MainStream` and select **Duplicate** from the shortcut menu.
2. In the **Name** window, type `SideStream`.
3. Click **OK**.
4. Double-click the newly created streamline, `SideStream`.
5. Configure the following setting(s):

Tab	Setting	Value
Geometry	Definition > **Start From**	 side inlet

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Color	**Mode**	Constant
	Color	(Red)

6. Click **Apply**.

   Red streamlines appear, starting from the side inlet.

7. For better view, select **Isometric View (Y up)**.

6.6.6. Examining Turbulence Kinetic Energy

Away from walls, turbulence kinetic energy has an influence on the level of mixing. A plane will be created to view the Turbulence Kinetic Energy variable within the domain.

Note:

This module has multiple changes compiled into single steps in preparation for other tutorials that provide fewer specific instructions.

1. Turn off the visibility of both the `MainStream` and the `SideStream` objects.
2. Create a plane named `Plane 1` that is normal to X and passing through the X = 0 Point. To do so, specific instructions follow.

   a. From the main menu, select **Insert** > **Location** > **Plane** and click **OK**.

   b. In the details view, set **Definition > Method** to `YZ Plane` and **X** to `0 [m]`.

   c. Click **Apply**.

3. Color the plane using the variable `Turbulence Kinetic Energy`, to show regions of high turbulence. To do so, apply the settings below.

Tab	Setting	Value
Color	**Mode**	Variable
	Variable	Turbulence Kinetic Energy

4. Click **Apply**.
5. Experiment with other variables to color this plane (for example, `Temperature` to show the temperature mixing of the two streams).

   Commonly used variables are in the drop-down menu. A full list of available variables can be viewed by clicking next to the **Variable** data box.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.6.7. Quitting CFD-Post

1. When you are finished, select **File** > **Quit** to exit CFD-Post.

2. If prompted by a dialog box, save the state at your discretion.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 7: Flow from a Circular Vent

This tutorial includes:

7.1. Tutorial Features

7.2. Overview of the Problem to Solve

7.3. Preparing the Working Directory

7.4. Defining the Case Using CFX-Pre

7.5. Obtaining the Solution Using CFX-Solver Manager

7.6. Viewing the Results Using CFD-Post

7.1. Tutorial Features

In this tutorial you will learn about:

- Setting up a transient problem in CFX-Pre.
- Using an opening type boundary in CFX-Pre.
- Making use of multiple configurations in CFX-Pre
- Modeling smoke using additional variables in CFX-Pre.
- Visualizing a smoke plume using an isosurface in CFD-Post.
- Creating an image for printing, and generating a movie in CFD-Post.

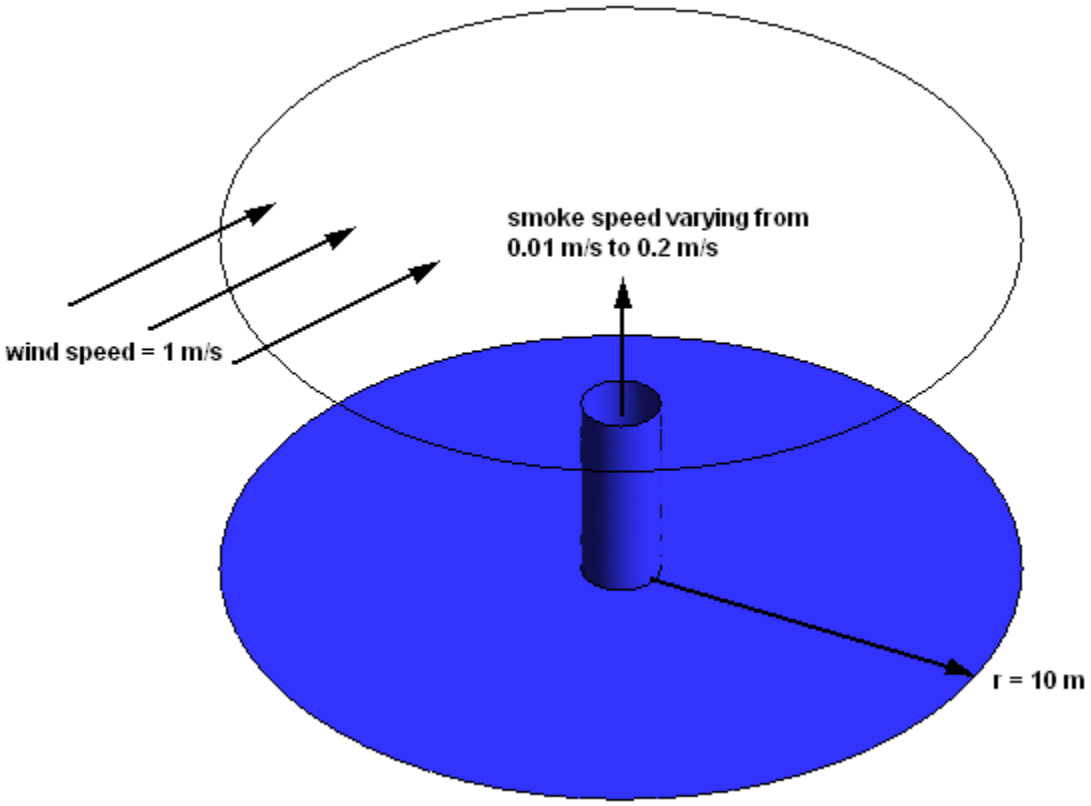
Component	Feature	Details
CFX-Pre	User Mode	General mode
	Analysis Type	Steady State
		Transient
	Fluid Type	General Fluid
	Domain Type	Single Domain
	Turbulence Model	k-Epsilon
	Boundary Conditions	Inlet (Subsonic)
		Opening
		Wall: No-Slip
	Timestep	Auto Time Scale
		Transient Example
	Configuration	Multiple

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Component	Feature	Details
	Transient Results File	
CFD-Post	Plots	Animation
		Isosurface
	Other	Auto Annotation
		Movie Generation
		Printing
		Time Step Selection
		Title/Text
		Transient Animation

7.2. Overview of the Problem to Solve

In this example, a chimney stack releases smoke that is dispersed into the atmosphere with an oncoming side wind of 1 m/s. The turbulence will be set to intensity and length scale with a value of 0.05, which corresponds to 5% turbulence, a medium level intensity, and with an eddy length scale value of 0.25 m. The goal of this tutorial is to model the dispersion of the smoke from the chimney stack over time. Unlike previous tutorials, which were steady-state, this example is time-dependent. Initially, no smoke is being released. Subsequently, the chimney starts to release smoke. As a postprocessing exercise, you produce an animation that illustrates how the plume of smoke develops with time.



If this is the first tutorial you are working with, it is important to review the following topics before beginning:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Running Ansys CFX Tutorials Using Ansys Workbench (p. 34)
- Changing the Display Colors (p. 37)

7.3. Preparing the Working Directory

1. Create a working directory.

   Ansys CFX uses a working directory as the default location for loading and saving files for a particular session or project.

2. Download the `circvent.zip` file here.

3. Unzip `circvent.zip` to your working directory.

   Ensure that the following tutorial input file is in your working directory:

   - `CircVentMesh.gtm`

4. Set the working directory and start CFX-Pre.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

7.4. Defining the Case Using CFX-Pre

This section describes the step-by-step definition of the flow physics in CFX-Pre for a simulation with two analyses. First is a steady-state analysis with no smoke being produced by the chimney. The second analysis takes the setup for the steady-state and adapts it for a transient analysis. The results from the steady-state analysis will be used as the initial guess for the transient analysis.

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** and click **OK**.
3. Select **File** > **Save Case As**.
4. Set **File name** to `CircVent`.
5. Click **Save**.

7.4.1. Importing the Mesh

1. Right-click `Mesh` and select **Import Mesh** > **CFX Mesh**.

   The **Import Mesh** dialog box appears.

2. Configure the following setting(s):

Setting	Value
File name	CircVentMesh.gtm

3. Click **Open**.

4. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)** from the shortcut menu.

7.4.2. Creating an Additional Variable

In this tutorial an Additional Variable (non-reacting scalar component) will be used to model the dispersion of smoke from the vent.

Note:

While smoke is not required for the steady-state simulation, including it here prevents you from having to set up time value interpolation in the transient analysis.

---

1. From the menu bar, select **Insert** > **Expressions, Functions and Variables** > **Additional Variable**.

2. Set **Name** to `smoke`.

3. Click **OK**.

4. Set **Variable Type** to `Volumetric`.

5. Set **Units** to `[kg m^-3]`.

6. Click **OK**.

7.4.3. Defining the Steady-State Analysis

The existing analysis will be set up as steady-state.

7.4.3.1. Renaming the Analysis

To rename the existing analysis:

1. From the **Outline** tree, right-click `Simulation` > `Flow Analysis 1` and click **Rename**.

2. Rename the analysis to `Steady State Analysis`.

7.4.3.2. Creating the Domain

You will create a fluid domain that includes support for smoke as an Additional Variable.

1. Select **Insert** > **Domain** from the menu bar, or click *Domain* , then set the name to `CircVent` and click **OK**.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Location and Type	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Location**	B1.P3
	Fluid and Particle Definitions	Fluid 1
	Fluid and Particle Definitions > Fluid 1 > **Material**	Air at 25 C
	Domain Models > Pressure > **Reference Pressure**	0 [atm]
Fluid Models	Heat Transfer > **Option**	None
	Additional Variable Models > **Additional Variable**	smoke
	Additional Variable Models > Additional Variable > **smoke**	(Selected)
	Additional Variable Models > Additional Variable > smoke > **Kinematic Diffusivity**	(Selected)
	Additional Variable Models > Additional Variable > smoke > Kinematic Diffusivity > **Kinematic Diffusivity**	1.0E-5 [m^2 s^-1] [a]

a. 1.0E-5 [m^2 s^-1] is a representative kinematic diffusivity value for smoke in air.

3. Click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.3.3. Creating the Boundaries

This is an example of external flow, since fluid is flowing *over* an object and not *through* an enclosure such as a pipe network (which would be an example of internal flow). In external flow problems, some inlets will be made sufficiently large that they do not affect the CFD solution. However, the length scale values produced by the **Default Intensity and AutoCompute Length Scale** option for turbulence are based on inlet size. They are appropriate for internal flow problems and particularly, cylindrical pipes. In general, you need to set the turbulence intensity and length scale explicitly for large inlets in external flow problems. If you do not have a value for the length scale, you can use a length scale based on a typical length of the object over which the fluid is flowing. In this case, you will choose a turbulence length scale that is one-tenth of the diameter of the vent.

For parts of the boundary where the flow direction changes, or is unknown, an opening boundary can be used. An opening boundary allows flow to both enter and leave the fluid domain during the course of the analysis.

7.4.3.3.1. Inlet Boundary

You will create the inlet boundary with velocity components set consistently with the problem description.

1. Select **Insert** > **Boundary** from the menu bar or click *Boundary* .
2. Set **Name** to `Wind`.
3. Click **OK**.
4. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Inlet
	Location	Wind
Boundary Details	Mass And Momentum > **Option**	Cart. Vel. Components
	Mass and Momentum > **U**	1 [m s^-1]
	Mass and Momentum > **V**	0 [m s^-1]
	Mass and Momentum > **W**	0 [m s^-1]
	Turbulence > **Option**	Intensity and Length Scale
	Turbulence	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Fractional Intensity**	0.05 [a]
	Turbulence > **Eddy Length Scale**	0.25 [m] [a]
	Additional Variables > smoke > **Option**	Value
	Additional Variables > smoke > **Add. Var. Value**	0 [kg m^-3] [b]
a. From the problem description. b. The smoke value that will be set up corresponds to no smoke at the inlet.		

5. Click **OK**.

Note:

The boundary marker vectors used to display boundary conditions (inlets, outlets, openings) are normal to the boundary surface regardless of the actual direction specification. To plot vectors in the direction of flow, select **Boundary Vector** under the **Plot Options** tab for the inlet boundary, and on the **Labels and Markers Options** tab (accessible from **Case Options** > **Labels and Markers** on the **Outline** tree view), ensure that **Settings** > **Show Boundary Markers** is selected and **Show Inlet Markers** is cleared.

7.4.3.3.2. Opening Boundary

You will create an opening boundary with pressure and flow direction specified. If fluid enters the domain through the opening, it should have turbulence intensity and length scale, as well as smoke concentration, set to the same values as for the inlet.

1. Select **Insert** > **Boundary** from the menu bar or click *Boundary* .
2. Set **Name** to `Atmosphere`.
3. Click **OK**.
4. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Opening
	Location	Atmosphere
Boundary Details	Mass And Momentum > **Option**	Opening Pres. and Dirn
	Mass and Momentum > **Relative Pressure**	0 [Pa]
	Flow Direction > **Option**	Normal to Boundary Condition
	Turbulence > **Option**	Intensity and Length Scale
	Turbulence > **Fractional Intensity**	0.05
	Turbulence > **Eddy Length Scale**	0.25 [m]
	Additional Variables > smoke > **Option**	Value
	Additional Variables > smoke > **Add. Var. Value**	0 [kg m^-3]

5. Click **OK**.

7.4.3.3.3. Inlet for the Vent

You will create the vent inlet boundary with a normal velocity of 0.01 m/s as prescribed in the problem description and no smoke release. The turbulence level for the inlet vent will be determined from turbulence intensity and eddy viscosity ratio.

1. Select **Insert** > **Boundary** from the menu bar or click *Boundary* .
2. Set **Name** to `Vent`.
3. Click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Inlet
	Location	Vent
Boundary Details	Mass And Momentum > **Normal Speed**	0.01 [m s^-1]
	Turbulence > **Option**	Intensity and Eddy Viscosity Ratio
	Turbulence > **Fractional Intensity**	0.05
	Turbulence > **Eddy Viscosity Ratio**	10
	Additional Variables > smoke > **Option**	Value
	Additional Variables > smoke > **Add. Var. Value**	0 [kg m^-3]

5. Click **OK**.

7.4.3.4. Setting Initial Values

For this tutorial, the automatic initial values are suitable. Review and apply the default settings:

1. Click *Global Initialization* .
2. Review the settings for velocity, pressure, turbulence and the smoke.
3. Click **OK**.

7.4.3.5. Setting Solver Control

CFX-Solver has the ability to calculate physical time step size for steady-state problems. If you do not know the time step size to set for your problem, you can use the `Auto Timescale` option.

1. Click *Solver Control* .

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Convergence Control > **Max. Iterations**	 75

3. Note that **Convergence Control** > **Fluid Timescale Control** > **Timescale Control** is set to `Auto Timescale`.

4. Click **OK**.

7.4.4. Defining the Transient Analysis

In this part of the tutorial, you will duplicate the steady-state analysis and adapt it to set up a transient flow analysis in CFX-Pre.

7.4.4.1. Creating the Analysis

1. In the **Outline** tree view, right-click `Simulation` > `Steady State Analysis` and select **Duplicate**.

2. Right-click `Simulation` > `Copy of Steady State Analysis` and select **Rename**.

3. Rename the analysis to `Transient Analysis`.

7.4.4.2. Modifying the Analysis Type

In this step you will set the new analysis to a type of transient. Later, you will set the concentration of smoke for the transient analysis to rise asymptotically to its final concentration with time, so it is necessary to ensure that the interval between the time steps is smaller at the beginning of the simulation than at the end.

1. In the **Outline** tree view, ensure that `Simulation` > `Transient Analysis` is expanded.

2. Right-click `Simulation` > `Transient Analysis` > `Analysis Type` and select **Edit**.

3. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Analysis Type > **Option**	 Transient
	Analysis Type > Time Duration > **Option**	 Total Time
	Analysis Type	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Time Duration > **Total Time**	 30 [s]
	Analysis Type > Time Steps > **Timesteps**	 4*0.25, 2*0.5, 2*1.0, 13*2.0 [s] [a]
	Analysis Type > Initial Time > **Option**	 Automatic with Value
	Analysis Type > Initial Time > **Time**	 0 [s]

a. This list specifies 4 time steps of 0.25 [s], then 2 time steps of 0.5 [s], and so on. Enter the list without the units, then set the units in the drop-down list. Do *NOT* click *Enter Expression* to enter lists of values.

4. Click **OK**.

7.4.4.3. Modifying the Boundary Conditions

The only boundary condition that needs altering for the transient analysis is the `Vent` boundary condition. In the steady-state calculation, this boundary had a small amount of air flowing through it. In the transient calculation, more air passes through the vent and there is a time-dependent concentration of smoke in the air. This is initially zero, but builds up to a larger value. The smoke concentration will be specified using the CFX Expression Language.

7.4.4.3.1. To Modify the Vent Inlet Boundary Condition

1. In the **Outline** tree view, ensure that `Simulation` > `Transient Analysis` > `CircVent` is expanded.
2. Right-click `Simulation` > `Transient Analysis` > `CircVent` > `Vent` and select **Edit**.
3. Configure the following setting(s):

Tab	Setting	Value
Boundary Details	Mass And Momentum > **Normal Speed**	 0.2 [m s^-1]

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Leave the **Vent** details view open for now.

You are going to create an expression for smoke concentration. The concentration is zero for time t=0 and builds up to a maximum of 1 kg m^-3.

4. Create a new expression by selecting **Insert** > **Expressions, Functions and Variables** > **Expression** from the menu bar. Set the name to `TimeConstant`.

5. Configure the following setting(s):

Name	Definition
TimeConstant	3 [s]

6. Click **Apply** to create the expression.

7. Create the following expressions with specific settings, remembering to click **Apply** after each is defined:

Name	Definition
FinalConcentration	1 [kg m^-3]
ExpFunction	FinalConcentration*abs(1-exp(-t/TimeConstant)) [a]

a. When entering this function, you can select most of the required items by right-clicking in the **Definition** window in the **Expression** details view instead of typing them. The names of the existing expressions are under the **Expressions** menu. The **exp** and **abs** functions are under **Functions > CEL**. The variable **t** is under **Variables**.

Note:

The `abs` function takes the modulus (or magnitude) of its argument. Even though the expression (1- exp (-t/TimeConstant)) can never be less than zero, the abs function is included to ensure that the numerical error in evaluating it near to zero will never make the expression evaluate to a negative number.

Next, you will visualize how the concentration of smoke issued from the vent varies with time.

7.4.4.3.2. Plotting Smoke Concentration

1. Double-click `ExpFunction` in the **Expressions** tree view.

2. Configure the following setting(s):

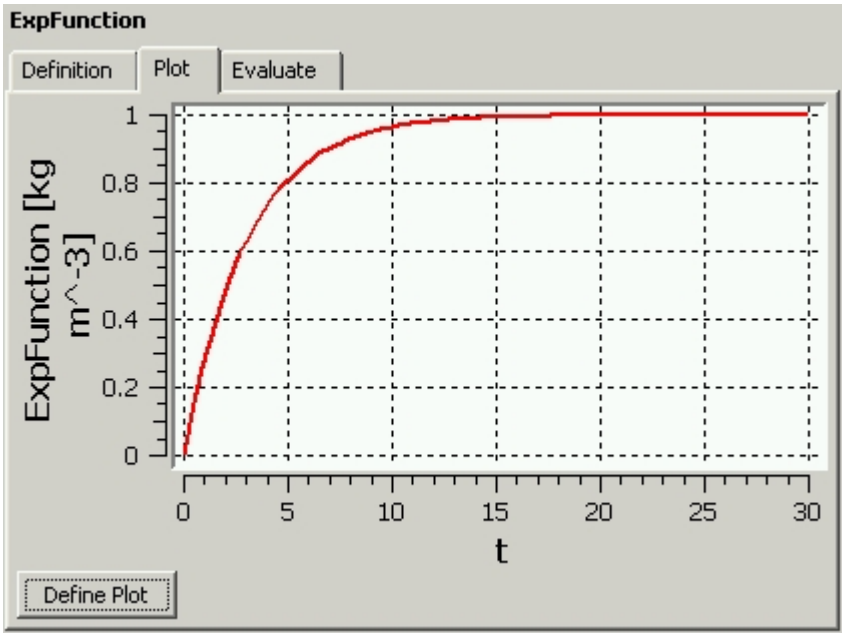
Tab	Setting	Value
Plot	**t**	(Selected)
	t	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Start of Range**	0 [s]
	t > **End of Range**	 30 [s]

3. Click **Plot Expression**.

   The button name then changes to **Define Plot**, as shown.



   As can be seen, the smoke concentration rises to its asymptotic value reaching 90% of its final value at around 7 seconds.

4. Click the **Boundary: Vent** tab.

   In the next step, you will apply the expression `ExpFunction` to the additional variable `smoke` as it applies to the `Transient Analysis` boundary `Vent`.

5. Configure the following setting(s):

Tab	Setting	Value
Boundary Details	Additional Variables > smoke > **Option**	 Value
	Additional Variables > smoke	

Tab	Setting	Value
	> **Add. Var. Value**	ExpFunction [a]
a. Click *Enter Expression* to enter text.		

6. Click **OK**.

7.4.4.4. Initialization Values

When the `Transient Analysis` is run, the initial values to the CFX-Solver will be taken from the results of the `Steady State Analysis`. The steady state and transient analyses will be sequenced by setting up the configurations of these analyses in a subsequent step. For the moment, you can leave all of the initialization data set for the `Transient Analysis` to **Automatic** and the initial values will be read automatically from the `Steady State Analysis` results. Therefore, there is no need to revisit the initialization settings.

7.4.4.5. Modifying the Solver Control

1. In the **Outline** tree view, ensure that `Simulation` > `Transient Analysis` > `Solver` is expanded.
2. Right-click `Simulation` > `Transient Analysis` > `Solver` > `Solver Control` and select **Edit**.
3. Set **Convergence Control** > **Max. Coeff. Loops** to 3.
4. Leave the other settings at their default values.
5. Click **OK** to set the solver control parameters.

7.4.4.6. Setting Output Control

To enable results to be viewed at different time steps, it is necessary to create transient results files at specified times. The transient results files do not have to contain all solution data. In this step, you will create minimal transient results files.

1. In the **Outline** tree view, double-click `Simulation` > `Transient Analysis` > `Solver` > `Output Control`.
2. Click the **Trn Results** tab.
3. In the **Transient Results** tree view, click *Add new item* , set **Name** to `Transient Results 1`, and click **OK**.
4. Configure the following setting(s) of `Transient Results 1`:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
Option	Selected Variables
Output Variables List	Pressure, Velocity, smoke [a]
Output Frequency > **Option**	Time List
Output Frequency > **Time List**	1, 2, 3 [s] [b]

a. Click the *Ellipsis* icon to select items if they do not appear in the drop-down list. Use the **Ctrl** key to select multiple items.

b. Do *not* click *Enter Expression* to enter lists of values. Enter the list without the units, then set the units in the drop-down list.

5. Click **Apply**.

6. In the **Transient Results** tree view, click *Add new item* set **Name** to `Transient Results 2`, and click **OK**.

   This creates a second transient results object. Each object can result in the production of many transient results files.

7. Configure the following setting(s) of `Transient Results 2`:

Setting	Value
Option	Selected Variables
Output Variables List	Pressure, Velocity, smoke
Output Frequency > **Option**	Time Interval
Output Frequency > **Time Interval**	4 [s] [a]

a. A transient results file will be produced every 4 s (including 0 s) and at 1 s, 2 s and 3 s. The files will contain no mesh, and data for only the three selected variables. This reduces the size of the minimal results files. A full results file is always written at the end of the run.

8. Click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.5. Configuring Simulation Control

With two types of analysis for this simulation, configuration control is used to sequence these analyses.

7.4.5.1. Configuration Control for the Steady State Analysis

To set up the `Steady State Analysis` so that it will start at the beginning of the simulation:

1. In the **Outline** tree view, ensure that `Simulation Control` is expanded.
2. Right-click `Simulation Control` > `Configurations` and select **Insert** > **Configuration**.
3. Set **Name** to `Steady State`.
4. Click **OK**.
5. Configure the following setting(s):

Tab	Setting	Value
General Settings	**Flow Analysis**	Steady State Analysis
	Activation Conditions > Activation Condition 1 > **Option**	 Start of Simulation

6. Click **OK**.

7.4.5.2. Configuration Control for the Transient Analysis

To set up the `Transient Analysis` so that it will start upon the completion of the `Steady State Analysis`:

1. Right-click `Simulation Control` > `Configurations` and select **Insert** > **Configuration**.
2. Set **Name** to `Transient`.
3. Click **OK**.
4. Configure the following setting(s):

Tab	Setting	Value
General Settings	**Flow Analysis**	Transient Analysis
	Activation Conditions > Activation Condition 1 > **Option**	 End of Configuration

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Activation Conditions > Activation Condition 1 > **Configuration Name**	 Steady State
Run Definition	**Configuration Execution Control**	Selected
Initial Values	**Initial Values Specification**	Selected
	Initial Values Specification > Initial Values > Initial Values 1 > **Option**	 Configuration Results
	Initial Values Specification > Initial Values > Initial Values 1 > **Configuration Name**	 Steady State

5. Click **OK**.

7.4.6. Writing the CFX-Solver Input (.mdef) File

1. In the **Outline** tree view, right-click `Simulation Control` and select **Write Solver Input File**.
2. Configure the following setting(s):

Setting	Value
File of type	CFX-Solver Input Files (*.mdef)
File name	CircVent.mdef

3. Click **Save**.

   This will create `CircVent.mdef` as well as a directory named `CircVent` that contains `SteadyState.cfg` and `Transient.cfg`.

4. Quit CFX-Pre, saving the case (`.cfx`) file.

7.5. Obtaining the Solution Using CFX-Solver Manager

You can obtain a solution to the steady-state and transient configurations by using the following procedure.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Start CFX-Solver Manager.
2. From the menu bar, select **File** > **Define Run**.
3. Configure the following setting(s):

Setting	Value
Solver Input File	CircVent.mdef
Edit Configuration	Global Settings

4. Click **Start Run**.

   CFX-Solver Manager will start with the solution of the steady-state configuration.

5. In the **Workspace** drop-down menu, select `SteadyState_001`. The residual plots for six equations will appear: U - Mom, V - Mom, W - Mom, P - Mass, K-TurbKE, and E-Diss.K (the three momentum conservation equations, the mass conservation equation and equations for the turbulence kinetic energy and turbulence eddy dissipation). The **Momentum and Mass** tab contains four of the plots and the other two are under **Turbulence Quantities**. The residual for the smoke equation is also plotted but registers no values since it is not initialized.
6. Upon the successful completion of the steady-state configuration, the solution of the transient configuration starts automatically. Notice that the text output generated by the CFX-Solver in the `Run Transient 001` **Workspace** will be more than you have seen for steady-state problems. This is because each time step consists of several inner (coefficient) iterations. At the end of each time step, information about various quantities is printed to the text output area. The residual for the smoke equation is now plotted under the **Additional Variables** tab.
7. Upon the successful completion of the combined steady-state and transient configurations, ensure that the check box beside **Post-Process Results** is cleared and click **OK** to close the message indicating the successful completion of the simulation.
8. In the CFX-Solver Manager, set **Workspace** to `Run CircVent 001`.
9. From the menu bar, select **Tools** > **Post-Process Results**.
10. On the **Start CFD-Post** dialog box, select **Shut down CFX-Solver Manager** and click **OK**.

7.6. Viewing the Results Using CFD-Post

In this part of the tutorial, you will:

- Create an isosurface to illustrate the pattern of smoke concentration.
- View results at different time steps.
- Animate the results to view the dispersion of smoke from the vent over time.
- Save the animation as an MPEG file.
- Use volume rendering to show the visibility of smoke with its transparency.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.6.1. Displaying Smoke Density Using an Isosurface

An isosurface is a surface of constant value of a variable. For instance, it could be a surface consisting of all points where the velocity is 1 [m s^-1]. In this case, you are going to create an isosurface of `smoke` concentration (`smoke` is the Additional Variable that you specified earlier).

1. In CFD-Post, right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)**.

   This ensures that the view is set to a position that is best suited to display the results.

2. From the menu bar, select **Insert** > **Location** > **Isosurface**, or under **Location** on the toolbar, click **Isosurface**.

3. Click **OK** to accept the default name.

4. Configure the following setting(s):

Tab	Setting	Value
Geometry	**Variable**	smoke
	Value	0.005 [kg m^-3]

5. Click **Apply**.

   - A bumpy surface is displayed, showing the smoke emanating from the vent.
   - The surface is rough because the mesh is coarse. For a smoother surface, you would re-run the problem with a smaller mesh length scale.
   - The surface will be a constant color because the default settings on the **Color** tab were used.
   - When **Color Mode** is set to either `Constant` or `Use Plot Variable` for an isosurface, the isosurface is displayed in one color.

6. In **Geometry**, experiment by changing the **Value** setting so that you can see the shape of the plume more clearly.

   Zoom in and rotate the geometry, as required.

7. When you have finished, set **Value** to `0.002 [kg m^-3]`.

8. Right-click a blank spot in the viewer and select **Predefined Camera** > **Isometric View (Z up)**.

7.6.2. Viewing the Results at Different Time Steps

When CFD-Post is loaded, the results that are immediately available are those at the final time step; in this case, at t = 30 s (this is nominally designated Final State). The **Timestep Selector** shows the **Configuration**, the step of the **Simulation**, the **Step** (outer loop) number, the **Time** (simulated time in seconds) of the configuration and the **Type** of results file that was saved at that time step for the configuration. You can see that **Partial** results files were saved (as requested in CFX-Pre) for all time steps in the transient configuration except for the last one.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click *Timestep Selector* .

2. Load the results for a **Time** value of 2 s by double-clicking the appropriate row in the **Timestep Selector**.

   After a short pause, the **Loaded Timestep** (located just below the title bar of the **Timestep Selector**) will be updated with the new time step number.

3. Load the time value of 4 s using the **Timestep Selector**.

   The smoke has now spread out even more, and is being carried by the wind.

4. Double-click some more time values to see how the smoke plume grows with time.

5. Finish by loading a time value of 12 s.

7.6.3. Generating Titled Image Files

You can produce titled image output from CFD-Post.

7.6.3.1. Adding a Title

First, you will add text to the viewer so that the printed output has a title.

1. Select **Insert** > **Text** from the menu bar or click *Create text* .

2. Click **OK** to accept default name.

3. In the **Text String** box, enter the following text.

```
Isosurface showing smoke concentration of 0.002 kg/m^3 after
```

   Note:

   Further text will be added at a later stage to complete this title.

4. Select **Embed Auto Annotation**.

5. Set **Type** to `Time Value`.

   In the text line, note that <aa> has been added to the end. This is where the time value will be placed.

6. Click **Apply** to create the title.

7. Click the **Location** tab to modify the position of the title.

   The default settings for text objects center text at the top of the screen. To experiment with the position of the text, change the settings on the **Location** tab.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. Under the **Appearance** tab, change **Color Mode** to `User Specified` and select a new color.
9. Click **Apply**.

7.6.3.2. JPEG output

CFD-Post can save images in several different formats. In this section you will save an image in JPEG format.

1. Select **File** > **Save Picture**, or click *Save Picture* .
2. Set **Format** to JPEG.
3. Click *Browse* next to the **File** data box.
4. Browse to the directory where you want the file saved.
5. Enter a name for the JPEG file.
6. Click **Save** to set the file name and directory.

   This sets the path and name for the file.
7. To save the file, click **Save** on the **Save Picture** dialog box.

   To view the file or make a hard copy, use an application that supports JPEG files.
8. Turn off the visibility of the text object to hide it.

7.6.4. Generating a Movie

You can generate an MPEG file to show the transient flow of the plume of smoke. To generate a movie file, you use the **Animation** dialog box.

1. In the **Timestep Selector**, ensure that a time value of 0 s is loaded.
2. Click *Animation* .

   The **Animation** dialog box appears.
3. Set **Type** to **Timestep Animation**.
4. Position the geometry so that you will be able to see the plume of smoke.
5. Click *More Animation Options* to show more animation settings.
6. Ensure that the *Repeat forever* button next to **Repeat** is not selected (not depressed).
7. Select **Save Movie**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. Set **Format** to `MPEG1`.
9. Click *Browse* next to **Save Movie**.
10. Set **File name** to `CircVent.mpg`.
11. If required, set the path location to a different directory.
12. Click **Save**.

    The movie file name (including path) has been set, but the animation has not yet been produced.

13. Click *Play the animation* .

    - The movie will be created as the animation proceeds.
    - This will be slow, because for each time step results will be loaded and objects will be created.
    - To view the movie file, you need to use a viewer that supports the MPEG format.

    Note:

    To explore additional animation options, click the **Options** button. On the **Advanced** tab of **Animation Options**, there is a check box called **Save Frames As Image Files**. By selecting this check box, the JPEG or PPM files used to encode each frame of the movie will persist after movie creation; otherwise, they will be deleted.

7.6.5. Viewing the Dispersion of Smoke at the Final Time Step

The final time step has the greatest dispersion of smoke, so you will load only that time step, then view the smoke using the Volume Rendering feature.

1. Select **File** > **Load Results**.
2. In the **Load Results File** dialog box, select **Load only the last results** (the other default settings should remain unchanged) and ensure that **File name** is set to **CircVent_001.mres**. Click **Open**.

    Note:

    A warning message appears. Click **OK**.

3. If necessary, right-click a blank spot in the viewer and select **Predefined Camera** > **Isometric View (Z up)**.
4. In the **Outline** tree view, clear **Isosurface 1** and **Text 1**.
5. Select **Insert** > **Volume Rendering** and set the **Name** to be `SmokeVolume`.
6. In the details view, set the following values:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Geometry	**Variable**	smoke
	Resolution	50
	Transparency	.2
Color	**Mode**	Variable
	Variable	smoke
	Color Map	Greyscale

Click **Apply**.

7. When you have finished, quit CFD-Post.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 8: Flow Around a Blunt Body

This tutorial includes:

8.1. Tutorial Features

8.2. Overview of the Problem to Solve

8.3. Preparing the Working Directory

8.4. Defining the Case Using CFX-Pre

8.5. Obtaining the Solution Using CFX-Solver Manager

8.6. Viewing the Results Using CFD-Post

8.1. Tutorial Features

In this tutorial you will learn about:

- Solving and postprocessing a case where the geometry has been omitted on one side of a symmetry plane.
- Using free-slip wall boundaries as a compromise between accurate flow modeling and computational grid size.
- Accurately modeling the near-wall flow using Shear Stress Transport (SST) turbulence model.
- Running the CFX-Solver in parallel (optional).
- Creating vector plots in CFD-Post with uniform spacing between the vectors.
- Creating a macro using power syntax in CFD-Post.

Component	Feature	Details
CFX-Pre	User Mode	General mode
	Analysis Type	Steady State
	Fluid Type	Constant Property Gas
	Domain Type	Single Domain
	Turbulence Model	Shear Stress Transport
	Boundary Conditions	Inlet (Subsonic)
		Outlet (Subsonic)
		Symmetry Plane
		Wall: No-Slip
		Wall: Free-Slip
	Timestep	Physical Time Scale

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

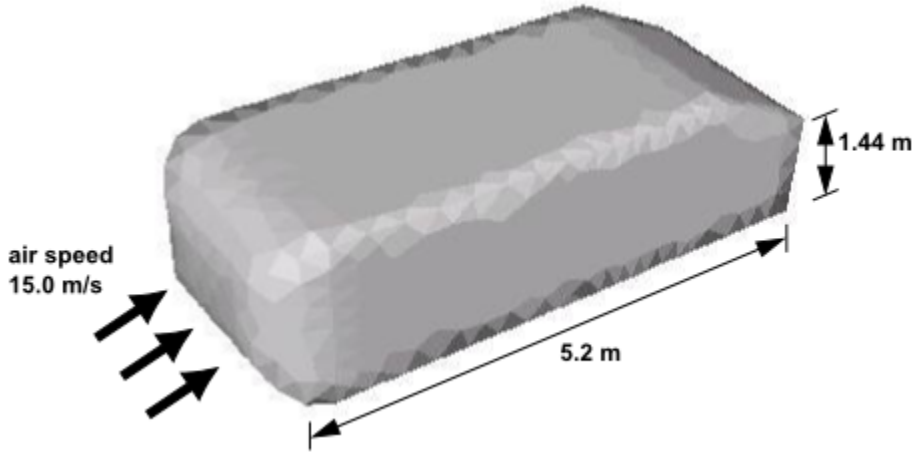
Component	Feature	Details
CFX-Solver Manager	Parallel Processing	
CFD-Post	Plots	Default Locators
		Outline Plot (Wireframe)
		Sampling Plane
		Streamline
		Vector
		Volume
	Other	Changing the Color Range
		Instancing Transformation
		Lighting Adjustment
		Symmetry
		Viewing the Mesh

8.2. Overview of the Problem to Solve

In this tutorial, a generic vehicle body is placed into an oncoming side wind of 15 m/s. The turbulence will be set to Intensity and Length scale with a value of 0.05, which corresponds to 5% turbulence, a medium level intensity, and with an Eddy Length scale value of 0.1 m.

The goal of this tutorial is to set up, simulate, and then visualize the flow around the body. Since both the geometry and the flow are symmetric about a vertical plane, you will only use half of the geometry to find the solution. You will use free-slip wall boundaries on some sides of the domain to model external flow. The flow near the body is modeled using Shear Stress Transport. Since compressibility effects are not expected to be significant, you can model this as incompressible flow.

Figure 8.1: External Air Flow Over a Generic Vehicle Body



Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- Running Ansys CFX Tutorials Using Ansys Workbench (p. 34)
- Changing the Display Colors (p. 37)

8.3. Preparing the Working Directory

1. Create a working directory.

   Ansys CFX uses a working directory as the default location for loading and saving files for a particular session or project.

2. Download the `blunt_body.zip` file here.

3. Unzip `blunt_body.zip` to your working directory.

   Ensure that the following tutorial input file is in your working directory:

   - `BluntBodyMesh.gtm`

4. Set the working directory and start CFX-Pre.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

8.4. Defining the Case Using CFX-Pre

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `BluntBody`.
5. Click **Save**.

8.4.1. Importing the Mesh

1. Right-click `Mesh` and select **Import Mesh** > **CFX Mesh**.

   The **Import Mesh** dialog box appears.

2. Configure the following setting(s):

Setting	Value
File name	BluntBodyMesh.gtm

3. Click **Open**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.4.2. Creating the Domain

The flow of air in the domain is expected to be turbulent. The Shear Stress Transport (SST) turbulence model with automatic wall function treatment will be used because of its highly accurate predictions of flow separation. To take advantage of the SST model, the boundary layer should be resolved with at least 10 mesh nodes. In order to reduce computational time, the mesh in this tutorial is much coarser than that.

As stated in the overview, this tutorial models incompressible flow. Incompressible flow is specified by choosing an incompressible fluid. You will use `Air at 25 C` as the incompressible fluid.

As part of the domain settings, you will specify a reference pressure. CFX requires this when modeling incompressible flow.

1. Click *Domain* , and set the name to `BluntBody`.

2. Configure the following setting(s) of `BluntBody`:

Tab	Setting	Value
Basic Settings	Location and Type > **Location**	 B1.P3
	Fluid and Particle Definitions	Fluid 1
	Fluid and Particle Definitions > Fluid 1 > **Material**	 Air at 25 C [a]
	Domain Models > Pressure > **Reference Pressure**	 1 [atm]
Fluid Models	Heat Transfer > **Option**	 None
	Turbulence > **Option**	 Shear Stress Transport

a. Click the *Ellipsis* icon if the required material does not appear in the drop-down menu.

3. Click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.4.3. Creating Composite Regions

An imported mesh may contain many 2D regions. For the purpose of creating boundary conditions, it can sometimes be useful to group several 2D regions together and apply a single boundary to the composite 2D region. In this case, you are going to create a **Union** between two regions that both require a free-slip wall boundary.

1. From the main menu, select **Insert** > **Regions** > **Composite Region**.
2. Set the name to `FreeWalls` and click **OK**.
3. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Dimension (Filter)**	2D

4. Click beside the **Region List** dialog box, to display the **Selection Dialog**. Hold down the **Ctrl** key and select `Free1` and `Free2`.
5. Click **OK** to confirm your selection.
6. Click **OK** to create the composite region.

8.4.4. Creating the Boundaries

The simulation requires inlet, outlet, wall (no slip and free-slip) and symmetry plane boundaries. The regions for these boundaries were defined when the mesh was created (except for the composite region just created for the free-slip wall boundary).

8.4.4.1. Inlet Boundary

1. Click *Boundary* .
2. Under **Name**, type `Inlet`.
3. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Inlet
	Location	Inlet
Boundary Details	Flow Regime > **Option**	Subsonic
	Mass and Momentum > **Option**	Normal Speed
	Mass and Momentum	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Normal Speed**	15 [m s^-1]
	Turbulence > **Option**	Intensity and Length Scale
	Turbulence > **Fractional Intensity**	0.05 [a]
	Turbulence > **Eddy Length Scale**	0.1 [m] [a]

a. From the problem description.

4. Click **OK**.

8.4.4.2. Outlet Boundary

1. Create a new boundary named `Outlet`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Outlet
	Location	Outlet
Boundary Details	Mass and Momentum > **Option**	Static Pressure
	Mass and Momentum > **Relative Pressure**	0 [Pa]

3. Click **OK**.

8.4.4.3. Free-Slip Wall Boundary

The top and side surfaces of the rectangular region will use free-slip wall boundaries.

- On free-slip walls the shear stress is set to zero so that the fluid is not retarded.
- The velocity normal to the wall is also set to zero.
- The velocity parallel to the wall is calculated during the solution.

This boundary is an approximation that may not accurately represent the true flow conditions. By using a free-slip wall boundary, the flow modeling will be less accurate but the computational grid

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

size can be reduced by modeling less of the surroundings. If this case were modeling a wind tunnel experiment, the geometry would match the size and shape of the wind tunnel and use no-slip walls. If this case were modeling a blunt body open to the atmosphere, a much larger domain would be used to minimize the effect of the far-field boundary, and the far-field boundary type would be set to either a free-slip wall or a pressure-specified entrainment opening.

You will apply a single boundary to both walls by using the composite region defined earlier.

1. Create a new boundary named `FreeWalls`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Wall
	Location	FreeWalls
Boundary Details	Mass and Momentum > **Option**	 Free Slip Wall

3. Click **OK**.

8.4.4.4. Symmetry Plane Boundary

1. Create a new boundary named `SymP`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Symmetry
	Location	SymP

3. Click **OK**.

8.4.4.5. Wall Boundary on the Blunt Body Surface

1. Create a new boundary named `Body`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Wall
	Location	Body
Boundary Details	Mass and Momentum > **Option**	 No Slip Wall

3. Click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The remaining 2D regions (in this case, just the low Z face) will be assigned the default boundary which is an adiabatic, no-slip wall condition. In this case, the name of the default boundary is **BluntBody Default**. Although the boundaries **Body** and **BluntBody Default** are identical (except for their locations), the **Body** boundary was created so that, during postprocessing, its location can be conveniently distinguished from the other adiabatic, no-slip wall surfaces.

8.4.5. Setting Initial Values

The initial conditions are consistent with inlet boundaries.

1. Click *Global Initialization* .
2. Configure the following setting(s):

Tab	Setting	Value
Global Settings	Initial Conditions > Cartesian Velocity Components > **Option**	Automatic with Value
	Initial Conditions > Cartesian Velocity Components > **U**	15 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > **V**	0 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > **W**	0 [m s^-1]

3. Click **OK**.

8.4.6. Setting Solver Control

1. Click *Solver Control* .
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Convergence Control > **Max. Iterations**	60

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Convergence Control > Fluid Timescale Control > **Timescale Control**	Physical Timescale
	Convergence Control > Fluid Timescale Control > **Physical Timescale**	2 [s][a]
	Convergence Criteria > **Residual Target**	1e-05

a. Based on the air speed and the size of the object.

3. Click **OK**.

8.4.7. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .
2. Configure the following setting(s):

Setting	Value
File name	BluntBody.def

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, **Solver Input File** is set.

4. If using stand-alone mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

8.5. Obtaining the Solution Using CFX-Solver Manager

This tutorial introduces the parallel solver capabilities of CFX.

Note:

The results produced will be identical, whether produced by a parallel or serial run.

If you do not want to solve this tutorial in parallel (on more than one processor) or you do not have a license to run the CFX-Solver in parallel, proceed to Obtaining a Solution in Serial (p. 196).

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you do not know if you have a license to run the CFX-Solver in parallel, you should either ask your system administrator, or query the license server (see the Ansys, Inc. Licensing Guide (which is installed with the Ansys License Manager) for details). Alternatively proceed to Obtaining a Solution in Serial (p. 196).

If you would like to solve this tutorial in parallel on the same machine, proceed to Obtaining a Solution with Local Parallel (p. 197).

If you would like to solve this tutorial in parallel across different machines, proceed to Obtaining a Solution with Distributed Parallel (p. 197).

8.5.1. Obtaining a Solution in Serial

When CFX-Solver Manager has started, you can obtain a solution to the CFD problem by using the following procedure.

1. Click **Start Run**.
2. When CFX-Solver is finished, select the check box next to **Post-Process Results**.
3. If using stand-alone mode, select the check box next to **Shut down CFX-Solver Manager**.
4. Click **OK**. Continue this tutorial from Viewing the Results Using CFD-Post (p. 200).

8.5.2. Obtaining a Solution in Parallel

8.5.2.1. Background to Parallel Running in CFX

Using the parallel capability of the CFX-Solver enables you to divide a large CFD problem so that it can run on more than one processor/machine at once. This saves time and, when multiple machines are used, avoids problems that arise when a CFD calculation requires more memory than a single machine has available. The partition (division) of the CFD problem is automatic.

A number of events occur when you set up a parallel run and then ask the CFX-Solver to calculate the solution:

- Your mesh will be divided into the number of partitions that you have chosen.
- The CFX-Solver runs separately on each of the partitions on the selected machine(s).
- The results that one CFX-Solver process calculates affects the other CFX-Solver processes at the interface between the different sections of the mesh.
- All of the CFX-Solver processes are required to communicate with each other and this is handled by the *leader* process.
- The leader process always runs on the machine that you are logged into when the parallel run starts. The other CFX-Solver processes are *follower* processes and may be run on other machines.
- After the problem has been solved, a single results file is written. It will be identical to a results file from the same problem run as a serial process, with one exception: an extra variable `Real`

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

`partition number` will be available for the parallel run. This variable will be used later in this tutorial during post processing.

8.5.2.2. Obtaining a Solution with Local Parallel

In CFX-Solver Manager, the **Define Run** dialog box should already be open.

To run in local parallel mode:

1. Leave **Type of Run** set to `Full`.

   If **Type of Run** was instead set to `Partitioner Only`, your mesh would be split into a number of partitions but would not be run in the CFX-Solver afterwards.

2. Set **Run Mode** to a parallel mode suitable for your configuration; for example, `Intel MPI Local Parallel`.

   This is the recommended method for most applications.

3. If required, click *Add Partition* to add more partitions.

   Ideally, the number of partitions should not exceed the number of available processor cores.

4. Select **Show Advanced Controls**.

5. Click the **Partitioner** tab at the top of the dialog box.

6. Use the default `MeTiS` partitioner.

   Your model will be divided into a number of sections, with each section running in its own CFX-Solver process. The default is the `MeTiS` partitioner because it produces more efficient partitions than either `Recursive Coordinate Bisection` or `User Specified Direction`.

7. Click **Start Run**.

8. When CFX-Solver is finished, select the check box next to **Post-Process Results**.

9. If using stand-alone mode, select the check box next to **Shut down CFX-Solver Manager**.

10. Click **OK**.

Continue this tutorial from Text Output when Running in Parallel (p. 199).

8.5.2.3. Obtaining a Solution with Distributed Parallel

Before running in Distributed Parallel mode, ensure that your system has been configured as described in the installation documentation.

In CFX-Solver Manager, the **Define Run** dialog box should already be open.

1. Leave **Type of Run** set to `Full`.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If **Type of Run** was instead set to `Partitioner Only`, your mesh would be split into a number of partitions but would not be run in the CFX-Solver afterwards.

2. Set **Run Mode** to a parallel mode suitable for your environment; for example, `Intel MPI Distributed Parallel`.

   The name of the machine that you are currently logged into should be in the **Host Name** list. You are going to run with two partitions on two different machines, so another machine must be added.

3. Click *Insert Host* to specify a new host machine.

   - The **Select Parallel Hosts** dialog box is displayed. This is where you choose additional machines to run your processes.
   - Your system administrator should have set up a hosts file containing a list of the machines that are available to run the parallel CFX-Solver.
   - The **Host Name** column displays names of available hosts.
   - The second column shows the number of processors on that machine.
   - The third shows the relative processor speed: a processor on a machine with a relative speed of 1 would typically be twice as fast as a machine with a relative speed of 0.5.
   - The last column displays operating system information.
   - This information is read from the hosts file; if any information is missing or incorrect your system administrator should correct the hosts file.

   Note:

   The # processors, relative speed and system information does not have to be specified to be able to run on a host.

4. Select the name of another machine in the **Host Name** list.

   Select a machine that you can log into.

5. Click **Add**.

   The name of the machine is added to the **Host Name** column.

   Note:

   Ensure that the machine that you are currently logged into is in the **Hosts Name** list in the **Define Run** dialog box.

6. **Close** the **Select Parallel Hosts** dialog box.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Select **Show Advanced Controls**.
8. Click the **Partitioner** tab at the top of the dialog box.
9. Use the default `MeTiS` partitioner.

   Your model will be divided into two sections, with each section running in its own CFX-Solver process. The default is the `MeTiS` partitioner because it produces more efficient partitions than either `Recursive Coordinate Bisection` or `User Specified Direction`.
10. Click **Start Run** to begin the parallel run.
11. Click **OK** on the dialog box.
12. When CFX-Solver is finished, select the check box next to **Post-Process Results**.
13. If using stand-alone mode, select the check box next to **Shut down CFX-Solver Manager**.
14. Click **OK**.

8.5.2.4. Text Output when Running in Parallel

The text output area shows what is being written to the CFX-Solver Output file. You will see information similar to the following:

```
+--------------------------------------------------------------------+
|                  Job Information at Start of Run                   |
+--------------------------------------------------------------------+

Run mode:       partitioning run

+------------------------------+------+--------+----------+----------+
| Host                         | Mesh |  PID   |     Job Started     |
|                              | Part |        | DD/MM/YY | hh:mm:ss |
+------------------------------+------+--------+----------+----------+
| fastmachine1                 |    1 |   5952 | 19/02/13 | 10:50:09 |
+------------------------------+------+--------+----------+----------+
```

This tells you that the information following is concerned with the partitioning. After the partitioning job has finished, you will find:

```


+--------------------------------------------------------------------+
|                      Partitioning Information                      |
+--------------------------------------------------------------------+

Partitioning information for domain: BluntBody

       +------------------+------------------------+-----------------+
       |     Elements     |        Vertices        |     Faces       |
+------+------------------+------------------------+-----------------+
| Part |   Number     %   |   Number     %   %Ovlp |  Number     %   |
+------+------------------+------------------------+-----------------+
| Full |    131878        |     37048              |    11318        |
+------+------------------+------------------------+-----------------+
|    1 |     67873   50.4 |     19431   50.4   4.0 |     5705   49.5 |
|    2 |     66865   49.6 |     19151   49.6   4.0 |     5820   50.5 |
+------+------------------+------------------------+-----------------+
| Sum  |    134738  100.0 |     38582  100.0   4.0 |    11525  100.0 |
+------+------------------+------------------------+-----------------+

+--------------------------------------------------------------------+
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
|                Partitioning CPU-Time Requirements                |
+------------------------------------------------------------------+

- Preparations                                        3.689E-01 seconds
- Low-level mesh partitioning                         5.599E-02 seconds
- Global partitioning information                     9.998E-03 seconds
- Element and face partitioning information           7.999E-03 seconds
- Vertex partitioning information                     0.000E+00 seconds
- Partitioning information compression                0.000E+00 seconds
- Summed CPU-time for mesh partitioning               4.609E-01 seconds


+------------------------------------------------------------------+
|                  Job Information at End of Run                   |
+------------------------------------------------------------------+

+--------------------------+------+----------+----------+-----------+
| Host                     | Mesh |     Job Finished    |    CPU    |
|                          | Part | DD/MM/YY | hh:mm:ss |  seconds  |
+--------------------------+------+----------+----------+-----------+
| fastmachine1             |    1 | 19/02/13 | 10:50:09 | 8.020E-01 |
+--------------------------+------+----------+----------+-----------+

Total wall clock time: 7.990E-01 seconds
            or: (          0:         0:         0:     0.799 )
                (       Days:     Hours:   Minutes:   Seconds )
```

This marks the end of the partitioning job. The CFX-Solver now begins to solve your parallel run:

```
+------------------------------------------------------------------+
|                  Job Information at Start of Run                 |
+------------------------------------------------------------------+

Run mode:       parallel run (Intel MPI)

+------------------------------+------+--------+----------+----------+
| Host                         | Mesh |  PID   |     Job Started     |
|                              | Part |        | DD/MM/YY | hh:mm:ss |
+------------------------------+------+--------+----------+----------+
| fastmchine1                  |    1 |   6044 | 19/02/13 | 10:50:11 |
|                              |    2 |   3276 | 19/02/13 | 10:50:11 |
+------------------------------+------+--------+----------+----------+
```

The machine that you are logged into runs the leader process, and controls the overall simulation. The second machine selected will run the follower process. If you had more than two processes, each additional process is run as a follower process.

The leader process in this example is running on the mesh partition number 1 and the follower is running on partition number 2. You can find out which nodes and elements are in each partition by using CFD-Post later on in the tutorial.

When the CFX-Solver finishes, the CFX-Solver Output file displays the job information and a dialog box to indicate completion of the run.

8.6. Viewing the Results Using CFD-Post

In CFD-Post, you will:

- Create an instance transform object to recreate the full geometry
- Create a vector plot that shows how the flow behaves around the body

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Create a pressure plot that shows the pressure distribution on the body
- Make a surface streamline that shows the path of air along the surface of the body
- Examine the values of the dimensionless wall distance y^+ to make sure that the mesh is sufficiently fine near the walls

8.6.1. Using Symmetry Plane to Display the Full Geometry

Earlier in this tutorial you used a symmetry plane boundary because the entire blunt body is symmetrical about a plane. Due to this symmetry, it was necessary to use only half of the full geometry to calculate the CFD results. However, for visualization purposes, it is helpful to use the full blunt body. CFD-Post is able to recreate the full data set from the half that was originally calculated. This is done by creating an **Instance Transform** object.

8.6.1.1. Manipulating the Geometry

You need to manipulate the geometry so that you will be able to see what happens when you use the symmetry plane. The CFD-Post features that you have used in earlier tutorials will not be described in detail. New features will be described in detail.

- Right-click a blank area in the viewer and select **Predefined Camera** > **View From +X**.

8.6.1.2. Creating an Instance Transform

Instance Transforms are used to visualize a full geometry representation in cases where the simulation took advantage of symmetry to solve for only part of the geometry. There are three types of transforms that you can use: Rotation, Translation, Reflection/Mirroring. In this tutorial, you will create a Reflection transform located on a plane.

1. Click **Location** > **Plane** and set the name to `Reflection Plane`.
2. Configure the following setting(s):

Tab	Setting	Value
Geometry	Definition > **Method**	ZX Plane
	Definition > **Y**	0.0 [m]
Render	**Show Faces**	(cleared)

3. Click **Apply**.

   This creates a plane at y=0, the same location as the symmetry plane defined in CFX-Pre. Now the instance transform can be created using this plane:

4. From the main menu, select **Insert** > **Instance Transform** and accept the default name.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Configure the following setting(s):

Tab	Setting	Value
Definition	**Instancing Info From Domain**	(Cleared)
	Apply Rotation	(Cleared)
	Apply Reflection	(Selected)
	Apply Reflection > **Plane**	 Reflection Plane

6. Click **Apply**.

8.6.1.3. Using the Reflection Transform

You can apply the new transform to graphics objects. For example, you can modify the display of the wireframe as follows:

1. Under the **Outline** tab, in `User Locations and Plots`, configure the following setting(s) of `Wireframe`:

Tab	Setting	Value
View	Apply Instancing Transform > **Transform**	 Instance Transform 1

2. Click **Apply**.

3. Zoom so that the geometry fills the Viewer.

   You will see the full blunt body.

In this case, you created a new instance transform and applied it to the wireframe. This caused only the wireframe object to be mirrored. If you had modified the default transform instead of creating a new one, then all graphics (including those not yet made) would be mirrored by default.

8.6.2. Creating Velocity Vectors

You are now going to create a vector plot to show velocity vectors behind the blunt body. You need to first create an object to act as a locator, which, in this case, will be a sampling plane. Then, create the vector plot itself.

8.6.2.1. Creating the Sampling Plane

A sampling plane is a plane with evenly spaced sampling points on it.

1. Right-click a blank area in the viewer and select **Predefined Camera** > **View From +Y**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This ensures that the changes can be seen.

2. Create a new plane named `Sample`.

3. Configure the following setting(s) to create a sampling plane that is parallel to the ZX plane and located at x= 6 m, y= 0.001 m and z= 1 m relative to blunt object:

Tab	Setting	Value
Geometry	Definition > **Method**	Point and Normal
	Definition > **Point**	6, -0.001, 1
	Definition > **Normal**	0, 1, 0
	Plane Bounds > **Type**	Rectangular
	Plane Bounds > **X Size**	2.5 [m]
	Plane Bounds > **Y Size**	2.5 [m]
	Plane Type	Sample
	Plane Type > **X Samples**	20
	Plane Type > **Y Samples**	20
Render	**Show Faces**	(Cleared)
	Show Mesh Lines	(Selected)

4. Click **Apply**.

   You can zoom in on the sampling plane to see the location of the sampling points (where lines intersect). There are a total of 400 (20 * 20) sampling points on the plane. A vector can be created at each sampling point.

5. Turn off the visibility of `Sample`.

8.6.2.2. Creating a Vector Plot Using Different Sampling Methods

1. Click *Vector* and accept the default name.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Configure the following setting(s):

Tab	Setting	Value
Geometry	Definition > **Locations**	Sample
	Definition > **Sampling**	Vertex
Symbol	**Symbol Size**	0.25

3. Click **Apply**.

4. Zoom until the vector plot is roughly the same size as the viewer.

   You should be able to see a region of recirculation behind the blunt body.

5. Ignore the vertices on the sampling plane and increase the density of the vectors by applying the following settings:

Tab	Setting	Value
Geometry	Definition > **Sampling**	Equally Spaced
	Definition > **# of Points**	1000

6. Click **Apply**.

7. Change the location of the Vector plot by applying the following setting:

Tab	Setting	Value
Geometry	Definition > **Locations**	SymP

8. Click **Apply**.

8.6.3. Displaying Pressure Distribution on Body and Symmetry Plane

1. Configure the following setting(s) of the boundary named Body:

Tab	Setting	Value
Color	**Mode**	Variable
	Variable	Pressure
View	Apply Instancing Transform	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Transform**	Instance Transform 1

2. Click **Apply**.

3. Configure the following setting(s) of SymP:

Tab	Setting	Value
Render	**Show Faces**	(Cleared)
	Show Mesh Lines	(Selected)

4. Click **Apply**.

   You will be able to see the mesh around the blunt body, with the mesh length scale decreasing near the body, but still coarse in the region of recirculation. By zooming in, you will be able to see the layers of inflated elements near the body.

8.6.4. Creating Surface Streamlines to Display the Path of Air along the Surface of the Body

In order to show the path of air along the surface of the blunt body, surface streamlines can be made as follows:

1. Turn off the visibility of Body, SymP and Vector 1.

2. Create a new plane named Starter.

3. Configure the following setting(s):

Tab	Setting	Value
Geometry	Definition > **Method**	 YZ Plane
	X	-0.1 [m]

4. Click **Apply**.

5. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)**.

   The plane appears just upstream of the blunt body.

6. Turn off the visibility of the plane.

   This hides the plane from view, although the plane still exists.

7. Click *Streamline* and click **OK** to accept the default name.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. Configure the following setting(s):

Tab	Setting	Value
Geometry	**Type**	Surface Streamline
	Definition > **Surfaces**	Body
	Definition > **Start From**	Locations
	Definition > **Locations**	Starter
	Definition > **Max Points**	100
	Definition > **Direction**	Forward

9. Click **Apply**.

The surface streamlines appear on half of the surface of the blunt body. They start near the upstream end because the starting points were formed by projecting nodes from the plane to the blunt body.

8.6.5. Moving Objects

In CFD-Post, you can reposition some locator objects directly in the viewer by using the mouse.

1. Turn on the visibility of the plane named `Starter`.

2. Select the **Single Select** mouse pointer from the **Selection Tools** toolbar.

3. In the viewer, click the `Starter` plane to select it, then drag it along the X axis.

   Notice that the streamlines are redrawn as the plane moves. The rate at which the streamlines are redrawn is dependent on your computer's speed. If the streamlines are updated infrequently, you may find it useful to move the mouse very slowly.

8.6.6. Creating a Surface Plot of y+

The velocity next to a no-slip wall boundary changes rapidly from a value of zero at the wall to the free stream value a short distance away from the wall. This layer of high velocity gradient is known as the boundary layer. Many meshes are not fine enough near a wall to accurately resolve the velocity profile in the boundary layer. Wall functions can be used in these cases to apply an assumed functional shape of the velocity profile. Other grids are fine enough that they do not require wall functions, and application of the latter has little effect. The majority of cases fall somewhere in between these two extremes, where the boundary layer is partially resolved by nodes near the wall and wall functions are used to supplement accuracy where the nodes are not sufficiently clustered near the wall.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

One indicator of the closeness of the first node to the wall is the dimensionless wall distance y^+. It is good practice to examine the values of y^+ at the end of your simulation. At the lower limit, a value of y^+ less than or equal to 11 indicates that the first node is within the laminar sublayer of the boundary flow. Values larger than this indicate that an assumed logarithmic shape of the velocity profile is being used to model the boundary layer portion between the wall and the first node. Ideally you should confirm that there are several nodes (3 or more) resolving the boundary layer profile. If this is not observed, it is highly recommended that more nodes be added near the wall surfaces in order to improve simulation accuracy. In this tutorial, a coarse mesh is used to reduce the run time. Thus, the grid is far too coarse to resolve any of the boundary layer profile, and the solution is not highly accurate.

A surface plot is one which colors a surface according to the values of a variable: in this case, y^+. A surface plot of y^+ can be obtained as follows:

1. Turn off the visibility of all previous plots.
2. Under the **Outline** tab, configure the following settings of `BluntBodyDefault`:

Tab	Setting	Value
Color	**Mode**	Variable
	Variable	Yplus [a]
View	Apply Instancing Transform > **Transform**	 Instance Transform 1

a. Click the *Ellipsis* ... icon to the right of the **Variable** dropdown menu to view a full list of variables, including **Yplus**.

3. Click **Apply**.
4. Under the **Outline** tab, configure the following settings of `Body`:

Tab	Setting	Value
Color	**Mode**	Variable
	Variable	Yplus
View	Apply Instancing Transform > **Transform**	 Instance Transform 1

5. Click **Apply**.

8.6.7. Viewing the Mesh Partitions (Parallel Only)

If you solved this tutorial in parallel, then an additional variable named `Real partition number` will be available in CFD-Post

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Create an isosurface of `Real partition number` equal to `1`.
2. Create a second isosurface of `Real partition number` equal to `1.999`.

The two isosurfaces show the edges of the two partitions. The gap between the two plots shows the overlap nodes. These were contained in both partitions 1 and 2.

When you have finished looking at the results, quit CFD-Post.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 9: Free Surface Flow Over a Bump

This tutorial includes:

9.1. Tutorial Features

9.2. Overview of the Problem to Solve

9.3. Preparing the Working Directory

9.4. Defining the Case Using CFX-Pre

9.5. Obtaining the Solution Using CFX-Solver Manager

9.6. Viewing the Results Using CFD-Post

9.7. Further Discussion

9.1. Tutorial Features

In this tutorial you will set up a 2D problem in which you:

- Import a mesh.
- Set up appropriate boundary conditions for a free surface simulation. (Free surface simulations are more sensitive to incorrect boundary and initial guess settings than other more basic models.)
- Use mesh adaption to refine the mesh where the volume fraction gradient is greatest. (The refined mesh aids in the development of a sharp interface between the liquid and gas.)

Component	Feature	Details
CFX-Pre	User Mode	General mode
	Analysis Type	Steady-state
	Fluid Type	General Fluid
	Domain Type	Single Domain
	Turbulence Model	k-Epsilon
	Heat Transfer	Isothermal
	Buoyant Flow	
	Multiphase	Homogeneous Model
	Boundary Conditions	Inlet
		Opening
		Outlet
		Symmetry Plane
		Wall: No Slip

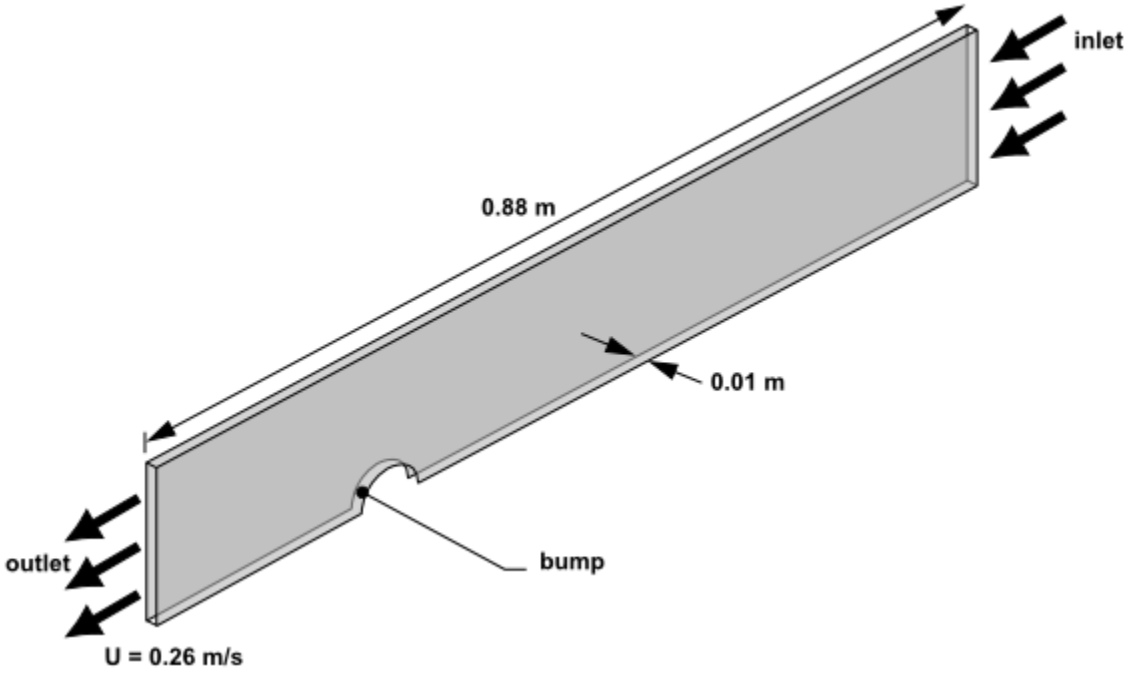
Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Component	Feature	Details
	CEL (CFX Expression Language)	
	Mesh Adaption	
	Timestep	Physical Time Scale
CFD-Post	Plots	Default Locators
		Isosurface
		Polyline
		Sampling Plane
		Vector
		Volume
	Other	Chart Creation
		Title/Text
		Viewing the Mesh

9.2. Overview of the Problem to Solve

This tutorial demonstrates the simulation of a free surface flow.

The geometry consists of a 2D channel in which the bottom of the channel is interrupted by a semicircular bump of radius 30 mm. The problem environment is composed of air at 1 Pa and isothermal water; the normal inlet speed is 0.26 m/s; the incoming water has a turbulence intensity of 5%. The flow upstream of the bump is subcritical. The downstream boundary conditions (the height of the water) were estimated for this tutorial; you can do this using an analytical 1D calculation or data tables for flow over a bump.



Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A mesh is provided. You will create a two-phase homogeneous setting and the expressions that will be used in setting initial values and boundary conditions. Later, you will use mesh adaption to improve the accuracy of the downstream simulation.

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- Running Ansys CFX Tutorials Using Ansys Workbench (p. 34)
- Changing the Display Colors (p. 37)

9.3. Preparing the Working Directory

1. Create a working directory.

   Ansys CFX uses a working directory as the default location for loading and saving files for a particular session or project.

2. Download the `bump2d.zip` file here.

3. Unzip `bump2d.zip` to your working directory.

   Ensure that the following tutorial input files are in your working directory:

   - `Bump2DExpressions.ccl`
   - `Bump2Dpatran.out`

4. Set the working directory and start CFX-Pre.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

9.4. Defining the Case Using CFX-Pre

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `Bump2D`.
5. Click **Save**.

9.4.1. Importing the Mesh

1. Right-click `Mesh` and select **Import Mesh** > **Other**.

   The **Import Mesh** dialog box appears.

2. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
Files of type	PATRAN Neutral (*out *neu)
Filename	Bump2Dpatran.out
Options > **Mesh units**	 m

3. Click **Open**.

4. To best orient the view, right-click a blank area in the viewer and select **Predefined Camera** > **View From -Z** from the shortcut menu.

9.4.2. Viewing the Region Labels

Enable the display of region labels so that you can see where you will define boundaries later in this tutorial:

1. In the **Outline** tree view, edit `Case Options` > `Labels and Markers`.

2. Configure the following setting(s):

Tab	Setting	Value
Settings	**Show Labels**	(Selected)
	Show Labels > **Show Primitive 3D Labels**	 (Selected)
	Show Labels > **Show Primitive 2D Labels**	 (Selected)

3. Click **OK**.

9.4.3. Creating Expressions for Initial and Boundary Conditions

Simulation of free surface flows usually requires defining boundary and initial conditions to set up appropriate pressure and volume fraction fields. You will need to create expressions using CEL (CFX Expression Language) to define these conditions.

In this simulation, the following conditions are set and require expressions:

- An inlet boundary where the volume fraction above the free surface is 1 for air and 0 for water, and below the free surface is 0 for air and 1 for water.

- A pressure-specified outlet boundary, where the pressure above the free surface is constant and the pressure below the free surface is a hydrostatic distribution. This requires you to know the approximate height of the fluid at the outlet. In this case, an analytical solution for 1D flow over a bump was used to determine the value for `DownH` in Creating Expressions in CEL (p. 213). The simulation is not sensitive to the exact outlet fluid height, so an approximation is sufficient. You

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

will examine the effect of the outlet boundary condition in the postprocessing section and confirm that it does not affect the validity of the results. It is necessary to specify such a boundary condition to force the flow downstream of the bump into the supercritical regime.

- An initial pressure field for the domain with a similar pressure distribution to that of the outlet boundary.

Either create expressions using the **Expressions** workspace or read in expressions from the example file provided:

- Creating Expressions in CEL (p. 213)
- Reading Expressions From a File (p. 214)

9.4.3.1. Creating Expressions in CEL

The expressions you create in this step are the same as the ones provided in Reading Expressions From a File (p. 214), so you can choose to follow either set of instructions.

1. Right-click `Expressions, Functions and Variables` > `Expressions` in the tree view and select **Insert** > **Expression**.
2. Set the name to `UpH` and click **OK** to create the upstream free surface height.
3. Set **Definition** to `0.069 [m]`, and then click **Apply**.
4. Use the same method to create the expressions listed in the table below. These are expressions for the downstream free surface height, the fluid density, the buoyancy reference density, the calculated density of the fluid (density - buoyancy reference density), the upstream volume fractions of air and water, the upstream pressure distribution, the downstream volume fractions of air and water, and the downstream pressure distribution.

Name	Definition
DownH	0.022 [m]
DenWater	997 [kg m^-3]
DenRef	1.185 [kg m^-3]
DenH	(DenWater - DenRef)
UpVFAir	step((y-UpH)/1[m])
UpVFWater	1-UpVFAir
UpPres	DenH*g*UpVFWater*(UpH-y)
DownVFAir	step((y-DownH)/1[m])
DownVFWater	1-DownVFAir
DownPres	DenH*g*DownVFWater*(DownH-y)

5. Proceed to Creating the Domain (p. 214).

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.4.3.2. Reading Expressions From a File

1. Select **File** > **Import** > **CCL**.
2. In the **Import CCL** dialog box, ensure that the **Append** option is selected.
3. Select `Bump2DExpressions.ccl`.
4. Click **Open**.
5. After the file has been imported, use the **Expressions** tree view to view the expressions that have been created.

9.4.4. Creating the Domain

Set up a homogeneous, two-fluid environment:

1. Edit `Case Options` > `General` in the **Outline** tree view and ensure that **Automatic Default Domain** is turned on. A domain named `Default Domain` should appear under the `Simulation` > `Flow Analysis 1` branch.
2. Double-click **Default Domain**.
3. Under **Fluid and Particle Definitions**, delete `Fluid 1` and create a new fluid named `Air`.
4. Confirm that the following settings are configured:

Tab	Setting	Value
Basic Settings	**Fluid and Particle Definitions**	Air
	Fluid and Particle Definitions > Air > **Material**	 Air at 25 C

5. Click *Add new item* and create a new fluid named `Water`.
6. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Fluid and Particle Definitions**	Water
	Fluid and Particle Definitions > Water > **Material**	 Water [a]
	Domain Models > Pressure	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Reference Pressure**	1 [atm]
	Domain Models > Buoyancy Model > **Option**	Buoyant
	Domain Models > Buoyancy Model > **Gravity X Dirn.**	0 [m s^-2]
	Domain Models > Buoyancy Model > **Gravity Y Dirn.**[b]	-g
	Domain Models > Buoyancy Model > **Gravity Z Dirn.**	0 [m s^-2]
	Domain Models > Buoyancy Model > **Buoy. Ref. Density** [c]	DenRef
Fluid Models	Multiphase > **Homogeneous Model** [d]	(Selected)
	Multiphase > Free Surface Model > **Option**	Standard
	Heat Transfer > **Option**	Isothermal
	Heat Transfer > **Fluid Temperature**	25 [C]
	Turbulence > **Option**	k-Epsilon

a. The models selected here describe how the fluids interact. No mass transfer between the phases occurs in this example. You do not need to model surface tension.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
b. You need to click *Enter Expression* beside the field first. c. Always set Buoyancy Reference Density to the density of the least dense fluid in free surface calculations. d. The homogeneous model solves for a single solution field.		

7. Click **OK**.

9.4.5. Creating the Boundaries

9.4.5.1. Inlet Boundary

1. Create a new boundary named `inflow`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Inlet
	Location	INFLOW
Boundary Details	Mass and Momentum > **Option**	Normal Speed
	Mass and Momentum > **Normal Speed**	0.26 [m s^-1]
	Turbulence > **Option**	Intensity and Length Scale
	Turbulence > **Fractional Intensity**	0.05
	Turbulence > **Eddy Length Scale**	UpH
Fluid Values	**Boundary Conditions**	Air
	Boundary Conditions > Air > Volume Fraction > **Volume Fraction**	UpVFAir
	Boundary Conditions	Water

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Boundary Conditions > Water > Volume Fraction > **Volume Fraction**	 UpVFWater

3. Click **OK**.

9.4.5.2. Outlet Boundary

1. Create a new boundary named `outflow`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Outlet
	Location	OUTFLOW
Boundary Details	Flow Regime > **Option**	 Subsonic
	Mass and Momentum > **Option**	 Static Pressure
	Mass and Momentum > **Relative Pressure**	 DownPres

3. Click **OK**.

9.4.5.3. Symmetry Boundaries

1. Create a new boundary named `front`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Symmetry [a]
	Location	FRONT

a. Symmetry, which makes a 3D problem into a 2D problem, can be used when geometry and mesh are invariant normal to the symmetry surface.

3. Click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Create a new boundary named `back`.
5. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Symmetry
	Location	BACK

6. Click **OK**.

9.4.5.4. Opening and Wall Boundaries

1. Create a new boundary named `top`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Opening
	Location	TOP
Boundary Details	Mass And Momentum > **Option**	Entrainment
	Mass And Momentum > **Relative Pressure**	0 [Pa]
	Turbulence > **Option**	Zero Gradient
Fluid Values	**Boundary Conditions**	Air
	Boundary Conditions > Air > Volume Fraction > **Volume Fraction**	1.0
	Boundary Conditions	Water
	Boundary Conditions > Water > Volume Fraction > **Volume Fraction**	0.0

3. Click **OK**.
4. Create a new boundary named `bottom`.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Wall
	Location	BOTTOM1, BOTTOM2, BOTTOM3
Boundary Details	Mass and Momentum > **Option**	No Slip Wall
	Wall Roughness > **Option**	Smooth Wall

6. Click **OK**.

9.4.6. Setting Initial Values

Set up the initial values to be consistent with the inlet boundary conditions:

1. Click *Global Initialization* .

2. Configure the following setting(s):

Tab	Setting	Value
Global Settings	Initial Conditions > Cartesian Velocity Components > **Option**	Automatic with Value
	Initial Conditions > Cartesian Velocity Components > **U**	0.26 [m s^-1] [a]
	Initial Conditions > Cartesian Velocity Components > **V**	0 [m s^-1] [a]
	Initial Conditions > Cartesian Velocity Components > **W**	0 [m s^-1] [a]
	Initial Conditions	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Static Pressure > **Option**	Automatic with Value
	Initial Conditions > Static Pressure > **Relative Pressure**	UpPres
Fluid Settings	**Fluid Specific Initialization**	Air
	Fluid Specific Initialization > Air > Initial Conditions > Volume Fraction > **Option**	Automatic with Value
	Fluid Specific Initialization > Air > Initial Conditions > Volume Fraction > **Volume Fraction**	UpVFAir
	Fluid Specific Initialization	Water
	Fluid Specific Initialization > Water > Initial Conditions > Volume Fraction > **Option**	Automatic with Value
	Fluid Specific Initialization > Water > Initial Conditions > Volume Fraction > **Volume Fraction**	UpVFWater

a. From the problem statement.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click **OK**.

9.4.7. Setting Mesh Adaption Parameters

To improve the resolution of the interface between the air and the water, set up the mesh adaption settings:

1. Click *Mesh Adaption* .

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Activate Adaption**	(Selected)
	Save Intermediate Files	(Cleared)
	Adaption Criteria > **Variables List**	Air.Volume Fraction
	Adaption Criteria > **Max. Num. Steps**	2
	Adaption Criteria > **Option**	Multiple of Initial Mesh
	Adaption Criteria > **Node Factor**	4
	Adaption Convergence Criteria > **Max. Iter. per Step**	100
Advanced Options	**Node Alloc. Parameter**	1.6
	Number of Levels	2

3. Click **OK**.

9.4.8. Setting the Solver Controls

Note:

Setting **Max. Iterations** to 200 (below) and **Number of (Adaption) Levels** to 2 with a **Max. Iter. per Step** of 100 time steps each (in the previous section), results in a total maximum number of time steps of 400 (2*100+200=400).

1. Click *Solver Control* .

2. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Basic Settings	Convergence Control > **Max. Iterations**	200
	Convergence Control > Fluid Timescale Control > **Timescale Control**	Physical Timescale
	Convergence Control > Fluid Timescale Control > **Physical Timescale**	0.25 [s] [a]
Advanced Options	**Multiphase Control**	(Selected)
	Multiphase Control > **Volume Fraction Coupling**	(Selected)
	Multiphase Control > Volume Fraction Coupling > **Option**	Coupled [b]

a. Estimated from the time it takes the water to flow over the bump.

b. The Coupled Volume Fraction solution algorithm typically converges better than the Segregated Volume Faction algorithm for buoyancy-driven problems. The Segregated Volume Faction algorithm would have required a significantly smaller timescale (0.05 [s]).

Note:

Selecting these options on the solver control activates the Coupled Volume Fraction solution algorithm. This algorithm typically converges better than the Segregated Volume Faction algorithm for buoyancy-driven problems such as this tutorial. The Segregated Volume Faction algorithm would have required a 0.05 second timescale, as compared with 0.25 seconds for the Coupled Volume Fraction algorithm.

3. Click **OK**.

9.4.9. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .

2. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
File name	`Bump2D.def`

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, **Solver Input File** is set.

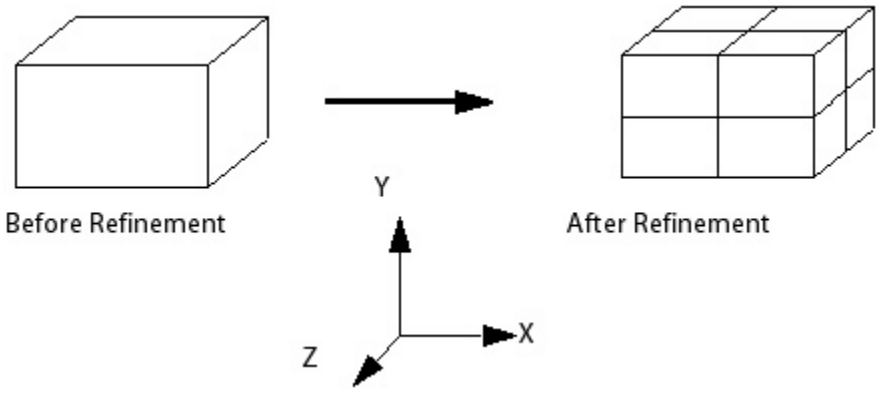
4. If using stand-alone mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

9.5. Obtaining the Solution Using CFX-Solver Manager

Click **Start Run**.

Within 100 iterations after CFX-Solver Manager has started, the first adaption step is performed. Information written to the CFX-Solver Output file includes the number of elements refined and the size of the new mesh.

After mesh refinement, there is a jump in the residual levels. This is because the solution from the old mesh is interpolated onto the new mesh. A new residual plot also appears for the W-Mom-Bulk equation. Hexahedral mesh elements are refined orthogonally, so the mesh is no longer 2D (it is more than 1 element thick in the Z direction). Convergence to the target residual level is achieved.



It is common for convergence in a residual sense to be difficult to obtain in a free surface simulation, due to the presence of small waves at the surface preventing the residuals from dropping to the target level. This is more frequently a problem in the subcritical flow regime, as the waves can travel upstream. In the supercritical regime, the waves tend to get carried downstream and out the domain. To satisfy convergence in these cases, monitor the value of a global quantity (for example, drag for flow around a ship's hull) to see when a steady-state value is reached.

Where there is no obvious global quantity to monitor, you should view the results to see where the solution is changing. You can do this by running transient (with time steps that are small enough to capture transient effects) for a few time steps, starting from a results file that you think is converged or from backup results files you have written at different time steps.

In both cases, look to see where the results are changing (this could be due to the presence of small transient waves). Also confirm that the value of quantities that you are interested in (for example, downstream fluid height for this case) has reached a steady-state value.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. When a dialog box is displayed at the end of the run, select **Post-Process Results**.
2. If using stand-alone mode, select **Shut down CFX-Solver Manager**.
3. Click **OK**.

9.6. Viewing the Results Using CFD-Post

Display the distribution of volume fraction of water in the domain:

1. To best orient the view, right-click a blank area in the viewer and select **Predefined Camera** > **View From -Z**.
2. Zoom in so the geometry fills the viewer.
3. In the tree view, edit `Bump2D_001` > `Default Domain` > `front`.
4. Configure the following setting(s):

Tab	Setting	Value
Color	**Mode**	Variable
	Variable	Water.Volume Fraction

5. Click **Apply**.
6. Observe the plot, then clear the check box next to `front`.

9.6.1. Creating Velocity Vector Plots

The next step involves creating a sampling plane upon which to display velocity vectors for `Water`.

1. Select **Insert** > **Location** > **Plane** to create a new plane named `Plane 1`.
2. Configure the following setting(s):

Tab	Setting	Value
Geometry	Definition > **Method**	XY Plane
	Plane Bounds > **Type**	Rectangular
	Plane Bounds > **X Size**	1.25 [m] [a]
	Plane Bounds > **Y Size**	0.3 [m]

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Plane Bounds > **X Angle**	 0 [degree]
	Plane Type	Sample
	X Samples	160 [b]
	Y Samples	40
Render	**Show Faces**	(Cleared)
	Show Mesh Lines	(Selected)

a. The Plane Bounds settings overlap the plane with the wireframe. You can experiment with other values and click **Apply** to see the results.

b. The Plane Samples settings produce square elements. You can experiment with other values and click **Apply** to see the results.

3. Click **Apply**.
4. Clear the check box next to `Plane 1`.
5. Create a new vector named `Vector 1`.
6. Configure the following setting(s):

Tab	Setting	Value
Geometry	Definition > **Locations**	 Plane 1
	Definition > **Variable** [a]	 Water.Velocity
Symbol	**Symbol Size**	0.5

a. Because fluids in a free-surface calculation share the same velocity field, only the velocity of the first non-vapor fluid is available. The other allowed velocities are superficial velocities. For details, see Further Postprocessing (p. 231).

7. Click **Apply**.
8. Configure the following setting(s):

Tab	Setting	Value
Geometry	Definition > **Variable**	 Air.Superficial Velocity

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Symbol	**Symbol Size**	0.15
	Normalize Symbols	(Selected)

9. Click **Apply**.

9.6.2. Viewing Mesh Refinement

In this section, you will view the surface mesh on one of the symmetry boundaries, create volume objects to show where the mesh was modified, and create a vector plot to visualize the added mesh nodes.

1. Clear the check box next to `Vector 1`.
2. Zoom in so the geometry fills the Viewer.
3. In **Outline** under `Default Domain`, edit `front`.
4. Configure the following setting(s):

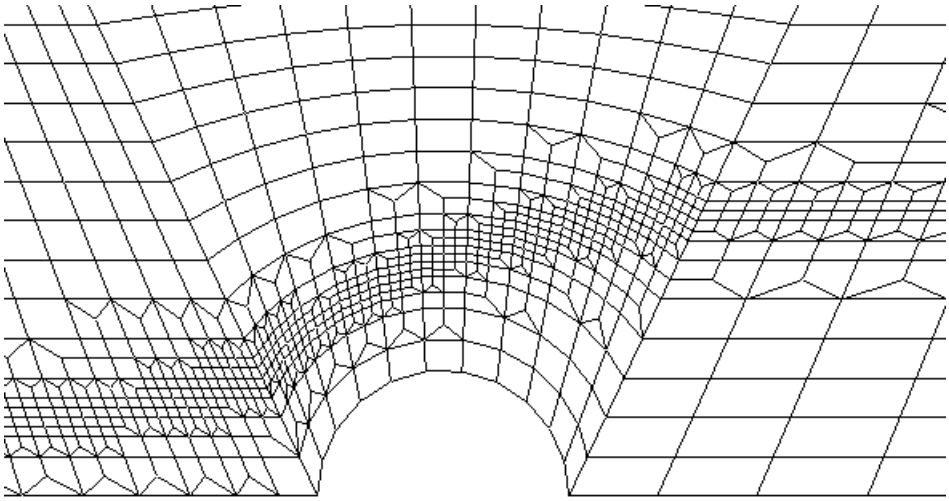
Tab	Setting	Value
Color	**Mode**	Constant
Render	**Show Faces**	(Cleared)
	Show Mesh Lines	(Selected)

5. Click **Apply**.

   - The mesh has been refined near the free surface.
   - In the transition region between different levels of refinement, tetrahedral and pyramidal elements are used because it is not possible to recreate hexahedral elements in CFX. Near the inlet, the aspect ratio of these elements increases.
   - Avoid performing mesh refinement on high-aspect-ratio hex meshes as this will produce high aspect ratio tetrahedral-elements and result in poor mesh quality.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 9.1: Mesh around the bump



6. Create a new volume named `first refinement elements`.

7. Configure the following setting(s):

Tab	Setting	Value
Geometry	Definition > **Method**	Isovolume
	Definition > **Variable**	Refinement Level [a]
	Definition > **Mode**	At Value
	Definition > **Value**	1
Render	**Show Faces**	(Cleared)
	Show Mesh Lines	(Selected)
	Show Mesh Lines > **Line Width**	2
	Show Mesh Lines > **Color Mode**	User Specified
	Show Mesh Lines > **Line Color**	(Green)

a. Click *More variables* ... to access the Refinement Level value.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. Click **Apply**.

   You will see a band of green, which indicates the elements that include nodes added during the first mesh adaption.

9. Create a new volume named `second refinement elements`.

10. Configure the following setting(s):

Tab	Setting	Value
Geometry	Definition > **Method**	Isovolume
	Definition > **Variable**	Refinement Level
	Definition > **Mode**	At Value
	Definition > **Value**	2
Color	**Color**	White
Render	**Show Faces**	(Selected)
	Show Mesh Lines	(Selected)
	Show Mesh Lines > **Line Width**	4
	Show Mesh Lines > **Color Mode**	User Specified
	Show Mesh Lines > **Line Color**	(Black)

11. Click **Apply**.

    You will see a band of white (with black lines); this indicates the elements that include nodes added during the second mesh adaption.

12. Zoom in to a region where the mesh has been refined.

    The Refinement Level variable holds an integer value at each node, which is either 0, 1, or 2 (because you used a maximum of two adaption levels).

    The nodal values of refinement level will be visualized next.

13. Create a new vector named `Vector 2`.

14. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Geometry	Definition > **Locations**	 Plane 1
	Definition > **Variable**	 (Any Vector Variable) [a]
Color	**Mode**	Variable
	Variable	Refinement Level
Symbol	**Symbol**	Cube
	Symbol Size	0.02
	Normalize Symbols	(Selected)

a. The variable's magnitude and direction do not matter because you will change the vector symbol to a cube with a normalized size.

15. Click **Apply**.

In `Vector 2`, Blue nodes (Refinement Level 0 according to the color legend) are part of the original mesh. Green nodes (Refinement Level 1) were added during the first adaption step. Red nodes (Refinement Level 2) were added during the second adaption step. Note that some elements contain combinations of blue, green, and red nodes.

9.6.3. Creating an Isosurface to Show the Free Surface

Later in this tutorial, you will create a chart to show the variation in free surface height along the channel. The data for the chart will be sampled along a polyline that follows the free surface. To make the polyline, you will use the intersection between one of the symmetry planes and an isosurface that follows the free surface. Start by creating an isosurface on the free surface:

1. Turn off the visibility for all objects except `Wireframe`.

2. Create a new isosurface named `Isosurface 1`.

3. Configure the following setting(s):

Tab	Setting	Value
Geometry	Definition > **Variable**	 Water.Volume Fraction
	Definition > **Value**	 0.5

4. Click **Apply**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating isosurfaces using this method is a good way to visualize a free surface in a 3D simulation.

5. Right-click any blank area in the viewer, select **Predefined Camera**, then select **Isometric View (Y up)**.

9.6.4. Creating a Polyline that Follows the Free Surface

Create a polyline along the isosurface that you created in the previous step:

1. Turn off the visibility of `Isosurface 1`.
2. Create a new polyline named `Polyline 1`.
3. Configure the following setting(s):

Tab	Setting	Value
Geometry	**Method**	Boundary Intersection
	Boundary List	front
	Intersect With	Isosurface 1

4. Click **Apply**.

   A green line is displayed that follows the high-Z edge of the isosurface.

9.6.5. Creating a Chart to Show the Height of the Surface

Create a chart that plots the free surface height using the polyline that you created in the previous step:

1. Create a new chart named `Chart 1`.

   The **Chart Viewer** is displayed.

2. Configure the following setting(s):

Tab	Setting	Value
General	**Title**	Free Surface Height for Flow over a Bump
Data Series	**Name**	free surface height
	Location	Polyline 1
X Axis	**Variable**	X
Y Axis	**Variable**	Y
Line Display	**Symbols**	Rectangle

3. Click **Apply**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

As discussed in Creating Expressions for Initial and Boundary Conditions (p. 212), an approximate outlet elevation is imposed as part of the boundary, even though the flow is supercritical. The chart illustrates the effect of this, in that the water level rises just before the exit plane. It is evident from this plot that imposing the elevation does not affect the upstream flow.

The chart shows a wiggle in the elevation of the free surface interface at the inlet. This is related to an over-specification of conditions at the inlet because both the inlet velocity and elevation were specified. For a subcritical inlet, only the velocity or the total energy should be specified. The wiggle is due to a small inconsistency between the specified elevation and the elevation computed by the solver to obtain critical conditions at the bump. The wiggle is analogous to one found if pressure and velocity were both specified at a subsonic inlet in a converging-diverging nozzle with choked flow at the throat.

9.6.6. Further Postprocessing

You may want to create some plots using the `<Fluid>.Superficial Velocity` variables. This is the fluid volume fraction multiplied by the fluid velocity and is sometimes called the volume flux. It is useful to use this variable for vector plots in separated multiphase flow, as you will only see a vector where a significant amount of that phase exists.

Tip:

You can right-click an existing vector plot and select a new vector variable.

9.7. Further Discussion

For supercritical free surface flows, the supercritical outlet boundary is usually the most appropriate boundary for the outlet because it does not rely on the specification of the outlet pressure distribution (which depends on an estimate of the free surface height at the outlet). The supercritical outlet boundary requires a relative pressure specification for the gas only; no pressure information is required for the liquid at the outlet. For this tutorial, the relative gas pressure at the outlet should be set to 0 Pa.

The supercritical outlet condition may admit multiple solutions. To find the supercritical solution, it is often necessary to start with a static pressure outlet condition (as previously done in this tutorial) or an average static pressure condition where the pressure is set consistent with an elevation to drive the solution into the supercritical regime. The outlet condition can then be changed to the supercritical option.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 10: Supersonic Flow Over a Wing

This tutorial includes:

10.1. Tutorial Features

10.2. Overview of the Problem to Solve

10.3. Preparing the Working Directory

10.4. Defining the Case Using CFX-Pre

10.5. Obtaining the Solution Using CFX-Solver Manager

10.6. Viewing the Results Using CFD-Post

10.1. Tutorial Features

In this tutorial you will learn about:

- Setting up a supersonic flow simulation.
- Using the Shear Stress Transport turbulence model to accurately resolve flow around a wing surface.
- Defining a custom vector to display pressure distribution.

Component	Feature	Details
CFX-Pre	User Mode	General mode
	Analysis Type	Steady State
	Fluid Type	Air Ideal Gas
	Domain Type	Single Domain
	Turbulence Model	Shear Stress Transport
	Heat Transfer	Total Energy
	Boundary Conditions	Inlet (Supersonic)
		Outlet (Supersonic)
		Symmetry Plane
		Wall: No-Slip
		Wall: Adiabatic
		Wall: Free-Slip
	Domain Interfaces	Fluid-Fluid (No Frame Change)
	Timestep	Maximum Timescale
CFD-Post	Plots	Contour
		Vector

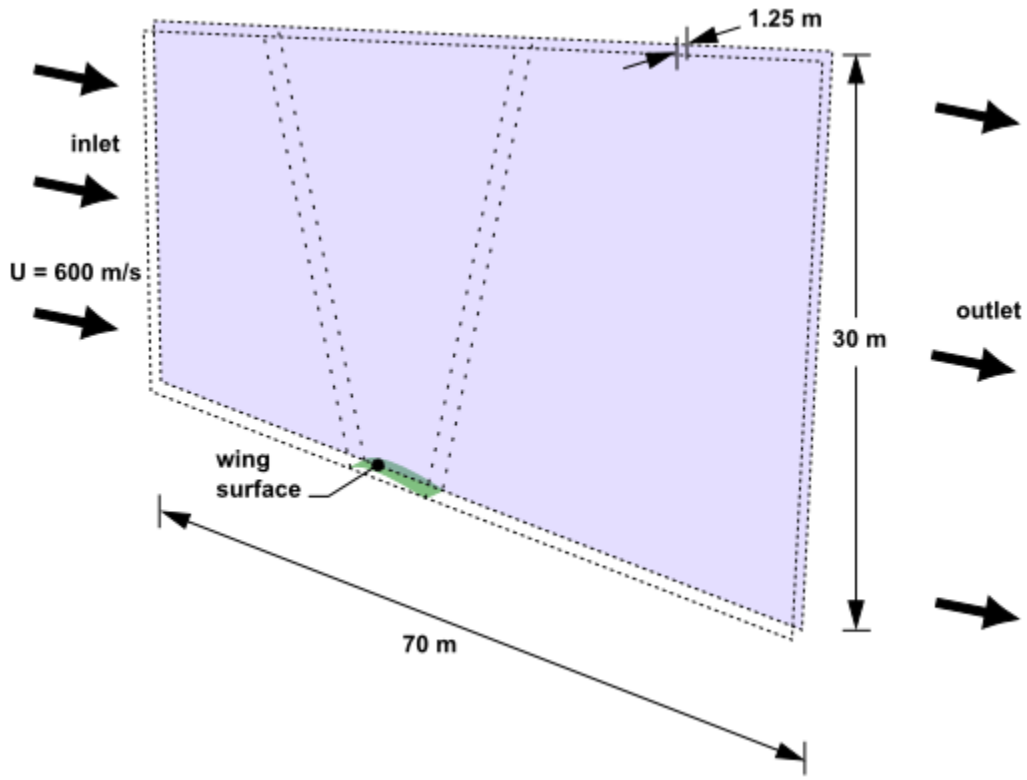
Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Component	Feature	Details
	Other	Variable Details View

10.2. Overview of the Problem to Solve

This example demonstrates the use of CFX in simulating supersonic flow over a symmetric NACA0012 airfoil at 0° angle of attack. A 2D section of the wing is modeled. A 2D hexahedral mesh is provided that you will import into CFX-Pre.

The environment is 300 K air at 1 atmosphere that passes the wing at 600 m/s. The turbulence intensity is low (.01) with an eddy length scale of .02 meters.



A mesh is provided. You will create a domain that contains three regions that will be connected by fluid-fluid interfaces. To solve the simulation, you will start with a conservative time scale that gradually increases towards the fluid residence time as the residuals decrease.

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- Running Ansys CFX Tutorials Using Ansys Workbench (p. 34)
- Changing the Display Colors (p. 37)

10.3. Preparing the Working Directory

1. Create a working directory.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys CFX uses a working directory as the default location for loading and saving files for a particular session or project.

2. Download the `wing_sps.zip` file here.

3. Unzip `wing_sps.zip` to your working directory.

   Ensure that the following tutorial input file is in your working directory:

   - `WingSPSMesh.out`

4. Set the working directory and start CFX-Pre.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

10.4. Defining the Case Using CFX-Pre

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `WingSPS`.
5. Click **Save**.

10.4.1. Importing the Mesh

1. Right-click `Mesh` and select **Import Mesh** > **Other**.

   The **Import Mesh** dialog box appears.

2. Configure the following setting(s):

Setting	Value
Files of type	PATRAN Neutral (*out *neu)
File name	WingSPSMesh.out
Options > **Mesh Units**	 m

3. Click **Open**.
4. To best orient the view, right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Y up)** from the shortcut menu.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.4.2. Creating the Domain

1. Edit `Case Options` > `General` in the **Outline** tree view and ensure that **Automatic Default Domain** is turned on. A domain named `Default Domain` should now appear under the `Simulation` branch.

2. Edit `Default Domain` and configure the following setting(s):

Tab	Setting	Value
Basic Settings	Location and Type > **Location**	 WING_Elements
	Fluid and Particle Definitions	Fluid 1
	Fluid and Particle Definitions > Fluid 1 > **Material**	 Air Ideal Gas
	Domain Models > Pressure > **Reference Pressure** [a]	 1 [atm]
Fluid Models	Heat Transfer > **Option**	 Total Energy [b]
	Turbulence > **Option**	 Shear Stress Transport

a. When using an ideal gas, it is important to set an appropriate reference pressure because some properties depend on the absolute pressure level.

b. The Total Energy model is appropriate for high-speed flows because it includes kinetic energy effects.

3. Click **OK**.

10.4.3. Creating the Boundaries

10.4.3.1. Creating an Inlet Boundary

1. Create a new boundary named `Inlet`.

2. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Basic Settings	**Boundary Type**	Inlet
	Location	INLET
Boundary Details	Flow Regime > **Option**	Supersonic
	Mass And Momentum > **Option**	Cart. Vel. & Pressure
	Mass And Momentum > **Rel. Static Pres.**	0 [Pa]
	Mass And Momentum > **U**	600 [m s^-1]
	Mass And Momentum > **V**	0 [m s^-1]
	Mass And Momentum > **W**	0 [m s^-1]
	Turbulence > **Option**	Intensity and Length Scale
	Turbulence > **Fractional Intensity**	0.01
	Turbulence > **Eddy Length Scale**	0.02 [m]
	Heat Transfer > **Static Temperature**	300 [K]

3. Click **OK**.

10.4.3.2. Creating an Outlet Boundary

1. Create a new boundary named `Outlet`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Outlet
	Location	OUTLET

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Boundary Details	Flow Regime > **Option**	 Supersonic

3. Click **OK**.

10.4.3.3. Creating the Symmetry Plane Boundaries

1. Create a new boundary named `SymP1`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Symmetry [a]
	Location	SIDE1

a. Symmetry, which can make a 3D problem into a 2D problem, can be used when geometry and mesh are invariant normal to the symmetry surface.

3. Click **OK**.
4. Create a new boundary named `SymP2`.
5. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Symmetry
	Location	SIDE2

6. Click **OK**.
7. Create a new boundary named `Bottom`.
8. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Symmetry
	Location	BOTTOM

9. Click **OK**.

10.4.3.4. Creating a Free Slip Boundary

1. Create a new boundary named `Top`.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Wall
	Location	TOP
Boundary Details	Mass And Momentum > **Option**	 Free Slip Wall

3. Click **OK**.

10.4.3.5. Creating a Wall Boundary

1. Create a new boundary named `WingSurface`.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Wall
	Location	WING_Nodes [a]
a. If particular items do not appear in the drop-down list, click the *Ellipsis* ... icon to see all available items.		

3. Click **OK**.

10.4.4. Creating Domain Interfaces

The imported mesh contains three regions that will be connected with domain interfaces.

1. Create a new domain interface named `Domain Interface 1`.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Interface Type**	Fluid Fluid
	Interface Side 1 > **Region List**	 Primitive 2D A
	Interface Side 2 > **Region List**	 Primitive 2D, Primitive 2D B

3. Click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.4.5. Setting Initial Values

For high-speed compressible flow, the CFX-Solver usually requires sensible initial conditions to be set for the velocity field.

1. Click *Global Initialization* .

2. Configure the following setting(s):

Tab	Setting	Value
Global Settings	Initial Conditions > Cartesian Velocity Components > **Option**	Automatic with Value
	Initial Conditions > Cartesian Velocity Components > **U**	600 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > **V**	0 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > **W**	0 [m s^-1]
	Initial Conditions > Temperature > **Option**	Automatic with Value
	Initial Conditions > Temperature > **Temperature**	300 [K]

3. Click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.4.6. Setting the Solver Controls

The residence time for the fluid is the length of the domain divided by the speed of the fluid; using values from the problem specification, the result is approximately:

70 [m] / 600 [m s^-1] = 0.117 [s]

In the next step, you will set a maximum timescale, then the solver will start with a conservative time scale that gradually increases towards the fluid-residence time as the residuals decrease.

1. Click *Solver Control* .

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Convergence Control > Fluid Timescale Control > **Maximum Timescale**	(Selected)
	Convergence Control > Fluid Timescale Control > Maximum Timescale > **Maximum Timescale**	0.1 [s]
	Convergence Criteria > **Residual Target**	1.0e-05

3. Click **OK**.

10.4.7. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .

2. Configure the following setting(s):

Setting	Value
File name	WingSPS.def

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, **Solver Input File** is set.

4. If using stand-alone mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.5. Obtaining the Solution Using CFX-Solver Manager

At this point, CFX-Solver Manager is running, and the **Define Run** dialog box is displayed, with the CFX-Solver input file set.

1. Click **Start Run**.
2. Select the check box next to **Post-Process Results** when the completion message appears at the end of the run.
3. If using stand-alone mode, select the check box next to **Shut down CFX-Solver Manager**.
4. Click **OK**.

10.6. Viewing the Results Using CFD-Post

The following topics will be discussed:

- Displaying Mach Information (p. 242)
- Displaying Pressure Information (p. 243)
- Displaying Temperature Information (p. 243)
- Displaying Pressure With User Vectors (p. 243)

10.6.1. Displaying Mach Information

The first view configured shows that the bulk of the flow over the wing has a Mach Number of over 1.5.

1. To best orient the view, select **View From -Z** by typing **Shift** +**Z**.
2. Zoom in so the geometry fills the Viewer.
3. Create a new contour named `SymP2Mach`.
4. Configure the following setting(s):

Tab	Setting	Value
Geometry	**Locations**	SymP2
	Variable	Mach Number
	Range	User Specified
	Min	1
	Max	2
	# of Contours	21

5. Click **Apply**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Clear the check box next to SymP2Mach.

10.6.2. Displaying Pressure Information

To display pressure information, create a contour plot that shows the pressure field:

1. Create a new contour named SymP2Pressure.
2. Configure the following setting(s):

Tab	Setting	Value
Geometry	**Locations**	SymP2
	Variable	Pressure
	Range	Global

3. Click **Apply**.
4. Clear the check box next to SymP2Pressure.

10.6.3. Displaying Temperature Information

You can confirm that a significant energy loss occurs around the wing's leading edge by plotting temperature on SymP2.

1. Create a new contour named SymP2Temperature.
2. Configure the following setting(s):

Tab	Setting	Value
Geometry	**Locations**	SymP2
	Variable	Temperature
	Range	Global

3. Click **Apply**.

   The contour shows that the temperature at the wing's leading edge is approximately 180 K higher than the inlet temperature.

4. Clear the check box next to SymP2Temperature.

10.6.4. Displaying Pressure With User Vectors

You can also create a user vector to show the pressure acting on the wing:

1. Create a new variable named Variable 1.
2. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Setting	Value
Variable 1	**Vector**	(Selected)
	X Expression	(Pressure+101325[Pa])*Normal X
	Y Expression	(Pressure+101325[Pa])*Normal Y
	Z Expression	(Pressure+101325[Pa])*Normal Z

3. Click **Apply**.
4. Create a new vector named `Vector 1`.
5. Configure the following setting(s):

Tab	Setting	Value
Geometry	**Locations**	WingSurface
	Variable	Variable 1
Symbol	**Symbol Size**	0.04

6. Click **Apply**.
7. Zoom in on the wing in order to see the created vector plot.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 11: Flow Through a Butterfly Valve

This tutorial includes:

11.1. Tutorial Features

11.2. Overview of the Problem to Solve

11.3. Preparing the Working Directory

11.4. Defining the Case Using CFX-Pre

11.5. Obtaining the Solution Using CFX-Solver Manager

11.6. Viewing the Results Using CFD-Post

11.1. Tutorial Features

In this tutorial you will learn about:

- Using a rough wall boundary in CFX-Pre to simulate the pipe wall
- Creating a fully developed inlet velocity profile using the CFX Expression Language
- Setting up a Particle Tracking simulation in CFX-Pre to trace sand particles
- Animating particle tracks in CFD-Post to trace sand particles through the domain
- Performing quantitative calculation of average static pressure in CFD-Post on the outlet boundary.

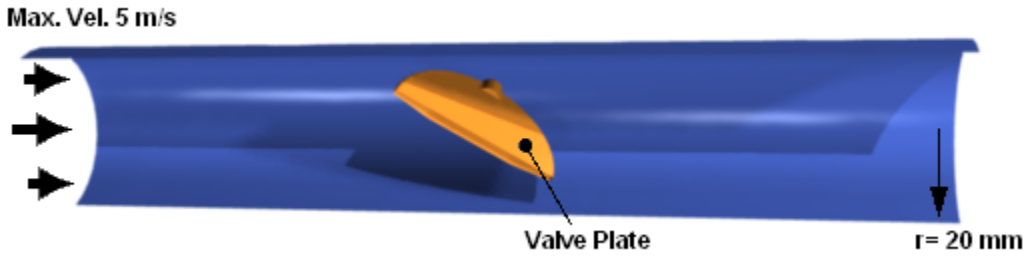
Component	Feature	Details
CFX-Pre	User Mode	General mode
	Analysis Type	Steady State
	Fluid Type	General Fluid
	Domain Type	Single Domain
	Turbulence Model	k-Epsilon
	Heat Transfer	None
	Particle Tracking	
	Boundary Conditions	Inlet (Profile)
		Inlet (Subsonic)
		Outlet (Subsonic)
		Symmetry Plane
		Wall: No-Slip
		Wall: Rough

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Component	Feature	Details
	CEL (CFX Expression Language)	
	Timestep	Auto Time Scale
CFD-Post	Plots	Animation
		Default Locators
		Particle Track
		Point
		Slice Plane
	Other	Changing the Color Range
		Movie Generation
		Particle Track Animation
		Quantitative Calculation
		Symmetry, Reflection Plane

11.2. Overview of the Problem to Solve

Pumps and compressors are commonplace. An estimate of the pumping requirement can be calculated based on the height difference between source and destination and head loss estimates for the pipe and any obstructions/joints along the way. Investigating the detailed flow pattern around a valve or joint however, can lead to a better understanding of why these losses occur. Improvements in valve/joint design can be simulated using CFD, and implemented to reduce pumping requirements and cost.



Flows can contain particulates that affect the flow and cause erosion to pipe and valve components. You can use the particle-tracking capability of CFX to simulate these effects.

In this example, water flows at 5 m/s through a 20 mm radius pipe that has a rough internal surface. The velocity profile is assumed to be fully developed at the pipe inlet. The flow, which is controlled by a butterfly valve set at an angle of 55° to the vertical axis, contains sand particles ranging in size from 50 to 500 microns. The equivalent sand grain roughness is 0.2 mm.

The reference temperature is 300 K; the reference pressure is 1 atm.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A mesh is provided. You will create sand particles and a domain that contains water; for one part of the simulation the water and sand will be fully coupled, and for the other part of the simulation they will be one-way coupled. To increase the accuracy of the simulation, the inlet will be given a velocity profile that simulates a fully-developed boundary layer.

To solve the simulation, you will create two sets of identical particles. The first set will be fully coupled to predict the effect of the particles on the continuous phase flow field and enable the particles to influence the flow field. The second set will be one-way coupled but will contain a much higher number of particles to provide a more accurate calculation of the particle volume fraction and local forces on walls, but without affecting the flow field.

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- Running Ansys CFX Tutorials Using Ansys Workbench (p. 34)
- Changing the Display Colors (p. 37)

11.3. Preparing the Working Directory

1. Create a working directory.

   Ansys CFX uses a working directory as the default location for loading and saving files for a particular session or project.

2. Download the `pipe_valve.zip` file here.

3. Unzip `pipe_valve.zip` to your working directory.

   Ensure that the following tutorial input file is in your working directory:

   - `PipeValveMesh.gtm`

4. Set the working directory and start CFX-Pre.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

11.4. Defining the Case Using CFX-Pre

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `PipeValve`.
5. Click **Save**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11.4.1. Importing the Mesh

1. Right-click `Mesh` and select **Import Mesh** > **CFX Mesh**.

   The **Import Mesh** dialog box appears.

2. Configure the following setting(s):

Setting	Value
File name	PipeValveMesh.gtm

3. Click **Open**.

11.4.2. Defining the Properties of the Sand

The material properties of the sand particles used in the simulation need to be defined. Heat transfer and radiation modeling are not used in this simulation, so the only properties that need to be defined are the density of the sand and the diameter range.

To calculate the effect of the particles on the continuous fluid, between 100 and 1000 particles are usually required. However, if accurate information about the particle volume fraction or local forces on wall boundaries is required, then a much larger number of particles must be modeled.

When you create the domain, choose either full coupling or one-way coupling between the particle and continuous phase. *Full coupling* is needed to predict the effect of the particles on the continuous phase flow field but has a higher CPU cost than one-way coupling; *one-way coupling* simply predicts the particle paths during postprocessing based on the flow field, but without affecting the flow field.

To optimize CPU usage, you can create two sets of identical particles. The first set should be fully coupled and around 200 particles will be used. This allows the particles to influence the flow field. The second set uses one-way coupling but contains 5000 particles. This provides a more accurate calculation of the particle volume fraction and local forces on walls. (These values are defined in the inlet boundary definition.)

For this tutorial you will create a "Sand Fully Coupled" boundary condition that has 200 particles moving with a mass flow rate of 0.01 kg/s and a "Sand One Way Coupled" boundary condition that has 5000 particles moving with a mass flow rate of 0.01 kg/s. In both cases the sand density is 2300 [kg m^-3]; particle diameters range from 50 e-6 m to 500 e-6 m, with an average diameter of 250 e-6 m and a standard deviation of 70 e-6 m. You will set a Finnie erosion model with a velocity power factor of 2 and a reference velocity of 1 m/s.

1. Click *Insert Material* then create a new material named `Sand Fully Coupled`.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Material Group**	Particle Solids
	Thermodynamic State	(Selected)

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Material Properties	Thermodynamic Properties > Equation of State > **Density**	2300 [kg m^-3] [a]
	Thermodynamic Properties > **Specific Heat Capacity**	(Selected)
	Thermodynamic Properties > Specific Heat Capacity > **Specific Heat Capacity**	0 [J kg^-1 K^-1] [b]
	Thermodynamic Properties > **Reference State**	(Selected)
	Thermodynamic Properties > Reference State > **Option**	Specified Point
	Thermodynamic Properties > Reference State > **Ref. Temperature**	300 [K] [a]

a. From the problem description.

b. `Specific Heat Capacity` is set to 0 because heat transfer is not modeled in this tutorial.

3. Click **OK**.
4. Under `Materials`, right-click `Sand Fully Coupled` and select **Duplicate** from the shortcut menu.
5. Rename the duplicate as `Sand One Way Coupled`.
6. `Sand One Way Coupled` is created with properties identical to `Sand Fully Coupled`.

11.4.3. Creating the Domain

Set up an environment that has water and sand defined in two ways; one in which the sand is fully coupled, and one in which the sand is one-way coupled:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Edit `Case Options` > `General` in the **Outline** tree view and ensure that **Automatic Default Domain** is turned on. A domain named `Default Domain` should appear under the `Simulation` > `Flow Analysis 1` branch.
2. Double-click `Default Domain`.
3. Under **Fluid and Particle Definitions**, delete `Fluid 1`.
4. Click *Add new item* to create a new fluid definition named `Water`.
5. Set **Fluid and Particle Definitions** > **Water** > **Material** to `Water`.
6. Create a new fluid definition named `Sand Fully Coupled`.
7. Under **Fluid and Particle Definitions** > **Sand Fully Coupled**, configure the following setting(s):

Setting	Value
Material	Sand Fully Coupled [a]
Morphology > **Option**	Particle Transport Solid
Morphology > **Particle Diameter Distribution**	(Selected)
Morphology > Particle Diameter Distribution > **Option**	Normal in Diameter by Mass
Morphology > Particle Diameter Distribution > **Minimum Diameter**	50e-6 [m]
Morphology > Particle Diameter Distribution > **Maximum Diameter**	500e-6 [m]
Morphology > Particle Diameter Distribution > **Mean Diameter**	250e-6 [m]
Morphology	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
> Particle Diameter Distribution > **Std. Deviation**	 70e-6 [m]
a. Click the *Ellipsis* icon to open the **Materials** dialog box, then select `Particle Solids` > `Sand Fully Coupled`.	

8. Create a new fluid definition named `Sand One Way Coupled`.
9. Under **Fluid and Particle Definitions** > **Sand One Way Coupled**, configure the following setting(s):

Setting	Value
Material	Sand One Way Coupled [a]
Morphology > **Option**	 Particle Transport Solid
Morphology > **Particle Diameter Distribution**	 (Selected)
Morphology > Particle Diameter Distribution > **Option**	 Normal in Diameter by Mass
Morphology > Particle Diameter Distribution > **Minimum Diameter**	 50e-6 [m]
Morphology > Particle Diameter Distribution > **Maximum Diameter**	 500e-6 [m]
Morphology > Particle Diameter Distribution > **Mean Diameter**	 250e-6 [m]
Morphology	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
> Particle Diameter Distribution > **Std. Deviation**	 70e-6 [m]
a. Click the *Ellipsis* icon to open the **Materials** dialog box, then select `Particle Solids` > `Sand One Way Coupled`.	

10. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Domain Models > Pressure > **Reference Pressure**	 1 [atm]
Fluid Models	Heat Transfer > **Option**	 None
	Turbulence > **Option**	 k-Epsilon[a]
Fluid Specific Models	**Fluid**	Sand Fully Coupled
	Fluid > Sand Fully Coupled > Erosion Model > **Option**	 Finnie
	Fluid > Sand Fully Coupled > Erosion Model > **Vel. Power Factor**	 2.0
	Fluid > Sand Fully Coupled > Erosion Model > **Reference Velocity**	 1 [m s^-1]
	Fluid	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Sand One Way Coupled**	(Selected)
	Fluid > Sand One Way Coupled > Erosion Model > **Option**	Finnie
	Fluid > Sand One Way Coupled > Erosion Model > **Vel. Power Factor**	2.0
	Fluid > Sand One Way Coupled > Erosion Model > **Reference Velocity**	1 [m s^-1]
Fluid Pair Models	**Fluid Pair**	Water | Sand Fully Coupled
	Fluid Pairs > Water | Sand Fully Coupled > **Particle Coupling**	Fully Coupled
	Fluid Pairs >Water | Sand Fully Coupled > Momentum Transfer > Drag Force > **Option**	Schiller Naumann [**b**]
	Fluid Pair	Water | Sand One Way Coupled
	Fluid Pairs > Water | Sand One Way Coupled > **Particle Coupling**	One-way Coupling
	Fluid Pairs	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Water | Sand One Way Coupled	
	> Momentum Transfer	
	> Drag Force	
	> **Option**	Schiller Naumann [b]
a. The turbulence model applies only to the continuous phase and not the particle phases. b. The Schiller Naumann drag model is appropriate for sparsely-distributed, solid spherical particles.		

11. Click **OK**.

11.4.4. Creating the Inlet Velocity Profile

In previous tutorials you have often defined a uniform velocity profile at an inlet boundary. This means that the inlet velocity near to the walls is the same as that at the center of the inlet. If you look at the results from these simulations, you will see that downstream of the inlet a boundary layer will develop, so that the downstream near wall velocity is much lower than the inlet near wall velocity.

You can simulate an inlet more accurately by defining an inlet velocity profile, so that the boundary layer is already fully developed at the inlet. The one seventh power law will be used in this tutorial to describe the profile at the pipe inlet. The equation for this is:

$$U = W_{\max}\left(1-\frac{r}{R_{\max}}\right)^{\frac{1}{7}} \tag{11.1}$$

where $W_{\max}$ is the pipe centerline velocity, $R_{\max}$ is the pipe radius, and r is the distance from the pipe centerline.

You can create a non-uniform (profile) boundary condition by doing one of the following:

- Creating an expression using CEL that describes the inlet profile. Using a CEL expression is the easiest way to create the profile.
- Creating a User CEL Function that uses a user subroutine (linked to the CFX-Solver during execution) to describe the inlet profile. The User CEL Function method is more complex, but is provided here as an example of how to use this feature.
- Loading a boundary condition profile file (a file that contains boundary condition profile data).

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Profiles created from data files are not used in this tutorial, but are used in the tutorial Flow in a Process Injection Mixing Pipe (p. 145).

Note:

For complex profiles, it may be necessary to use a User CEL Function or a boundary condition profile file.

Use a CEL expression to define the velocity profile for the inlet boundary:

1. Click *Insert Expression* and create the following expressions using Equation 11.1 (p. 254) and values from the problem description:

Name	Definition
Rmax	20 [mm]
Wmax	5 [m s^-1]
Wprof	Wmax*(abs(1-r/Rmax)^0.143)

In the definition of `Wprof`, the variable r (radius) is a CFX System Variable defined as:

$$r=\sqrt{x^2+y^2} \tag{11.2}$$

In this equation, x and y are defined as directions 1 and 2 (X and Y for Cartesian coordinate frames) respectively, in the selected reference coordinate frame.

2. Continue with the tutorial at Creating the Boundary Conditions (p. 255).

11.4.5. Creating the Boundary Conditions

11.4.5.1. Inlet Boundary

1. Create a new boundary named `inlet`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Inlet
	Location	inlet
Boundary Details	Mass And Momentum > **Option**	Cart. Vel. Components
	Mass And Momentum > **U**	0 [m s^-1]
	Mass And Momentum > **V**	0 [m s^-1]

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Mass And Momentum > **W**	Wprof [a]
Fluid Values [b]	**Boundary Conditions**	Sand Fully Coupled
	Boundary Conditions > Sand Fully Coupled > Particle Behavior > **Define Particle Behavior**	(Selected)
	Boundary Conditions > Sand Fully Coupled > Mass and Momentum > **Option**	Cart. Vel. Components [c]
	Boundary Conditions > Sand Fully Coupled > Mass And Momentum > **U**	0 [m s^-1]
	Boundary Conditions > Sand Fully Coupled > Mass And Momentum > **V**	0 [m s^-1]
	Boundary Conditions > Sand Fully Coupled > Mass And Momentum > **W**	Wprof [a]
	Boundary Conditions > Sand Fully Coupled > Particle Position	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Option**	Uniform Injection
	Boundary Conditions > Sand Fully Coupled > Particle Position > Number of Positions > **Option**	Direct Specification
	Boundary Conditions > Sand Fully Coupled > Particle Position > Number of Positions > **Number**	200
	Boundary Conditions > Sand Fully Coupled > Particle Mass Flow > **Mass Flow Rate**	0.01 [kg s^-1]
	Boundary Conditions	Sand One Way Coupled
	Boundary Conditions > Sand One Way Coupled > Particle Behavior > **Define Particle Behavior**	(Selected)
	Boundary Conditions > Sand One Way Coupled > Mass and Momentum > **Option**	Cart. Vel. Components [c]
	Boundary Conditions > Sand One Way Coupled	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Mass And Momentum > **U**	0 [m s^-1]
	Boundary Conditions > Sand One Way Coupled > Mass And Momentum > **V**	0 [m s^-1]
	Boundary Conditions > Sand One Way Coupled > Mass And Momentum > **W**	Wprof [a]
	Boundary Conditions > Sand One Way Coupled > Particle Position > **Option**	Uniform Injection
	Boundary Conditions > Sand One Way Coupled > Particle Position > Number of Positions > **Option**	Direct Specification
	Boundary Conditions > Sand One Way Coupled > Particle Position > Number of Positions > **Number**	5000
	Boundary Conditions > Sand One Way Coupled > Particle Position > Particle Mass Flow Rate	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Mass Flow Rate**	0.01 [kg s^-1]

a. Use the **Expressions** details view to enter `Wprof`.

b. Do *not* select **Particle Diameter Distribution**. The diameter distribution was defined when creating the domain; this option would override those settings for this boundary only.

c. Instead of manually specifying the same velocity profile as the fluid, you can also select the Zero Slip Velocity option.

3. Click **OK**.

One-way coupled particles are tracked as a function of the fluid flow field. The latter is not influenced by the one-way coupled particles. The fluid flow will therefore be influenced by the 0.01 [kg s^-1] flow of two-way coupled particles, but not by the 0.01 [kg s^-1] flow of one-way coupled particles.

11.4.5.2. Outlet Boundary

1. Create a new boundary named `outlet`.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Outlet
	Location	outlet
Boundary Details	Flow Regime > **Option**	Subsonic
	Mass and Momentum > **Option**	Average Static Pressure
	Mass and Momentum > **Relative Pressure**	0 [Pa]

3. Click **OK**.

11.4.5.3. Symmetry Plane Boundary

1. Create a new boundary named `symP`.

2. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Basic Settings	**Boundary Type**	Symmetry [a]
	Location	symP

a. Symmetry can be used when geometry and mesh are invariant normal to the symmetry surface.

3. Click **OK**.

11.4.5.4. Pipe Wall Boundary

1. Create a new boundary named `pipe wall`.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Wall
	Location	pipe wall
Boundary Details	Wall Roughness > **Option**	Rough Wall
	Wall Roughness > **Sand Grain Roughness**	0.2 [mm] [a]
Fluid Values	**Boundary Conditions**	Sand Fully Coupled
	Boundary Conditions > Sand Fully Coupled > Velocity > **Option**	Restitution Coefficient
	Boundary Conditions > Sand Fully Coupled > Velocity > **Perpendicular Coeff.**	0.8 [b]
	Boundary Conditions > Sand Fully Coupled > Velocity > **Parallel Coeff.**	1

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
a. From the problem description. Make sure that you change the units to millimeters. The thickness of the first element should be of the same order as the roughness height. b. This value would typically come from experimental or reference data.		

3. Apply the same setting values for `Sand One Way Coupled` as for `Sand Fully Coupled`.
4. Click **OK**.

11.4.5.5. Editing the Default Boundary

1. In the **Outline** tree view, edit the boundary named `Default Domain Default`.
2. Configure the following setting(s):

Tab	Setting	Value
Fluid Values	**Boundary Conditions**	Sand Fully Coupled
	Boundary Conditions > Sand Fully Coupled > Velocity > **Perpendicular Coeff.**	0.9 [a]
	Boundary Conditions	Sand One Way Coupled
	Boundary Conditions > Sand One Way Coupled > Velocity > **Perpendicular Coeff.**	0.9
a. This value would typically come from experimental or reference data. For this tutorial, the pipe wall and butterfly valve are considered to be made of different materials, so their perpendicular coefficients are different.		

3. Click **OK**.

11.4.6. Setting Initial Values

Set up the initial values to be consistent with the inlet boundary conditions:

1. Click *Global Initialization* .

2. Configure the following setting(s):

Tab	Setting	Value
Global Settings	Initial Conditions > Cartesian Velocity Components > **Option**	Automatic with Value
	Initial Conditions > Cartesian Velocity Components > Option > **U**	0 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > Option > **V**	0 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > Option > **W**	Wprof

3. Click **OK**.

11.4.7. Setting the Solver Controls

1. Click *Solver Control* .

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Advection Scheme > **Option**	High Resolution
Particle Control	Particle Integration > **Max. Particle Intg. Time Step**	(Selected)
	Particle Integration	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Max. Particle Intg. Time Step > **Value**	 1e+10 [s]
	Particle Termination Control	(Selected)
	Particle Termination Control > **Maximum Tracking Time**	 (Selected)
	Particle Termination Control > Maximum Tracking Time > **Value**	 10 [s]
	Particle Termination Control > **Maximum Tracking Distance**	 (Selected)
	Particle Termination Control > Maximum Tracking Distance > **Value**	 10 [m]
	Particle Termination Control > **Max. Num. Integration Steps**	 (Selected)
	Particle Termination Control > Max. Num. Integration Steps > **Value**	 10000 [a]
a. This value controls the number of mesh elements a particle is allowed to cross and therefore must take into account the size and density of the mesh.		

Note:

The numeric values in the preceding table are all designed to put a high upper limit on the amount of processing that will be done. For example, the tracking time of 10 seconds would enable a particle to get caught in an eddy for a reasonable amount of time.

3. Click **OK**.

11.4.8. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Configure the following setting(s):

Setting	Value
File name	`PipeValve.def`

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, **Solver Input File** is set.

4. If using stand-alone mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

11.5. Obtaining the Solution Using CFX-Solver Manager

When CFX-Pre has shut down and CFX-Solver Manager has started, you can obtain a solution to the CFD problem by using the procedure that follows.

1. Ensure the **Define Run** dialog box is displayed and click **Start Run**.
2. Select the check box next to **Post-Process Results** when the completion message appears at the end of the run.
3. If using stand-alone mode, select the check box next to **Shut down CFX-Solver Manager**.
4. Click **OK**.

11.6. Viewing the Results Using CFD-Post

In this section, you will first plot erosion on the valve surface and side walls due to the sand particles. You will then create an animation of particle tracks through the domain.

11.6.1. Erosion Due to Sand Particles

An important consideration in this simulation is erosion to the pipe wall and valve due to the sand particles. A good indication of erosion is given by the `Erosion Rate Density` parameter, which corresponds to pressure and shear stress due to the flow.

1. Edit the object named `Default Domain Default`.
2. Configure the following setting(s) using the *Ellipsis* as required for variable selection:

Tab	Setting	Value
Color	**Mode**	Variable
	Variable	Sand One Way Coupled.Erosion Rate Density [a]
	Range	User Specified
	Min	0 [kg m^-2 s^-1]

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Max	25 [kg m^-2 s^-1] [b]

a. This is statistically better than `Sand Fully Coupled.Erosion Rate Density` because many more particles were calculated for `Sand One Way Coupled`.

b. This range is used to gain a better resolution of the wall shear stress values around the edge of the valve surfaces.

3. Click **Apply**.

   As can be seen, the highest values occur on the edges of the valve where most particles strike. Erosion of the low Z side of the valve would occur more quickly than for the high Z side.

11.6.2. Displaying Erosion on the Pipe Wall

Set the user specified range for coloring to resolve areas of stress on the pipe wall near of the valve:

1. Ensure that the check box next to `Res PT for Sand Fully Coupled` is cleared.
2. Clear the check box next to `Default Domain Default`.
3. Edit the object named `pipe wall`.
4. Configure the following setting(s):

Tab	Setting	Value
Color	**Mode**	Variable
	Variable	Sand One Way Coupled.Erosion Rate Density
	Range	User Specified
	Min	0 [kg m^-2 s^-1]
	Max	25 [kg m^-2 s^-1]

5. Click **Apply**.
6. Optionally, fill the check box next to `Default Domain Default` to see how sand particles have deflected off the butterfly valve then to the pipe wall.

11.6.3. Creating Particle Tracks

Default particle track objects are created at the start of the session. One particle track is created for each set of particles in the simulation. You are going to make use of the default object for `Sand Fully Coupled`.

The default object draws 25 tracks as lines from the inlet to outlet. The **Info** tab shows information about the total number of tracks, the index range, and the track numbers that are drawn.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Turn off the visibility for all objects except `Wireframe`.
2. Edit the object named `Res PT for Sand Fully Coupled`.
3. Configure the following setting(s):

Tab	Setting	Value
Geometry	**Max Tracks**	20 [a]
Color	**Mode**	Variable
	Variable	Sand Fully Coupled.Velocity w
Symbol	**Show Symbols**	(Selected)
	Show Symbols > **Max Time**	0 [s]
	Show Symbols > **Min Time**	0 [s]
	Show Symbols > **Interval**	0.07 [s]
	Show Symbols > **Symbol**	Ball
	Show Symbols > **Scale**	1.2

a. This value improves the resolution of the tracks generated.

4. Click **Apply**.
5. Right-click a blank area anywhere in the viewer, select **Predefined Camera** from the shortcut menu and select **View From +X** to view the particle tracks.

   Symbols can be seen at the start of each track.

11.6.4. Creating a Particle Track Animation

The following steps describe how to create a particle tracking animation using **Sweep Animation**. Similar effects can be achieved in more detail using the **Keyframe Animation** option, which allows full control over all aspects on an animation.

1. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Y up)** from the shortcut menu.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Right-click an edge of the flat side on the half cylinder and select **Reflect/Mirror** from the shortcut menu. Click **X Axis** to choose it as the normal direction.

   Note:

   Alternatively, you can apply **Reflect/Mirror**, by double-clicking `Default Domain` to open the details view. In the **Instancing** tab enable **Apply Reflection** and select **Method** to `YZ Plane`. Click **Apply**.

3. Select **Tools** > **Animation** or click *Animation* .

   The **Animation** dialog box appears.

4. Set **Type** to **Sweep Animation**.

5. Select `Res PT for Sand Fully Coupled`:

6. Click **Options** to display the **Animation Options** dialog box, then clear **Override Symbol Settings** to ensure the symbol type and size are kept at their specified settings for the animation playback. Click **OK**.

   Note:

   The arrow pointing downward in the bottom right corner of the Animation Window will reveal the **Options** button if it is not immediately visible.

7. Select **Loop**.

8. Clear *Repeat forever* and ensure that **Repeat** is set to `1`.

9. Select **Save Movie**.

10. Set **Format** to `MPEG1`.

11. Click *Browse* and enter `tracks.mpg` as the filename.

12. Click *Play the animation* .

13. If prompted to overwrite an existing movie, click **Overwrite**.

    The animation plays and builds an `.mpg` file.

14. Close the **Animation** dialog box.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11.6.5. Determining Minimum, Maximum, and Average Pressure Values

On the outlet boundary you created in CFX-Pre, you set the **Average Static Pressure** to `0.0 [Pa]`. To see the effect of this:

1. From the main menu select **Tools** > **Function Calculator**.

   The **Function Calculator** is displayed. It enables you to perform a wide range of quantitative calculations on your results.

   Note:

   You should use Conservative variable values when performing calculations and Hybrid values for visualization purposes. Conservative values are set by default in CFD-Post but you can manually change the setting for each variable in the Variables Workspace, or the settings for all variables by using the Function Calculator.

2. Set **Function** to `maxVal`.
3. Set **Location** to `outlet`.
4. Set **Variable** to `Pressure`.
5. Click **Calculate**.

   The result is the maximum value of pressure at the outlet.

6. Perform the calculation again using `minVal` to obtain the minimum pressure at the outlet.
7. Select `areaAve`, and then click **Calculate**.

   - This calculates the area weighted average of pressure.
   - The average pressure is approximately zero, as specified by the boundary.

11.6.6. Other Features

The geometry was created using a symmetry plane. In addition to the **Reflect/Mirror** option from the shortcut menu, you also can display the other half of the geometry by creating a YZ Plane at X = 0 and then editing the `Default Transform` object to use this plane as a reflection plane.

When you have finished viewing the results, quit CFD-Post.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 12: Flow in a Catalytic Converter

This tutorial includes:

12.1. Tutorial Features

12.2. Overview of the Problem to Solve

12.3. Preparing the Working Directory

12.4. Defining the Case Using CFX-Pre

12.5. Obtaining the Solution Using CFX-Solver Manager

12.6. Viewing the Results Using CFD-Post

12.1. Tutorial Features

In this tutorial you will learn about:

- Using multiple meshes in CFX-Pre.
- Joining meshes together using static fluid-fluid domain interfaces between the inlet/outlet flanges and the central catalyst body.
- Applying a source of resistance using a directional loss model.
- Creating a chart to show pressure drop versus Z coordinate in CFD-Post.
- Exporting data from a line locator to a file.

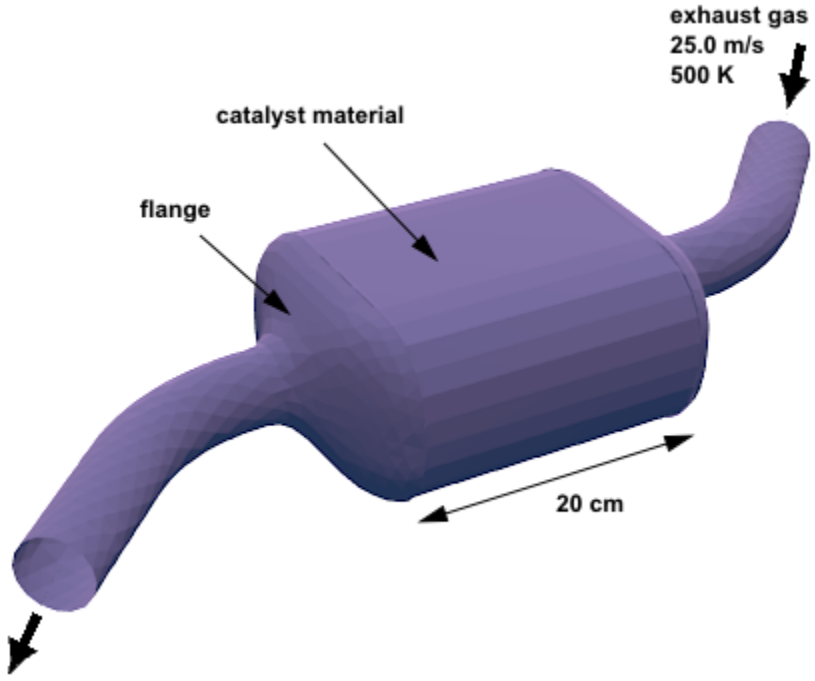
Component	Feature	Details
CFX-Pre	User Mode	General mode
	Analysis Type	Steady State
	Domain Type	Multiple Domain (Fluid, Porous)
	Fluid Type	Ideal Gas
	Turbulence Model	k-Epsilon
	Heat Transfer	Thermal Energy
	Boundary Conditions	Inlet (Subsonic)
		Outlet (Subsonic)
		Wall: No-Slip
	Domain Interfaces	Fluid-Porous
	Timestep	Physical Time Scale
CFD-Post	Plots	Contour

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Component	Feature	Details
		Default Locators
		Outline Plot (Wireframe)
		Polyline
		Slice Plane
		Vector
	Other	Chart Creation
		Data Export
		Title/Text
		Viewing the Mesh

12.2. Overview of the Problem to Solve

Catalytic converters are used on most vehicles on the road today. They reduce harmful emissions from internal combustion engines (such as oxides of nitrogen, hydrocarbons, and carbon monoxide) that are the result of incomplete combustion. Most new catalytic converters are the honeycomb ceramic type and are usually coated with platinum, rhodium, or palladium.



In this tutorial, you will model a catalytic converter in order to determine the pressure drop and heat transfer through it when air enters the inlet at 25 m/s and 500 K, and exits the outlet at a static pressure of 1 atm. For simplicity, you will not model chemical reactions.

You are provided with a mesh for the passageways inside a pipe-and-flange structure. You will use this mesh, and a copy of it, to model the pipe and flange portions of the flow field, at both ends of the catalytic converter.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For the housing, you are provided with a hexahedral mesh that was created in ICEM-Hexa. This mesh fills the entire 3D volume of the housing.

To model the presence of the honeycomb structure that exists in the housing, you will model porosity and apply resistance to the flow. The honeycomb structure has a porosity of 70%, which means that 70% of the total volume is available for fluid flow, while the other 30% is occupied by the solid material that makes up the honeycomb structure. The solid component of the structure will be steel. The honeycomb structure is lined up with the flow to prevent flow travel in the transverse direction. To model resistance to the flow, you will apply a streamwise quadratic resistance coefficient of 650 kg m^-4. To reduce the amount of transverse flow, apply a quadratic resistance coefficient of 6500 kg m^-4 in the transverse direction. These given resistance coefficients are based on the superficial flow velocity, rather than the true flow velocity.

The Inlet boundary has a static temperature of 500 K. You will model heat transfer through the solid material in the porous domain. The heat transfer between the air and steel within the porous domain is modeled using an interfacial area density of 360 m^-1 and a heat transfer coefficient of 50 W m^-2 K^-1. Thermal energy is lost to the environment through the midsection walls of the catalytic converter; the rate of heat loss is defined by the heat transfer coefficient (20 W m^-2 K^-1) and the outside temperature (40 °C).

You will first import the mesh for the housing. You will then import a mesh for one of the two flanges. You will then produce another flange mesh by transforming the first. You will create one porous domain for the housing, and one fluid domain for both flanges. You will model a honeycomb structure inside the housing by specifying a porosity and applying a directional momentum loss model.

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- Running Ansys CFX Tutorials Using Ansys Workbench (p. 34)
- Changing the Display Colors (p. 37)

12.3. Preparing the Working Directory

1. Create a working directory.

   Ansys CFX uses a working directory as the default location for loading and saving files for a particular session or project.

2. Download the `catconv.zip` file here.

3. Unzip `catconv.zip` to your working directory.

   Ensure that the following tutorial input files are in your working directory:

   - `CatConvHousing.hex`
   - `CatConvMesh.gtm`
   - `CatConv.ccl`

4. Set the working directory and start CFX-Pre.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

12.4. Defining the Case Using CFX-Pre

This tutorial requires that you are working with SI units. In particular, some coordinates used in the specification of a mesh transformation (performed later in this tutorial) are given without units, and CFX-Pre assumes that the working units apply. Before continuing, ensure that your units are set to SI as follows:

1. Select **Edit** > **Options**.

   The **Options** dialog box appears.

2. In the tree on the left side of the dialog box, select `Common` > `Units`.

3. Set **System** to `SI`.

4. Click **OK**.

5. Select **File** > **New Case**.

6. Select **General** and click **OK**.

7. Select **File** > **Save Case As**.

8. Under **File name**, type `CatConv`.

9. Click **Save**.

12.4.1. Importing the Meshes and CCL File

The mesh for this catalytic converter is composed of three distinct parts:

- The outlet section (pipe and flange).
- The inlet section (pipe and flange).

  You will copy and rotate the outlet section through 180 degrees to create this section.

- The catalyst (or monolith).

You will import a CFX Command Language (CCL) file defining mathematical expressions for this case. Next you will import the catalyst housing and a generic inlet/outlet section from provided files.

12.4.1.1. Importing the Required Expressions From a CCL File

The CCL file you are going to import contains expressions that will be used to define parameters in the simulation. These can be seen in the table below:

Expression	Value
AreaDen	360 [m^-1]

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Expression	Value
HTC	50 [W m^-2 K^-1]
HTCoutside	20 [W m^-2 K^-1]
L	0.4 [m]
Porosity	0.7
Tinlet	500[K]
Toutside	40 [C]

Import the CCL File to define relevant expressions:

1. Select **File** > **Import** > **CCL**.

   The **Import CCL** dialog box appears.

2. Select `CatConv.ccl`

3. Click **Open**.

4. Expand the `Expressions` section in the **Outline** tree to see a list of the expressions that have been imported.

12.4.1.2. Importing the Housing Mesh

The first mesh that you will import, `CatConvHousing.hex`, is a hexahedral mesh for the catalyst housing. The mesh was originally created in ICEM-Hexa using mesh units of centimeters. Because this type of mesh file does not specify the mesh units, you must specify them manually. The imported mesh has a width in the X direction of 21 cm and a length in the Z direction of 20 cm.

1. Right-click `Mesh` and select **Import Mesh** > **Other**.

   The **Import Mesh** dialog box appears.

2. Configure the following setting(s):

Setting	Value
Files of type	All Types(*)
Mesh Format	ICEM CFD
File name	CatConvHousing.hex
Options > **Mesh Units**	 cm

3. Click **Open**.

Later in this tutorial, you will create a porous domain for the housing in order to simulate flow through a honeycomb structure.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12.4.1.3. Importing the Pipe and Flange Mesh

The second mesh that you will import, `CatConvMesh.gtm`, is a mesh for a pipe and flange. The mesh has units of centimeters. Because this type of mesh file does specify the mesh units, there is no need to specify them manually.

1. Right-click `Mesh` and select **Import Mesh** > **CFX Mesh**.

   The **Import Mesh** dialog box appears.

2. Configure the following setting(s):

Setting	Value
File name	CatConvMesh.gtm

3. Click **Open**.

You now have a pipe and flange on the outlet end of the housing. In the next step, you will create a transformed copy of the pipe and flange for the inlet end.

12.4.1.4. Creating a Second Pipe and Flange Mesh

Create a transformed copy of the pipe and flange mesh:

1. Right-click `CatConvMesh.gtm` and select **Transform Mesh**.

   The **Mesh Transformation Editor** dialog box appears.

2. Configure the following setting(s):

Setting	Value
Transformation	Rotation
Rotation Option	Rotation Axis
From	0, 0, 0.16
To	0, 1, 0.16 [a]
Rotation Angle Option	Specified
Rotation Angle	180 [degree]
Multiple Copies	(Selected)
Multiple Copies > # of Copies	1

a. This specifies an axis located at the center of the housing parallel to the Y axis.

3. Click **Apply**.

Later in this tutorial, you will create a fluid domain for both pipe and flange sections.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12.4.1.5. Creating a Single Region for Both Pipe and Flange Meshes

The outlet pipe and flange region is `B1.P3`. The inlet pipe and flange region is `B1.P3 2`.

There are three basic options for creating fluid domains on these regions:

- Create two similar domains: one that applies to `B1.P3`, and one that applies to `B1.P3 2`.
- Create one domain that applies to both `B1.P3` and `B1.P3 2`.
- Create one domain that applies to one composite region, the latter referring to `B1.P3` and `B1.P3 2`.

For demonstration purposes, you will create a composite region and use it as the location for a single fluid domain.

Create a single region that includes both pipe-flange regions:

1. Create a new composite region by selecting **Insert** > **Regions** > **Composite Region**.
2. In the **Insert Region** dialog box, set the name to `CatConverter`.
3. Click **OK**.
4. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Dimension (Filter)**	3D
	Region List	B1.P3, B1.P3 2

5. Click **OK**.

12.4.2. Creating the Fluid Domain

For this simulation you will use a thermal energy heat transfer model and assume turbulent flow. Create the fluid domain using the composite region that you created earlier:

1. Create a new domain by selecting **Insert** > **Domain**, or click *Domain* .
2. In the **Insert Domain** dialog box, set the name to `Pipes`.
3. Click **OK**.
4. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Location and Type > **Location**	CatConverter
	Location and Type	

Tab	Setting	Value
	> **Domain Type**	Fluid Domain
	Fluid and Particle Definitions	Fluid 1
	Fluid and Particle Definitions > Fluid 1 > **Material**	Air Ideal Gas
	Domain Models > Pressure > **Reference Pressure**	1 [atm]
Fluid Models	Heat Transfer > **Option**	Thermal Energy

5. Click **OK**.

12.4.3. Creating the Porous Domain

The catalyst-coated honeycomb structure will be modeled using a porous domain with a directional source of quadratic resistance, as described in the problem description. The streamwise directional resistance is aligned with the Z axis.

For quadratic resistances, the pressure drop is modeled using:

$$\frac{\partial p}{\partial x_i} = -K_Q \left|\boldsymbol{U}\right| U_i \tag{12.1}$$

where K_Q is the quadratic resistance coefficient, U_i is the local velocity in the i direction, and $\frac{\partial p}{\partial x_i}$ is the pressure drop gradient in the i direction.

1. Create a new domain by selecting **Insert** > **Domain**, or click *Domain* .
2. In the **Insert Domain** dialog box, set the name to `Housing`.
3. Click **OK**.
4. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Location and Type > **Location**	LIVE [a]
	Location and Type > **Domain Type**	Porous Domain

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Fluid and Particle Definitions	Fluid 1
	Fluid and Particle Definitions > Fluid 1 > **Material**	 Air Ideal Gas
	Solid Definitions	(Add a new solid named `Steel`)
	Solid Definitions > Steel > **Material**	 Steel
	Domain Models > Pressure > **Reference Pressure**	 1 [atm]
Fluid Models	Heat Transfer > **Option**	 Thermal Energy
Solid Models	Heat Transfer > **Option**	 Thermal Energy
Porosity Settings	Volume Porosity > **Option**	 Value
	Volume Porosity > **Volume Porosity**	 Porosity [**d**]
	Loss Model > **Option**	 Directional Loss
	Loss Model > **Loss Velocity Type**	 Superficial [**b**]
	Loss Model > Directional Loss > Streamwise Direction	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Option**	Cartesian Components
	Loss Model > Directional Loss > Streamwise Direction > **X Component**	0
	Loss Model > Directional Loss > Streamwise Direction > **Y Component**	0
	Loss Model > Directional Loss > Streamwise Direction > **Z Component**	-1
	Loss Model > Directional Loss > **Streamwise Loss > Option**	Linear and Quadratic Coeffs.
	Loss Model > Directional Loss > Streamwise Loss > **Quadratic Resistance Coefficient**	(Selected)
	Loss Model > Directional Loss > Streamwise Loss > Quadratic Resistance Coefficient > **Quadratic Coefficient**	650 [kg m^-4] [c]
	Loss Model > Directional Loss	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Transverse Loss > **Option**	Streamwise Coeff. Multiplier
	Loss Model > Directional Loss > Transverse Loss > **Multiplier**	10 [c]
	Fluid Solid Area Density > **Interfacial Area Den.**	AreaDen [d]
	Fluid Solid Heat Transfer > **Heat Trans. Coeff.**	HTC [d]

a. This is the entire housing section as predefined in the mesh.

b. Superficial velocity is the velocity at which the flow would travel if the porosity of the domain were 100%. It is less than the true velocity.

c. From the problem description.

d. In order to enter an expression, you must first click *Enter Expression* .

5. Click **OK**.

12.4.4. Creating and Editing the Boundaries

Create the inlet and outlet boundaries using the values given in the problem description.

12.4.4.1. Creating the Inlet Boundary

1. Create a new boundary in domain `Pipes` named `Inlet`.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Inlet
	Location	PipeEnd 2
Boundary Details [a]	Mass and Momentum > **Normal Speed**	25 [m s^-1] [b]

Tab	Setting	Value
	Heat Transfer > **Static Temperature**	 Tinlet
a. The default level of turbulence is suitable for this application. b. From the problem description.		

3. Click **OK**.

12.4.4.2. Creating the Outlet Boundary

Set up the outlet with a static pressure boundary:

1. Create a new boundary in domain `Pipes` named `Outlet`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Outlet
	Location	PipeEnd
Boundary Details	Mass and Momentum > **Option**	 Static Pressure
	Mass and Momentum > **Relative Pressure**	 0 [Pa]

3. Click **OK**.

   The remaining outer surfaces are automatically assigned to the default (no slip wall) boundaries: `Housing Default` and `Pipes Default`.

12.4.4.3. Editing the Housing Default Boundary

In order to model the heat transfer through the `Housing` domain, several parameters from the `Housing Default` boundary need to be modified.

1. In the **Outline** tree, right-click `Housing Default` and select **Edit**.
2. Configure the following setting(s):

Tab	Setting	Value
Boundary Details	Heat Transfer > **Option**	 Heat Transfer Coefficient

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Heat Transfer > **Heat Trans. Coeff.**	HTCoutside
	Heat Transfer > **Outside Temperature**	Toutside
Solid Values	Boundary Conditions > Steel > Heat Transfer > **Option**	Heat Transfer Coefficient
	Boundary Conditions > Steel > Heat Transfer > **Heat Trans. Coeff.**	HTCoutside
	Boundary Conditions > Steel > Heat Transfer > **Outside Temperature**	Toutside

3. Click **OK**.

12.4.5. Creating the Domain Interfaces

You will next create a pair of domain interfaces to model the connection between the fluid and porous domains. The meshes on the interfaces are dissimilar, so a *General Grid Interface* (GGI) connection method is required. Domain interfaces are capable of modeling changes in reference frame as well as other changes that are not applicable to this simulation.

Two interfaces are required, one to connect the inlet flange to the catalyst housing and one to connect the outlet flange to the catalyst housing.

1. Create a new domain interface by selecting **Insert** > **Domain Interface**, or click *Domain Interface* .

2. In the **Insert Domain Interface** dialog box, set the name to `InletSide`.

3. Click **OK**.

4. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Basic Settings	**Interface Type**	Fluid Porous
	Interface Side 1 > **Domain (Filter)**	 Pipes
	Interface Side 1 > **Region List**	 FlangeEnd 2
	Interface Side 2 > **Domain (Filter)**	 Housing
	Interface Side 2 > **Region List**	 INLET
Mesh Connection	Mesh Connection Method > Mesh Connection > **Option**	 GGI

5. Click **OK**.

6. Create a similar domain interface named `OutletSide` that connects `FlangeEnd` (in domain `Pipes`) to `OUTLET` (in domain `Housing`).

12.4.6. Setting Initial Values

A sensible guess for the initial velocity is the expected velocity through the catalyst housing. You can assume incompressible flow and apply conservation of mass to obtain an approximate velocity of 2.8 [m s^-1] through the housing based on the following known information:

- The inlet velocity: 25 [m s^-1]
- The cross-sectional area of the inlet and housing, which can be determined using the function calculator in CFD-Post: 0.001913 m^2 and 0.024039 m^2 respectively
- The porosity of the honeycomb structure: 70%

1. Click *Global Initialization* .

2. Configure the following setting(s):

Tab	Setting	Value
Global Settings	Initial Conditions > Cartesian Velocity Components > **Option**	 Automatic with Value

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Initial Conditions > Cartesian Velocity Components > **U**	0 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > **V**	0 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > **W**	-2.8 [m s^-1]

3. Click **OK**.

12.4.7. Setting Solver Control

Assuming velocities of 25 m/s in the inlet and outlet pipes, and 2.8 m/s in the catalyst housing, an approximate fluid residence time of 0.1 s can be calculated. A sensible time step is 1/4 to 1/2 of the fluid residence time. In this case, use a time step of 0.04 s.

For the convergence criteria, an RMS value of at least 1e-05 is usually required for adequate convergence, but the default value of 1e-04 is sufficient for demonstration purposes.

1. Click *Solver Control* .

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Convergence Control > Fluid Timescale Control > **Timescale Control**	Physical Timescale
	Convergence Control > Fluid Timescale Control > **Physical Timescale**	0.04 [s]

3. Click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12.4.8. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .
2. Configure the following setting(s):

Setting	Value
File name	CatConv.def

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, **Solver Input File** is set.

4. If using stand-alone mode, quit CFX-Pre, saving the simulation (.cfx) file at your discretion.

12.5. Obtaining the Solution Using CFX-Solver Manager

At this point, CFX-Solver Manager is running.

1. Ensure that the **Define Run** dialog box is displayed.
2. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. At the end of the run, a dialog box is displayed stating that the simulation has ended.

3. Select **Post-Process Results**.
4. If using stand-alone mode, select **Shut down CFX-Solver Manager**.
5. Click **OK**.

12.6. Viewing the Results Using CFD-Post

In this section, you will inspect the GGI interface to see the lack of node alignment that characterizes this type of interface. You will also create plots showing the distribution of temperature and pressure on a flat plane that intersects the catalytic converter. You will then make a chart showing pressure versus Z coordinate using data from a polyline that runs around the perimeter of the catalytic converter. Finally, you will export data from the polyline to a file. Such a file could be used in other programs, or could be loaded into CFD-Post (for example, to use as data for a chart line).

The topics in this section include:

12.6.1. Viewing the Mesh on a GGI Interface
12.6.2. Creating User Locations
12.6.3. Creating Plots
12.6.4. Exporting Polyline Data

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12.6.1. Viewing the Mesh on a GGI Interface

In this section, you will examine a GGI interface. As a preliminary step, do the following:

1. When CFD-Post opens, if you see the Domain Selector dialog box, ensure that both domains are selected, then click **OK**.
2. Edit `User Locations and Plots` > `Wireframe`.
3. Set **Edge Angle** to `10 [degree]` and click `Apply` to see more of the mesh surface.
4. Turn off the visibility of `User Locations and Plots` > `Wireframe`.
5. Right-click a blank area in the viewer and select **Predefined Camera** > **View From -Z**.

In the **Outline** tree view, four interface sides are listed. There are two sides to the interface between the housing and the inlet. Similarly, there are two sides to the corresponding interface on the outlet side.

Examine the interface on the inlet side to see the nature of the GGI connection:

1. In the **Outline** tree view, edit `InletSide Side 1`.
2. Configure the following setting(s):

Tab	Setting	Value
Render	**Show Faces**	(Cleared)
	Show Mesh Lines	(Selected)
	Show Mesh Lines > **Color Mode**	User Specified
	Show Mesh Lines > **Line Color**	(Red)

3. Click **Apply**.
4. In the **Outline** tree view, edit `InletSide Side 2`.
5. Configure the following setting(s):

Tab	Setting	Value
Render	**Show Faces**	(Cleared)
	Show Mesh Lines	(Selected)
	Show Mesh Lines > **Color Mode**	User Specified
	Show Mesh Lines > **Line Color**	(Green)

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Click **Apply**.

7. Click *Fit View* to re-center and re-scale the geometry.

You can now see the tetrahedral/prism and hexahedral mesh on each side of the GGI interface. This interface was used to produce a connection between dissimilar meshes before the solution was calculated. Notice that there are more tetrahedral/prism elements than hexahedral elements and that the extent of the two meshes is not quite the same (this is most noticeable on the curved edges).

12.6.2. Creating User Locations

In this section, you will create locators that you will use to make plots.

To make it easier to see the locators, adjust the view as follows:

1. Turn on the visibility of `User Locations and Plots` > `Wireframe`.

2. Turn off the visibility of `Pipes` > `InletSide Side 1` and `Housing` > `InletSide Side 2`.

12.6.2.1. Creating a Slice Plane

Later in this tutorial, you will produce a contour plot and a vector plot to observe pressure changes. Both of these plots require a slice plane locator.

Create a slice plane through the geometry as follows:

1. Right-click a blank area in the viewer and select **Predefined Camera** > **View From +Y**.

2. Create a new plane named `Plane 1`.

3. Configure the following setting(s):

Tab	Setting	Value
Geometry	Definition > **Method**	 ZX Plane
	Definition > **Y**	 0.0 [m]
Color	**Mode**	Variable
	Variable	Steel.Temperature[a]
	Range	Global

a. If particular items do not appear in the drop-down list, click the *Ellipsis* icon to see all available items.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Click **Apply**. Notice the temperature distribution in the steel throughout the catalytic converter housing.

5. To see the temperature distribution in the fluid, change **Variable** to **Temperature** and click **Apply**.

6. Turn off the visibility of `User Locations and Plots` > `Plane 1` after you have analyzed the air temperature variation on `Plane 1`.

12.6.2.2. Creating a User Surface

You will create a user surface to observe the different characteristics of steel and air at the catalytic converter wall. Because the materials have different properties, the characteristics of each will vary slightly.

1. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Y Up)**.

2. Click **Location**

and select **User Surface**.

3. Click **OK** to accept the default name.

4. Configure the following setting(s):

Tab	Setting	Value
Geometry	**Method**	Transformed Surface
	Surface Name	Housing Default
Color	**Mode**	Variable
	Variable	Steel.Temperature
	Range	Local

5. Click **Apply**.

   Observe that the steel temperature decreases as it reaches the outlet pipe.

6. Configure the following setting(s) to show the air temperature at the wall:

Tab	Setting	Value
Color	**Variable**	Temperature
	Range	Local

7. Click **Apply**.

8. Configure the following setting(s) to show the steel heat flux at the wall:

Tab	Setting	Value
Color	**Variable**	Steel.Wall Heat Flux
	Range	Local

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9. Click **Apply**.

   The values of the heat flux are negative because heat flow is directed away from the catalyst housing. The magnitude of heat flux should be greatest where the inlet pipe meets the housing body.

10. Configure the following setting(s) to show the air heat flux at the wall:

Tab	Setting	Value
Color	**Variable**	Wall Heat Flux
	Range	Local

11. Click **Apply**.

    The values of the heat flux are negative because heat flow is directed away from the catalyst converter. The magnitude of heat flux should be greatest where the inlet pipe meets the housing body.

12. Turn off the visibility of `User Locations and Plots` > `User Surface 1`.

12.6.2.3. Creating a Polyline

Later in this tutorial, you will make a chart that plots data from a polyline.

The **Method** used to create the polyline can be `From File`, `Boundary Intersection`, or `From Contour`. If you select `From File`, you must specify a file containing point definitions in the required format.

In this tutorial, you will use the `Boundary Intersection` method. This creates a polyline from the intersecting line between a boundary object and a location (for example, between a wall and a plane). The points on the polyline are where the intersecting line cuts through a surface mesh edge.

You will be able to see the polyline following the intersecting line between the wall, inlet and outlet boundaries and the slice plane.

1. Create a new polyline named `Polyline 1`.

2. Configure the following setting(s):

Tab	Setting	Value
Geometry	**Method**	Boundary Intersection
	Boundary List	Housing Default, Pipes Default [a]
	Intersect With	Plane 1
Color	**Mode**	Constant
	Color	(Yellow)
Render	**Line Width**	3

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
a. Click the *Ellipsis* icon to select multiple items using the **Ctrl** key.		

3. Click **Apply**.
4. Turn off the visibility of `User Locations and Plots` > `Polyline 1`.

12.6.3. Creating Plots

In this section, you will make plots on the slice plane and polyline locators.

12.6.3.1. Creating a Contour Plot of Pressure

You will now create a contour plot to observe the pressure change throughout the main body of the catalytic converter:

1. Right-click a blank area in the viewer and select **Predefined Camera** > **View From +Y**.
2. Clear `Plane 1` in the **Outline** tab if you have not already done so.
3. Create a new contour plot named `Contour 1`.
4. Configure the following setting(s):

Tab	Setting	Value
Geometry	**Locations**	Plane 1
	Variable	Pressure
	Range	Global
	# of Contours	30
Render	**Show Contour Bands**	(Cleared)

5. Click **Apply**.

   From the contour plot, you can see that the pressure falls steadily through the main body of the catalytic converter.

12.6.3.2. Creating a Vector Plot on the Slice Plane

Create a vector plot to display the recirculation zone:

1. Create a new vector plot named `Vector 1`.
2. Configure the following setting(s):

Tab	Setting	Value
Geometry	**Locations**	Plane 1

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Symbol	**Symbol Size**	0.1
	Normalize Symbols	(Selected)

3. Click **Apply**.

Notice that the flow separates from the walls, where the inlet pipe expands into the flange, setting up a recirculation zone. The flow is uniform through the catalyst housing.

12.6.3.3. Creating a Chart of Pressure versus the Z Coordinate

In this section, you will make a chart to see if the pressure drop is, as expected, linear by plotting a line graph of pressure against the z-coordinate. In this case you will use CFD-Post to produce the graph, but you could also export the data, then read it into any standard plotting package.

1. Create a new chart named `Chart 1`.

2. Configure the following setting(s):

Tab	Setting	Value
General	**Title**	Pressure Drop through a Catalytic Converter
Data Series	**Name**	Pressure Drop
	Data Source > **Location**	Polyline 1
X Axis	Data Selection > **Variable**	Z
Y Axis	Data Selection > **Variable**	Pressure
Line Display	**Pressure Drop**	(Selected) [a]
	Line Display > **Line Style** [b]	None
	Line Display > **Symbols** [b]	Rectangle
Chart Display	Sizes > **Symbol**	3

a. Click the `Pressure Drop` entry in the **Specify the properties of each line** list to select it.

b. These options will only be revealed if you have successfully selected `Pressure Drop`.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click **Apply**.

   Through the main body of the catalytic converter you can see that the pressure drop is linear. This is in the region from approximately Z=0.06 to Z=0.26. The two lines show the pressure on each side of the wall. You can see a noticeable difference in pressure between the two walls on the inlet side of the housing (at around Z=0.26).

4. If required, in the **Outline** tree view, select `Contour 1` and `Vector 1`.

5. Click the **3D Viewer** tab, then right-click a blank area and select **Predefined Camera** > **View From +Y**.

   You should now see that the flow enters the housing from the inlet pipe at a slight angle, producing a higher pressure on the high X wall of the housing.

12.6.4. Exporting Polyline Data

You can export data from a polyline for use in other software.

Export data as follows:

1. From the main menu, select **File** > **Export** > **Export**.

2. Configure the following setting(s):

Tab	Setting	Value
Options	**Locations**	Polyline 1
	Export Geometry Information	(Selected) [a]
	Select Variables	Pressure
Formatting	**Precision**	3

a. This ensures that X, Y, and Z are written to the output file.

3. Click **Save**.

   The file `export.csv` will be written to the current working directory in a comma-separated variable format. This file can be opened in any text editor. You can use the exported data file to plot charts in other software such as a Microsoft Excel spreadsheet.

4. When finished, quit CFD-Post.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 13: Non-Newtonian Fluid Flow in an Annulus

This tutorial includes:

13.1. Tutorial Features

13.2. Overview of the Problem to Solve

13.3. Background Theory

13.4. Preparing the Working Directory

13.5. Defining the Case Using CFX-Pre

13.6. Obtaining the Solution Using CFX-Solver Manager

13.7. Viewing the Results Using CFD-Post

13.1. Tutorial Features

In this tutorial you will learn about:

- Defining a non-Newtonian fluid.
- Using the Moving Wall feature to apply a rotation to the fluid at a wall boundary.

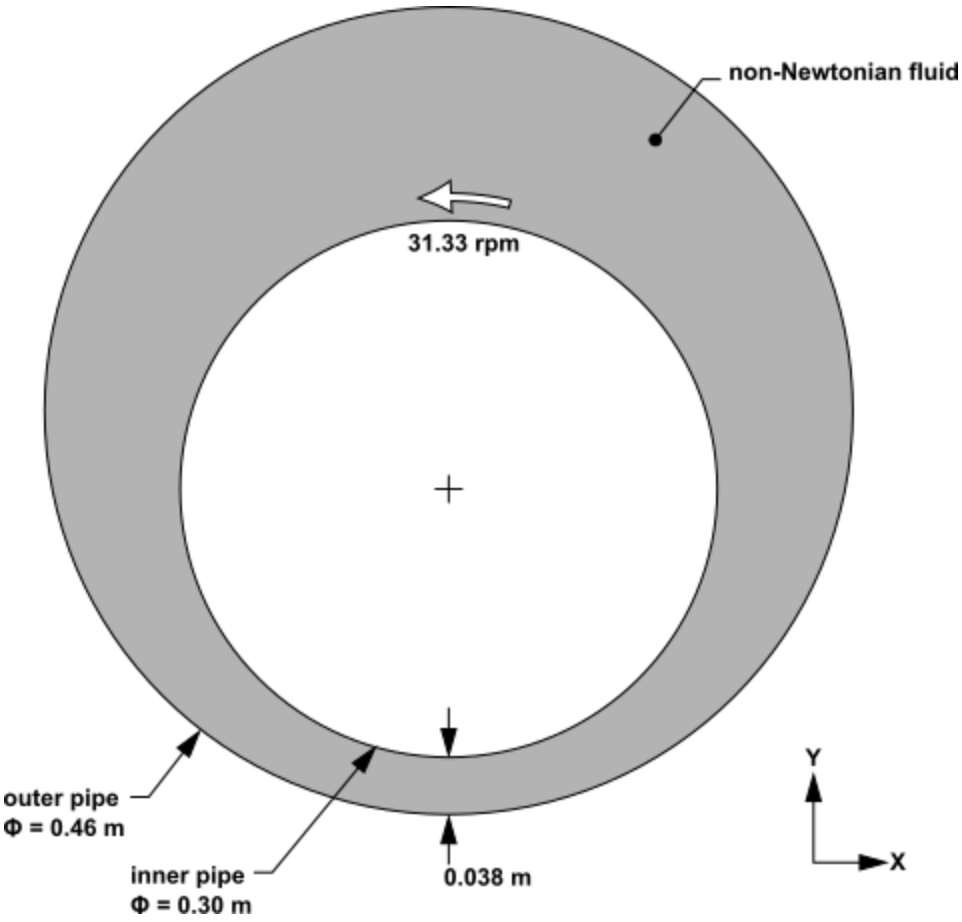
Component	Feature	Details
CFX-Pre	User Mode	General mode
	Analysis Type	Steady State
	Fluid Type	General Fluid
	Domain Type	Single Domain
	Turbulence Model	Laminar
	Heat Transfer	None
	Boundary Conditions	Symmetry Plane
		Wall: No-Slip
		Wall: Moving
	Timestep	Auto Time Scale
CFD-Post	Plots	Sampling Plane
		Vector

13.2. Overview of the Problem to Solve

In this tutorial, a shear-thickening liquid rotates in a 2D eccentric annular pipe gap. The outer pipe remains stationary while the inner pipe rotates at a constant rate about its own axis, which is the Z axis. Both pipes have nonslip surfaces.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The fluid used in this simulation has material properties that are not a function of temperature. The ambient pressure is 1 atmosphere.



The shear-thickening liquid that is used in this tutorial obeys the Ostwald de Waele model with a viscosity consistency of 10.0 kg m^{-1} s^{-1}, a Power Law index of 1.5, and a time constant of 1 s. This model is assumed to be valid for shear-strain rates ranging from 1.0E-3 s^{-1} to 100 s^{-1}. The fluid has a density of 1.0E4 kg m^{-3}. The viscosity is plotted over this range in Figure 13.2: Apparent Viscosity of a Shear-thickening Fluid (p. 295).

13.3. Background Theory

A Newtonian fluid is a fluid for which shear stress is linearly proportional to shear-strain rate, with temperature held constant. For such a fluid, the dynamic viscosity is constant and equal to the shear stress divided by the shear-strain rate.

A non-Newtonian fluid is a fluid for which the shear stress in not linearly proportional to the shear-strain rate. For such fluids, the *apparent viscosity* is the ratio of shear stress to shear-strain rate for a given shear-strain rate.

A shear-thickening fluid is a type of non-Newtonian fluid for which the apparent viscosity increases with increasing shear-strain rate.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.1: Shear Stress of a Shear-thickening Fluid

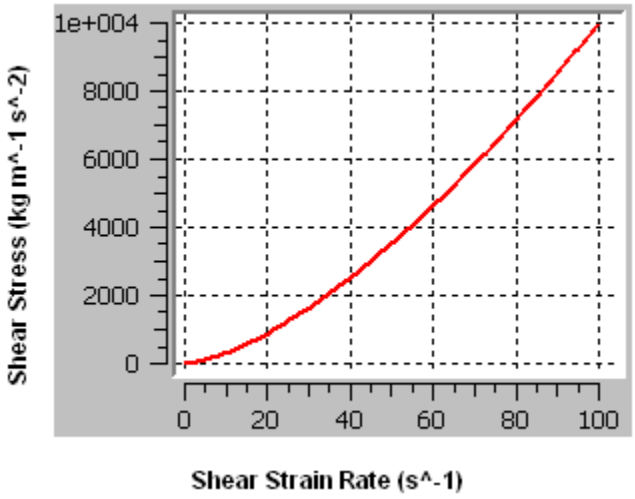
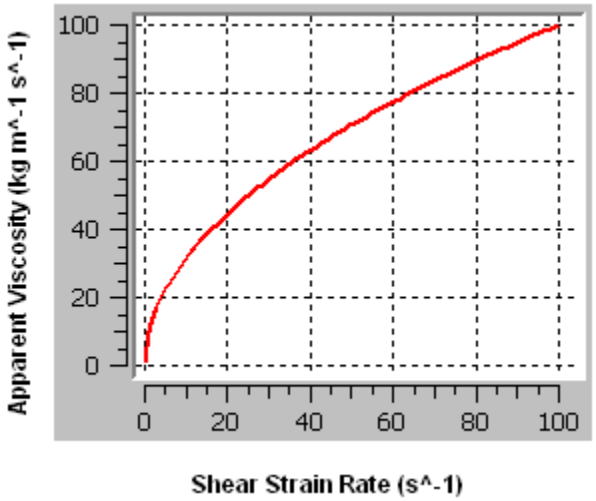


Figure 13.2: Apparent Viscosity of a Shear-thickening Fluid



This tutorial involves a shear thickening fluid that obeys the Ostwald de Waele model between apparent viscosity and shear-strain rate:

$$\mu = K\left(\lambda\dot{\gamma}\right)^{n-1} \tag{13.1}$$

where μ is the apparent viscosity, K is the viscosity consistency, $\dot{\gamma}$ is the shear-strain rate, λ is a normalizing time constant, and n is the Power Law index. Note that the units for K are not tied to the value of n because the quantity in parentheses is dimensionless.

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- Running Ansys CFX Tutorials Using Ansys Workbench (p. 34)
- Changing the Display Colors (p. 37)

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.4. Preparing the Working Directory

1. Create a working directory.

   Ansys CFX uses a working directory as the default location for loading and saving files for a particular session or project.

2. Download the `non_newton.zip` file here.

3. Unzip `non_newton.zip` to your working directory.

   Ensure that the following tutorial input file is in your working directory:

   - `NonNewtonMesh.gtm`

4. Set the working directory and start CFX-Pre.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

13.5. Defining the Case Using CFX-Pre

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `NonNewton`.
5. Click **Save**.

13.5.1. Importing the Mesh

1. Right-click `Mesh` and select **Import Mesh** > **CFX Mesh**.

   The **Import Mesh** dialog box appears.

2. Configure the following setting(s):

Setting	Value
File name	NonNewtonMesh.gtm

3. Click **Open**.

13.5.2. Creating the Fluid

As stated in the problem description, the shear-thickening liquid that is used in this tutorial obeys the Ostwald de Waele model with a viscosity consistency (K) of 10.0 kg m^{-1} s^{-1}, a Power Law index

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(n) of 1.5, and a time constant of 1 s. This model is assumed to be valid for shear-strain rates ranging from 1.0E-3 s^{-1} to 100 s^{-1}. The fluid has a density of 1.0E4 kg m^{-3}.

1. Create a new material named `myfluid`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Thermodynamic State**	(Selected)
	Thermodynamic State > **Thermodynamic State**	Liquid
Material Properties	Thermodynamic Properties > Equation of State > **Molar Mass**	1.0 [kg kmol^-1] [a]
	Thermodynamic Properties > Equation of State > **Density**	1.0E+4 [kg m^-3]
	Transport Properties > **Dynamic Viscosity**	(Selected)
	Transport Properties > Dynamic Viscosity > **Option**	Non Newtonian Model

a. While this is not the correct molar mass, it is not a value used by CFX-Solver in this particular case.

3. Configure the following setting(s) under **Transport Properties** > **Dynamic Viscosity** > **Non Newtonian Viscosity Model**:

Setting	Value
Option	Ostwald de Waele
Viscosity Consistency	10.0 [kg m^-1 s^–1]
Min. Shear Strn. Rate	0.001 [s^-1]
Max. Shear Strn. Rate	100 [s^-1]

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
Time Constant	1 [s]
Power Law Index	1.5

4. Click **OK**.

13.5.3. Creating the Domain

The flow is expected to be laminar because the Reynolds number, based on the rotational speed, the maximum width of the pipe gap, and a representative viscosity (calculated using the shear-strain rate in the widest part of the gap, assuming a linear velocity profile), is approximately 30, which is well within the laminar-flow range.

From the problem description, the ambient pressure is 1 atmosphere.

Create a fluid domain that uses the non-Newtonian fluid you created in the previous section, and specify laminar flow with a reference pressure of 1 atmosphere:

1. Click *Domain* and set the name to `NonNewton`.

2. Configure the following setting(s) of `NonNewton`:

Tab	Setting	Value
Basic Settings	**Location**	B8
	Fluid and Particle Definitions	Fluid 1
	Fluid and Particle Definitions > Fluid 1 > **Material**	 myfluid
Fluid Models	Heat Transfer > **Option**	 None
	Turbulence > **Option**	 None (Laminar)

3. Click **OK**.

13.5.4. Creating the Boundaries

The inner and outer pipes both have nonslip surfaces. A rotating-wall boundary is required for the inner pipe. For the outer pipe, which is stationary, the default boundary is suitable. By not explicitly creating a boundary for the outer pipe, the latter receives the default wall boundary.

This tutorial models 2D flow in a pipe gap, where the latter is infinite in the Z direction. The flow domain models a thin 3D slice (in fact, just one layer of mesh elements) that has two surfaces of

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

constant-Z coordinate that each require a boundary. Symmetry boundary conditions are suitable in this case, since there is no pressure gradient or velocity gradient in the Z direction.

13.5.4.1. Wall Boundary for the Inner Pipe

From the problem description, the inner pipe rotates at 31.33 rpm about the Z axis. Create a wall boundary for the inner pipe that indicates this rotation:

1. Create a new boundary named `rotwall`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Wall
	Location	rotwall
Boundary Details	Mass And Momentum > **Option**	No Slip Wall
	Mass And Momentum > **Wall Velocity**	(Selected)
	Mass And Momentum > Wall Velocity > **Option**	Rotating Wall
	Mass And Momentum > Wall Velocity > **Angular Velocity**	31.33 [rev min^-1]
	Mass And Momentum > Axis Definition > **Option**	Coordinate Axis
	Mass And Momentum > Axis Definition > **Rotation Axis**	Global Z

3. Click **OK**.

13.5.4.2. Symmetry Plane Boundary

In order to simulate the presence of an infinite number of identical 2D slices while ensuring that the flow remains 2D, apply a symmetry boundary on the high-Z and low-Z sides of the domain:

1. Create a new boundary named `SymP1`.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Symmetry
	Location	SymP1

3. Click **OK**.
4. Create a new boundary named SymP2.
5. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Symmetry
	Location	SymP2

6. Click **OK**.

   The outer annulus surfaces will default to the no-slip stationary wall boundary.

13.5.5. Setting Initial Values

A reasonable guess for the initial velocity field is a value of zero throughout the domain. In this case, the problem converges adequately and quickly with such an initial guess. If this were not the case, you could, in principle, create and use CEL expressions to specify a better approximation of the steady-state flow field based on the information given in the problem description.

Set a static initial velocity field:

1. Click *Global Initialization* .
2. Configure the following setting(s):

Tab	Setting	Value
Global Settings	Initial Conditions > Cartesian Velocity Components > **Option**	Automatic with Value
	Initial Conditions > Cartesian Velocity Components > **U**	0 [m s^-1]
	Initial Conditions > Cartesian Velocity Components	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **V**	0 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > **W**	 0 [m s^-1]

3. Click **OK**.

13.5.6. Setting Solver Control

Because this flow is low-speed, laminar, and because of the nature of the geometry, the solution converges very well. For this reason, set the solver control settings for a high degree of accuracy and a high degree of convergence.

1. Click *Solver Control* .

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Advection Scheme > **Option**	 Specified Blend Factor
	Advection Scheme > **Blend Factor**	 1.0 [a]
	Convergence Control > **Max. Iterations**	 50
	Convergence Criteria > **Residual Type**	 RMS
	Convergence Criteria > **Residual Target**	 1e-05 [b]

a. This is the most accurate but least robust advection scheme.

b. This target demands a solution with a very high degree of convergence.

3. Click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.5.7. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .
2. Configure the following setting(s):

Setting	Value
File name	NonNewton.def

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, **Solver Input File** is set.

4. If using stand-alone mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

13.6. Obtaining the Solution Using CFX-Solver Manager

When CFX-Pre has shut down and CFX-Solver Manager has started, you can obtain a solution to the CFD problem by following the instructions below:

1. Ensure that the **Define Run** dialog box is displayed.
2. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. At the end of the run, a dialog box is displayed stating that the simulation has ended.

3. Select **Post-Process Results**.
4. If using stand-alone mode, select **Shut down CFX-Solver Manager**.
5. Click **OK**.

13.7. Viewing the Results Using CFD-Post

The following steps instruct you on how to create a vector plot showing the velocity values in the domain.

1. Right-click a blank area in the viewer and select **Predefined Camera** > **View From -Z** from the shortcut menu.
2. Create a new plane named `Plane 1`.

   This plane will be used as a locator for a vector plot. To produce regularly-spaced sample points, create a circular sample plane, centered on the inner pipe, with a radius sufficient to cover the entire domain, and specify a reasonable number of sample points in the radial and theta directions. Note that the sample points are generated over the entire plane, and only those that are in the domain are usable in a vector plot.

3. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Geometry	Definition > **Method**	Point and Normal
	Definition > **Point**	0, 0, 0.015 [a]
	Definition > **Normal**	0, 0, 1
	Plane Bounds > **Type**	Circular
	Plane Bounds > **Radius**	0.3 [m]
	Plane Type	Sample
	Plane Type > **R Samples**	32
	Plane Type > **Theta Samples**	24
Render	**Show Faces**	(Cleared)
	Show Mesh Lines	(Selected)
	Show Mesh Lines > **Color Mode**	User Specified
	Line Color	(Choose green, or some other color, to distinguish the sample plane from the `Wireframe` object.)

a. This is the point on the axis of the inner pipe, in the middle of the domain in the Z direction.

4. Click **Apply**.

5. Examine the sample plane. The sample points are located at the line intersections. Note that many of the sample points are outside the domain. Only those points that are in the domain are usable for positioning vectors in a vector plot.

6. Turn off the visibility of `Plane 1`.

7. Create a new vector plot named `Vector 1` on `Plane 1`.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. Configure the following setting(s):

Tab	Setting	Value
Geometry	Definition > **Locations**	Plane 1
	Definition > **Sampling**	Vertex [a]
	Definition > **Reduction**	Reduction Factor
	Definition > **Factor**	1.0 [b]
	Definition > **Variable**	Velocity
	Definition > **Boundary Data**	Hybrid [c]
Symbol	**Symbol Size**	3 [d]

a. This causes the vectors to be located at the nodes of the sample plane you created previously. Note that the vectors can alternatively be spaced using other options that do not require a sample plane.

b. A reduction factor of 1.0 causes no reduction in the number of vectors so that there will be one vector per sample point.

c. The hybrid values are modified at the boundaries for postprocessing purposes.

d. Because CFD-Post normalizes the size of the vectors based on the largest vector, and because of the large variation of velocity in this case, the smallest velocity vectors would normally be too small to see clearly.

9. Click **Apply**.

In CFX-Pre, you created a shear-thickening liquid that obeys the Ostwald de Waele model for shear-strain rates ranging from 1.0E-3 s^{-1} to 100 s^{-1}. The values of dynamic viscosity, which are a function of the shear-strain rate, were calculated as part of the solution. You can post-process the solution using these values, which are stored in the `Dynamic Viscosity` variable. For example, you can use this variable to color graphics objects.

Color `Plane 1` using the `Dynamic Viscosity` variable:

1. Turn on the visibility of `Plane 1`.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Edit `Plane 1`.

3. Configure the following setting(s):

Tab	Setting	Value
Color	**Mode**	Variable
	Variable	Dynamic Viscosity
Render	**Show Faces**	(Selected)

4. Click **Apply**

Try plotting `Shear Strain Rate` on the same plane. Note that the distribution is somewhat different than that of `Dynamic Viscosity`, as a consequence of the nonlinear relationship (see Figure 13.2: Apparent Viscosity of a Shear-thickening Fluid (p. 295)).

When you have finished, quit CFD-Post.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 14: Flow in an Axial Turbine Stage

This tutorial includes:

14.1. Tutorial Features

14.2. Overview of the Problem to Solve

14.3. Preparing the Working Directory

14.4. Simulating the Stage with the Steady-State Frozen Rotor Model

14.5. Simulating the Stage with the Transient Rotor-Stator Model

14.6. Live Monitoring of the Solution in CFD-Post

14.1. Tutorial Features

In this tutorial you will learn about:

- Using the Turbo Wizard in CFX-Pre to quickly specify a turbomachinery application.
- Multiple Frames of Reference and Generalized Grid Interface.
- Using a Frozen Rotor interface between the rotor and stator domains.
- Modifying an existing simulation.
- Setting up a transient calculation.
- Using a Transient Rotor-Stator interface condition to replace a Frozen Rotor interface.
- Creating a transient animation showing domain movement in CFD-Post.

Component	Feature	Details
CFX-Pre	User Mode	Turbo Wizard
	Analysis Type	Steady State
		Transient
	Fluid Type	Ideal Gas
	Domain Type	Multiple Domain
		Rotating Frame of Reference
	Turbulence Model	k-Epsilon
	Heat Transfer	Total Energy
	Boundary Conditions	Inlet (Subsonic)
		Outlet (Subsonic)

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

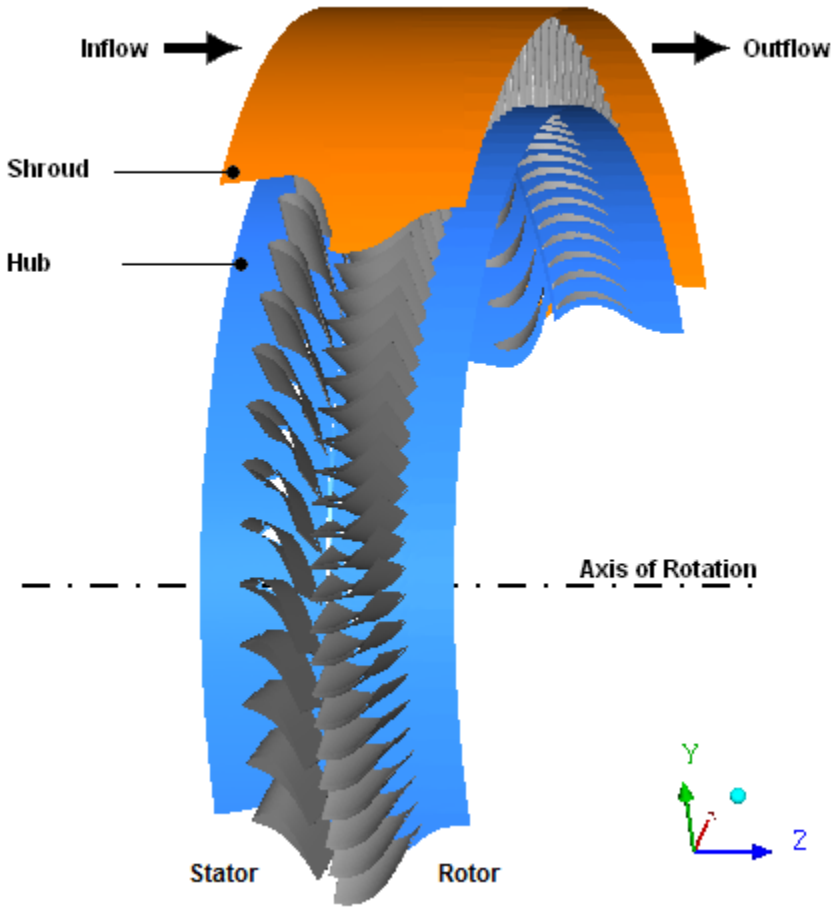
Component	Feature	Details
		Wall: No-Slip
		Wall: Adiabatic
	Domain Interfaces	Frozen Rotor
		Periodic
		Transient Rotor Stator
	Timestep	Physical Time Scale
		Transient Example
	Transient Results File	
CFX-Solver Manager	Restart	
	Parallel Processing	
CFD-Post	Plots	Animation
		Isosurface
		Surface Group
	Turbo Post	
	Other	Changing the Color Range
		Chart Creation
		Instancing Transformation
		Movie Generation
		Quantitative Calculation
		Time Step Selection
		Transient Animation

14.2. Overview of the Problem to Solve

The goal of this tutorial is to set up a transient calculation of an axial turbine stage.

The stage contains 60 stator blades and 113 rotor blades. The following figure shows approximately half of the full geometry. The Inflow and Outflow labels show the location of the modeled section in Figure 14.1: Geometry subsection (p. 310).

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The geometry to be modeled consists of a single stator blade passage and two rotor blade passages. This is an approximation to the full geometry since the ratio of rotor blades to stator blades is close to, but not exactly, 2:1. In the stator blade passage a 6° section is being modeled (360°/60 blades), while in the rotor blade passage, a 6.372° section is being modeled (2*360°/113 blades). This produces a pitch ratio at the interface between the stator and rotor of 0.942. As the flow crosses the interface, it is scaled to enable this type of geometry to be modeled. This results in an approximation of the inflow to the rotor passage. Furthermore, the flow across the interface will not appear continuous due to the scaling applied.

You should always try to obtain a pitch ratio as close to 1 as possible in your model to minimize approximations, but this must be weighed against computational resources. A full machine analysis can be performed (modeling all rotor and stator blades), which always eliminates any pitch change, but will require significant computational time. For this geometry, a 1/4 machine section (28 rotor blades, 15 stator blades) would produce a pitch change of 1.009, but this would require a model about 15 times larger than in this tutorial example.

In this example, the rotor rotates about the Z axis at 523.6 rad/s while the stator and shroud are stationary. The tip gap between the rotor blades and the shroud is not included in the mesh. Periodic boundaries are used to enable only a small section of the full geometry to be modeled.

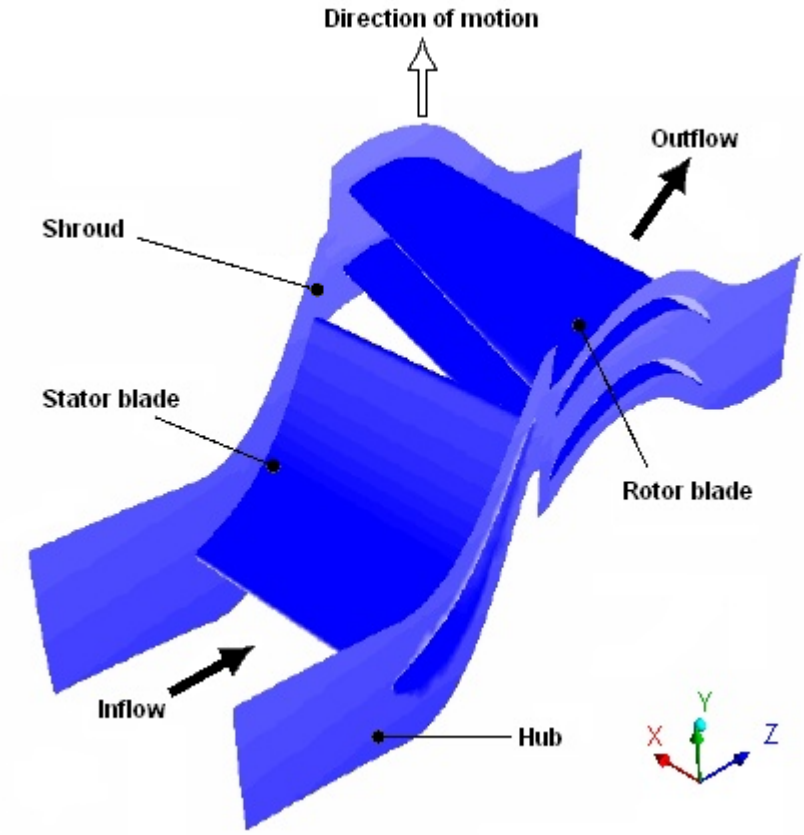
The important parameters of this problem are:

- Total pressure = 0.265 bar

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Static Pressure = 0.0662 bar
- Total temperature = 328.5 K

Figure 14.1: Geometry subsection



The overall approach of this tutorial is to run a transient simulation using the Transient Rotor-Stator interface model initialized with the results of a steady-state simulation that uses the Frozen Rotor interface model. First, you will define a steady-state Frozen Rotor simulation using the Turbomachinery Wizard. The results of this simulation will be viewed using the Turbo-Post feature. Next, you will rerun the steady-state Frozen Rotor simulation using an Exit Corrected Mass Flow Rate boundary condition. You will modify a copy of this steady-state Frozen Rotor simulation to define the transient simulation with the Transient Rotor-Stator interface model. After running the Transient Rotor-Stator simulation, you will create an animation showing domain movement.

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- Running Ansys CFX Tutorials Using Ansys Workbench (p. 34)
- Changing the Display Colors (p. 37)

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.3. Preparing the Working Directory

1. Create a working directory.

   Ansys CFX uses a working directory as the default location for loading and saving files for a particular session or project.

2. Download the `axial.zip` file here.

3. Unzip `axial.zip` to your working directory.

   Ensure that the following tutorial input files are in your working directory:

   - `rotor.grd`
   - `stator.gtm`

4. Set the working directory and start CFX-Pre.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

14.4. Simulating the Stage with the Steady-State Frozen Rotor Model

You will first create the Frozen Rotor simulation.

14.4.1. Defining the Case Using CFX-Pre

The Turbomachinery wizard is designed to simplify the setup of turbomachinery simulations.

1. In CFX-Pre, select **File** > **New Case**.
2. Select **Turbomachinery** and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `AxialIni`.
5. Click **Save**.

14.4.1.1. Basic Settings

1. In the **Basic Settings** panel, configure the following setting(s):

Setting	Value
Machine Type	Axial Turbine
Analysis Type > **Type**	 Steady State

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click **Next**.

14.4.1.2. Component Definition

Two new components are required. As they are created, meshes are imported.

1. Right-click in the blank area and select **Add Component** from the shortcut menu.
2. Create a new component of type `Stationary`, named `S1`.
3. Configure the following setting(s):

Setting	Value
Mesh > **File**	 stator.gtm [a]
a. Click *Browse* to open the **Import Mesh** dialog box. You may have to select the `CFX Mesh (*gtm *cfx)` option under **Files of type**.	

4. Create a new component of type `Rotating`, named `R1`.
5. Configure the following setting(s):

Setting	Value
Component Type > **Value**	 523.6 [radian s^-1]
Mesh > **File**	 rotor.grd [a]
a. Click *Browse* to open the **Import Mesh** dialog box. You may have to select the `CFX-TASCflow (*grd)` option under **Files of type**. Ensure that **Options** > **Mesh Units** is set to `m`.	

Note:

The components must be ordered as above (stator then rotor) in order for the interface to be created correctly. The order of the two components can be changed by right-clicking on S1 and selecting **Move Component Up**.

When a component is defined, Turbo Mode will automatically select a list of regions that correspond to certain boundary types. This information should be reviewed in the **Region Information** section to ensure that all is correct. This information will be used to help set up boundary conditions and interfaces. The upper case turbo regions that are selected (for example, HUB)

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

correspond to the region names in the CFX-TASCflow grd file. CFX-TASCflow turbomachinery meshes use these names consistently.

6. Click **Passages and Alignment** > **Edit**.

7. Configure the following setting(s):

Setting	Value
Passage and Alignment > Passages/Mesh > **Passages per Mesh**	2
Passage and Alignment > **Passages to Model**	2
Passage and Alignment > **Passages in 360**	113

8. Click **Passages and Alignment** > **Done**.

9. Configure the following setting(s):

Setting	Value
Wall Configuration	(Selected)
Wall Configuration > **Tip Clearance at Shroud**	Yes [a]
Wall Configuration > **Tip Clearance at Hub**	No [a]
a. These settings configure the slip conditions on the walls such that the shroud is counter-rotating with respect to the rotor, but the hub is not.	

10. Click **Next**.

14.4.1.3. Physics Definition

In this section, you will set properties of the fluid domain and some solver parameters.

1. In the **Physics Definition** panel, configure the following setting(s):

Setting	Value
Fluid	Air Ideal Gas
Model Data	

Setting	Value
> **Reference Pressure**	0.25 [atm]
Model Data > **Heat Transfer**	Total Energy
Model Data > **Turbulence**	k-Epsilon
Inflow/Outflow Boundary Templates > **P-Total Inlet Mass Flow Outlet**	(Selected)
Inflow/Outflow Boundary Templates > Inflow > **P-Total**	0 [atm]
Inflow/Outflow Boundary Templates > Inflow > **T-Total**	340 [K]
Inflow/Outflow Boundary Templates > Inflow > **Flow Direction**	Normal to Boundary
Inflow/Outflow Boundary Templates > Outflow > **Mass Flow**	Per Component
Inflow/Outflow Boundary Templates > Outflow > **Mass Flow Rate**	0.06 [kg s^-1]
Interface > **Default Type**	Frozen Rotor
Solver Parameters	(Selected)
Solver Parameters > **Advection Scheme**	High Resolution
Solver Parameters > **Convergence Control**	Physical Timescale
Solver Parameters	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
> **Physical Timescale**	0.002 [s] [a]

a. This time scale is approximately equal to 1 / ω, which is often appropriate for rotating machinery applications.

2. Click **Next**.

14.4.1.4. Interface Definition

CFX-Pre will try to create appropriate interfaces using the region names presented previously in the **Region Information** section. In this case, you should see that a periodic interface has been generated for both the rotor and the stator. These are required when modeling a small section of the true geometry. An interface is also required to connect the two components together across the frame change.

1. Review the various interfaces but do not change them.

2. Click **Next**.

14.4.1.5. Boundary Definition

CFX-Pre will try to create appropriate boundary conditions using the region names presented previously in the **Region Information** section. In this case, you should see a list of boundary conditions that have been generated. They can be edited or deleted in the same way as the interface connections that were set up earlier.

1. Review the various boundary definitions but do not change them.

2. Click **Next**.

14.4.1.6. Final Operations

1. Set **Operation** to `Enter General Mode`.

2. Click **Finish**.

14.4.1.7. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .

2. Configure the following setting(s):

Setting	Value
File name	AxialIni.def [a]

a. You do not need to set the full path unless you are saving the solver file somewhere other than the working directory.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, **Solver Input File** is set.

4. If using stand-alone mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

14.4.2. Obtaining the Solution Using CFX-Solver Manager

Compared to previous tutorials, the mesh for this tutorial contains many more nodes (although it is still too coarse to perform a high quality CFD simulation). This results in a corresponding increase in solution time for the problem. Solving this problem in parallel is recommended, if possible.

More detailed information about setting up CFX to run in parallel is available in Flow Around a Blunt Body (p. 187).

You can solve this example using Serial, Local Parallel or Distributed Parallel.

- Obtaining a Solution in Serial (p. 316)
- Obtaining a Solution With Local Parallel (p. 316)
- Obtaining a Solution with Distributed Parallel (p. 317)

14.4.2.1. Obtaining a Solution in Serial

If you do not have a license to run CFX in parallel you can run in serial by clicking the **Start Run** button when CFX-Solver Manager has opened up. Solution time in serial is approximately 45 minutes on a 1 GHz processor.

1. Click **Start Run** on the **Define Run** dialog box.

   CFX-Solver runs and attempts to obtain a solution. At the end of the run, a dialog box is displayed stating that the simulation has ended.

2. Select **Post-Process Results**.

3. If using stand-alone mode, select **Shut down CFX-Solver Manager**.

4. Click **OK**.

When you are finished, proceed to Viewing the Results Using CFD-Post (p. 317).

14.4.2.2. Obtaining a Solution With Local Parallel

To run in local parallel mode:

1. Set **Run Mode** to a parallel mode suitable for your environment; for example, `Intel MPI Local Parallel`.

2. If required, click *Add Partition* to add more partitions.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

By default, two partitions are assigned. Ideally, the number of partitions should not exceed the number of available processor cores.

3. Click **Start Run**.
4. Select the check box next to **Post-Process Results** when the completion message appears at the end of the run.
5. If using stand-alone mode, select the check box next to **Shut down CFX-Solver Manager**.
6. Click **OK**.

When you are finished, proceed to Viewing the Results Using CFD-Post (p. 317).

14.4.2.3. Obtaining a Solution with Distributed Parallel

1. Set **Run Mode** to a parallel mode suitable for your environment; for example, `Intel MPI Distributed Parallel`.

   One partition should already be assigned to the host that you are logged into.

2. Click *Insert Host* to specify a new parallel host.
3. In **Select Parallel Hosts**, select another host name (this should be a machine that you can log into using the same user name).
4. Click **Add**, and then **Close**.

   The names of the two selected machines should be listed in the **Host Name** column of the **Define Run** dialog box.

5. Click **Start Run**.
6. Select the check box next to **Post-Process Results** when the completion message appears at the end of the run.
7. If using stand-alone mode, select the check box next to **Shut down CFX-Solver Manager**.
8. Click **OK**.

14.4.2.4. Viewing the Results Using CFD-Post

When CFD-Post starts, the **Domain Selector** dialog box might appear. If it does, ensure that both the R1 and S1 domains are selected, then click **OK** to load the results from these domains.

The Turbo-Post feature will be demonstrated in the following sections. This feature is designed to greatly reduce the effort taken to post-process turbomachinery simulations.

14.4.2.4.1. Initializing Turbo-Post

When initializing turbo components, span, a (axial), r (radial), and Theta coordinates are generated for each component. Therefore, after entering the Turbo workspace and initializing the turbo components, you will be ready to start using the turbo-specific features offered in the Turbo

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

workspace immediately. These features include Turbo Tree View, Turbo Surface, Turbo Line and Turbo Plots.

To initialize Turbo-Post, the properties of each component must be set up. This includes information about the inlet, outlet, hub, shroud, blade, and periodic regions. It also includes information about the number of instances of each turbo component needed to represent the full geometry around the rotation axis, and the number of blade passages in the mesh for each turbo component.

1. Click the **Turbo** tab.

   The **Turbo Initialization** dialog box is displayed, and asks you whether you want to auto-initialize all components.

   Note:

   If you do not see the **Turbo Initialization** dialog box, or as an alternative to using that dialog box, you can initialize all components by clicking the **Initialize All Components** button, which is visible initially by default, or after double-clicking the **Initialization** object in the **Turbo** tree view.

2. Click **Yes**.

   In this case, the initialization works without problems. If there was a problem initializing a component, this would likely be indicated in the tree view.

14.4.2.4.2. Viewing Three Domain Passages

Next, you will create an instancing transformation to plot three domain passages; three blade passages for the stator and six blade passages for the rotor. It was chosen to create three times the geometry that was used in the simulation to help visualize the variation of pressure. Seeing neighboring passages will give a better understanding of the pressure variation through the stage.

The instancing properties of each domain have already been entered during Initialization. In the next steps, you will create a surface group plot to color the blade and hub surfaces with the same variable.

1. From the main menu, select **Insert** > **Location** > **Surface Group**.
2. Click **OK** to accept the default name.
3. Configure the following setting(s):

Tab	Setting	Value
Geometry	**Locations**	R1 Blade, R1 Hub, S1 Blade, S1 Hub
Color	**Mode**	Variable
	Variable	Pressure

4. Click **Apply**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Click the **Turbo** tab.
6. Open **Plots** > **3D View** for editing.
7. Configure the following setting(s):

Tab	Setting	Value
3D View	Graphical Instancing > **Domain**	R1
	Graphical Instancing > **# of Copies**	3

8. Click **Apply**.
9. Configure the following setting(s):

Tab	Setting	Value
3D View	Graphical Instancing > **Domain**	S1
	Graphical Instancing > **# of Copies**	3

10. Click **Apply**.
11. Click the **Outline** tab to see the surface group.

14.4.2.4.3. Blade Loading Turbo Chart

In this section, you will create a plot of pressure around the stator blade at a given spanwise location.

1. Click the **Turbo** tab.
2. In the **Turbo** tree view, double-click `Blade Loading`.
3. Click **Apply**.

   This profile of the pressure curve is typical for turbomachinery applications.
4. When you are finished viewing the chart, return to the **Outline** tab and keep CFD-Post open.

14.4.3. Simulating the Stage with the Exit Corrected Mass Flow Rate Boundary Condition

You will now use a different outlet boundary condition, namely the **Exit Corrected Mass Flow Rate** specification for the `R1 Outlet` boundary, that will enable you to specify the mass flow at the

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

outlet at corrected conditions. The Reference Pressure and Reference Temperature will be set to International Standard Atmosphere (ISA) conditions (1 atm and 288.15 K).

The exit corrected mass flow is defined as:

$$\dot{m}_{cor} = \dot{m} \frac{\sqrt{\tilde{T}_{o1} / T_{ref}}}{\tilde{P}_{o1} / P_{ref}} \tag{14.1}$$

where, $\tilde{P}_{o1}$ and $\tilde{T}_{o1}$ are mass averaged values of total pressure and temperature in the stationary frame at the outlet.

The intent of this section is to obtain the same operating point with the exit corrected mass flow boundary condition as with the previous setup that used the mass flow boundary condition at the outlet. To do this you need to calculate, from the previous case's results, the mass averaged values of total pressure and temperature at the outlet, in the stationary frame. These values along with the mass flow (0.06 kg s^-1) and the reference values mentioned above are substituted into Equation 14.1 (p. 320). The resulting value for the exit corrected mass flow is 0.3745 kg s^-1. You may want to verify this by doing the calculation yourself.

1. Ensure that the following tutorial input file is in your working directory:

   - `AxialIni_001.res`

2. Set the working directory and start CFX-Pre if it is not already open.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

3. Open the results file named `AxialIni_001.res`.

4. Save the case as `AxialIni_Cor.cfx` in your working directory.

5. In the tree view, open `R1 Outlet` for editing.

6. Configure the following setting(s):

Tab	Setting	Value
Boundary Details	Mass And Momentum > **Option**	Exit Corrected Mass Flow Rate
	Mass And Momentum > **Mass Flow Rate**	0.3745 [kg s^-1]
	Mass And Momentum > **Ref. Temperature**	288.15 [K]
	Mass And Momentum > **Reference Pressure**	1 [atm] [a]

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
a. The Ref. Temperature and Reference Pressure values should be the default settings but if they are not, ensure that they are the values given in the table.		

7. Click **OK**.

14.4.3.1. Writing the CFX-Solver Input (.def) File

1. Follow one of the next two procedures as appropriate:

   Procedure 1: If **Simulation Control** > **Execution Control** does not appear in the outline tree view:

   1. Click *Define Run* .
   2. Configure the following setting(s):

Setting	Value
File name	AxialIni_Cor.def

   3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, **Solver Input File** is set.

   Procedure 2: If **Simulation Control** > **Execution Control** appears in the outline tree view:

   1. Click *Execution Control* .
   2. Configure the following setting(s):

Tab	Setting	Value
Run Definition	Input File Settings > **Solver Input File**	 AxialIni_Cor.def

   3. Click **OK**.
   4. Click *Define Run* .

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, **Solver Input File** is set.

2. If using stand-alone mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.4.3.2. Obtaining the Solution Using CFX-Solver Manager

As in the previous setup of the steady-state simulation for this case, solving the problem in parallel is recommended, if possible.

1. Ensure that the **Define Run** dialog box is displayed.
2. Select **Run Mode** > **Intel MPI Local Parallel** and click **Start Run**.
3. In CFX-Solver Manager, select **Workspace** > **New Monitor** in the main menu and click **OK** to accept the default name.
4. Configure the following setting(s):

Tab	Setting	Value
Plot Lines	Flow > Boundary > R1 Outlet > **P-Mass Boundary Flow on R1 Outlet (R1)**	(Selected)

5. Click **OK**.
6. When the dialog box is displayed stating that the simulation has ended, click **OK**.

Note:

The resulting plot of the P-Mass Boundary Flow on R1 Outlet (R1) vs. Accumulated Time Step converges to –0.06 on the Y axis. The exit corrected mass flow rate entered (0.3745 kg s^-1) for this run yields the same physical mass flow as used in the previous steady-state setup; this is expected. The value is negative because the flow is leaving the domain.

If you would like to convince yourself that the results are identical, you can proceed to the following section and do a case comparison in CFD-Post. If you do not want to compare the cases, proceed to Simulating the Stage with the Transient Rotor-Stator Model (p. 324).

14.4.4. Comparing the Two Cases Using CFD-Post

In this section you will compare the domain passages of the two cases.

1. With `AxialIni_001.res` already loaded in CFD-Post, select **File** > **Load Results**.
2. In the **Load Results File** dialog box, select **Keep current cases loaded**, then select the file `AxialIni_Cor_001.res`. Click **Open**. If the **Domain Selector** dialog box appears, ensure that both the R1 and S1 domains are selected, and then click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. You should see the two cases beside one another in the **3D Viewer**. Click the synchronize active view icon .

4. Right-click within the 3D view and select **Predefined Camera** > **Isometric View (Y up)** to orient the views. Click *Fit View* to scale the blades appropriately within the **3D Viewer**.

5. Next, you will create an instancing transformation to plot three domain passages; three blade passages for the stator and six blade passages for the rotor, as you did for the previous case.

   The instancing properties of each domain have already been entered during Initialization. In the next steps, you will create a surface group plot to color the blade and hub surfaces with the same variable.

6. Select **Insert** > **Location** > **Surface Group** from the main menu.

7. Click **OK** to accept the default name.

8. Configure the following setting(s):

Tab	Setting	Value
Geometry	Locations > **AxialIni_Cor_001**	 R1 Blade, R1 Hub, S1 Blade, S1 Hub [a]
Color	**Mode**	Variable
	Variable	Pressure
a. Click the ... button beside the drop-down box to see all available items.		

9. Click **Apply**.

10. Click the **Turbo** tab.

11. Open **Plots** > **3D View** for editing.

12. Configure the following setting(s):

Tab	Setting	Value
3D View	Graphical Instancing > **Domain**	 R1 (AxialIni_Cor_001)
	Graphical Instancing > **# of Copies**	 3

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13. Click **Apply**.

Note:

If the image from the results of `AxialIni_Cor_001` is not displayed in the **3D Viewer**, select it from the drop-down list at the top of the viewer.

14. Configure the following setting(s):

Tab	Setting	Value
3D View	Graphical Instancing > **Domain**	 S1 (AxialIni_Cor_001)
	Graphical Instancing > **# of Copies**	 3

15. Click **Apply**.

16. Click the **Outline** tab to see the surface group.

17. To see both images, click the viewport icon and select . You may need to turn on the visibility of `Surface Group 1` in the tree view to compare the two surfaces.

After you have finished your comparison, close CFD-Post.

14.5. Simulating the Stage with the Transient Rotor-Stator Model

You will now create the Transient Rotor-Stator simulation. The existing steady-state Frozen Rotor simulation is modified to define the Transient Rotor-Stator simulation. If you have not already completed the Frozen Rotor simulation, refer to Simulating the Stage with the Steady-State Frozen Rotor Model (p. 311) before proceeding with the Transient Rotor-Stator simulation.

14.5.1. Defining the Case Using CFX-Pre

This section describes the step-by-step definition of the flow physics in CFX-Pre.

This step involves opening the original simulation and saving it to a different location.

1. Ensure that the following file is in your working directory:

   - `AxialIni_001.res`

2. Set the working directory and start CFX-Pre if it is not already running.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Open the results file named `AxialIni_001.res`.

4. Save the case as `Axial.cfx` in your working directory.

14.5.1.1. Modifying the Physics Definition

You need to modify the domain to define a transient simulation. You are going to run for a time interval such that the rotor blades pass through 1 pitch (6.372°) using 10 time steps. This is generally too few time steps to obtain high quality results, but is sufficient for tutorial purposes. The time step size is calculated as follows:

Note:

$$\text{RotationalSpeed} = 523.6 \text{ rad/s}$$

$$\text{Rotor Pitch Modeled} = 2 \cdot \left(2\pi \text{ rad} / 113\right) \approx 0.1112 \text{ rad}$$

$$\text{Time to pass through 1 pitch} \approx \left(0.1112 \text{ rad}\right) / \left(523.6 \text{ rad/s}\right) \approx 2.124\,\text{e}{-4}\,\text{s}$$

Since 10 time steps are used over this interval each time step should be 2.124e-5 s.

1. Select **Tools** > **Turbo Mode**.

   Basic Settings is displayed.

2. Configure the following setting(s):

Setting	Value
Analysis Type > **Type**	Transient
Analysis Type > **Total Time**	2.124e-4 [s]
Analysis Type > **Time Steps**	2.124e-5 [s] [a]

a. This time step size will be used until the total time is reached, giving 10 time steps, each 2.124e-5 s.

3. Click **Next**.

   Component Definition is displayed.

4. Click **Next**.

   Physics Definition is displayed.

5. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
Fluid	Air Ideal Gas
Interface > **Default Type**	 Transient Rotor Stator

Note:

A Transient Rotor-Stator calculation often runs long enough for the blades to pass through more than one pitch. In these cases, it may be useful to look at variable data averaged over the time interval required to complete 1 pitch. You can then compare data for each pitch rotation to see if a "steady state" has been achieved, or if the flow is still developing.

6. Click **Next**.

   A warning message is displayed.

7. Click **Yes**.

   Interface Definition is displayed.

8. Click **Next**.

   Boundary Definition is displayed.

9. Click **Next**.

   Final Operations is displayed.

10. Ensure that **Operation** is set to `Enter General Mode`.

11. Click **Finish**.

Initial values are required, but will be supplied later using a results file.

14.5.1.2. Setting Output Control

1. Click *Output Control* .

2. Click the **Trn Results** tab.

3. In the **Transient Results** tree view, click *Add new item* , set **Name** to `Transient Results 1`, and click **OK**.

4. Configure the following setting(s):

Setting	Value
Option	Selected Variables

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
Output Variables List	Pressure, Velocity, Velocity in Stn Frame [a]
Output Frequency > **Option**	Time Interval
Output Frequency > **Time Interval**	2.124e-5 [s]

a. Use the **Ctrl** key to select more than one variable.

5. Click **OK**.

14.5.1.3. Modifying Execution Control

1. Click *Execution Control* .
2. Configure the following setting(s):

Tab	Setting	Value
Run Definition	Input File Settings > **Solver Input File**	Axial.def [a]

a. You do not need to set the full path unless you are saving the solver file somewhere other than the working directory.

3. Confirm that the rest of the execution control settings are set appropriately.
4. Click **OK**.

14.5.1.4. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .

   A warning will appear, due to a lack of initial values.

   Initial values are required, but will be supplied later using a results file.

2. Click **Yes**.
3. If using stand-alone mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.5.2. Obtaining the Solution Using CFX-Solver Manager

When the CFX-Solver Manager has started you will need to specify an initial values file before starting the CFX-Solver.

14.5.2.1. Serial Solution

If you do not have a license, or do not want to run CFX in parallel, you can run it in serial. Solution time in serial is similar to the first part of this tutorial.

1. Under the **Initial Values** tab, select **Initial Values Specification**.
2. Under **Initial Values Specification** > **Initial Values**, select `Initial Values 1`.
3. Under **Initial Values Specification** > **Initial Values** > **Initial Values 1 Settings** > **File Name**, click *Browse* .
4. Select `AxialIni_001.res` from your working directory.
5. Click **Open**.
6. Under **Initial Values Specification** > **Use Mesh From**, select `Solver Input File`.
7. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. At the end of the run, a dialog box is displayed stating that the simulation has ended.
8. Select **Post-Process Results**.
9. If using stand-alone mode, select **Shut down CFX-Solver Manager**.
10. Click **OK**, then continue this tutorial from Monitoring the Run (p. 328).

14.5.2.2. Parallel Solution

Follow the first 6 steps of the serial procedure above (in Serial Solution (p. 328)), then perform the local parallel or distributed parallel procedure from the first part of this tutorial (in Obtaining the Solution Using CFX-Solver Manager (p. 316)).

14.5.2.3. Monitoring the Run

During the solution, look for the additional information that is provided for Transient Rotor-Stator runs. Each time the rotor is rotated to its next position, the number of degrees of rotation and the fraction of a pitch moved is given. You should see that after 10 time steps the rotor has been moved through 1 pitch.

14.5.3. Viewing the Results Using CFD-Post

To examine the transient interaction between the rotor and stator, you are going to create a blade-to-blade animation of pressure. A turbo surface will be used as the basis for this plot.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.5.3.1. Initializing Turbo-Post

All the preprocessing will be done during the initialization of the turbo components. Only a few steps will therefore be required to display a surface of constant span and to create a turbo surface later on in this tutorial.

1. Click the **Turbo** tab.

   The **Turbo Initialization** dialog box is displayed and asks you whether you want to auto-initialize all components.

   Note:

   If you do not see the **Turbo Initialization** dialog box, or as an alternative to using that dialog box, you can initialize all components by clicking the **Initialize All Components** button, which is visible initially by default, or after double-clicking the **Initialization** object in the **Turbo** tree view.

2. Click **Yes**.

   Both components (domains) are initialized based on the automatically selected turbo regions. When the process is complete, a green turbine icon appears next to each component entry in the list. Also, the viewer displays a green background mesh for each initialized component, visible when a component is selected.

3. Double-click `Component 1 (S1)` and review the automatically-selected turbo regions and other data in the details view.

4. Double-click `Component 2 (R1)` and review the automatically-selected turbo regions and other data in the details view (including the **Passages per Component** setting on the **Instancing** tab, which should have a value of 2).

14.5.3.2. Displaying a Surface of Constant Span

1. In the **Turbo** tree view, double-click `Blade-to-Blade`.

2. Click **Apply**.

   A surface of constant span appears, colored by pressure.

14.5.3.3. Using Multiple Turbo Viewports

1. In the **Turbo** tree view, double-click **Initialization**.

2. Click **Three Views**.

   Left view is **3D View**, top right is **Blade-to-Blade** and bottom right is **Meridional** view.

3. Click **Single View**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.5.3.4. Creating a Turbo Surface at Mid-Span

1. Select **Insert** > **Location** > **Turbo Surface**, and accept the default name.
2. Set **Method** to `Constant Span` and **Value** to `0.5`.
3. Under the **Color** tab, set **Mode** to `Variable`, **Variable** to `Pressure`, **Range** to `User Specified`, **Min** to `-10000 [Pa]`, and **Max** to `-7000 [Pa]`.
4. Click **Apply**.

14.5.3.5. Setting up Instancing Transformations

Next, you will use instancing transformations to view a larger section of the model. The properties for each domain have already been entered during the initialization phase, so only the number of instances must be set.

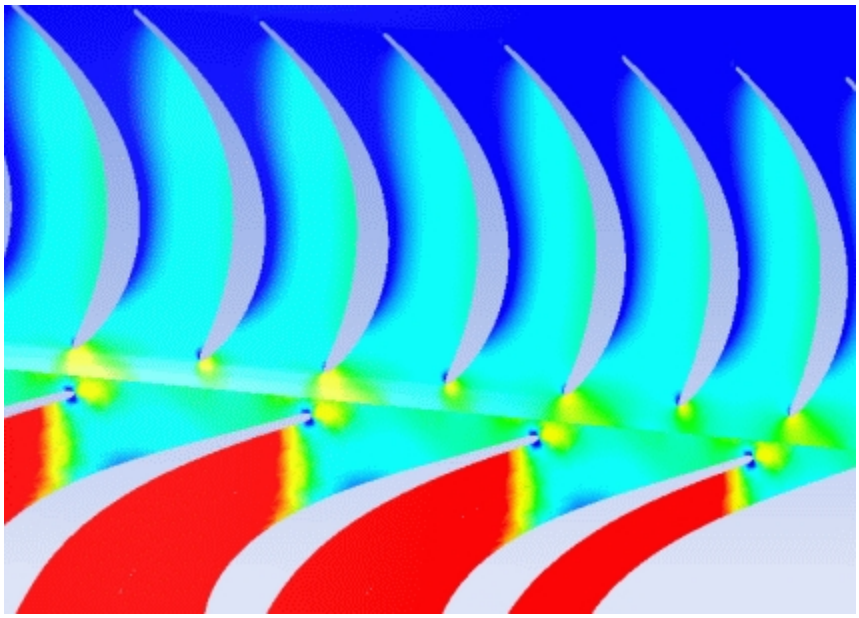
1. In the **Turbo** tree view, double-click the **3D View** object.
2. In the **Graphical Instancing** section of the form, set **# of Copies** to 6 for R1.
3. Click **Apply**.
4. In the **Graphical Instancing** section of the form, set **# of Copies** to 6 for S1.
5. Click **Apply**.
6. Return to the **Outline** tab and ensure that the turbo surface is visible again.

14.5.3.6. Animating the Movement of the Rotor Relative to the Stator

Start by loading the first time step:

1. Click *Timestep Selector* .
2. Select time value 0.
3. Click **Apply** to load the time step. The rotor blades move to their starting positions. These positions are exactly 1 pitch from the blade positions that were previously displayed, so the blades will appear to be in the same positions.
4. Turn off the visibility of `Wireframe`.
5. Position the geometry as shown below, ready for the animation. During the animation the rotor blades will move to the right. Make sure you have at least two rotor blades out of view to the left side of the viewer. They will come into view during the animation.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



6. Click *Animation* .

   The **Animation** dialog box appears.

7. Set **Type** to **Keyframe Animation**.

8. Click *New* to create `KeyFrameNo1`.

9. Select `KeyframeNo1`, then set **# of Frames** to 9, then press **Enter** while in the **# of Frames** box.

   Tip:

   Be sure to press **Enter** and confirm that the new number appears in the list before continuing.

10. Use the **Timestep Selector** to load the final time step.

11. In the **Animation** dialog box, click *New* to create `KeyframeNo2`.

12. Click *More Animation Options* to expand the **Animation** dialog box.

13. Click **Options** and set **Transient Case** to `TimeValue Interpolation`. Click **OK**.

    The animation now contains a total of 11 frames (9 intermediate frames plus the two Keyframes), one for each of the available time values.

14. In the expanded **Animation** dialog box, select **Save Movie**.

15. Set **Format** to `MPEG1`.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16. Click *Browse* , next to the **Save Movie** box and then set the file name to an appropriate file name.

17. If frame 1 is not loaded (shown in the **F:** text box at the bottom of the **Animation** dialog box), click *To Beginning* to load it.

    Wait for CFD-Post to finish loading the objects for this frame before proceeding.

18. Click *Play the animation* .

    - It takes a while for the animation to be completed.
    - To view the movie file, you will need to use a media player that supports the MPEG format.

    You will be able to see from the animation, and from the plots created previously, that the flow is not continuous across the interface. This is because a pitch change occurs. The relatively coarse mesh and the small number of time steps used in the transient simulation also contribute to this. The movie was created with a narrow pressure range compared to the global range, which exaggerates the differences across the interface.

14.5.3.7. Further Postprocessing

You can produce a report for the turbine as follows:

1. Click **File** > **Report** > **Report Templates**.
2. In the **Report Templates** dialog box, select **Turbine Report**, then click **Load**.

   The report will be generated automatically.
3. Click the **Report Viewer** tab (located below the viewer window).

   A report appears.

Note that a valid report depends on valid turbo initialization.

14.6. Live Monitoring of the Solution in CFD-Post

In this section, you will export a surface from CFD-Post and then use this surface to create a Monitor Surface in CFX-Pre. This will enable you to see the solution update in CFD-Post as the solver progresses.

14.6.1. Creating and Exporting Surfaces in CFD-Post

1. Load the `AxialIni_001.res` file created earlier.
2. Click the **Turbo** tab.
3. A dialog box will ask if you want to auto-initialize all turbo components. Click **Yes**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

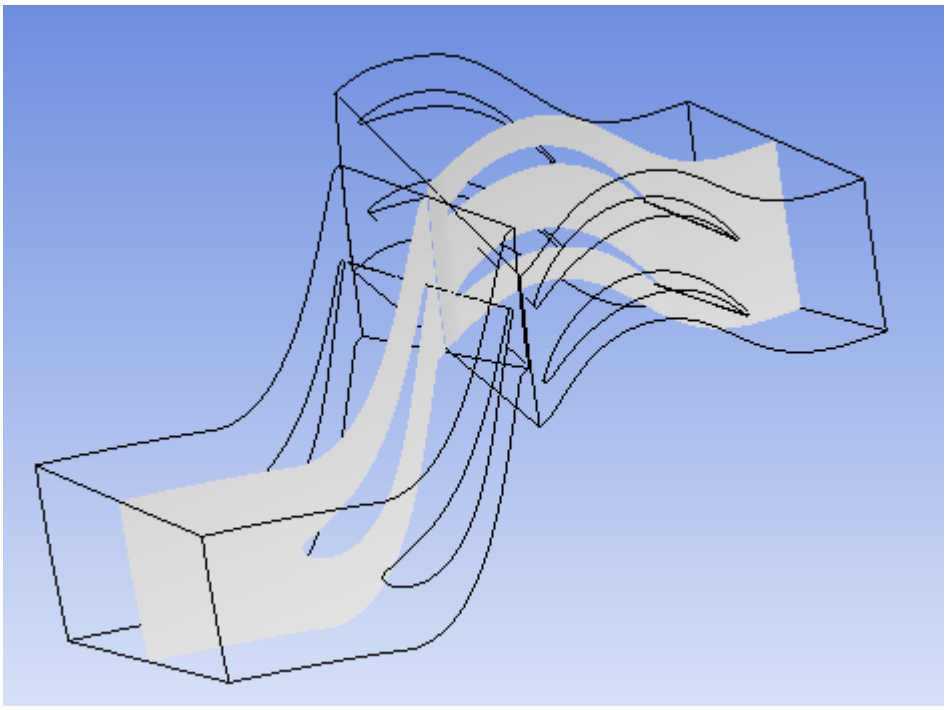
14.6.1.1. Creating a Turbo Surface at Mid-Span of the Rotor and Stator

1. Select **Insert** > **Location** > **Turbo Surface**.
2. Set the **Name** to `R1 MidSpan` and click **OK**.
3. Under the **Geometry** tab, set **Domains** to `R1`.
4. Ensure that **Method** is set to `Constant Span` and **Value** is set to `0.5` to create a surface at the mid-span of the rotor.
5. Click **Apply**.

A surface of constant span appears in the rotor.

1. Select **Insert** > **Location** > **Turbo Surface**.
2. Set the **Name** to `S1 MidSpan` and click **OK**.
3. Under the **Geometry** tab, set **Domains** to `S1`.
4. Ensure that **Method** is set to `Constant Span` and **Value** is set to `0.5` to create a surface at the mid-span of the rotor.
5. Click **Apply**.

A surface of constant span appears in the stator. You now have two surfaces appearing in the viewer.



14.6.1.2. Exporting the Surfaces

1. Click **File** > **Export** > **Export**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Export** dialog box appears.

2. Set **File** to `R1 MidSpan.csv`.
3. Set **Type** to `Geometry Only`.
4. Set **Locations** to `R1 MidSpan`.
5. Click **Save**.
6. Repeat steps 1-5 for the stator surface naming the file `S1 MidSpan.csv`.

The surfaces have now been exported to your working directory and you can now close CFD-Post.

14.6.2. Defining Monitor Surfaces in CFX-Pre

This section describes importing the surface into CFX-Pre and using it to define a rotating Monitor Surface.

1. Ensure that the following file is in your working directory:
   - `Axial.cfx`
2. Set the working directory and start CFX-Pre if it is not already running.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).
3. Open the file named `Axial.cfx`.
4. Save the case as `AxialMonitoring.cfx` in your working directory.

14.6.2.1. Importing the User Surfaces

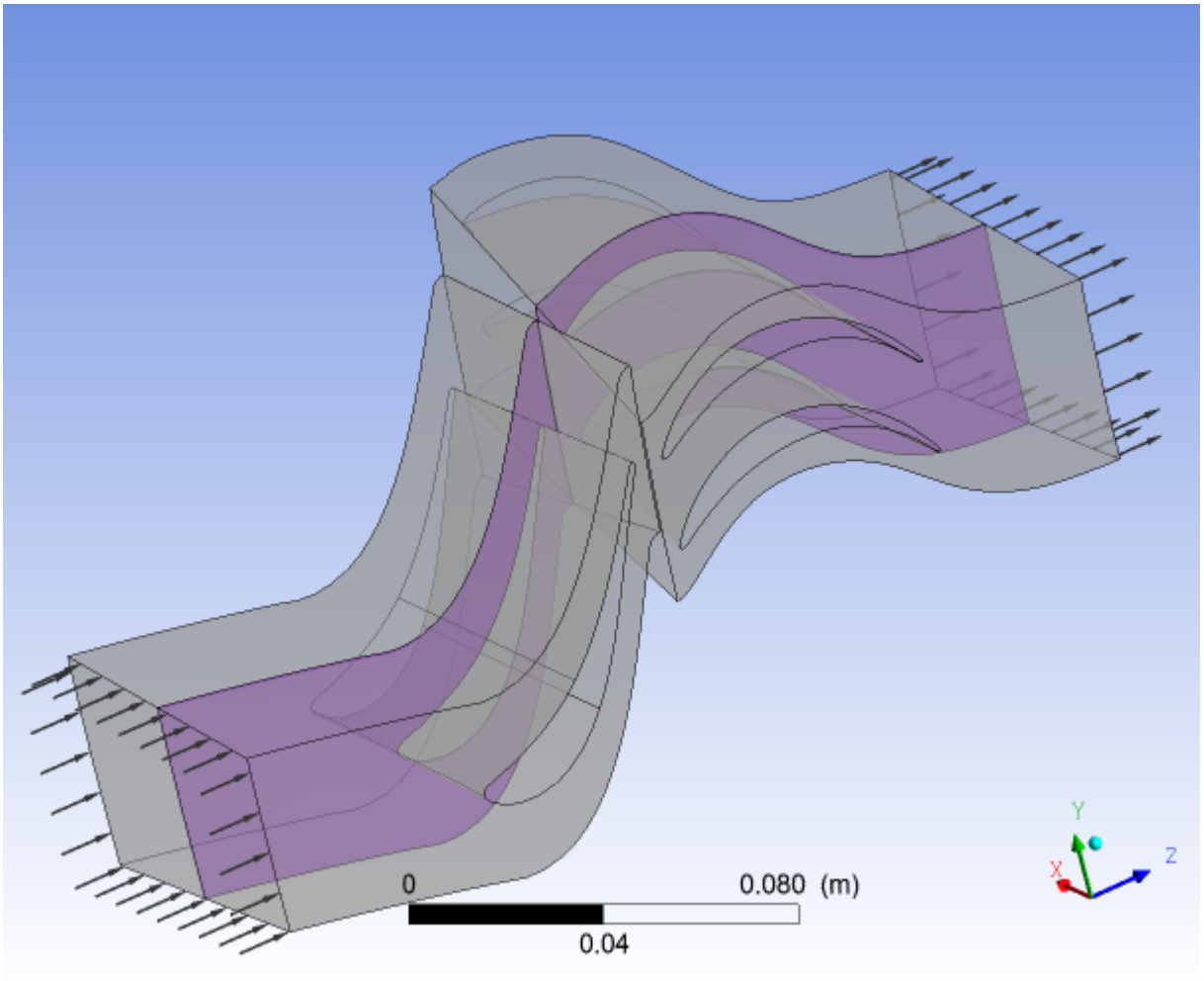
In this section, you will load the previously created surfaces into CFX-Pre.

1. Select **Insert** > **User Locations** > **User Surface**.

   The **Insert User Surface** dialog box appears.
2. Set the **Name** to `R1 MidSpan`.
3. Click *Browse* .
4. Select `R1 MidSpan.csv` from your working directory.
5. Click **Open**.
6. Select **Visibility** so you can see the surface in the graphics viewer.
7. Click **OK**.
8. Repeat steps 1-7, naming the user surface "`S1 MidSpan`" and selecting `S1 MidSpan.csv` from your working directory.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can now see the two surfaces you created in CFD-Post in the CFX-Pre viewer.



14.6.2.2. Creating a Rotating Coordinate Frame

In this section, you will create a coordinate frame that rotates at the same speed as the rotor. This will be used later in the Monitor Surface definition.

1. Select **Insert** > **Coordinate Frame** and accept the default name.
2. Configure the following setting(s):

Setting	**Value**
Option	Axis points
Frame Motion	(selected)
Frame Motion > **Option**	Rotating
Frame Motion > **Angular Velocity**	523.6 [rad/s]

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click **OK**.

14.6.2.3. Defining the Monitor Surfaces

1. Click *Output Control* .
2. Click the **Monitor** tab.
3. In the **Monitor Surfaces** group box, click *Add new item* , set **Name** to `S1 MonitorSurface`, and click **OK**.
4. Configure the following setting(s):

Setting	Value
Output Variables List	Pressure, Velocity
Output Location	S1 MidSpan
Output Frequency > **Option**	 Every Timestep

5. Click **Apply**.

Now you will create a Monitor Surface for the rotor.

1. In the **Monitor Surfaces** group box, click *Add new item* , set **Name** to `R1 MonitorSurface`, and click **OK**.
2. Configure the following setting(s):

Setting	Value
Coordinate Frame	Coord 1 [**a**]
Output Variables List	Pressure, Velocity
Output Location	R1 MidSpan
Output Frequency > **Option**	 Every Timestep
a. By specifying the rotating coordinate frame, the monitor surface will rotate at the same speed as the blades, allowing for simpler post-processing.	

3. Click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.6.2.4. Reducing Solution Data

A benefit of using Monitor Surfaces is that solution data can be written for only those surfaces. This enables you to significantly reduce the size of any solution files by disabling the CFX-Solver from writing any volume data.

1. Click *Output Control* .
2. Click the **Results** tab.
3. Set **Option** to `None`.
4. Click **OK**.

14.6.2.5. Defining the Number of Timesteps

1. Right-click `Analysis Type` in the **Outline** tree view and select **Edit**.
2. Set **Time Steps** > **Timesteps** to `2.124e-6 [s]`.

   Reducing the timestep size by a factor of 10 will make the simulation run for 100 timesteps. This allows more time to view the Monitor Surface in CFD-Post.
3. Click **OK**.

14.6.2.6. Modifying Execution Control

1. Right-click `Execution Control` in the **Outline** tree view and select **Edit**.
2. Set **Input File Settings** > **Solver Input File** to `AxialMonitoring.def`.

   This ensures that you do not overwrite any previously created `.def` file.
3. Click **OK**.

14.6.2.7. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .

   A warning will appear, due to a lack of initial values.

   Initial values are required, but will be supplied later using a results file.
2. Click **Yes**.
3. If using stand-alone mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

14.6.3. Starting the Solution Using CFX-Solver Manager

When the CFX-Solver Manager has started you will need to specify an initial values file before starting the CFX-Solver.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.6.3.1. Serial Solution

- Ensure that **Solver Input File** is set to `AxialMonitoring.def`.

1. Under the **Initial Values** tab, select **Initial Values Specification**.
2. Under **Initial Values Specification** > **Initial Values**, select `Initial Values 1`.
3. Under **Initial Values Specification** > **Initial Values** > **Initial Values 1 Settings** > **File Name**, click *Browse* .
4. Select `AxialIni_001.res` from your working directory.
5. Click **Open**.
6. Under **Initial Values Specification** > **Use Mesh From**, select `Solver Input File`.
7. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution.
8. After the run has started, **Select Tools** > **Monitor Run with CFD-Post**.

   CFD-Post opens and loads the solution data.

14.6.4. Monitoring the Solution in CFD-Post

In this section, you will create results objects and then allow CFD-Post to update and see the solution progress in real time.

14.6.4.1. Creating a Text Label to View the Current Timestep

1. Select **Insert** > **Text** and accept the default name.
2. Select **Embed Auto Annotation**.
3. Set **Type** to `Timestep`.
4. Click **Apply**.

The current timestep will be displayed at the top of the viewer.

14.6.4.2. Creating a Contour

The Monitor Surface you defined in CFX-Pre has been pre-loaded into CFD-Post. You can view it in the **Outline** table under **Other Locations**.

Select **Other Locations** > **R1 MonitorSurface** to unhide it

1. Double-click **R1 MonitorSurface**.
2. Under the **Color** tab set the **Mode** to `Variable`.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Set the **Variable** to `Pressure`.
4. Set the **Range** to `Local`.
5. Click **Apply**.

The contour is defined.

14.6.4.3. Live Monitoring of the Solution

1. Select **Monitor** > **Start Auto Update**.

   The contour now shows data and will regularly update to the data associated with the latest timestep generated by the solver. Notice how the rotor and Monitor Surface rotate together.

   Object creation is prohibited in CFD-Post during Auto Update. You can look in CFX-Solver Manager to see if a timestep is almost complete.

2. Select **Monitor** > **Stop Auto Update**.

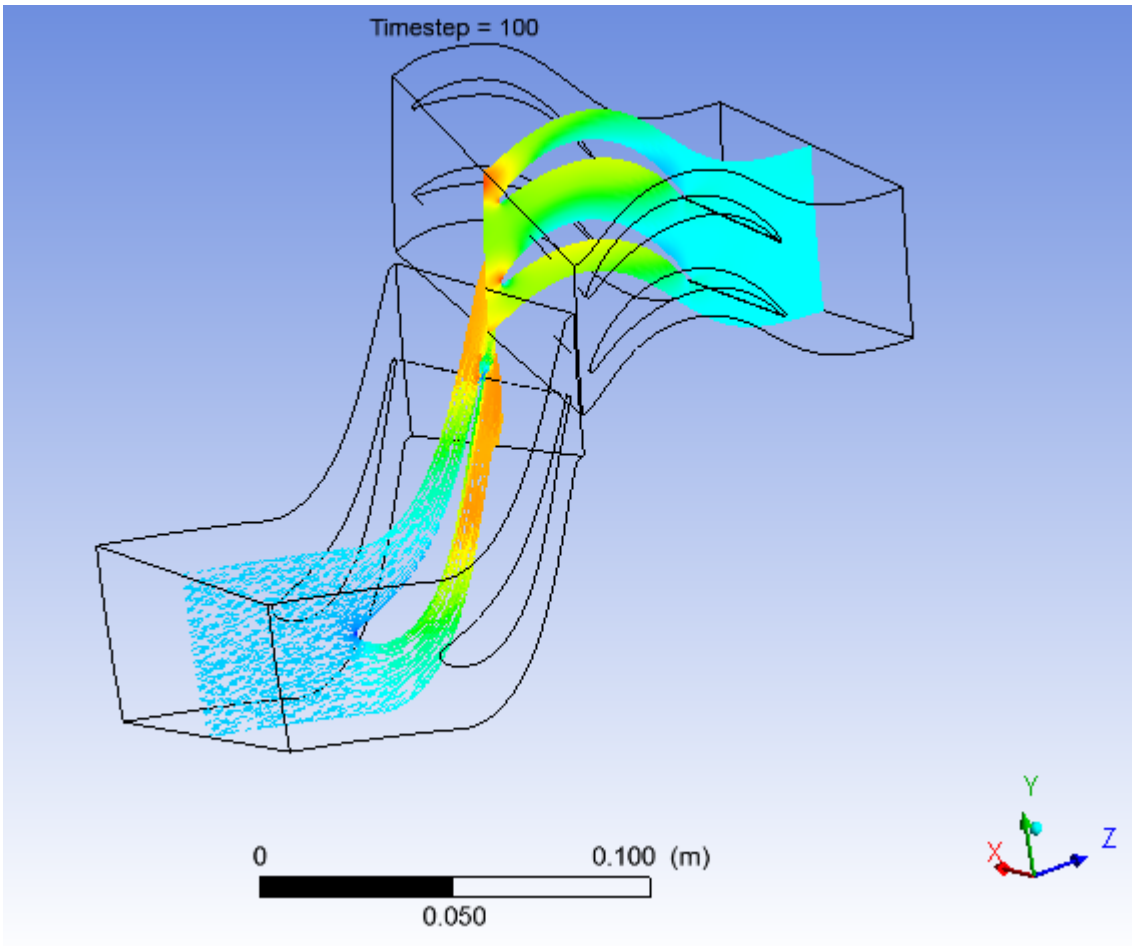
   This stops the Auto Update and you can once again create and edit objects.

14.6.4.4. Creating a Vector

1. Select **Insert** > **Vector** and accept the default name.
2. Under the **Geometry** tab, set the **Locations** to `S1 MonitorSurface`.
3. Under the **Color** tab , set the **Range** to `Local`.
4. Click **Apply**.

The vector plot appears on the stator Monitor Surface. Now both results object are visible.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



14.6.4.5. Resuming Live Monitoring of the Solution

1. Wait until another timestep has completed and then select **Monitor** > **Update Once**.

   This will load the most recently written solution data (most recent timestep in this case).

2. Select **Monitor** > **Start Auto Update** again.

   You can watch the solution progress up until the run is finished.

3. In CFX-Solver Manager, you can stop the run by selecting **Workspace** > **Stop Current Run** at any time.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 15: Reacting Flow in a Mixing Tube

This tutorial includes:

15.1. Tutorial Features

15.2. Overview of the Problem to Solve

15.3. Preparing the Working Directory

15.4. Defining the Case Using CFX-Pre

15.5. Obtaining the Solution Using CFX-Solver Manager

15.6. Viewing the Results Using CFD-Post

15.1. Tutorial Features

In this tutorial you will learn about:

- Creating and using a multicomponent fluid in CFX-Pre.
- Using CEL to model a reaction in CFX-Pre.
- Using an algebraic Additional Variable to model a scalar distribution.
- Using a subdomain as the basis for component sources.

Component	Feature	Details
CFX-Pre	User Mode	General mode
	Analysis Type	Steady State
	Fluid Type	Variable Composition Mixture
	Domain Type	Single Domain
	Turbulence Model	k-Epsilon
	Heat Transfer	Thermal Energy
	Particle Tracking	Component Source
	Boundary Conditions	Inlet (Subsonic)
		Outlet (Subsonic)
		Symmetry Plane
		Wall: Adiabatic
	Additional Variables	
	CEL (CFX Expression Language)	
	Timestep	Physical Time Scale

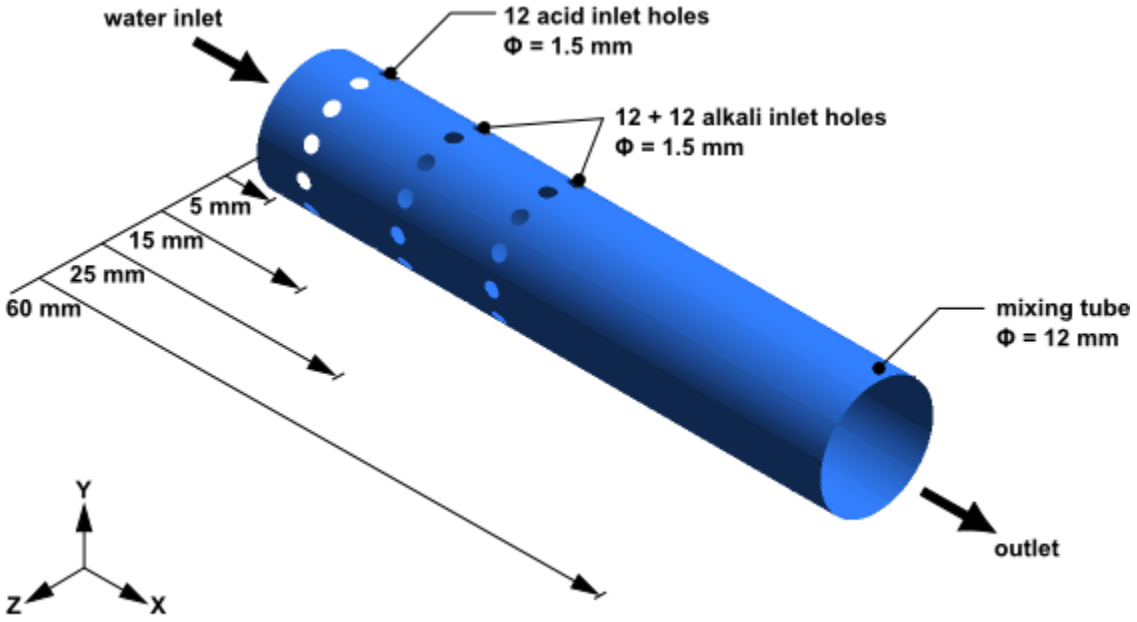
Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Component	Feature	Details
CFD-Post	Plots	Isosurface
		Slice Plane

15.2. Overview of the Problem to Solve

Reaction engineering is one of the core components in the chemical industry. Optimizing reactor design leads to higher yields, lower costs and, as a result, higher profit.

This example demonstrates the capability of Ansys CFX to model basic reacting flows using a multicomponent fluid and CEL expressions.



The geometry consists of a mixing tube with three rings with twelve holes in each ring.

The main inlet has water entering at 2 m/s with a temperature of 300 K. The pressure at the outlet is 1 atm.

Through the ring of holes nearest the inlet, a solution of dilute sulfuric acid enters at 2 m/s with a temperature of 300 K. Through each of the two other rings of holes, a solution of dilute sodium hydroxide enters at 2.923 m/s with a temperature of 300 K. The properties of the solution of sulfuric acid are shown in Table 15.1: Properties of the Dilute Sulfuric Acid Solution (p. 342):

Table 15.1: Properties of the Dilute Sulfuric Acid Solution

Property	Value
Molar mass	19.517 kg kmol^-1
Density	1078 kg m^-3
Specific heat capacity	4190 J kg^-1 K^-1

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Property	Value
Dynamic Viscosity	0.001 kg m^-1 s^-1
Thermal Conductivity	0.6 W m^-1 K^-1

Through the remaining two rings of holes, a solution of dilute sodium hydroxide (an alkali) enters with a temperature of 300 K. The properties of the solution of sodium hydroxide are shown in Table 15.2: Properties of the Dilute Sodium Hydroxide Solution (p. 343).

Table 15.2: Properties of the Dilute Sodium Hydroxide Solution

Property	Value
Molar mass	18.292 kg kmol^-1
Density	1029 kg m^-3
Specific heat capacity	4190 J kg^-1 K^-1
Dynamic Viscosity	0.001 kg m^-1 s^-1
Thermal Conductivity	0.6 W m^-1 K^-1

The acid and alkali undergo an exothermic reaction to form a solution of sodium sulfate (a type of salt) and water according to the reaction:

$$H_2SO_4 + 2NaOH \rightarrow Na_2SO_4 + 2H_2O$$

Mixing the acid and alkali solutions in a stoichiometric ratio (and enabling them to react completely) would result in a salt water solution that would include water from each of the original solutions plus water produced during the reaction. The properties of this salt water product are shown in Table 15.3: Properties of the Salt Water Product (p. 343).

Table 15.3: Properties of the Salt Water Product

Property	Value
Molar mass	18.600 kg kmol^-1
Density	1031 kg m^-3
Specific heat capacity	4190 J kg^-1 K^-1
Dynamic Viscosity	0.001 kg m^-1 s^-1
Thermal Conductivity	0.6 W m^-1 K^-1

The heat of reaction is 460 kJ per kg of dilute acid solution.

The flow is assumed to be fully turbulent and turbulence is assumed to have a significant effect on the reaction rate.

After running the simulation, you will plot the distribution of pH in the tube, and determine the extent to which the pH is neutralized at the outlet. You will also plot mass fraction distributions of acid, alkali and product.

In order to reduce memory requirements and solution time, only a 30° slice of the geometry will be modeled, and symmetry boundary conditions will be applied to represent the remaining geometry.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The reaction between acid and alkali is represented as a single-step irreversible liquid-phase reaction:

$$A+B \rightarrow C$$

Reagent A (dilute sulfuric acid) is injected through a ring of holes near the start of the tube. As it flows along the tube it reacts with Reagent B (dilute sodium hydroxide), which is injected through a further two rings of holes downstream. The product, C, remains in solution.

You will create a variable-composition mixture that contains water, the reactants, and the product. To model the reaction, you will use CEL expressions to govern the mass sources for the acid, alkali and product components. You will also use CEL expressions to govern the thermal energy source. Providing mass and energy sources over a volume requires a subdomain. Because the reaction may occur anywhere in the domain, you will create a subdomain that occupies the entire flow domain.

Note:

You can also model this type of reaction using a reacting mixture as your fluid. For details, see Combustion and Radiation in a Can Combustor (p. 471).

To model the pH, you will create an algebraic Additional Variable that is governed by a CEL expression for pH. The Additional Variable will be available in the solution results for analysis during postprocessing.

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- Running Ansys CFX Tutorials Using Ansys Workbench (p. 34)
- Changing the Display Colors (p. 37)

15.3. Preparing the Working Directory

1. Create a working directory.

   Ansys CFX uses a working directory as the default location for loading and saving files for a particular session or project.

2. Download the `reactor.zip` file here.

3. Unzip `reactor.zip` to your working directory.

   Ensure that the following tutorial input files are in your working directory:

   - `ReactorExpressions.ccl`
   - `ReactorMesh.gtm`

4. Set the working directory and start CFX-Pre.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15.4. Defining the Case Using CFX-Pre

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `Reactor`.
5. Click **Save**.

15.4.1. Importing the Mesh

1. Right-click `Mesh` and select **Import Mesh** > **CFX Mesh**.

   The **Import Mesh** dialog box appears.

2. Configure the following setting(s):

Setting	Value
File name	ReactorMesh.gtm

3. Click **Open**.

15.4.2. Creating a Multicomponent Fluid

In addition to providing template fluids, CFX allows you to create custom fluids for use in all your CFX models. A custom fluid may be defined as a pure substance, but may also be defined as a mixture, consisting of a number of transported fluid components. This type of fluid model is useful for applications involving mixtures, reactions, and combustion.

In order to define custom fluids, CFX-Pre provides the **Material** details view. This tool allows you to define your own fluids as pure substances, fixed composition mixtures or variable composition mixtures using a range of template property sets defined for common materials.

The mixing tube application requires a fluid made up from four separate materials (or components). The components are the reactants and products of a simple chemical reaction together with a neutral carrier liquid. You are first going to define the materials that take part in the reaction (acid, alkali and product) as pure substances. The neutral carrier liquid is water, and is already defined. Finally, you will create a variable composition mixture consisting of these four materials. This is the fluid that you will use in your simulation. A variable composition mixture (as opposed to a fixed composition mixture) is required because the proportion of each component will change throughout the simulation due to the reaction.

15.4.2.1. Acid Properties

The properties of the dilute sulfuric acid solution were stated in the problem description.

1. Create a new material named `acid`.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Option**	Pure Substance
	Thermodynamic State	(Selected)
	Thermodynamic State > **Thermodynamic State**	Liquid
Material Properties	**Option**	General Material
	Thermodynamic Properties > Equation of State > **Option**	Value
	Thermodynamic Properties > Equation of State > **Molar Mass**	19.517 [kg kmol^-1] [a]
	Thermodynamic Properties > Equation of State > **Density**	1078 [kg m^-3]
	Thermodynamic Properties > **Specific Heat Capacity**	(Selected)
	Thermodynamic Properties > Specific Heat Capacity > **Option**	Value
	Thermodynamic Properties > Specific Heat Capacity > **Specific Heat Capacity**	4190 [J kg^-1 K^-1]
	Transport Properties > **Dynamic Viscosity**	(Selected)
	Transport Properties > Dynamic Viscosity	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Option**	Value
	Transport Properties > Dynamic Viscosity > **Dynamic Viscosity**	 0.001 [kg m^-1 s^-1]
	Transport Properties > **Thermal Conductivity**	 (Selected)
	Transport Properties > Thermal Conductivity > **Option**	 Value
	Transport Properties > Thermal Conductivity > **Thermal Conductivity**	 0.6 [W m^-1 K^-1]
a. The Molar Masses for the three materials do not affect the solution except through the post-processed variables Molar Concentration and Molar Fraction.		

3. Click **OK**.

15.4.2.2. Alkali Properties

The properties of the dilute sodium hydroxide solution were stated in the problem description.

1. Create a new material named `alkali`.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Option**	Pure Substance
	Thermodynamic State	(Selected)
	Thermodynamic State > **Thermodynamic State**	 Liquid
Material Properties	**Option**	General Material
	Thermodynamic Properties	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Equation of State > **Option**	 Value
	Thermodynamic Properties > Equation of State > **Molar Mass**	 18.292 [kg kmol^-1]
	Thermodynamic Properties > Equation of State > **Density**	 1029 [kg m^-3]
	Thermodynamic Properties > **Specific Heat Capacity**	 (Selected)
	Thermodynamic Properties > Specific Heat Capacity > **Option**	 Value
	Thermodynamic Properties > Specific Heat Capacity > **Specific Heat Capacity**	 4190 [J kg^-1 K^-1]
	Transport Properties > **Dynamic Viscosity**	 (Selected)
	Transport Properties > Dynamic Viscosity > **Option**	 Value
	Transport Properties > Dynamic Viscosity > **Dynamic Viscosity**	 0.001 [kg m^-1 s^-1]
	Transport Properties > **Thermal Conductivity**	 (Selected)
	Transport Properties > Thermal Conductivity	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Option**	Value
	Transport Properties > Thermal Conductivity > **Thermal Conductivity**	 0.6 [W m^-1 K^-1]

3. Click **OK**.

15.4.2.3. Reaction Product Properties

The properties of the salt water product were stated in the problem description.

1. Create a new material named `product`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Option**	Pure Substance
	Thermodynamic State	(Selected)
	Thermodynamic State > **Thermodynamic State**	 Liquid
Material Properties	**Option**	General Material
	Thermodynamic Properties > Equation of State > **Option**	 Value
	Thermodynamic Properties > Equation of State > **Molar Mass**	 18.600 [kg kmol^-1]
	Thermodynamic Properties > Equation of State > **Density**	 1031 [kg m^-3]
	Thermodynamic Properties > **Specific Heat Capacity**	 (Selected)

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Thermodynamic Properties > Specific Heat Capacity > **Option**	Value
	Thermodynamic Properties > Specific Heat Capacity > **Specific Heat Capacity**	4190 [J kg^-1 K^-1]
	Transport Properties > **Dynamic Viscosity**	(Selected)
	Transport Properties > Dynamic Viscosity > **Option**	Value
	Transport Properties > Dynamic Viscosity > **Dynamic Viscosity**	0.001 [kg m^-1 s^-1]
	Transport Properties > **Thermal Conductivity**	(Selected)
	Transport Properties > Thermal Conductivity > **Option**	Value
	Transport Properties > Thermal Conductivity > **Thermal Conductivity**	0.6 [W m^-1 K^-1]

3. Click **OK**.

15.4.2.4. Fluid Properties

Define a variable composition mixture by combining water with the three materials you have defined: `acid`, `alkali`, `product`.

1. Create a new material named `mixture`.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Option**	Variable Composition Mixture
	Material Group	User, Water Data
	Materials List	Water, acid, alkali, product
	Thermodynamic State	(Selected)
	Thermodynamic State > **Thermodynamic State**	 Liquid

3. Click **OK**.

15.4.3. Creating an Additional Variable to Model pH

You are going to use an Additional Variable to model the distribution of pH in the mixing tube. You can create Additional Variables and use them in selected fluids in your domain.

1. Create a new Additional Variable named `MixturePH`.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Variable Type**	Specific
	Units	[ ]
	Tensor Type	Scalar

3. Click **OK**.

This Additional Variable is now available for use when you create or modify a domain. You will set other properties of the Additional Variable, including how it is calculated, when you apply it to the domain later in this tutorial.

15.4.4. Formulating the Reaction and pH as Expressions

This section includes:

- Stoichiometric Ratio (p. 352)
- Reaction Source Terms (p. 354)
- Calculating pH (p. 355)
- Loading the Expressions to Model the Reaction and pH (p. 357)

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The first section shows a derivation for the mass-based stoichiometric ratio of alkali solution to acid solution. This ratio is used for calculating various quantities throughout this tutorial.

The second subsection (Reaction Source Terms (p. 354)) shows you how reactions and reaction kinetics can be formulated using the Eddy Break Up (EBU) model.

The third subsection (Calculating pH (p. 355)), shows you how pH is calculated.

In the fourth subsection (Loading the Expressions to Model the Reaction and pH (p. 357)) you will use a provided file to load CEL expressions for the reaction source terms and the pH.

15.4.4.1. Stoichiometric Ratio

The mass-based stoichiometric ratio of alkali solution to acid solution is a quantity that is used in several calculations in this tutorial. It represents the mass ratio of alkali solution to acid solution which leads to complete reaction with no excess alkali or acid (that is, neutral pH). This section of the tutorial shows you how to calculate the stoichiometric ratio, and introduces other quantities that are used in this tutorial.

The alkali solution contains water and sodium hydroxide. In the alkali solution, it is assumed that the sodium hydroxide molecules completely dissociate into ions according to the following reaction:

$$\mathrm{NaOH} \Rightarrow \mathrm{Na^+} + \mathrm{OH^-}$$

The acid solution contains water and sulfuric acid. In the acid solution, it is assumed that the sulfuric acid molecules completely dissociate into ions according to the following reaction:

$$\mathrm{H_2SO_4} \Rightarrow 2\mathrm{H^+} + \mathrm{SO_4^{2-}}$$

The $\mathrm{Na^+}$ ions and $\mathrm{SO_4^{2-}}$ ions react to form sodium sulfate (a type of salt) and water according to the reaction:

$$2\mathrm{Na^+} + 2\mathrm{OH^-} + 2\mathrm{H^+} + \mathrm{SO_4^{2-}} \Rightarrow \mathrm{Na_2SO_4} + 2\mathrm{H_2O}$$

Note that this reaction requires the ions from two molecules of sodium hydroxide and the ions from one molecule of sulfuric acid. The stoichiometric ratio for the dry alkali and acid molecules is 2-to-1.

Instead of modeling dry molecules of alkali and acid, this tutorial models solutions that contain these molecules (in dissociated form) plus water. The calculations used to model the alkali-acid reactions, and to measure the pH, require a mass-based stoichiometric ratio, i, that expresses the mass ratio between the alkali solution and the acid solution required for complete reaction of all of the (dissociated) alkali and acid molecules within them.

Using $\mathrm{X^{2-}}$ to denote $\mathrm{SO_4^{2-}}$ and $\mathrm{Y^+}$ to denote $\mathrm{Na^+}$, the ratio i can be computed as the ratio of the following two masses:

- The mass of alkali solution required to contain 2 kmol of $\mathrm{Y^+}$
- The mass of acid solution required to contain 1 kmol of $\mathrm{X^{2-}}$

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A formula for calculating i is:

$$i = \frac{\left(\frac{2 \text{ kmol } Y^+}{\beta \text{ kmol } Y^+/\text{kg}}\right)}{\left(\frac{1 \text{ kmol } X^{2-}}{\alpha \text{ kmol } X^{2-}/\text{kg}}\right)} = \frac{2\alpha}{\beta} \quad (15.1)$$

where:

- β is the concentration of NaOH in kmol/kg solution (equal to the concentration of Y^+ in kmol/kg solution).
- α is the concentration of H_2SO_4 in kmol/kg solution (equal to the concentration of X^{2-} in kmol/kg solution).

The molar mass of the alkali solution (given as 18.292 kg/kmol solution) is a weighted average of the molar masses of water (18.015 kg/kmol) and dry sodium hydroxide (39.9971 kg/kmol), with the weighting in proportion to the number of each type of molecule in the solution. You can compute the fraction of the molecules in the solution that are sodium hydroxide as:

$$\begin{aligned} \mathrm{NaOH} \text{ fraction} &= \frac{\text{molar mass alkali solution} - \text{molar mass } H_2O}{\text{molar mass NaOH} - \text{molar mass } H_2O} \\ &= \frac{18.292 \text{ kg/kmol} - 18.015 \text{ kg/kmol}}{39.9971 \text{ kg/kmol} - 18.015 \text{ kg/kmol}} \\ &= 0.012601 \text{ kmol NaOH/kmol solution} \end{aligned}$$

β can then be calculated as follows:

$$\beta = \frac{0.012601 \text{ kmol NaOH/kmol solution}}{18.292 \text{ kg solution/kmol solution}} = 0.689\text{E-}3 \text{ kmol NaOH/kg solution}$$

The molar mass of the acid solution (given as 19.517 kg/kmol solution) is a weighted average of the molar masses of water (18.015 kg/kmol) and dry sulfuric acid (98.07848 kg/kmol), with the weighting in proportion to the number of each type of molecule in the solution. You can compute the fraction of the molecules in the solution that are sulfuric acid as:

$$\begin{aligned} H_2SO_4 \text{ fraction} &= \frac{\text{molar mass acid solution} - \text{molar mass } H_2O}{\text{molar mass } H_2SO_4 - \text{molar mass } H_2O} \\ &= \frac{19.517 \text{ kg/kmol} - 18.015 \text{ kg/kmol}}{98.07848 \text{ kg/kmol} - 18.015 \text{ kg/kmol}} \\ &= 0.01876 \text{ kmol } H_2SO_4/\text{kmol solution} \end{aligned}$$

α can then be calculated as follows:

$$\alpha = \frac{0.01876 \text{ kmol } H_2SO_4/\text{kmol solution}}{19.517 \text{ kg solution/kmol solution}} = 0.961\text{E-}3 \text{ kmol } H_2SO_4/\text{kg solution}$$

Substituting the values for β and α into Equation 15.1 (p. 353) yields the mass-based stoichiometric ratio of alkali solution to acid solution: $i = 2.79$.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15.4.4.2. Reaction Source Terms

The reaction and reaction rate are modeled using a basic Eddy Break Up formulation for the component and energy sources. For example, the transport equation for the mass fraction of acid solution is:

$$d\frac{\partial}{\partial t}\left(\rho\, m f_{\text{acid}}\right)+\nabla\cdot\left(\rho\, U\, m f_{\text{acid}}\right)-\nabla\cdot\left(\rho\, D_{\text{A}}\,\nabla m f_{\text{acid}}\right) = -4\,\rho\,\frac{\varepsilon}{k}\,min\left(m f_{\text{acid}}, \frac{mf_{\text{alkali}}}{i}\right) \tag{15.2}$$

where t is time, U is velocity, ρ is the local density of the variable composition mixture, $m f_{\text{acid}}$ is the mass fraction of the acid solution in the mixture, D_{A} is the kinematic diffusivity of the acid solution through the mixture, and i is the stoichiometric ratio of alkali solution to acid solution based on mass fraction. The right-hand side represents the mass source term that is applied to the transport equation for the acid solution. The left-hand side consists of the transient, advection and diffusion terms.

In addition to specifying the sources for the acid solution and alkali solution, source coefficients will also be used in order to enhance solution convergence. For details, see the technical note at the end of this section.

The reaction rate is computed as:

$$\text{ReactionRate}=\frac{4\rho\epsilon}{k}$$

where k is the turbulence kinetic energy, and ϵ is the turbulence eddy dissipation. Note that the reaction rate appears on the right-hand side of Equation 15.2 (p. 354). The reaction rate is also used to govern the rate of thermal energy production according to the relation:

$$\text{HeatSource}=\text{Heat of Reaction}\times 4\,\rho\,\frac{\varepsilon}{k}\,min\left(m f_{\text{acid}}, \frac{m f_{\text{alkali}}}{i}\right)$$

From the problem description, the heat of reaction is 460 kJ per kg of acid solution.

Note:

This is a technical note, for reference only.

A source is fully specified by an expression for its value S.

A source coefficient C is optional, but can be specified to provide convergence enhancement or stability for strongly-varying sources. The value of C may affect the rate of convergence but should not affect the converged results.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If no suitable value is available for C, the solution time scale or time step can still be reduced to help improve convergence of difficult source terms.

Important:

C must never be positive.

An optimal value for C when solving an individual equation for a positive variable φ with a source S whose strength decreases with increasing φ is

$$C=\frac{\partial S}{\partial \varphi}$$

Where this derivative cannot be computed easily,

$$C=\frac{S}{\varphi}$$

may be sufficient to ensure convergence. (This is the form used for the acid solution and alkali solution mass source coefficients in this tutorial.)

Another useful formula for C is

$$C=-\frac{\rho}{\tau}$$

where τ is a local estimate for the source time scale. Provided that the source time scale is not excessively short compared to flow or mixing time scales, this may be a useful approach for controlling sources with positive feedback ($\partial S / \partial \varphi > 0$) or sources that do not depend directly on the solved variable φ.

15.4.4.3. Calculating pH

The pH (or acidity) of the mixture is a function of the volume-based concentration of $\mathrm{H^+}$ ions. The latter can be computed using the following two equations, which are based on charge conservation and equilibrium conditions, respectively:

$$[\mathrm{H^+}]+[\mathrm{Y^+}]=2[\mathrm{X^{2-}}]+[\mathrm{OH^-}]$$

$$[\mathrm{H^+}]\,[\mathrm{OH^-}]=K_{\mathrm{W}}$$

(where K_{W} is the constant for the self-ionization of water (1.0E-14 $\mathrm{kmol^2\ m^{-6}}$)).

You can substitute one equation into the other to obtain the following quadratic equation:

$$[\mathrm{H^+}]([\mathrm{H^+}]+[\mathrm{Y^+}]-2[\mathrm{X^{2-}}])=K_{\mathrm{W}}$$

which can be rearranged into standard quadratic form as:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$[H^+]^2+([Y^+]-2[X^{2-}])[H^+]-K_W=0$$

The quadratic equation can be solved for $[H^+]$ using the equation $[H^+]=\frac{-b+\sqrt{\max[(b^2-4ac),0]}}{2a}$ where $a=1$, $b=[Y^+]-2[X^{2-}]$ and $c=-K_W$.

The volume-based concentrations of X^{2-} and Y^+ are required to calculate b, and can be calculated from the mass fractions of the components using the following expressions:

$$[X^{2-}]=\alpha\rho\left(mf_{acid}+\frac{mf_{prod}}{1+i}\right)$$

$$[Y^+]=\beta\rho\left(mf_{alkali}+\frac{imf_{prod}}{1+i}\right)$$

where:

- $[X^{2-}]$ is the concentration of X^{2-} in kmol/m^3.
- $[Y^+]$ is the concentration of Y^+ in kmol/m^3.
- α is the concentration of H_2SO_4 in kmol/kg solution (equal to the concentration of X^{2-} in kmol/kg solution).
- β is the concentration of $NaOH$ in kmol/kg solution (equal to the concentration of Y^+ in kmol/kg solution).
- ρ is the local density of the variable composition mixture.
- i is the mass-based stoichiometric ratio of alkali solution to acid solution.

Note that the second expression above can be re-written by substituting for β using Equation 15.1 (p. 353). The result is:

$$[Y^+]=2\alpha\rho\left(\frac{mf_{alkali}}{i}+\frac{mf_{prod}}{1+i}\right)$$

After solving for the concentration of H^+ ions, the pH can be computed as:

$$pH=-\log10[H^+]$$

In order to set a limit on pH for calculation purposes, the following relation will be used in this tutorial:

$$pH=-\log10(\max[[H^+],10^{-15}])$$

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15.4.4.4. Loading the Expressions to Model the Reaction and pH

Load the expressions required to model the reaction sources and pH:

1. Select **File** > **Import** > **CCL**.
2. Ensure that **Import Method** is set to **Append**.
3. Select `ReactorExpressions.ccl`, which should be in your working directory.
4. Click **Open**.

Observe the expressions listed in the tree view of CFX-Pre. Some expressions are used to support other expressions. The main expressions are:

Expression Name	Description	Supporting Expressions
pH	The pH of the mixture.	Hions, a, b, c, Yions, Xions, alpha, i
HeatSource	The thermal energy released from the reaction.	HeatReaction, Rate
AcidSource	The rate of production of acid due to the reaction (always negative or zero).	Rate
AcidSourceCoeff	The source coefficient for AcidSource (to enhance convergence).	AcidSource
AlkaliSource	The rate of production of alkali due to the reaction (always negative or zero).	Rate
AlkaliSourceCoeff	The source coefficient for AlkaliSource (to enhance convergence).	AlkaliSource
ProductSource	The rate of production of salt water product (always positive or zero).	Rate

Note that the expressions do not refer to a particular fluid since there is only a single fluid (which happens to be a multicomponent fluid). In a multiphase simulation you must prefix variables with a fluid name, for example `Mixture.acid.mf` instead of `acid.mf`.

15.4.5. Creating the Domain

In this section, you will create a fluid domain that contains the variable composition mixture and the Additional Variable that you created earlier. The Additional Variable will be set up as an algebraic equation with values calculated from the CEL expression for pH.

1. Edit `Case Options` > `General` in the **Outline** tree view and ensure that **Automatic Default Domain** is turned on.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A domain named `Default Domain` should now appear under the `Simulation` branch.

2. Edit `Default Domain`.

3. Under the **Fluid and Particle Definitions** setting, delete `Fluid 1` and create a new fluid definition called `Mixture`.

4. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Location and Type > **Location**	B1.P3
	Location and Type > **Domain Type**	Fluid Domain
	Fluid and Particle Definitions	Mixture
	Fluid and Particle Definitions > Mixture > **Material**	mixture
	Domain Models > Pressure > **Reference Pressure**	1 [atm]
Fluid Models	Heat Transfer > **Option**	Thermal Energy
	Component Models > **Component**	acid
	Component Models > Component > acid > **Option**	Transport Equation
	Component Models > Component > acid > **Kinematic Diffusivity**	(Selected)
	Component Models > Component	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> acid > Kinematic Diffusivity > **Kinematic Diffusivity**	 0.001 [m^2 s^-1]

5. Use the same **Option** and **Kinematic Diffusivity** settings for `alkali` and `product` as you have just set for `acid`.

6. For `Water`, set **Option** to `Constraint` as follows:

Tab	Setting	Value
Fluid Models	Component Models > **Component**	 Water
	Component Models > Component > Water > **Option**	 Constraint

One component must always use `Constraint`. This is the component used to balance the mass fraction equation; the sum of the mass fractions of all components of a fluid must equal unity.

7. Configure the following setting(s) to apply the Additional Variable that you created earlier:

Tab	Setting	Value
Fluid Models	Additional Variable Models > Additional Variable > **MixturePH**	 (Selected)
	Additional Variable Models > Additional Variable > MixturePH > **Option**	 Algebraic Equation [a]
	Additional Variable Models > Additional Variable > MixturePH	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Add. Var. Value**	pH

a. The other possible options either involve a transport equation to transport the Additional Variable in the flow field, or a Vector Algebraic Equation, which is for vector quantities. The Algebraic Equation is suitable because it allows the calculation of pH as a function of existing variables and expressions.

8. Click **OK**.

15.4.6. Creating a Subdomain to Model the Chemical Reactions

To provide the correct modeling for the chemical reaction you need to define mass fraction sources for the fluid components `acid`, `alkali`, and `product`. To do this, you need to create a subdomain where the relevant sources can be specified. In this case, sources need to be provided within the entire domain of the mixing tube since the reaction occurs throughout the domain.

1. Ensure that you have loaded the CEL expressions from the provided file.

   The expressions should be listed in the tree view.

2. Create a new subdomain named `sources`.

3. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Location**	B1.P3 [a]
Sources	**Sources**	(Selected)
	Sources > **Equation Sources**	acid.mf
	Sources > Equation Sources > **acid.mf**	(Selected)
	Sources > Equation Sources > acid.mf > **Option**	Source
	Sources > Equation Sources	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> acid.mf > **Source**	AcidSource
	Sources > Equation Sources > acid.mf > **Source Coefficient**	(Selected)
	Sources > Equation Sources > acid.mf > Source Coefficient > **Source Coefficient**	AcidSourceCoeff
	Sources > **Equation Sources**	alkali.mf
	Sources > Equation Sources > **alkali.mf**	(Selected)
	Sources > Equation Sources > alkali.mf > **Option**	Source
	Sources > Equation Sources > alkali.mf > **Source**	AlkaliSource
	Sources > Equation Sources > alkali.mf > **Source Coefficient**	(Selected)
	Sources > Equation Sources	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> alkali.mf > Source Coefficient > **Source Coefficient**	AlkaliSourceCoeff
	Sources > **Equation Sources**	Energy
	Sources > Equation Sources > **Energy**	(Selected)
	Sources > Equation Sources > Energy > **Option**	Source
	Sources > Equation Sources > Energy > **Source**	HeatSource
	Sources > **Equation Sources**	product.mf
	Sources > Equation Sources > **product.mf**	(Selected)
	Sources > Equation Sources > product.mf > **Option**	Source
	Sources > Equation Sources > product.mf > **Source**	ProductSource
	Sources	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Equation Sources > product.mf > **Source Coefficient**	(Selected)
	Sources > Equation Sources > product.mf > Source Coefficient > **Source Coefficient**	0 [kg m^-3 s^-1]
a. This is the 3D region that fills the domain.		

4. Click **OK**.

15.4.7. Creating the Boundary Conditions

Add boundary conditions for all boundaries except the mixing tube wall; the latter will receive the default wall condition. Many of the required settings were given in the problem description. Since the fluid in the domain is a multicomponent fluid, you can control which component enters at each inlet by setting mass fractions appropriately. Note that water is the constraint material; its mass fraction is computed as unity minus the sum of the mass fractions of the other components.

15.4.7.1. Water Inlet Boundary

Create a boundary for the water inlet using the given information:

1. Create a new boundary named `InWater`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Inlet
	Location	InWater
Boundary Details	Mass and Momentum > **Option**	Normal Speed
	Mass and Momentum > **Normal Speed**	2 [m s^-1]
	Heat Transfer > **Option**	Static Temperature

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Heat Transfer > **Static Temperature**	 300 [K]

3. Leave mass fractions for all components set to zero. Since `Water` is the constraint fluid, it will be automatically given a mass fraction of 1 on this inlet.

4. Click **OK**.

15.4.7.2. Acid Inlet Boundary

Create a boundary for the acid solution inlet hole using the given information:

1. Create a new boundary named `InAcid`.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Inlet
	Location	InAcid
Boundary Details	Mass and Momentum > **Option**	 Normal Speed
	Mass and Momentum > **Normal Speed**	 2 [m s^-1]
	Heat Transfer > **Option**	 Static Temperature
	Heat Transfer > **Static Temperature**	 300 [K]
	Component Details	acid
	Component Details > acid > **Mass Fraction**	 1.0
	Component Details	alkali
	Component Details > alkali > **Mass Fraction**	 0
	Component Details	product
	Component Details	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> product > **Mass Fraction**	 0

3. Click **OK**.

15.4.7.3. Alkali Inlet Boundary

Create a boundary for the alkali solution inlet holes using the given information:

1. Create a new boundary named `InAlkali`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Inlet
	Location	InAlkali
Boundary Details	Mass and Momentum > **Option**	 Normal Speed
	Mass and Momentum > **Normal Speed**	 2.923 [m s^-1]
	Heat Transfer > **Option**	 Static Temperature
	Heat Transfer > **Static Temperature**	 300 [K]
	Component Details > **acid**	 (Selected)
	Component Details > acid > **Mass Fraction**	 0
	Component Details > **alkali**	 (Selected)
	Component Details > alkali > **Mass Fraction**	 1

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Component Details > **product**	 (Selected)
	Component Details > product > **Mass Fraction**	 0

3. Click **OK**.

15.4.7.4. Outlet Boundary

Create a subsonic outlet at 1 atm (which is the reference pressure that was set in the domain definition):

1. Create a new boundary named `out`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Outlet
	Location	out
Boundary Details	Mass and Momentum > **Option**	 Static Pressure
	Mass and Momentum > **Relative Pressure**	 0 [Pa]

3. Click **OK**.

15.4.7.5. Symmetry Boundary

The geometry models a 30° slice of the full geometry. Create two symmetry boundaries, one for each side of the geometry, so that the simulation models the entire geometry.

1. Create a new boundary named `sym1`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Symmetry
	Location	sym1

3. Click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Create a new boundary named `sym2`.

5. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Symmetry
	Location	sym2

6. Click **OK**.

Note that, in this case, a periodic interface can be used as an alternative to the symmetry boundary conditions.

15.4.7.6. Default Wall Boundary

The default adiabatic wall boundary applies automatically to the remaining unspecified boundary, which is the mixer wall. The default boundary is a smooth, no-slip, adiabatic wall.

15.4.8. Setting Initial Values

The values for `acid`, `alkali`, and `product` will be initialized to 0. Since `Water` is the constrained component, it will make up the remaining mass fraction which, in this case, is 1.

Since the inlet velocity is 2 m/s, a reasonable guess for the initial velocity is 2 m/s.

1. Click *Global Initialization* .

2. Configure the following setting(s):

Tab	Setting	Value
Global Settings	Initial Conditions > Cartesian Velocity Components > **Option**	Automatic with Value
	Initial Conditions > Cartesian Velocity Components > **U**	2 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > **V**	0 [m s^-1]
	Initial Conditions > Cartesian Velocity Components	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **W**	0 [m s^-1]
	Initial Conditions > **Component Details**	acid
	Initial Conditions > Component Details > acid > **Option**	Automatic with Value
	Initial Conditions > Component Details > acid > **Mass Fraction**	0
	Initial Conditions > **Component Details**	alkali
	Initial Conditions > Component Details > alkali > **Option**	Automatic with Value
	Initial Conditions > Component Details > alkali > **Mass Fraction**	0
	Initial Conditions > **Component Details**	product
	Initial Conditions > Component Details > product > **Option**	Automatic with Value
	Initial Conditions	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Component Details > product > **Mass Fraction**	 0

3. Click **OK**.

15.4.9. Setting Solver Control

1. Click *Solver Control* .
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Advection Scheme > **Option**	 High Resolution
	Convergence Control > **Max. Iterations**	 50
	Convergence Control > Fluid Timescale Control > **Timescale Control**	 Physical Timescale
	Convergence Control > Fluid Timescale Control > **Physical Timescale**	 0.01 [s] [a]

a. The length of mixing tube is 0.06 m and inlet velocity is 2 m/s. An estimate of the dynamic timescale is 0.03 s. An appropriate time step would be between 1/4 and 1/2 of this estimate.

3. Click **OK**.

15.4.10. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .
2. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
File name	Reactor.def

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, **Solver Input File** is set.

4. If using stand-alone mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

15.5. Obtaining the Solution Using CFX-Solver Manager

When CFX-Solver Manager has started, obtain a solution to the CFD problem as follows:

1. Ensure that the **Define Run** dialog box is displayed.
2. Select **Double Precision**.

   This provides the precision required to evaluate the expression for pH.

3. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. At the end of the run, a dialog box is displayed stating that the simulation has ended.

4. Select **Post-Process Results**.
5. If using stand-alone mode, select **Shut down CFX-Solver Manager**.
6. Click **OK**.

15.6. Viewing the Results Using CFD-Post

To see the nature and extent of the reaction process, examine the pH, the mass fractions, and turbulence quantities on a plane as follows:

1. Create an XY slice plane through Z = 0.
2. Turn off the visibility of the plane you just created.
3. Create contour plots of the following variables on that plane:

   - `MixturePH`
   - `acid.Mass Fraction`
   - `alkali.Mass Fraction`
   - `product.Mass Fraction`
   - `Turbulence Kinetic Energy`

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- `Turbulence Eddy Dissipation`

4. Create an expression for `Turbulence Eddy Dissipation`/`Turbulence Kinetic Energy`, then create a variable using the expression (only variables can be plotted) and create a contour plot using that variable. This quantity is an indicator of the reaction rate — it represents 1 / mixing timescale.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 16: Heat Transfer from a Heating Coil

This tutorial includes:

16.1. Tutorial Features

16.2. Overview of the Problem to Solve

16.3. Preparing the Working Directory

16.4. Defining the Case Using CFX-Pre

16.5. Obtaining the Solution using CFX-Solver Manager

16.6. Viewing the Results Using CFD-Post

16.1. Tutorial Features

In this tutorial you will learn about:

- Creating and using a solid domain as a heating coil in CFX-Pre.
- Creating a domain interface.
- Modeling conjugate heat transfer in CFX-Pre.
- Using electricity to power a heat source.
- Plotting temperature on a cylindrical locator in CFD-Post.
- Lighting in CFD-Post.

Component	Feature	Details
CFX-Pre	User Mode	General mode
	Analysis Type	Steady State
	Fluid Type	General Fluid
	Domain Type	Multiple Domain
	Turbulence Model	k-Epsilon
		Shear Stress Transport
	Heat Transfer	Thermal Energy
	Heat Transfer Modeling	Conjugate Heat Transfer (via Electrical Resistance Heating)
	Boundary Conditions	Inlet (Subsonic)
		Opening
		Wall: No-Slip

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Component	Feature	Details
		Wall: Adiabatic
	CEL (CFX Expression Language)	
	Timestep	Physical Time Scale
CFD-Post	Plots	Contour
		Cylindrical Locator
		Isosurface
		Temperature Profile Chart
	Other	Changing the Color Range
		Expression Details View
		Lighting Adjustment
		Variable Details View
		Exporting Results to Ansys

16.2. Overview of the Problem to Solve

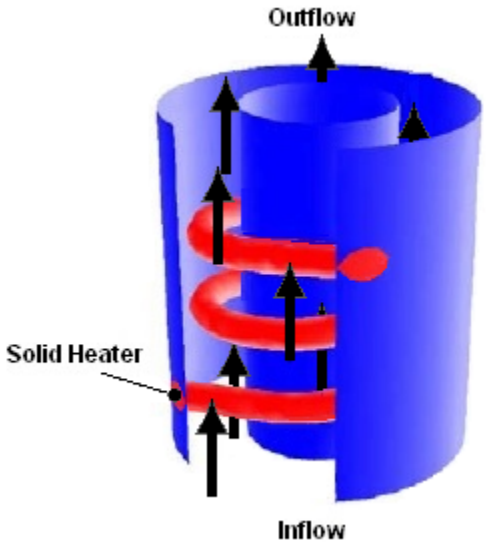
This tutorial demonstrates the capability of Ansys CFX to model conjugate heat transfer. In this tutorial, a heat exchanger is used to model the transfer of thermal energy from an electrically-heated solid copper coil to the water flowing around it.

There is a fluid domain for the water and a solid domain for the coil. The fluid domain is an annular region that envelops the coil, and has water at an initial temperature of 300 K flowing through it at 0.4 m/s. The copper coil has a 4.4 V difference in electric potential from one end to the other end and is given an initial temperature of 550 K. Assume that the copper has a uniform electrical conductivity of 59.6E+06 S/m and that there is a 1 mm thick calcium carbonate deposit on the heating coil.

The material parameters for the calcium carbonate deposit are:

- Molar Mass = `100.087[kg kmol^-1]`
- Density = `2.71[g cm^-3]`
- Specific Heat Capacity = `0.9[J g^-1 K^-1]`
- Thermal Conductivity = `3.85[W m^-1 K^-1]`

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- Running Ansys CFX Tutorials Using Ansys Workbench (p. 34)
- Changing the Display Colors (p. 37)

16.3. Preparing the Working Directory

1. Create a working directory.

   Ansys CFX uses a working directory as the default location for loading and saving files for a particular session or project.

2. Download the `cfx_heating_coil.zip` file here.

3. Unzip `cfx_heating_coil.zip` to your working directory.

   Ensure that the following tutorial input file is in your working directory:

   - `HeatingCoilMesh.gtm`

4. Set the working directory and start CFX-Pre.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

16.4. Defining the Case Using CFX-Pre

In this tutorial, you will create a solid copper coil with a 1 mm thick calcium carbonate deposit.

1. In CFX-Pre, select **File** > **New Case**.

2. Select **General** and click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Select **File** > **Save Case As**.
4. Under **File name**, type `HeatingCoil`.
5. If you are notified the file already exists, click **Overwrite**.
6. Click **Save**.

16.4.1. Importing the Mesh

1. Right-click `Mesh` and select **Import Mesh** > **CFX Mesh**.

   The **Import Mesh** dialog box appears.

2. Configure the following setting(s):

Setting	Value
File name	HeatingCoilMesh.gtm

3. Click **Open**.
4. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)** from the shortcut menu.

16.4.2. Editing the Material Properties

1. Expand `Materials` in the tree view, right-click `Copper` and select **Edit**.
2. Configure the following setting(s) of `Copper`:

Tab	Setting	Value
Material Properties	**Electromagnetic Properties**	Expand the **Electromagnetic Properties** frame [a]
	Electromagnetic Properties > **Electrical Conductivity**	(Selected)
	Electromagnetic Properties > Electrical Conductivity > **Electrical Conductivity**	59.6E+06 [S m^-1]

a. Expand a section by clicking *Roll Down* +.

3. Click **OK** to apply these settings to `Copper`.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.4.3. Defining the Calcium Carbonate Deposit Material

Create a new material definition that will be used to model the calcium carbonate deposit on the heating coil:

1. Click *Material* and name the new material `Calcium Carbonate`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Material Group**	User [a]
	Thermodynamic State	(Selected)
	Thermodynamic State > **Thermodynamic State**	Solid
Material Properties	Thermodynamic Properties > Equation of State > **Molar Mass**	100.087 [kg kmol^-1]
	Thermodynamic Properties > Equation of State > **Density**	2.71 [g cm^-3] [b]
	Thermodynamic Properties > **Specific Heat Capacity**	(Selected)
	Thermodynamic Properties > Specific Heat Capacity > **Specific Heat Capacity**	0.9 [J g^-1 K^-1] [b]
	Transport Properties > **Thermal Conductivity**	(Selected) [c]
	Transport Properties > Thermal Conductivity > **Thermal Conductivity**	3.85 [W m^-1 K^-1]

a. The material properties for Calcium Carbonate defined in this table came directly from the Overview of the Problem to Solve (p. 374) section at the beginning of this tutorial.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
b. Make sure that you change the units to those indicated. c. You may need to first expand the **Transport Properties** frame by clicking *Roll Down* .		

3. Click **OK** to apply these settings.

16.4.4. Creating the Domains

This simulation requires both a fluid domain and a solid domain. First, you will create a fluid domain for the annular region of the heat exchanger.

16.4.4.1. Creating a Fluid Domain

The fluid domain will include the region of fluid flow but exclude the solid copper heating coil.

1. Click *Domain* and set the name to `WaterZone`.
2. Configure the following setting(s) of `WaterZone`:

Tab	Setting	Value
Basic Settings	Location and Type > **Location**	Annulus [a]
	Fluid and Particle Definitions	Fluid 1
	Fluid and Particle Definitions > Fluid 1 > **Material**	Water
	Domain Models > Pressure > **Reference Pressure**	1 [atm]
Fluid Models	Heat Transfer > **Option**	Thermal Energy
Initialization	**Domain Initialization**	(Selected)
a. This region name may be different depending on how the mesh was created. You should pick the region that forms the exterior surface of the volume surrounding the coil.		

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click **OK** to apply these settings to `WaterZone`.

16.4.4.2. Creating a Solid Domain

Since you know that the copper heating element will be much hotter than the fluid, you can initialize the temperature to a reasonable value. The initialization option that is set when creating a domain applies only to that domain.

Create the solid domain as follows:

1. Create a new domain named `SolidZone`.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Location and Type > **Location**	Coil [a]
	Location and Type > **Domain Type**	Solid Domain
	Solid Definitions	Solid 1
	Solid Definitions > Solid 1 > Solid 1 > **Material**	Copper
Solid Models	Heat Transfer > **Option**	Thermal Energy
	Electromagnetic Model	(Selected)
	Electromagnetic Model > Electric Field Model > **Option**	Electric Potential
Initialization	Domain Initialization > Initial Conditions > Temperature > **Option**	Automatic with Value
	Domain Initialization	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Initial Conditions > Temperature > **Temperature**	 550 [K]

a. This region name may be different depending on how the mesh was created. You should pick the region that forms the coil.

3. Click **OK** to apply these settings.

16.4.5. Creating the Boundaries

You will now set the boundary conditions using the values given in the problem description.

16.4.5.1. Heating Coil Boundaries

In order to pass electricity through the heating coil, you are going to specify a voltage of 0 [V] at one end of the coil and 4.4 [V] at the other end:

1. Click *Boundary* and select `in SolidZone` from the drop-down menu that appears.
2. Name this new boundary `Ground` and click **OK**.
3. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Wall
	Location	Coil End 1[a]
Boundary Details	Electric Field > **Option**	 Voltage
	Electric Field > **Voltage**	 0 [V]

a. You will need to click *Multi-select from extended list* ... to see a list of all regions.

4. Click **OK** to apply these settings.
5. Create a similar boundary named `Hot` at the other end of the coil, Coil End 2, and apply a voltage of 4.4[V].

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.4.5.2. Inlet Boundary

You will now create an inlet boundary for the cooling fluid (Water).

1. Create a new boundary in the `WaterZone` domain named `inflow`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Inlet
	Location	inflow
Boundary Details	Mass and Momentum > **Option**	Normal Speed
	Mass and Momentum > **Normal Speed**	0.4 [m s^-1]
	Heat Transfer > **Option**	Static Temperature
	Heat Transfer > **Static Temperature**	300 [K]

3. Click **OK** to apply these settings.

16.4.5.3. Opening Boundary

An opening boundary is appropriate for the exit in this case because, at some stage during the solution, the coiled heating element will cause some recirculation at the exit. At an opening boundary you need to set the temperature of fluid that enters through the boundary. In this case it is useful to base this temperature on the fluid temperature at the outlet, since you expect the fluid to be flowing mostly out through this opening.

1. Insert a new expression by clicking *Expression* .
2. Name this new expression `OutletTemperature` and press the **Enter** key to continue.
3. In the **Definition** entry box, type the formula `areaAve(T)@outflow`
4. Click **Apply**.
5. Close the **Expressions** view by clicking *Close* at the top of the tree view.
6. Create a new boundary in the `WaterZone` domain named `outflow`.
7. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Basic Settings	**Boundary Type**	Opening
	Location	outflow
Boundary Details	Mass and Momentum > **Option**	Opening Pres. and Dirn
	Mass and Momentum > **Relative Pressure**	0 [Pa]
	Heat Transfer > **Option**	Static Temperature
	Heat Transfer > **Static Temperature**	OutletTemperature [a]
a. In order to enter an expression, you need to click *Enter Expression* .		

8. Click **OK** to apply these settings.

A default no slip, adiabatic wall boundary named `WaterZone Default` will be applied automatically to the remaining unspecified external boundaries of the `WaterZone` domain.

Two more boundary conditions are generated automatically when a domain interface is created to connect the fluid and solid domains. The domain interface is discussed in the next section.

16.4.6. Creating the Domain Interface

1. Click *Domain Interface* from the row of icons located along the top of the screen.
2. Accept the default name `Domain Interface 1` by clicking **OK**.
3. Configure the following setting(s) of `Domain Interface 1`:

Tab	Setting	Value
Basic Settings	**Interface Type**	Fluid Solid
	Interface Side 1 > **Domain (Filter)**	WaterZone
	Interface Side 1 > **Region List**	coil surface
	Interface Side 2	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Domain (Filter)**	SolidZone
	Interface Side 2 > **Region List**	 F22.33, F30.33, F31.33, F32.33, F34.33, F35.33
Additional Interface Models	**Heat Transfer**	(Selected)
	Heat Transfer > Interface Model > **Option**	 Thin Material
	Heat Transfer > Interface Model > **Material**	 Calcium Carbonate
	Heat Transfer > Interface Model > **Thickness**	 1 [mm] [a]
a. Make sure that you change the units to those indicated.		

4. Click **OK** to apply these settings.

16.4.7. Setting Solver Control

1. Click *Solver Control* .

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Convergence Control > Fluid Timescale Control > **Timescale Control**	 Physical Timescale
	Convergence Control > Fluid Timescale Control > **Physical Timescale**	 2 [s]

For the **Convergence Criteria**, an RMS value of at least 1e-05 is usually required for adequate convergence, but the default value is sufficient for demonstration purposes.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click **OK** to apply these settings.

16.4.8. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .
2. Configure the following setting(s):

Setting	Value
File name	HeatingCoil.def

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, **Solver Input File** is set.

4. If using stand-alone mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

16.5. Obtaining the Solution using CFX-Solver Manager

1. Ensure that the **Define Run** dialog box is displayed.
2. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. At the end of the run, a dialog box is displayed stating that the simulation has ended.

   While the calculations proceed, you can see residual output for various equations in both the text area and the plot area. Use the tabs to switch between different plots (for example, **Heat Transfer**, **Turbulence (KE)**, and so on) in the plot area. You can view residual plots for the fluid and solid domains separately by editing the workspace properties (under **Workspace** > **Workspace Properties**).

3. Select **Post-Process Results**.
4. If using stand-alone mode, select **Shut down CFX-Solver Manager**.
5. Click **OK**.

16.6. Viewing the Results Using CFD-Post

The following topics will be discussed:

16.6.1. Heating Coil Temperature Range
16.6.2. Creating a Cylindrical Locator
16.6.3. Specular Lighting
16.6.4. Moving the Light Source

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.6.1. Heating Coil Temperature Range

To grasp the effect of the calcium carbonate deposit, it is beneficial to compare the temperature range on either side of the deposit.

1. When CFD-Post opens, if you see the **Domain Selector** dialog box, ensure that both domains are selected, then click **OK**.
2. Create a new contour named `Contour 1`.
3. Configure the following setting(s):

Tab	Setting	Value
Geometry	**Location**	Domain Interface 1 Side 1 [a]
	Variable	Temperature
	Range	Local
	Boundary Data > **Hybrid**	 (Selected)
a. This is the deposit side that is in contact with the water.		

4. Click **Apply**.
5. Take note of the temperature range displayed below the **Range** drop-down box. The temperature on the outer surface of the deposit should range from around 380 [K] to 740 [K].
6. Change the contour location to `Domain Interface 1 Side 2` (the deposit side that is in contact with the coil) and click **Apply**.

   Notice how the temperature ranges from around 420 [K] to 815 [K] on the inner surface of the deposit.

16.6.2. Creating a Cylindrical Locator

Next, you will create a cylindrical locator close to the outside wall of the annular domain. This can be done by using an expression to specify radius and locating a particular radius with an isosurface.

16.6.2.1. Expression

1. Create a new expression by clicking *Expression* .
2. Set the name of this new expression to `expradius` and press the **Enter** key to continue.
3. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
Definition	(x^2 + y^2)^0.5

4. Click **Apply**.

16.6.2.2. Variable

1. Create a new variable by clicking *Variable* .
2. Set the name of this new variable to `radius` and press the **Enter** key to continue.
3. Configure the following setting(s):

Setting	Value
Expression	expradius

4. Click **Apply**.

16.6.2.3. Isosurface of the variable

1. Insert a new isosurface by clicking *Location* > **Isosurface**.
2. Accept the default name `Isosurface 1` by clicking **OK**.
3. Configure the following setting(s):

Tab	Setting	Value
Geometry	Definition > **Variable**	radius
	Definition > **Value**	0.8 [m] [a]
Color	**Mode**	Variable
	Variable	Temperature
	Range	User Specified [b]
	Min	299 [K]
	Max	309 [K]
Render	**Show Faces**	(Selected)

a. The maximum radius is 1 m, so a cylinder locator at a radius of 0.8 m is suitable.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
b. The full temperature range is much larger due to temperature extremes on a small fraction of the isosurface. By neglecting those extreme temperatures, more colors are used over the range of interest.		

4. Click **Apply**.

5. Turn off the visibility of `Contour 1` so that you have an unobstructed view of `Isosurface 1`.

Note:

The default range legend now displayed is that of the isosurface and not the contour. The default legend is set according to what is being edited in the details view.

16.6.2.4. Creating a Temperature Profile Chart

For a quantitative analysis of the temperature variation through the water and heating coil, it is beneficial to create a temperature profile chart.

First, you will create a line that passes through two turns of the heating coil. You can then graphically analyze the temperature variance along that line by creating a temperature chart.

1. Insert a line by clicking *Location*

> **Line**.

2. Accept the default name `Line 1` by clicking **OK**.

3. Configure the following setting(s) of `Line 1`

Tab	Setting	Value
Geometry	Definition > **Point 1**	-0.75, 0, 0
	Definition > **Point 2**	-0.75, 0, 2.25
	Line Type > **Sample**	(Selected)
	Line Type > **Samples**	200

4. Click **Apply**.

5. Create a new chart by clicking *Chart* .

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Name this chart `Temperature Profile` and press the **Enter** key to continue.
7. Click the **Data Series** tab.
8. Set **Data Source** > **Location** to `Line 1`.
9. Click the **Y Axis** tab.
10. Set **Data Selection** > **Variable** to `Temperature`.
11. Click **Apply**.

You can see from the chart that the temperature spikes upward when entering the deposit region and is at its maximum at the center of the coil turns.

16.6.3. Specular Lighting

Specular lighting is on by default. Specular lighting allows glaring bright spots on the surface of an object, depending on the orientation of the surface and the position of the light. You can disable specular lighting as follows:

1. Click the **3D Viewer** tab at the bottom of the viewing pane.
2. Edit `Isosurface 1` in the **Outline** tree view.

Tab	Setting	Value
Render	Show Faces > **Specular**	 (Cleared)

3. Click **Apply**.

16.6.4. Moving the Light Source

To move the light source, click within the **3D Viewer**, then press and hold **Shift** while pressing the arrow keys left, right, up or down.

Tip:

If using the stand-alone version, you can move the light source by positioning the mouse pointer in the viewer, holding down the **Ctrl** key, and dragging using the right mouse button.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 17: Multiphase Flow in a Mixing Vessel

This tutorial includes:

17.1. Tutorial Features

17.2. Overview of the Problem to Solve

17.3. Preparing the Working Directory

17.4. Defining the Case Using CFX-Pre

17.5. Obtaining the Solution Using CFX-Solver Manager

17.6. Viewing the Results Using CFD-Post

17.1. Tutorial Features

In this tutorial you will learn about:

- Setting up a multiphase flow simulation involving air and water.
- Importing meshes that have CFX-4 and CFX Mesh file formats.
- Setting up a simulation using multiple frames of reference.
- Using a fluid dependent turbulence model to set different turbulence options for each fluid.
- Specifying buoyant flow.
- Specifying a degassing outlet boundary to enable air, but not water, to escape from the boundary.
- Connecting two domains (one for a tank and one for an impeller inside the tank) via Frozen Rotor interfaces.
- Modeling rotational periodicity using periodic boundary conditions.
- Using periodic GGI interfaces where the mesh does not match exactly.
- Using thin surfaces for blade and baffle surfaces.

Component	Feature	Details
CFX-Pre	User Mode	General mode
	Analysis Type	Steady State
	Fluid Type	General Fluid
	Domain Type	Multiple Domain
		Rotating Frame of Reference

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Component	Feature	Details
	Turbulence Model	Dispersed Phase Zero Equation
		Fluid-Dependent
		k-Epsilon
	Heat Transfer	None
	Buoyant Flow	
	Multiphase	
	Boundary Conditions	Inlet (Subsonic)
		Outlet (Degassing)
		Wall: Thin Surface
		Wall: (Slip Depends on Volume Fraction)
	Domain Interfaces	Frozen Rotor
		Periodic
		Thin Surface Partners
	Output Control	
	Timestep	Physical Time Scale
CFD-Post	Plots	Default Locators
		Slice Plane
	Other	Quantitative Calculation

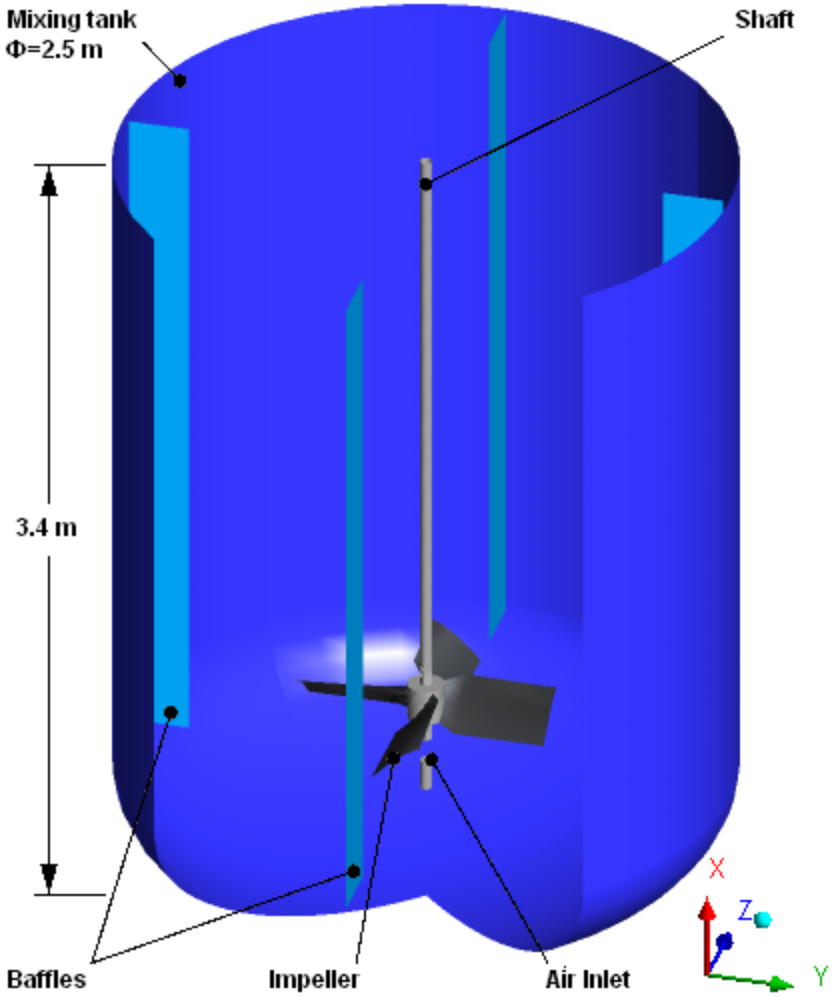
17.2. Overview of the Problem to Solve

This example simulates the mixing of water and air in a mixing vessel. The geometry consists of a mixing tank vessel, an air injection pipe, four baffles, a rotating impeller, and a shaft that runs vertically through the vessel. The impeller rotates at 84 rpm about the X axis (in the counterclockwise direction, when viewed from above). Air is injected into the vessel through an inlet pipe located below the impeller at a speed of 5 m/s. The inlet pipe diameter is 2.48 cm. Assume that both the water and air remain at a constant temperature of 25°C and that the air is incompressible, with a density equal to that at 25°C and 1 atmosphere. Also assume that the air bubbles are 3 mm in diameter.

Examine the steady-state distribution of air in the tank. Also calculate the torque and power required to turn the impeller at 84 rpm.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 17.1: Cut-away Diagram of the Mixer



The figure above shows the full geometry with part of the tank walls and one baffle cut away. The symmetry of the vessel allows a 1/4 section of the full geometry to be modeled.

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- Running Ansys CFX Tutorials Using Ansys Workbench (p. 34)
- Changing the Display Colors (p. 37)

17.3. Preparing the Working Directory

1. Create a working directory.

   Ansys CFX uses a working directory as the default location for loading and saving files for a particular session or project.

2. Download the `multiphase_mixer.zip` file here.

3. Unzip `multiphase_mixer.zip` to your working directory.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ensure that the following tutorial input files are in your working directory:

- `MixerImpellerMesh.gtm`
- `MixerTank.geo`

4. Set the working directory and start CFX-Pre.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

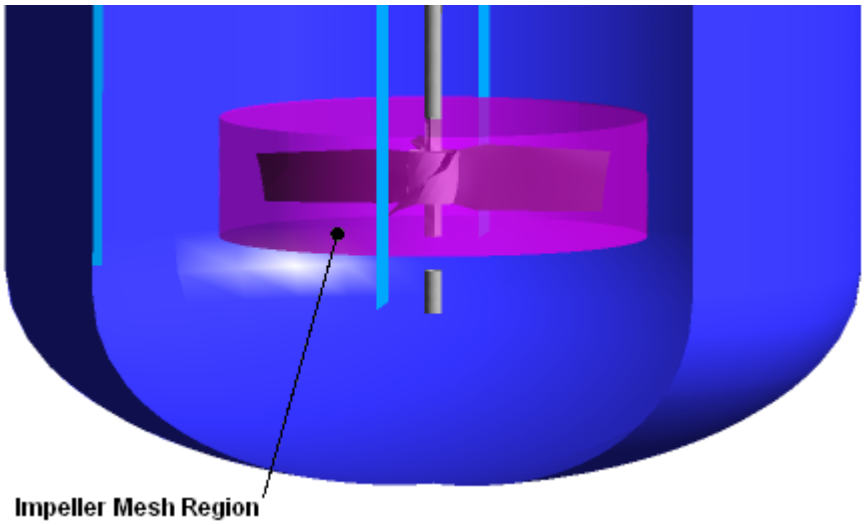
17.4. Defining the Case Using CFX-Pre

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `MultiphaseMixer`.
5. Click **Save**.

17.4.1. Importing the Meshes

In this tutorial, two mesh files are provided: one for the mixer tank excluding the impeller, and one for the impeller. These meshes fit together to occupy the entire tank. The region occupied by the impeller mesh is indicated in Figure 17.2: Impeller Mesh Region (p. 392).

Figure 17.2: Impeller Mesh Region



Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Next, you will import the mesh for the mixer tank, followed by the mesh for the impeller. The impeller mesh, as provided, is not located in the correct spatial position relative to the tank mesh. After importing the impeller mesh, you will move it to the correct position.

Note:

This simulation involves the use of two domains: a stationary fluid domain on the main 3D region of the tank mesh and a rotating fluid domain on the main 3D region of the impeller mesh. It is not necessary to use separate meshes in this type of simulation, as long as there are 3D regions available for locating these two domains.

17.4.1.1. Importing the Mixer Tank Mesh

The mixer tank mesh is provided as a CFX-4 mesh file (`*.geo`). Import it as follows:

1. Right-click `Mesh` and select **Import Mesh** > **Other**.

   The **Import Mesh** dialog box appears.

2. Configure the following setting(s):

Setting	Value
Files of type	CFX-4 (*geo)
File name	MixerTank.geo
Options > **Mesh Units**	 m
Advanced Options > CFX-4 Options > Create 3D Regions on > **Fluid Regions (USER3D, POROUS)**	 (Cleared) [a]
a. In this case, the mesh file contains USER3D regions that you do not need.	

3. Click **Open**.

17.4.1.2. Importing the Impeller Mesh

The impeller mesh is provided as a CFX Mesh file (`*.gtm`). Import it as follows:

1. Right-click `Mesh` and select **Import Mesh** > **CFX Mesh**.

   The **Import Mesh** dialog box appears.

2. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
File name	MixerImpellerMesh.gtm

3. Click **Open**.

4. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (X up)** to view the mesh assemblies.

17.4.1.3. Relocating the Impeller Mesh

In the next step you will move the impeller mesh to its correct position.

1. Right-click `MixerImpellerMesh.gtm` and select **Transform Mesh**.

   The **Mesh Transformation Editor** dialog box appears.

2. Configure the following setting(s):

Setting	Value
Transformation	Translation
Method	Deltas
Dx, Dy, Dz	0.275, 0, 0

3. Click **Apply** then **Close**.

17.4.1.4. Viewing the Mesh at the Tank Periodic Boundary

1. In the **Outline** workspace, expand the tree to show `MixerTank.geo` > `Principal 3D Regions` > `Primitive 3D` > `Principal 2D Regions`.

2. Click the primitive region `BLKBDY_TANK_PER2`.

You can now see the mesh on one of the periodic regions of the tank. To reduce the solution time for this tutorial, the mesh used is very coarse. This is not a suitable mesh to obtain accurate results, but it is sufficient for demonstration purposes.

Note:

If you do not see the surface mesh, highlighting may be turned off. If highlighting is disabled, toggle *Highlighting* . The default highlight type will show the surface mesh for any selected regions. If you see a different highlighting type, you can alter it by selecting **Edit** > **Options** and browsing to **CFX-Pre** > **Graphics Style**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

17.4.2. Creating the Domains

The mixer requires two domains: a rotating impeller domain and a stationary tank domain. Both domains contain water as a continuous phase and air as a dispersed phase. The domains will model turbulence, buoyancy, and forces between the fluids.

17.4.2.1. Rotating Domain for the Impeller

As stated in the problem description, the impeller rotates at 84 rpm.

1. Click *Domain* and set the name to `impeller`.
2. Under the **Fluid and Particle Definitions** setting, delete `Fluid 1`.
3. Click *Add new item* and name it `Air`
4. Click *Add new item* and name it `Water`
5. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Location and Type > **Location**	 Main
	Fluid and Particle Definitions	Air
	Fluid and Particle Definitions > Air > **Material**	 Air at 25 C
	Fluid and Particle Definitions > Air > Morphology > **Option**	 Dispersed Fluid
	Fluid and Particle Definitions > Air > Morphology > **Mean Diameter**	 3 [mm]
	Fluid and Particle Definitions	Water
	Fluid and Particle Definitions > Water	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Material**	Water
	Domain Models > Pressure > **Reference Pressure**	1 [atm]
	Domain Models > Buoyancy Model > **Option**	Buoyant
	Domain Models > Buoyancy Model > **Gravity X Dirn.**	-9.81 [m s^-2]
	Domain Models > Buoyancy Model > **Gravity Y Dirn.**	0 [m s^-2]
	Domain Models > Buoyancy Model > **Gravity Z Dirn.**	0 [m s^-2]
	Domain Models > Buoyancy Model > **Buoy. Ref. Density**	997 [kg m^-3] [a]
	Domain Models > **Domain Motion > Option**	Rotating
	Domain Models > Domain Motion > **Angular Velocity**	84 [rev min ^-1] [b]
	Domain Models > Domain Motion > Axis Definition > **Rotation Axis**	Global X

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Fluid Models	Multiphase > **Homogeneous Model**	(Cleared) [c]
	Multiphase > **Free Surface Model > Option**	None
	Heat Transfer > **Homogeneous Model**	(Cleared)
	Heat Transfer > **Option**	Isothermal
	Heat Transfer > **Fluid Temperature**	25 [C]
	Turbulence > **Homogeneous Model**	(Cleared)
	Turbulence > **Option**	Fluid Dependent
Fluid Pair Models	**Fluid Pair**	Air | Water
	Fluid Pair > Air | Water > **Surface Tension Coefficient**	(Selected)
	Fluid Pair > Air | Water > Surface Tension Coefficient > **Surf. Tension Coeff.**	0.073 [N m^-1] [d]
	Fluid Pair > Air | Water > Momentum Transfer > Drag Force > **Option**	Grace
	Fluid Pair	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Air | Water > Momentum Transfer > Drag Force > **Volume Fraction Correction Exponent**	(Selected)
	Fluid Pair > Air | Water > Momentum Transfer > Drag Force > Volume Fraction Correction Exponent > **Value**	4 [e]
	Fluid Pair > Air | Water > Momentum Transfer > Non-drag forces > Turbulent Dispersion Force > **Option**	Favre Averaged Drag Force
	Fluid Pair > Air | Water > Momentum Transfer > Non-drag forces > Turbulent Dispersion Force > **Dispersion Coeff.**	1
	Fluid Pair > Air | Water > Turbulence Transfer > **Option**	Sato Enhanced Eddy

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
		Viscosity [f]

a. This buoyancy reference density value should always be used for dilute dispersed multiphasic flow.

b. Ensure that you are using the correct unit.

c. Disabling the homogeneous model allows you to specify a unique velocity field for each fluid.

d. This enables the Grace drag model.

e. A positive value is appropriate for large bubbles.

f. This models particle-induced turbulence.

6. Click **OK**.

17.4.2.2. Stationary Domain for the Main Tank

Next, you will create a stationary domain for the main tank by copying the properties of the existing impeller domain.

1. Right-click `impeller` and select **Duplicate** from the shortcut menu.
2. Rename the duplicated domain to `tank` and then open it for editing.
3. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Location and Type > **Location**	Primitive 3D
	Domain Models > Domain Motion > **Option**	Stationary

4. Click **OK**.

17.4.3. Creating the Boundaries

The following boundary conditions will be set:

- An inlet through which air enters the mixer.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- A degassing outlet, so that only the gas phase can leave the domain.
- Thin surfaces for the baffle.
- A wall for the hub and the portion of the shaft that is in the rotating domain. This wall will be rotating, and therefore stationary relative to the rotating domain.
- A wall for the portion of the shaft in the stationary domain. This wall will be rotating relative to the stationary domain.

When the default wall boundary is generated, the internal 2D regions of an imported mesh are ignored, while the regions that form domain boundaries are included.

Note:

The blade surfaces of the impeller will be modeled using domain interfaces later in the tutorial.

17.4.3.1. Air Inlet Boundary

1. Create a new boundary in the domain `tank` named `Airin`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Inlet
	Location	INLET_DIPTUBE
Boundary Details	Mass And Momentum > **Option**	 Fluid Dependent
Fluid Values	**Boundary Conditions**	Air
	Boundary Conditions > Air > Velocity > **Option**	 Normal Speed
	Boundary Conditions > Air > Velocity > **Normal Speed**	 5 [m s^-1]
	Boundary Conditions > Air > Volume Fraction	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Option**	Value
	Boundary Conditions > Air > Volume Fraction > **Volume Fraction**	1
	Boundary Conditions	Water
	Boundary Conditions > Water > Velocity > **Option**	Normal Speed
	Boundary Conditions > Water > Velocity > **Normal Speed**	5 [m s^-1]
	Boundary Conditions > Water > Volume Fraction > **Option**	Value
	Boundary Conditions > Water > Volume Fraction > **Volume Fraction**	0

3. Click **OK**.

17.4.3.2. Degassing Outlet Boundary

Create a degassing outlet to represent the free surface where air bubbles escape. The continuous phase (water) sees this boundary as a free-slip wall and does not leave the domain. The dispersed phase (air) sees this boundary as an outlet.

1. Create a new boundary in the domain `tank` named `LiquidSurface`.

2. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Basic Settings	**Boundary Type**	Outlet
	Location	WALL_LIQUID_SURFACE
Boundary Details	Mass And Momentum > **Option**	Degassing Condition

3. Click **OK**.

Note that no pressure is specified for this boundary. The solver will compute a pressure distribution on this fixed-position boundary to represent the surface height variations that would occur in the real flow.

17.4.3.3. Thin Surface for the Baffle

In CFX-Pre, thin surfaces can be created by specifying wall boundary conditions on both sides of internal 2D regions. Both sides of the baffle regions will be specified as walls in this case.

1. Create a new boundary in the domain `tank` named `Baffle`.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Wall
	Location	WALL_BAFFLES [a]
Boundary Details	Mass And Momentum > **Option**	Fluid Dependent
	Wall Contact Model > **Option**	Use Volume Fraction
Fluid Values	**Boundary Conditions**	Air
	Boundary Conditions > Air > Mass and Momentum > **Option**	Free Slip Wall [b]
	Boundary Conditions	Water
	Boundary Conditions > Water > Mass and Momentum	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Option**	No Slip Wall

a. The WALL_BAFFLES region includes the surfaces on both sides of the baffle. You can confirm this by examining WALL_BAFFLES in the region selector.

b. The `Free Slip Wall` condition can be used for the gas phase since the contact area with the walls is near zero for low gas phase volume fractions.

3. Click **OK**.

17.4.3.4. Wall Boundary for the Shaft

You will now set up a boundary for the portions of the shaft that are in the tank domain. Since the tank domain is not rotating, you need to specify a moving wall on the shaft to account for the shaft's rotation.

Part of the shaft is located directly above the air inlet, so the volume fraction of air in this location will be high and the assumption of zero contact area for the gas phase is not physically correct. In this case, a no slip boundary is more appropriate than a free slip condition for the air phase. When the volume fraction of air in contact with a wall is low, a free slip condition is more appropriate for the air phase.

In cases where it is important to correctly model the dispersed phase slip properties at walls for all volume fractions, you can declare both fluids as no slip, but set up an expression for the dispersed phase wall area fraction. The expression should result in an area fraction of zero for dispersed phase volume fractions from 0 to 0.3, for example, and then linearly increase to an area fraction of 1 as the volume fraction increases to 1.

1. Create a new boundary in the domain `tank` named `TankShaft`.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Wall
	Location	WALL_SHAFT, WALL_SHAFT_CENTER
Boundary Details	Mass and Momentum > **Option**	Fluid Dependent
	Wall Contact Model > **Option**	Use Volume Fraction
Fluid Values	**Boundary Conditions**	Air
	Boundary Conditions	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Air > Mass And Momentum > **Option**	No Slip Wall
	Boundary Conditions > Air > Mass And Momentum > **Wall Velocity**	(Selected)
	Boundary Conditions > Air > Mass And Momentum > **Wall Velocity > Option**	Rotating Wall
	Boundary Conditions > Air > Mass And Momentum > Wall Velocity > **Angular Velocity**	84 [rev min ^-1]
	Boundary Conditions > Air > Mass And Momentum > Wall Velocity > Axis Definition > **Option**	Coordinate Axis
	Boundary Conditions > Air > Mass And Momentum > Wall Velocity > Axis Definition > **Rotation Axis**	Global X

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Select `Water` and set the same values as for `Air`.

4. Click **OK**.

17.4.3.5. Required Boundary in the Impeller Domain

1. Create a new boundary in the domain `impeller` named `HubShaft`.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Wall
	Location	Hub,Shaft
Boundary Details	Mass And Momentum > **Option**	 Fluid Dependent
	Wall Contact Model > **Option**	 Use Volume Fraction
Fluid Values	**Boundary Conditions**	Air
	Boundary Conditions > Air > Mass And Momentum > **Option**	 Free Slip Wall
	Boundary Conditions	Water
	Boundary Conditions > Water > Mass and Momentum > **Option**	 No Slip Wall

3. Click **OK**.

17.4.3.6. Modifying the Default Wall Boundary

As mentioned previously, when the volume fraction of air in contact with a wall is low, a free slip condition is more appropriate for the air phase.

1. In the tree view, open `tank Default` for editing.

2. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Boundary Details	Mass and Momentum > **Option**	 Fluid Dependent
	Wall Contact Model > **Option**	 Use Volume Fraction
Fluid Values	**Boundary Conditions**	Air
	Boundary Conditions > Air > Mass And Momentum > **Option**	 Free Slip Wall
	Boundary Conditions	Water
	Boundary Conditions > Water > Mass And Momentum > **Option**	 No Slip Wall

3. Click **OK**.

   It is not necessary to set the default boundary in the impeller domain since the remaining surfaces will be assigned interface conditions in the next section.

17.4.4. Creating the Domain Interfaces

The following interfaces will be set:

- Blade thin surface interface.
- Rotational periodic domain interfaces for the periodic faces of the tank and impeller.
- Frozen Rotor interfaces between the impeller and tank domains.

17.4.4.1. Modeling the Blade Using a Domain Interface

You can model thin surfaces using either wall boundaries or domain interfaces. There are some differences between domain interfaces and ordinary wall boundaries; for example, CFX-Pre automatically detects the matching domain boundary regions when setting up a domain interface.

Previously, the thin surface representation of the tank baffle was modeled using boundary conditions. For demonstrational purposes, you will use a domain interface to model the thin surface representation of the impeller blade (even though using wall boundary conditions would also work).

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Create a new domain interface named `Blade Thin Surface`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Interface Type**	Fluid Fluid
	Interface Side 1 > **Domain (filter)**	impeller
	Interface Side 1 > **Region List**	Blade
	Interface Side 2 > **Domain (filter)**	impeller
	Interface Side 2 > **Region List**	Solid 3.3 2, Solid 3.6 2
Additional Interface Models	Mass And Momentum > **Option**	Side Dependent [a]
a. This option allows you to set a fluid dependent treatment on each side of the interface.		

3. Click **OK**.

   Two boundaries named `Blade Thin Surface Side 1` and `Blade Thin Surface Side 2` are created automatically.

4. In the tree view, open `Blade Thin Surface Side 1` for editing.
5. Configure the following setting(s):

Tab	Setting	Value
Boundary Details	Mass and Momentum > **Option**	Fluid Dependent
	Wall Contact Model > **Option**	Use Volume Fraction
Fluid Values	**Boundary Conditions**	Air
	Boundary Conditions > Air	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Mass And Momentum > **Option**	 Free Slip Wall
	Boundary Conditions	Water
	Boundary Conditions > Water > Mass And Momentum > **Option**	 No Slip Wall

6. Click **OK**.
7. In the tree view, open `Blade Thin Surface Side 2` for editing.
8. Apply the same settings as for `Blade Thin Surface Side 1`.
9. Click **OK**.

17.4.4.2. Rotational Periodic Interfaces

Periodic domain interfaces can either be one-to-one or GGI interfaces. One-to-one transformations occur for topologically similar meshes whose nodes match within a given tolerance. One-to-one periodic interfaces are more accurate and reduce CPU and memory requirements. Here, you will choose the `Automatic` mesh connection method, to let Ansys CFX choose between one-to-one and GGI.

1. Create a new domain interface named `ImpellerPeriodic`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Interface Type**	Fluid Fluid
	Interface Side 1 > **Domain (Filter)**	 impeller
	Interface Side 1 > **Region List**	 Periodic1
	Interface Side 2 > **Domain (Filter)**	 impeller
	Interface Side 2 > **Region List**	 Periodic2
	Interface Models	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Option**	Rotational Periodicity
	Interface Models > Axis Definition > **Option**	Coordinate Axis
	Interface Models > Axis Definition > **Rotation Axis**	Global X
Mesh Connection	Mesh Connection Method > Mesh Connection > **Option**	Automatic

3. Click **OK**.

1. Create a new domain interface named `TankPeriodic`.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Interface Type**	Fluid Fluid
	Interface Side 1 > **Domain (Filter)**	tank
	Interface Side 1 > **Region List**	BLKBDY_TANK_PER1
	Interface Side 2 > **Domain (Filter)**	tank
	Interface Side 2 > **Region List**	BLKBDY_TANK_PER2
	Interface Models > **Option**	Rotational Periodicity
	Interface Models > Axis Definition > **Option**	Coordinate Axis
	Interface Models	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Axis Definition > **Rotation Axis**	 Global X
Mesh Connection	Mesh Connection Method > Mesh Connection > **Option**	 Automatic

3. Click **OK**.

17.4.4.3. Frozen Rotor Interfaces

You will now create three Frozen Rotor interfaces for the regions connecting the two domains. In this case three separate interfaces are created. You should not try to create a single domain interface for multiple surfaces that lie in different planes.

1. Create a new domain interface named `Top`.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Interface Type**	Fluid Fluid
	Interface Side 1 > **Domain (Filter)**	 impeller
	Interface Side 1 > **Region List**	 Top
	Interface Side 2 > **Domain (Filter)**	 tank
	Interface Side 2 > **Region List**	 BLKBDY_TANK_TOP
	Interface Models > **Option**	 General Connection
	Interface Models > Frame Change/Mixing Model > **Option**	 Frozen Rotor

3. Click **OK**.

4. Create a new domain interface named `Bottom`.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Interface Type**	Fluid Fluid
	Interface Side 1 > **Domain (Filter)**	impeller
	Interface Side 1 > **Region List**	Bottom
	Interface Side 2 > **Domain (Filter)**	tank
	Interface Side 2 > **Region List**	BLKBDY_TANK_BOT
	Interface Models > **Option**	General Connection
	Interface Models > Frame Change/Mixing Model > **Option**	Frozen Rotor

6. Click **OK**.

7. Create a new domain interface named `Outer`.

8. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Interface Type**	Fluid Fluid
	Interface Side 1 > **Domain (Filter)**	impeller
	Interface Side 1 > **Region List**	Outer
	Interface Side 2 > **Domain (Filter)**	tank
	Interface Side 2 > **Region List**	BLKBDY_TANK_OUTER
	Interface Models > **Option**	General Connection

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Interface Models > Frame Change/Mixing Model > **Option**	 Frozen Rotor

9. Click **OK**.

17.4.5. Setting Initial Values

You will set the initial volume fraction of air to 0, and enable the initial volume fraction of water to be computed automatically. Since the volume fractions must sum to unity, the initial volume fraction of water will be 1.

It is important to understand how the velocity is initialized in this tutorial. Here, both fluids use `Automatic` for the **Cartesian Velocity Components** option. When the `Automatic` option is used, the initial velocity field will be based on the velocity values set at inlets, openings, and outlets. In this tutorial, the only boundary that has a set velocity value is the inlet, which specifies a velocity of 5 [m s^-1] for both phases. Without setting the **Velocity Scale** parameter, the resulting initial guess would be a uniform velocity of 5 [m s^-1] in the X direction throughout the domains for both phases. This is clearly not suitable since the water phase is enclosed by the tank. When the boundary velocity conditions are not representative of the expected domain velocities, the **Velocity Scale** parameter should be used to set a representative domain velocity. In this case the velocity scale for water is set to zero, causing the initial velocity for the water to be zero. The velocity scale is not set for air, resulting in an initial velocity of 5 [m s^-1] in the X direction for the air. This should not be a problem since the initial volume fraction of the air is zero everywhere.

1. Click *Global Initialization* .

2. Configure the following setting(s):

Tab	Setting	Value
Fluid Settings	**Fluid Specific Initialization**	Air
	Fluid Specific Initialization > Air > Initial Conditions > Volume Fraction > **Option**	 Automatic with Value
	Fluid Specific Initialization > Air > Initial Conditions	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Volume Fraction > **Volume Fraction**	 0
	Fluid Specific Initialization	Water
	Fluid Specific Initialization > Water > Initial Conditions > Cartesian Velocity Components > **Option**	 Automatic
	Fluid Specific Initialization > Water > Initial Conditions > Cartesian Velocity Components > **Velocity Scale**	 (Selected)
	Fluid Specific Initialization > Water > Initial Conditions > Cartesian Velocity Components > Velocity Scale > **Value**	 0 [m s^-1]

3. Click **OK**.

17.4.6. Setting Solver Control

Generally, two different time scales exist for multiphase mixers. The first is a small time scale based on the rotational speed of the impeller, typically taken as $1 / \omega$, resulting in a time scale of 0.11 s for this case. The second time scale is usually larger and based on the recirculation time of the continuous phase in the mixer.

Using a time step based on the rotational speed of the impeller will be more robust, but convergence will be slow since it takes time for the flow field in the mixer to develop. Using a larger time step reduces the number of iterations required for the mixer flow field to develop, but reduces robustness. You will need to experiment to find an optimum time step.

1. Click *Solver Control* .

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Advection Scheme > **Option**	High Resolution
	Convergence Control > **Max. Iterations**	100
	Convergence Control > Fluid Timescale Control > **Timescale Control**	Physical Timescale
	Convergence Control > Fluid Timescale Control > **Physical Timescale**	1 [s] [a]
	Convergence Criteria	(Default) [b]
Advanced Options	**Global Dynamic Model Control**	(Selected)
	Multiphase Control	(Selected)
	Multiphase Control > **Volume Fraction Coupling**	(Selected)
	Multiphase Control > Volume Fraction Coupling > **Option**	Coupled

a. This is an aggressive time step for this case.

b. The default is an RMS value of 1.0E-04. If you are using a maximum edge length of 0.005 m or less to produce a finer mesh, use a target residual of 1.0E-05 to obtain a more accurate solution.

3. Click **OK**.

17.4.7. Adding Monitor Points

You can monitor the value of an expression during the solver run so that you can view the volume fraction of air in the tank (the gas hold up). The gas hold up is often used to judge convergence in these types of simulations by converging until a steady-state value is achieved.

1. Create the following expressions:

```
TankAirHoldUp = volumeAve(Air.vf)@tank
ImpellerAirHoldUp = volumeAve(Air.vf)@impeller
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
TotalAirHoldUp = (volume()@tank * TankAirHoldUp +
                  volume()@impeller * ImpellerAirHoldUp) /
                  (volume()@tank + volume()@impeller)
```

2. Click *Output Control* .
3. Configure the following setting(s):

Tab	Setting	Value
Monitor	**Monitor Objects**	(Selected)

4. Create a new **Monitor Points and Expressions** item named `Total Air Holdup`.
5. Configure the following setting(s) of `Total Air Holdup`:

Setting	Value
Option	Expression
Expression Value	TotalAirHoldUp

6. Click **OK**.

17.4.8. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .
2. Configure the following setting(s):

Setting	Value
File name	MultiphaseMixer.def

3. Click **Save**.

   If you are notified the file already exists, click **Overwrite**.
4. Click **OK**.
5. If using stand-alone mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

17.5. Obtaining the Solution Using CFX-Solver Manager

Start the simulation from CFX-Solver Manager:

1. Ensure that the **Define Run** dialog box is displayed.
2. Click **Start Run**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CFX-Solver runs and attempts to obtain a solution. This can take a long time depending on your system.

3. Select the check box next to **Post-Process Results** when the completion message appears at the end of the run.

4. If using stand-alone mode, select the check box next to **Shut down CFX-Solver Manager**.

5. Click **OK**.

17.6. Viewing the Results Using CFD-Post

When CFD-Post starts, the **Domain Selector** dialog box might appear. If it does, ensure that both the impeller and tank domains are selected, then click **OK** to load the results from these domains. When the mixer geometry appears in the viewer, orient the view so that the X axis points up as follows:

- Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (X up)**.

You will create some plots showing the distributions of velocity and other variables. You will also calculate the torque and power required to turn the impeller at 84 rpm.

17.6.1. Creating a Plane Locator

Create a vertical plane that extends from the shaft to the tank wall at a location far from the baffle. This plane will be used as a locator for various plots, such as velocity vector plots and plots showing the distribution of air.

1. Create a new plane named `Plane 1`.

2. Configure the following setting(s):

Tab	Setting	Value
Geometry	Definition > **Method**	Three Points
	Definition > **Point 1**	1, 0, 0
	Definition > **Point 2**	0, 1, -0.9
	Definition > **Point 3**	0, 0, 0

3. Click **Apply**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

17.6.2. Plotting Velocity

Recall that the homogeneous multiphase option was not used when specifying the domain settings (see the setting for **Fluid Models** > **Multiphase Options** > **Homogeneous Model** in Rotating Domain for the Impeller (p. 395)). As a consequence, the air and water velocity fields may differ from each other. Plot the velocity of water, then air on `Plane 1`:

1. Create a new vector plot named `Vector 1`.
2. Configure the following setting(s):

Tab	Setting	Value
Geometry	Definition > **Locations**	 Plane 1
	Variable	Water.Velocity in Stn Frame [a]
Symbol	**Symbol Size**	0.2
	Normalize Symbols	(Selected)

a. Using this variable, instead of Water.Velocity, results in the velocity vectors appearing to be continuous at the interface between the rotating and stationary domains. Velocity variables that do not include a frame specification always use the local reference frame.

3. Click **Apply**.
4. Turn off the visibility of `Plane 1` to better see the vector plot.
5. Observe the vector plot (in particular, near the top of the tank). Note that the water is not flowing out of the domain.
6. Change the variable to `Air.Velocity in Stn Frame` and click **Apply**.

   Observe this vector plot, noting how the air moves upward all the way to the water surface, where it escapes.
7. Turn off the visibility of `Vector 1` in preparation for the next plots.

17.6.3. Plotting Pressure Distribution

Color `Plane 1` to see the pressure distribution:

1. Turn on the visibility of `Plane 1`.
2. Configure the following setting(s) of `Plane 1`:

Tab	Setting	Value
Color	**Mode**	Variable

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Variable	Pressure
	Range	Local

3. Click **Apply**.

   Note that the pressure field computed by the solver excludes the hydrostatic pressure corresponding to the specified buoyancy reference density. The pressure field including this hydrostatic component (as well as the reference pressure) can by visualized by plotting `Absolute Pressure`.

17.6.4. Plotting Volume Fractions

To see the distribution of air, color `Plane 1` by the volume fraction of air:

1. Configure the following setting(s) of `Plane 1`:

Tab	Setting	Value
Color	**Mode**	Variable
	Variable	Air.Volume Fraction
	Range	User Specified
	Min	0
	Max	0.04

2. Click **Apply**.

The user-specified range was made much narrower than the **Global** and **Local** ranges in order to better show the variation.

17.6.5. Plotting Shear Strain Rate and Shear Stress

Areas of high shear strain rate or shear stress are typically also areas where the highest mixing occurs.

To see where the most of the mixing occurs, color `Plane 1` by shear strain rate.

1. Configure the following setting(s) of `Plane 1`:

Tab	Setting	Value
Color	**Variable**	Air.Shear Strain Rate
	Range	User Specified
	Min	0 [s^-1]
	Max	15 [s^-1]

2. Click **Apply**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The user-specified range was made much narrower than the **Global** and **Local** ranges in order to better show the variation.

3. Modify the coloring of the `MultiphaseMixer_001` > `tank` > `tank Default` object by applying the following settings:

Tab	Setting	Value
Color	**Mode**	Variable
	Variable	Water.Wall Shear
	Range	Local

4. Click **Apply**.

The legend for this plot shows the range of wall shear values.

The global maximum wall shear stress is much higher than the maximum value on the default walls. The global maximum values occur on the `TankShaft` boundary directly above the inlet. Although these values are very high, the shear force exerted on this boundary is small since the contact area fraction of water is very small there.

17.6.6. Calculating Torque and Power Requirements

Calculate the torque and power required to spin the impeller at 84 rpm:

1. Select **Tools** > **Function Calculator** from the main menu or click *Function Calculator* .

2. Configure the following setting(s):

Tab	Setting	Value
Function Calculator	**Function**	torque
	Location	Blade Thin Surface Side 1
	Axis	Global X
	Fluid	All Fluids

3. Click **Calculate** to find the torque about the X axis imparted by both fluids on location `Blade Thin Surface Side 1`.

4. Repeat the calculation, setting **Location** to `Blade Thin Surface Side 2`.

The sum of these two torques is approximately -72 [N m] about the X axis. Multiplying by -4 to find the torque required by all of the impeller blades gives a required torque of approximately 288 [N m] about the X axis. You could also include the contributions from the locations `HubShaft` and `Tank Shaft`; however in this case their contributions are negligible.

The power requirement is simply the required torque multiplied by the rotational speed (84 rpm = 8.8 rad/s): Power = 288 N m * 8.8 rad/s = 2534.4 W.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Remember that this value is the power requirement for the work done on the fluids; it does not account for any mechanical losses, motor efficiencies and so on. Also note that the accuracy of these results is significantly affected by the coarseness of the mesh. You should not use a mesh of this length scale to obtain accurate quantitative results.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 18: Gas-Liquid Flow in an Airlift Reactor

This tutorial includes:

18.1. Tutorial Features

18.2. Overview of the Problem to Solve

18.3. Preparing the Working Directory

18.4. Defining the Case Using CFX-Pre

18.5. Obtaining the Solution Using CFX-Solver Manager

18.6. Viewing the Results Using CFD-Post

18.7. Further Discussion

18.1. Tutorial Features

In this tutorial you will learn about:

- Setting up a multiphase flow simulation involving air and water.
- Using a fluid dependent turbulence model to set different turbulence options for each fluid.
- Specifying buoyant flow.
- Specifying a degassing outlet boundary to enable air, but not water, to escape from the boundary.
- Using face culling in CFD-Post to turn off the visibility of one side of a surface.

Component	Feature	Details
CFX-Pre	User Mode	General mode
	Analysis Type	Steady State
	Fluid Type	General Fluid
	Domain Type	Single Domain
	Turbulence Model	Dispersed Phase Zero Equation
		Fluid-Dependent Turbulence Model
		k-Epsilon
	Heat Transfer	None
	Buoyant Flow	
	Multiphase	
	Boundary Conditions	Inlet (Subsonic)

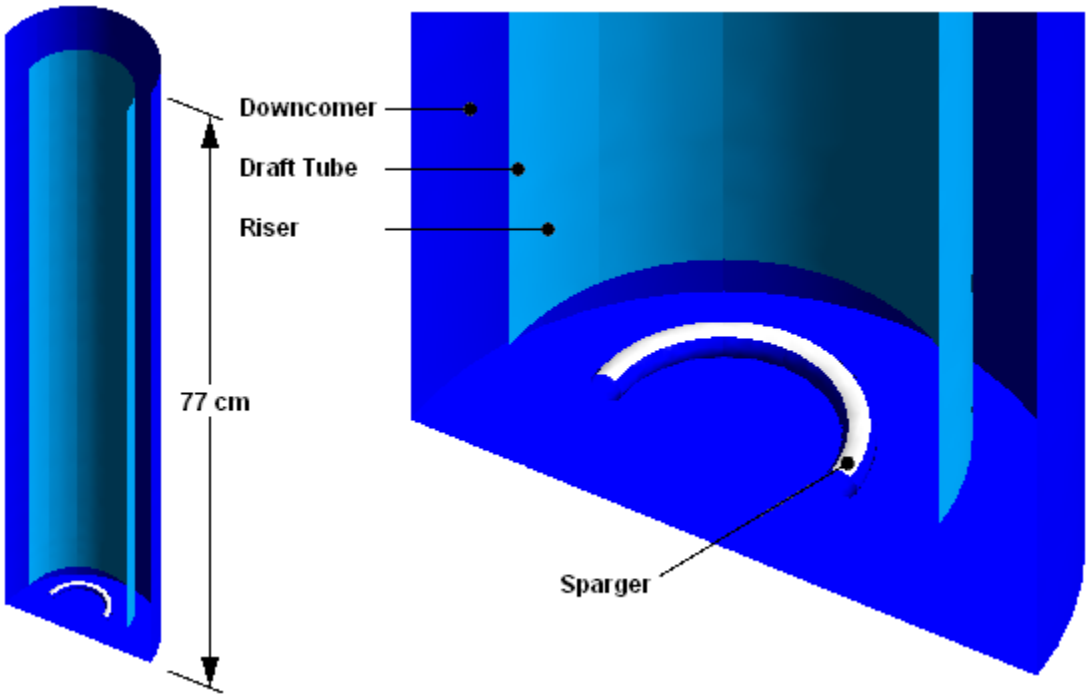
Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Component	Feature	Details
		Outlet (Degassing)
		Symmetry Plane
		Wall: (Slip Depends on Volume Fraction)
	Timestep	Physical Time Scale
CFD-Post	Plots	Default Locators
		Vector
	Other	Changing the Color Range
		Symmetry

18.2. Overview of the Problem to Solve

This tutorial demonstrates the Eulerian-Eulerian multiphase model in CFX by simulating an airlift reactor. Airlift reactors are tall gas-liquid contacting vessels and are often used in processes where gas absorption is important (for example, bioreactors to dissolve oxygen in broths) and to limit the exposure of microorganisms to excessive shear imparted by mechanically driven mixers.

Figure 18.1: Cut-away Diagram of the Airlift Reactor



This tutorial models the dispersion of air bubbles in water. Air is supplied through a sparger at the bottom of the vessel and the rising action of the bubbles provides gentle agitation of the water. An internal tube (draft tube) directs recirculation of the flow. The airlift reactor is shown in a cut-away diagram in Figure 18.1: Cut-away Diagram of the Airlift Reactor (p. 422).

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Simple airlift reactors that are without a draft tube tend to develop irregular flow patterns and poor overall mixing. The draft tube in the airlift reactor helps to establish a regular flow pattern in the column and to achieve better uniformity in temperature, concentration, and pH in the liquid phase, but sometimes at the expense of decreased mass transfer from gas to liquid.

This tutorial also demonstrates the use of pairs of internal wall boundaries to model thin 3D features. In this case, a pair of wall boundaries is used to model the draft tube. Other applications include baffles and guide vanes. In the postprocessing section of this tutorial, you will learn how to use face culling to hide one side of a boundary. This technique enables you to independently color each boundary of a pair of back-to-back boundaries (located at the same position in 3D space, but with opposite orientation).

The airlift reactor that is modeled here is very similar to the laboratory bench scale prototype used by García-Calvo and Letón.

A formal analysis of this simulation involving a finer mesh is available at the end of this tutorial. For details, see Further Discussion (p. 441).

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- Running Ansys CFX Tutorials Using Ansys Workbench (p. 34)
- Changing the Display Colors (p. 37)

18.3. Preparing the Working Directory

1. Create a working directory.

   Ansys CFX uses a working directory as the default location for loading and saving files for a particular session or project.

2. Download the `bubble_column.zip` file here.

3. Unzip `bubble_column.zip` to your working directory.

   Ensure that the following tutorial input file is in your working directory:

   - `BubbleColumnMesh.gtm`

4. Set the working directory and start CFX-Pre.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

18.4. Defining the Case Using CFX-Pre

1. In CFX-Pre, select **File** > **New Case**.

2. Select **General** and click **OK**.

3. Select **File** > **Save Case As**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Under **File name**, type `BubbleColumn`.
5. Click **Save**.

18.4.1. Importing the Mesh

1. Right-click `Mesh` and select **Import Mesh** > **CFX Mesh**.

   The **Import Mesh** dialog box appears.

2. Configure the following setting(s):

Setting	Value
File name	BubbleColumnMesh.gtm

3. Click **Open**.

18.4.2. Creating the Domain

1. Edit `Case Options` > `General` in the **Outline** tree view and ensure that **Automatic Default Domain** is turned on. A domain named `Default Domain` should now appear under the `Simulation` branch.
2. Double-click **Default Domain**.
3. In the **Basic Settings** tab, under **Fluid and Particle Definitions**, delete `Fluid 1` and create a new fluid definition called `Air`.
4. Use the button to create a new fluid named `Water`.
5. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Location and Type > **Location**	 B1.P3, B2.P3
	Fluid and Particle Definitions	Air
	Fluid and Particle Definitions > Air > **Material**	 Air at 25 C
	Fluid and Particle Definitions > Air > Morphology	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Option**	Dispersed Fluid
	Fluid and Particle Definitions > Air > Morphology > **Mean Diameter**	6 [mm]
	Fluid and Particle Definitions	Water
	Fluid and Particle Definitions > Water > **Material**	Water
	Domain Models > Pressure > **Reference Pressure**	1 [atm]
	Domain Models > Buoyancy Model > **Option**	Buoyant
	Domain Models > Buoyancy Model > **Gravity X Dirn.**	0 [m s^-2]
	Domain Models > Buoyancy Model > **Gravity Y Dirn.**	-9.81 [m s^-2]
	Domain Models > Buoyancy Model > **Gravity Z Dirn.**	0 [m s^-2]
	Domain Models > Buoyancy Model > **Buoy. Ref. Density**	997 [kg m^-3] [a]
Fluid Models	Multiphase	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Homogeneous Model**	(Cleared) [b]
	Multiphase > **Free Surface Model > Option**	None
	Heat Transfer > **Homogeneous Model**	(Cleared)
	Heat Transfer > **Option**	Isothermal
	Heat Transfer > **Fluid Temperature**	25 [C]
	Turbulence > **Homogeneous Model**	(Cleared)
	Turbulence > **Option**	Fluid Dependent [c]
Fluid Pair Models	**Fluid Pair**	Air | Water
	Fluid Pair > Air | Water > **Surface Tension Coefficient**	(Selected)
	Fluid Pair > Air | Water > Surface Tension Coefficient > **Surf. Tension Coeff.**	0.072 [N m^-1] [d]
	Fluid Pair > Air | Water > Momentum Transfer > Drag Force > **Option**	Grace
	Fluid Pair > Air | Water	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Momentum Transfer > Drag Force > **Volume Fraction Correction Exponent**	(Selected)
	Fluid Pair > Air | Water > Momentum Transfer > Drag Force > Volume Fraction Correction Exponent > **Value**	2 [e]
	Fluid Pair > Air | Water > Momentum Transfer > Non-drag forces > Turbulent Dispersion Force > **Option**	Favre Averaged Drag Force
	Fluid Pair > Air | Water > Momentum Transfer > Non-drag forces > Turbulent Dispersion Force > **Dispersion Coeff.**	1.0
	Fluid Pair > Air | Water > Turbulence Transfer > **Option**	Sato Enhanced Eddy Viscosity [f]

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
a. This buoyancy reference density value should always be used for dilute dispersed multiphasic flow. b. Disabling the homogeneous model allows you to specify a unique velocity field for each fluid. c. The fluid-specific turbulence settings are defined in the **Fluid Specific Models** tab. They are set to default values. d. This enables the Grace drag model. e. A positive value is appropriate for large bubbles. f. This models particle-induced turbulence.		

6. Click **OK**.

18.4.3. Creating the Boundary Conditions

For this simulation of the airlift reactor, the required boundary conditions are:

- An inlet for air on the sparger.
- A degassing outlet for air at the liquid surface.
- A pair of wall boundaries for the draft tube.
- An exterior wall for the outer wall, base and sparger tube.
- Symmetry planes.

18.4.3.1. Inlet Boundary

At the sparger, create an inlet boundary that injects air at 0.3 m/s with a volume fraction of 0.25:

1. Create a new boundary named `Sparger`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Inlet
	Location	Sparger
Boundary Details	Mass And Momentum > **Option**	Fluid Dependent
Fluid Values	**Boundary Conditions**	Air
	Boundary Conditions	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Air > Velocity > **Option**	Normal Speed
	Boundary Conditions > Air > Velocity > **Normal Speed**	0.3 [m s^-1]
	Boundary Conditions > Air > Volume Fraction > **Option**	Value
	Boundary Conditions > Air > Volume Fraction > **Volume Fraction**	0.25
	Boundary Conditions	Water
	Boundary Conditions > Water > Velocity > **Option**	Normal Speed
	Boundary Conditions > Water > Velocity > **Normal Speed**	0 [m s^-1]
	Boundary Conditions > Water > Volume Fraction > **Option**	Value
	Boundary Conditions > Water	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Volume Fraction > **Volume Fraction**	 0.75

3. Click **OK**.

18.4.3.2. Outlet Boundary

Create a degassing outlet boundary at the top of the reactor:

1. Create a new boundary named `Top`.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Outlet
	Location	Top
Boundary Details	Mass And Momentum > **Option**	 Degassing Condition

3. Click **OK**.

18.4.3.3. Draft Tube Boundaries

The draft tube is an infinitely thin surface that requires a wall boundary on both sides; if only one side has a boundary then CFX-Solver will fail.

The `Free Slip` condition can be used for the gas phase since the contact area with the walls is near zero for low gas phase volume fractions.

The required boundary settings are the same for both sides of the draft tube. From the point of view of solving the simulation, you could therefore define a single boundary and choose both sides of the tube as the location. However, the postprocessing section of this tutorial requires the use of separate boundaries in order to illustrate the use of face culling (a visualization technique), so you will create two wall boundaries instead.

Start by creating a wall boundary for the outer side of the draft tube:

1. Create a new boundary named `DraftTube Downcomer Side`.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Wall
	Location	DraftTube

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Boundary Details	Mass And Momentum > **Option**	 Fluid Dependent
	Wall Roughness > **Option**	 Smooth Wall
	Wall Contact Model > **Option**	 Use Volume Fraction
Fluid Values	**Boundary Conditions**	Air
	Boundary Conditions > Air > Mass And Momentum > **Option**	 Free Slip Wall
	Boundary Conditions	Water
	Boundary Conditions > Water > Mass And Momentum > **Option**	 No Slip Wall

3. Click **OK**.

Now create a boundary named `DraftTube Riser Side` using the same settings, but located on `F10.B1.P3` (the riser side of the draft tube).

18.4.3.4. Symmetry Plane Boundary

To simulate the full geometry, create symmetry plane boundary conditions on the `Symmetry1` and `Symmetry2` locators:

1. Create a new boundary named `SymP1`.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Symmetry
	Location	Symmetry1

3. Click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Create a new boundary named SymP2.

5. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Symmetry
	Location	Symmetry2

6. Click **OK**.

18.4.3.5. Modifying the Default Boundary

The remaining external regions are assigned to the default wall boundary. As for the draft tube boundary, set the air phase to use the free slip wall condition:

1. Edit Default Domain Default.

2. Configure the following setting(s):

Tab	Setting	Value
Boundary Details	Mass And Momentum > **Option**	 Fluid Dependent
Fluid Values	**Boundary Conditions**	Air
	Boundary Conditions > Air > Mass And Momentum > **Option**	 Free Slip Wall
	Boundary Conditions	Water
	Boundary Conditions > Water > Mass And Momentum > **Option**	 No Slip Wall

3. Click **OK**.

The boundary specifications are now complete.

18.4.4. Setting Initial Values

It often helps to set an initial velocity for a dispersed phase that is different to that of the continuous phase. This results in a nonzero drag between the phases that can help stability at the start of a simulation.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For some airlift reactor problems, improved convergence can be obtained by using CEL (CFX Expression Language) to specify a nonzero volume fraction for air in the riser portion and a value of zero in the downcomer portion. This should be done if two solutions are possible (for example, if the flow could go up the downcomer and down the riser).

Set the initial values:

1. Click *Global Initialization* .

   Since a single pressure field exists for a multiphase calculation, do not set pressure values on a per-fluid basis.

2. Configure the following setting(s):

Tab	Setting	Value
Fluid Settings	**Fluid Specific Initialization**	Air
	Fluid Specific Initialization > Air > Initial Conditions > Cartesian Velocity Components > **Option**	Automatic with Value
	Fluid Specific Initialization > Air > Initial Conditions > Cartesian Velocity Components > **U**	0 [m s^-1]
	Fluid Specific Initialization > Air > Initial Conditions > Cartesian Velocity Components > **V**	0.3 [m s^-1]
	Fluid Specific Initialization > Air > Initial Conditions > Cartesian Velocity Components	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **W**	0 [m s^-1]
	Fluid Specific Initialization > Air > Initial Conditions > Volume Fraction > **Option**	Automatic
	Fluid Specific Initialization	Water [a]
	Fluid Specific Initialization > Water > Initial Conditions > Cartesian Velocity Components > **Option**	Automatic with Value
	Fluid Specific Initialization > Water > Initial Conditions > Cartesian Velocity Components > **U**	0 [m s^-1]
	Fluid Specific Initialization > Water > Initial Conditions > Cartesian Velocity Components > **V**	0 [m s^-1]
	Fluid Specific Initialization > Water > Initial Conditions > Cartesian Velocity Components > **W**	0 [m s^-1]
	Fluid Specific Initialization > Water	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Initial Conditions > Volume Fraction > **Option**	Automatic with Value
	Fluid Specific Initialization > Water > Initial Conditions > Volume Fraction > **Volume Fraction**	1 [b]

a. Since no water enters or leaves the domain, a stationary initial guess is recommended.

b. The volume fractions must sum to unity over all fluids. Since a value has been set for water, the volume fraction of air will be calculated as the remaining difference, in this case, 0.

3. Click **OK**.

18.4.5. Setting Solver Control

1. Click *Solver Control* .

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Advection Scheme > **Option**	High Resolution
	Convergence Control > **Max. Iterations**	200
	Convergence Control > Fluid Timescale Control > **Timescale Control**	Physical Timescale
	Convergence Control > Fluid Timescale Control > **Physical Timescale**	1 [s]

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Convergence Criteria > **Residual Type**	MAX
	Convergence Criteria > **Residual Target**	1.0E-05 [a]
	Convergence Criteria > **Conservation Target**	(Selected)
	Convergence Criteria > Conservation Target > **Value**	0.01
Advanced Options	Global Dynamic Model Control > **Multiphase Control**	(Selected)
	Global Dynamic Model Control > Multiphase Control > **Volume Fraction Coupling**	(Selected)
	Global Dynamic Model Control > Multiphase Control > Volume Fraction Coupling > **Option**	Coupled [b]

a. If you are using a maximum edge length of `0.005` m or less to produce a finer mesh, a target residual of 1.0E-05 helps obtain a more accurate solution.

b. Volume Fraction Coupling is recommended when changes in volume fraction have a large effect on momentum transport. This is the case in this tutorial, because it employs a model for the turbulent dispersion in which the force is proportional to the volume fraction gradient. Other cases where Volume Fraction Coupling is recommended include those in which solid pressure forces are used and those in which gravitational forces are significant (for example, free surface flows).

3. Click **OK**.

18.4.6. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Configure the following setting(s):

Setting	Value
File name	BubbleColumn.def

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, **Solver Input File** is set.

4. If using stand-alone mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

18.5. Obtaining the Solution Using CFX-Solver Manager

Start the simulation from CFX-Solver Manager:

Note:

If you are using a fine mesh for a formal quantitative analysis of the flow in the reactor, the solution time will be significantly longer than for the coarse mesh. You can run the simulation in parallel to reduce the solution time. For details, see Obtaining a Solution in Parallel (p. 196).

1. Ensure that the **Define Run** dialog box is displayed.
2. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. This can take a long time depending on your system.

3. Select the check box next to **Post-Process Results** when the completion message appears at the end of the run.
4. If using stand-alone mode, select the check box next to **Shut down CFX-Solver Manager**.
5. Click **OK**.

18.6. Viewing the Results Using CFD-Post

You will first create plots of velocity and volume fraction. You will then display the entire geometry.

Because the simulation in this tutorial is conducted on a coarse grid, the results are only suitable for a qualitative demonstration of the multiphase capability of Ansys CFX.

The following topics will be discussed:

18.6.1. Creating Water Velocity Vector Plots
18.6.2. Creating Volume Fraction Plots
18.6.3. Displaying the Entire Airlift Reactor Geometry

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

18.6.1. Creating Water Velocity Vector Plots

1. Right-click a blank area in the viewer and select **Predefined Camera** > **View From -Z**.
2. Create a new vector plot named `Vector 1`.
3. Configure the following setting(s):

Tab	Setting	Value
Geometry	Definition > **Locations**	SymP1
	Definition > **Variable**	Water.Velocity
Color	**Range**	User Specified
	Min	0 [m s^-1]
	Max	1 [m s^-1]
Symbol	**Symbol Size**	0.3

4. Click **Apply**.
5. In the tree view, right-click `Vector 1`, select **Duplicate**, and click **OK** to accept the default name, `Vector 2`.
6. Edit `Vector 2`.
7. On the **Geometry** tab, set **Definition** > **Variable** to `Air.Velocity` and click **Apply**.
8. Compare `Vector 1` and `Vector 2` by toggling the visibility of each one.

   Zoom in as required.

   Observe that the air flows upward, leaving the tank, at the degassing outlet.

Note that the air rises faster than the water in the riser and descends slower than the water in the downcomer.

18.6.2. Creating Volume Fraction Plots

Plot the volume fraction of air on the symmetry plane `SymP1`:

1. Right-click a blank area in the viewer and select **Predefined Camera** > **View From -Z**.
2. Turn on the visibility of `SymP1`.
3. Edit `SymP1`.
4. Configure the following setting(s) of `SymP1`:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Color	**Mode**	Variable
	Variable	Air.Volume Fraction
	Range	User Specified
	Min	0
	Max	0.025

5. Click **Apply**.

   Observe the volume fraction values throughout the domain.

6. Turn off the visibility of SymP1.

Next, plot the volume fraction of air on each side of the draft tube:

1. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Y up)**.
2. Turn on the visibility of DraftTube Downcomer Side.
3. Modify DraftTube Downcomer Side by applying the following settings:

Tab	Setting	Value
Color	**Mode**	Variable
	Variable	Air.Volume Fraction
	Range	User Specified
	Min	0
	Max	0.025

4. Click **Apply**.
5. Rotate the plot in the viewer to see both sides of DraftTube Downcomer Side.

   Notice that the plot appears on both sides of the DraftTube Downcomer Side boundary. When viewing plots on internal surfaces, you must ensure that you are viewing the correct side. You will make use of the face culling rendering feature to turn off the visibility of the plot on the side of the boundary for which the plot does not apply.

   The DraftTube Downcomer Side boundary represents the side of the internal surface in the downcomer (in this case, outer) region of the reactor. To confirm this, you could make a vector plot of the variable Normal (representing the face normal vectors) on the locator DraftTube Downcomer Side; the fluid is on the side opposite the normal vectors.

   In this case, you need to turn off the "front" faces of the plot; The front faces are, by definition, on the same side of the plot as the normal vectors.

6. Modify DraftTube Downcomer Side by applying the following settings:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Render	Show Faces > **Face Culling**	 Front Faces

7. Click **Apply**.

8. Rotate the image in the viewer to see the effect of face culling on `DraftTube Downcomer Side`. You should see that the color appears on the downcomer side only.

9. Turn on the visibility of `DraftTube Riser Side`.

10. Color the `DraftTube Riser Side` object using the same color and rendering settings as for `DraftTube Downcomer Side`.

    The normal vectors for `DraftTube Riser Side` point opposite to those of `DraftTube Downcomer Side`, so the faces on `DraftTube Riser Side` are plotted only on the riser (in this case, inner) side of the airlift reactor.

Rotating the geometry will now correctly show the air volume fraction on each side of the draft tube.

Face culling was needed to prevent interference between the plots on each side of the draft tube. To demonstrate this, try turning off face culling for `DraftTube Downcomer Side` and watch the effect on the riser side. You might notice that the plot from the downcomer side interferes with, or even completely overrides, the riser-side plot. Results may vary, which is why face culling should always be used to prevent possible interference.

18.6.3. Displaying the Entire Airlift Reactor Geometry

Display the entire airlift reactor geometry by expanding `User Locations and Plots` and double-clicking the `Default Transform` object:

1. Configure the following setting(s) of `Default Transform`:

Tab	Setting	Value
Definition	**Instancing Info From Domain**	(Cleared)
	Number of Graphical Instances	12
	Apply Rotation	(Selected)
	Apply Rotation > **Method**	 Principal Axis
	Apply Rotation > **Axis**	 Y
	Apply Rotation > **Number of Passages**	 12

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click **Apply**.

18.7. Further Discussion

A formal analysis of this airlift reactor was carried out on a finer grid (having 21000+ nodes and a maximum edge length of 0.005 m).

The analysis showed a region of air bubble recirculation at the top of the reactor on the downcomer side. This was confirmed by zooming in on a vector plot of Air.Velocity on SymP1 near the top of the downcomer. A similar plot of Water.Velocity revealed no recirculation of the water.

Other results of the simulation:

- Due to their large 0.006 m diameter, the air bubbles quickly attained a significant terminal slip velocity (that is, the terminal velocity relative to water). The resulting terminal slip velocity, obtained using the Grace drag model, is consistent with the prediction by Maneri and Mendelson and the prediction by Baker and Chao. These correlations predict a terminal slip velocity of about 0.23 m s^{-1} to 0.25 m s^{-1} for air bubbles of the diameter specified.
- The values of gas hold-up (the average volume fraction of air in the riser), the superficial gas velocity (the rising velocity, relative to the reactor vessel, of gas bubbles in the riser, multiplied by the gas hold-up), and the liquid velocity in the downcomer agree with the results reported by García-Calvo and Letón, for gas hold-up values of 0.03 or less. At higher values of gas hold-up, the multifluid model does not account for pressure-volume work transferred from gas to liquid due to isothermal expansion of the bubbles. The simulation therefore tends to under-predict both the superficial gas velocity in the riser, and the liquid velocity in the downcomer for gas hold-up values greater than 0.03.

> **Note:**
>
> Multiphase results files contain the vector variable Fluid.Superficial Velocity defined as Fluid.Volume Fraction multiplied by Fluid.Velocity. This is sometimes also referred to as the fluid volume flux. The components of this vector variable are available as scalar variables (for example, Fluid.Superficial Velocity X).

Many reference texts on airlift reactors cite the Hughmark correlation as a standard for gas hold-up and superficial gas velocity in airlift reactors. However, the Hughmark correlation should not be used when liquid flow is concurrent with gas at velocities exceeding 0.1 m/s. In the airlift reactor described in this tutorial, the liquid velocity in the riser clearly exceeds 0.2 m/s, and the Hughmark correlation is therefore not applicable.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 19: Air Conditioning Simulation

This tutorial includes:

19.1. Tutorial Features

19.2. Overview of the Problem to Solve

19.3. Preparing the Working Directory

19.4. Defining the Case Using CFX-Pre

19.5. Obtaining the Solution Using CFX-Solver Manager

19.6. Viewing the Results Using CFD-Post

19.7. Further Discussion

Important:

You must have the required Fortran compiler installed and set in your system path in order to run this tutorial. For details on which Fortran compiler is required for your platform, see the applicable Ansys, Inc. installation guide. If you are not sure which Fortran compiler is installed on your system, try running the `cfx5mkext` command (found in `<CFXROOT>/bin`) from the command line and read the output messages.

19.1. Tutorial Features

In this tutorial you will learn about:

- Importing CEL expressions.
- @REGION CEL syntax.
- Setting up a user CEL function.
- Setting up a monitor point to observe the temperature at a prescribed location.
- Setting up an interface with conditional connection control.
- Using the Monte Carlo radiation model with a directional source of radiation.
- Postprocessing a transient simulation.

Component	Feature	Details
CFX-Pre	User Mode	General mode
	Analysis Type	Transient
	Fluid Type	General Fluid

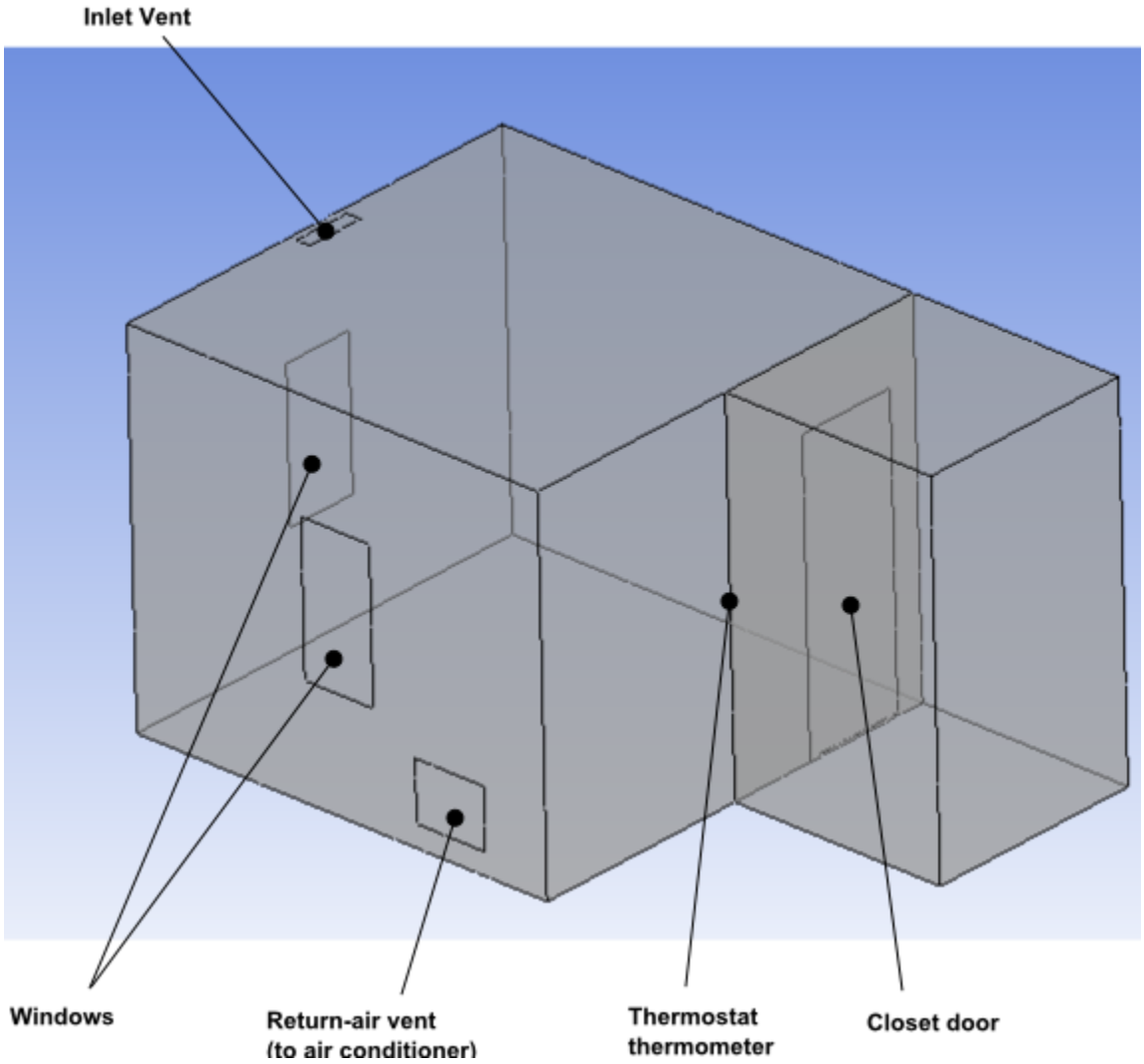
Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Component	Feature	Details
	Domain Type	Single Domain
	Turbulence Model	k-Epsilon
	Heat Transfer	Thermal Energy
	Radiation	Monte Carlo
	Buoyant Flow	
	Boundary Conditions	Boundary Profile Visualization
		Inlet (Profile)
		Outlet (Subsonic)
		Wall: No-Slip
		Wall: Adiabatic
		Wall: Fixed Temperature
	Domain Interface	Conditional GGI
	Output Control	Transient Results Files
		Monitor Points
	CEL (CFX Expression Language)	User CEL Function
CFD-Post	Plots	Animation
		Isosurface
		Point
		Slice Plane
	Other	Text Label with Auto Annotation
		Changing the Color Range
		Legend
		Time Step Selection
		Transient Animation with Movie Generation

19.2. Overview of the Problem to Solve

This tutorial simulates a room with a thermostat-controlled air conditioner adjacent to a closet. The closet door is initially closed and opens during the simulation.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The thermostat switches the air conditioner on and off based on the following data:

- A set point of 22°C (the temperature at or above which the air conditioner turns on)
- A temperature tolerance of 1°C (the amount by which cooling continues below the set point before the air conditioner turns off)
- The temperature at a wall-mounted thermometer

Air flows in steadily from an inlet vent on the ceiling, and flows out through a return-air vent near the floor. When the air conditioner is turned on, the incoming air temperature is reduced compared to the outgoing air temperature. When the air conditioner is turned off, the incoming air temperature is set equal to the outgoing air temperature.

Two windows enable sunlight to enter and heat the room. The walls (except for the closet door and part of the wall surrounding the closet door) and windows are assumed to be at a constant 26 °C. The simulation is transient, and continues long enough to enable the air conditioner to cycle on and off.

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Running Ansys CFX Tutorials Using Ansys Workbench (p. 34)
- Changing the Display Colors (p. 37)

19.3. Preparing the Working Directory

1. Create a working directory.

   Ansys CFX uses a working directory as the default location for loading and saving files for a particular session or project.

2. Download the `hvac.zip` file here.

3. Unzip `hvac.zip` to your working directory.

   Ensure that the following tutorial input files are in your working directory:

   - `HVAC_expressions.ccl`
   - `HVACMesh.gtm`
   - `TStat_Control.F`

   Note:

   You must have a Fortran compiler installed on your system to perform this tutorial.

4. Set the working directory and start CFX-Pre.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

19.4. Defining the Case Using CFX-Pre

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type HVAC.
5. Click **Save**.

19.4.1. Importing the Mesh

1. Right-click `Mesh` and select **Import Mesh** > **CFX Mesh**.

   The **Import Mesh** dialog box appears.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Configure the following setting(s):

Setting	Value
File name	HVACMesh.gtm

3. Click **Open**.

4. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)** from the shortcut menu.

19.4.2. Importing CEL Expressions

This tutorial uses several CEL expressions to store parameters and to evaluate other quantities that are required by the simulation. Import all of the expressions from the provided file:

1. Select **File** > **Import** > **CCL**.

2. Ensure that **Import Method** is set to **Append**.

3. Select HVAC_`expressions.ccl`, which should be in your working directory.

4. Click **Open**.

The table below lists the expressions, along with the definition and information for each expression:

Expression Name	Expression Definition	Information
ACOn	Thermostat Function(TSensor,TSet,TTol,atstep)	On/off status of the air conditioner (determined by calling a user CEL function with the thermometer temperature, thermostat set point, and temperature tolerance).
Cool TempCalc	TVentOut - (HeatRemoved / (MassFlow * 1.004 [kJ kg^-1 K^-1]))	Temperature of air at the return-air vent (determined by a CEL function).
Flowrate	0.06 [m^3 s^-1]	Volumetric flow rate of air entering the room.
HeatRemoved	1000 [J s^-1]	Rate of thermal energy removal when the air conditioner is on.
MassFlow	1.185 [kg m^-3] * Flowrate	Mass flow rate of air entering the room.
TIn	ACOn*CoolTempCalc+(1-ACOn)*TVentOut	Temperature of inlet vent air (a function of the air conditioner on/off status, return-air vent temperature, thermal energy removal rate, and mass flow rate).
TSensor	probe(T)@Thermometer	Thermometer temperature (determined by a CEL function that gets temperature data from a monitor point).
TSet	22 [C]	Thermometer set point.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Expression Name	Expression Definition	Information
TTol	1 [K]	Temperature tolerance.
TVentOut	areaAve(T)@REGION:VentOut	Temperature of outlet vent air.
XCompInlet	5*(x-0.05 [m]) / 1 [m]	Direction vector components for guiding the inlet vent air in a diverging manner as it enters the room.
ZCompInlet	-1+XCompInlet	
tStep	3 [s]	Time step size.
tTotal	225 [s]	Total time.

The CEL function that evaluates the thermometer temperature relies on a monitor point that you will create later in this tutorial.

The CEL expression for the air conditioner on/off status requires a compiled Fortran subroutine and a user CEL function that uses the subroutine. These are created next, starting with the compiled subroutine.

Note:

The expression for the return-air vent temperature, `TVentOut`, makes use of @REGION CEL syntax, which indicates the mesh region named `VentOut`, rather than a boundary named `VentOut`.

19.4.3. Compiling the Fortran Subroutine for the Thermostat

A Fortran subroutine that simulates the thermostat is provided with the tutorial input files. Before the subroutine can be used, it must be compiled for your platform.

You can compile the subroutine at any time before running CFX-Solver. The operation is performed at this point in the tutorial so that you have a better understanding of the values you need to specify in CFX-Pre when creating a User CEL Function. The `cfx5mkext` command is used to compile the subroutine as described below.

1. Ensure that file `TStat_Control.F` is in your working directory.

   You can examine the contents of this file in any text editor to gain a better understanding of the subroutine it contains.

2. Select **Tools** > **Command Editor**.

3. Type the following command in the **Command Editor** dialog box (make sure you do not miss the semicolon at the end of the line):

   ```
   ! system ("cfx5mkext TStat_Control.F") == 0 or die "cfx5mkext failed";
   ```

   - This is equivalent to executing the following at a command prompt:

     ```
     cfx5mkext TStat_Control.F
     ```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The `!` indicates that the following line is to be interpreted as power syntax and not CCL. Everything after the `!` symbol is processed as Perl commands.
- `system` is a Perl function to run a system command.
- The "`== 0 or die`" will cause an error message to be returned if, for some reason, there is an error in processing the command.

4. Click **Process** to compile the subroutine.

Note:

You can use the `-double` option to compile the subroutine for use with double precision CFX-Solver executables. That is:

```
cfx5mkext -double TStat_Control.F
```

A subdirectory will have been created in your working directory whose name is system dependent (for example, on Linux it is named `linux`). This subdirectory contains the shared object library.

Note:

If you are running problems in parallel over multiple platforms then you will need to create these subdirectories using the `cfx5mkext` command for each different platform.

- You can view more details about the `cfx5mkext` command by running:

```
cfx5mkext -help
```

- You can set a Library Name and Library Path using the `-name` and `-dest` options respectively.
- If these are not specified, the default Library Name is that of your Fortran file and the default Library Path is your current working directory.
- Close the **Command Editor** dialog box.

19.4.4. Creating a User CEL Function for the Thermostat

The expression for the air conditioner on/off status is named `ACOn`. This expression consists of a call to a user CEL function, `Thermostat Function`, which must be created.

Before you create the user CEL function, you will first create a user routine that holds basic information about the compiled Fortran subroutine. You will then create a user function, `Thermostat Function`, so that it is associated with the user routine.

Create the user routine:

1. From the main menu, select **Insert** > **Expressions, Functions and Variables** > **User Routine** or click *User Routine* .

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Set the name to `Thermostat Routine`.
3. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Option**	User CEL Function
	Calling Name	ac_on [a]
	Library Name	TStat_Control [b]
	Library Path	(Working Directory) [c]

a. This is the name of the subroutine within the Fortran file. Always use lower case letters for the calling name, even if the subroutine name in the Fortran file is in upper case.

b. This is the name passed to the `cfx5mkext` command by the -name option. If the -name option is not specified, a default is used. The default is the Fortran file name without the `.F` extension.

c. Set this to your working directory.

4. Click **OK**.

Create the user CEL function:

1. From the main menu, select **Insert** > **Expressions, Functions and Variables** > **User Function** or click *User Function* .
2. Set the name to `Thermostat Function`.
3. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Option**	User Function
	User Routine Name	Thermostat Routine
	Argument Units	[K], [K], [K], [] [a]
	Result Units	[] [b]

a. These are the units for the four input arguments: TSensor, TSet, TTol, and atstep.

b. The result will be a dimensionless flag with a value of 0 or 1.

4. Click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

19.4.5. Setting the Analysis Type

This is a transient simulation. The total time and time step are specified by expressions that you imported earlier. Use these expressions to set up the `Analysis Type` information:

1. Right-click `Analysis Type` in the **Outline** tree view and select **Edit**.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Analysis Type > **Option**	Transient
	Analysis Type > Time Duration > **Option**	Total Time
	Analysis Type > Time Duration > **Total Time**	tTotal
	Analysis Type > Time Steps > **Option**	Timesteps
	Analysis Type > Time Steps > **Timesteps**	tStep
	Analysis Type > Initial Time > **Option**	Automatic with Value
	Analysis Type > Initial Time > **Time**	0 [s]

3. Click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

19.4.6. Creating the Domain

The domain that models the room air should model buoyancy, given the expected temperature differences, air speeds, and the size and geometry of the room. The domain must model radiation, since directional radiation (representing sunlight) will be emitted from the windows.

Create the domain:

1. Edit `Case Options` > `General` in the **Outline** tree view and ensure that **Automatic Default Domain** is turned on.

   A domain named `Default Domain` should appear under the `Simulation` branch.

2. Edit `Default Domain` and configure the following setting(s):

Tab	Setting	Value
Basic Settings	Location and Type > **Location**	B54,B64
	Fluid and Particle Definitions	Fluid 1
	Fluid and Particle Definitions > Fluid 1 > **Material**	Air Ideal Gas
	Domain Models > Pressure > **Reference Pressure**	1 [atm]
	Domain Models > Buoyancy Model > **Option**	Buoyant
	Domain Models > Buoyancy Model > **Gravity X Dirn.**	0 [m s^-2]
	Domain Models > Buoyancy Model > **Gravity Y Dirn.**	0 [m s^-2]
	Domain Models > Buoyancy Model > **Gravity Z Dirn.**	-g

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Domain Models > Buoyancy Model > **Buoy. Ref. Density**	 1.2 [kg m^-3]
Fluid Models	Heat Transfer > **Option**	 Thermal Energy
	Thermal Radiation > **Option**	 Monte Carlo
	Thermal Radiation > **Number of Histories**	 100000

3. Click **OK**.

Note:

When using a statistical radiation model like Monte Carlo, the results can usually be improved (at the expense of increased computational time) by increasing the number of histories.

19.4.7. Creating the Boundaries

In this section you will define several boundaries:

- An inlet vent that injects air into the room.
- A return-air vent that lets air leave the room.
- Fixed-temperature windows that emit directed radiation.
- Fixed-temperature walls.

19.4.7.1. Inlet Boundary

Create a boundary for the inlet vent, using the previously-loaded expressions for mass flow rate, flow direction, and temperature:

1. Create a boundary named `Inlet`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Inlet
	Location	Inlet

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Boundary Details	Mass and Momentum > **Option**	Mass Flow Rate
	Mass and Momentum > **Mass Flow Rate**	MassFlow
	Flow Direction > **Option**	Cartesian Components
	Flow Direction > **X Component**	XCompInlet
	Flow Direction > **Y Component**	0
	Flow Direction > **Z Component**	ZCompInlet
	Heat Transfer > **Option**	Static Temperature
	Heat Transfer > **Static Temperature**	TIn
Plot Options	**Boundary Vector**	(Selected)
	Boundary Vector > **Profile Vec. Comps.**	Cartesian Components

3. Click **OK**.

The viewer shows the inlet velocity profile applied at the inlet, which uses the expressions `XCompInlet` and `ZCompInlet` to specify a diverging flow pattern.

Note:

Ignore the physics errors that appear. They will be fixed by setting up the rest of the simulation. The error concerning the expression `TIn` is due to a reference to `Thermometer`, which does not yet exist. A monitor point named `Thermometer` will be created later as part of the output control settings.

19.4.7.2. Outlet Boundary

Create a boundary for the return-air vent, specifying a relative pressure of 0 Pa:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Create a boundary named `VentOut`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Outlet
	Location	VentOut
Boundary Details	Mass and Momentum > **Option**	Average Static Pressure
	Mass and Momentum > **Relative Pressure**	0 [Pa]

3. Click **OK**.

19.4.7.3. Window Boundary

To model sunlight entering the room, the windows are required to emit directional radiation. To approximate the effect of the outdoor air temperature, assume that the windows have a fixed temperature of 26°C.

Create a boundary for the windows using a fixed temperature of 26°C and apply a radiation source of 600 W m^-2 in the (1, 1, -1) direction:

1. Create a boundary named `Windows`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Wall
	Location	Window1,Window2
Boundary Details	Heat Transfer > **Option**	Temperature
	Heat Transfer > **Fixed Temperature**	26 [C]
Sources	**Boundary Source**	(Selected)
	Boundary Source > **Sources**	(Selected)

3. Create a new radiation source item by clicking *Add new item* and accepting the default name.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Configure the following setting(s) of `Radiation Source 1`:

Setting	Value
Option	Directional Radiation Flux
Radiation Flux	600 [W m^-2]
Direction > **Option**	Cartesian Components
Direction > **X Component**	1
Direction > **Y Component**	1
Direction > **Z Component**	-1

5. Configure the following setting(s):

Tab	Setting	Value
Plot Options	**Boundary Vector**	(Selected)
	Boundary Vector > **Profile Vec. Comps.**	Cartesian Components in Radiation Source 1

6. Click **OK**.

   The direction of the radiation is shown in the viewer.

19.4.7.4. Default Wall Boundary

The default boundary for any undefined surface in CFX-Pre is a no-slip, smooth, adiabatic wall. For this simulation, assume that the walls have a fixed temperature of 26°C. A more detailed simulation would model heat transfer through the walls.

For radiation purposes, assume that the default wall is a perfectly absorbing and emitting surface (emissivity = 1).

Set up the default wall boundary:

1. Edit the boundary named `Default Domain Default`.

2. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Boundary Details	Heat Transfer > **Option**	Temperature
	Heat Transfer > **Fixed Temperature**	26 [C]
	Thermal Radiation > **Option**	Opaque
	Thermal Radiation > **Emissivity**	1

3. Click **OK**.

Because this boundary is opaque with an emissivity of 1, all of the radiation is absorbed and none of the radiation is reflected. With no reflected radiation, the **Diffuse Fraction** setting has no effect. For lower values of emissivity, some radiation is reflected, and the **Diffuse Fraction** setting controls the fraction of the reflected radiation that is diffuse, with the remainder being specular (directional).

19.4.8. Closet Wall Interface

1. Click *Domain Interface* , set **Name** to `ClosetWall`, and click **OK**.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Interface Side 1 > **Region List**	F51.54
	Interface Side 2 > **Region List**	F51.64
Additional Interface Models	Mass And Momentum > **Option**	No Slip Wall
	Heat Transfer	(Selected)
	Heat Transfer > **Option**	Conservative Interface Flux
	Heat Transfer > Interface Model > **Option**	Thin Material

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Heat Transfer > Interface Model > **Material**	Building Board Hardwood [a]
	Heat Transfer > Interface Model > **Thickness**	12 [cm]
	Thermal Radiation	(Selected)
	Thermal Radiation > **Option**	Opaque
	Thermal Radiation > **Emissivity**	1

a. Click the *Ellipsis* icon to open the **Material** dialog box. Click *Import Library Data*. In the **Select Library Data to Import** dialog box, select `Building Board Hardwood` under the `CHT Solids` branch and click **OK**. In the **Material** dialog box, select `Building Board Hardwood` under the `CHT Solids` branch and click **OK**.

3. Click **OK**.

19.4.9. Creating a Logical Expression for the Conditional GGI Interface

Create an expression named `tOpen`:

1. In the **Outline** tree view, expand **Expressions, Functions and Variables**, right-click **Expressions** and select **Insert** > **Expression**
2. Give the new expression the name: `tOpen`
3. In the **Definition** area, type `t>150[s]`.
4. Click **Apply**.

This expression will be used to indicate the point at which the closet door is to open during the simulation. In the next section, you will set up the door interface, and indicate that the interface should change from closed to open when the expression `tOpen` evaluates to `1[]` (true).

19.4.10. Creating a Conditional GGI Interface for the Closet Door

Create the interface:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click *Domain Interface* , set **Name** to `ClosetDoor`, and click **OK**.

2. Configure the following settings:

Tab	Setting	Value
Basic Settings	Interface Side 1 > **Region List**	 F53.54
	Interface Side 2 > **Region List**	 F53.64
Additional Interface Models	Mass And Momentum > **Option**	 Conservative Interface Flux
	Conditional Connection Control	(Selected)
	Conditional Connection Control > **Option**	 Irreversible State Change
	Conditional Connection Control > **Logical Expression**	 tOpen
	Conditional Connection Control > **Initial State**	 Closed

3. Click **OK**.

Next, set the properties of the closet door that will apply when the door is closed:

1. Edit the boundary named `ClosetDoor Side 1`.

2. Configure the following setting(s):

Tab	Setting	Value
Nonoverlap Conditions	**Nonoverlap Conditions**	(Selected)
	Nonoverlap Conditions > **Boundary Type**	 Wall
	Nonoverlap Conditions > Mass and Momentum > **Option**	 No Slip Wall

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Nonoverlap Conditions > Heat Transfer > **Option**	Adiabatic
	Nonoverlap Conditions > Thermal Radiation > **Option**	Opaque
	Nonoverlap Conditions > Thermal Radiation > **Emissivity**	1

3. Click **OK**.
4. Do the same for `ClosetDoor Side 2`.

19.4.11. Creating Space Under the Closet Door

Create an interface under the closet door so that air can pass freely under the door. This connection will prevent the air in the closet from becoming isolated, so that the pressure level in the closet will be constrained.

1. Click *Domain Interface* , set **Name** to `DoorSpace`, and click **OK**.
2. Configure the following settings:

Tab	Setting	Value
Basic Settings	Interface Side 1 > **Region List**	F52.54
	Interface Side 2 > **Region List**	F52.64

3. Click **OK**.

19.4.12. Setting Initial Values

1. Click *Global Initialization* .
2. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Global Settings	Initial Conditions > **Velocity Type**	Cartesian
	Initial Conditions > Cartesian Velocity Components > **Option**	Automatic with Value
	Initial Conditions > Cartesian Velocity Components > **U**	0 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > **V**	0 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > **W**	0 [m s^-1]
	Initial Conditions > Static Pressure > **Option**	Automatic with Value
	Initial Conditions > Static Pressure > **Relative Pressure**	0 [Pa]
	Initial Conditions > Temperature > **Option**	Automatic with Value
	Initial Conditions > Temperature > **Temperature**	21.8 [C]
	Initial Conditions	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Turbulence > **Option**	Intensity and Length Scale
	Initial Conditions > Turbulence > Fractional Intensity > **Option**	Automatic with Value
	Initial Conditions > Turbulence > Fractional Intensity > **Value**	0.05
	Initial Conditions > Turbulence > Eddy Length Scale > **Option**	Automatic with Value
	Initial Conditions > Turbulence > Eddy Length Scale > **Value**	0.25 [m]
	Initial Conditions > Radiation Intensity > **Option**	Automatic with Value
	Initial Conditions > Radiation Intensity > **Blackbody Temperature**	(Selected)
	Initial Conditions > Radiation Intensity > Blackbody Temperature	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Blackbody Temp.**	21.8 [C] [a]

a. The initial blackbody temperature of the air should be set to the initial temperature of the air.

3. Click **OK**.

19.4.13. Setting Solver Control

In a typical transient simulation, there should be sufficient coefficient loops per time step to achieve convergence. In order to reduce the time required to run this particular simulation, reduce the maximum number of coefficient loops per time step to 3:

1. Click *Solver Control* .

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Convergence Control > **Max. Coeff. Loops**	 3

3. Click **OK**.

19.4.14. Setting Output Control

Set up the solver to output transient results files that record pressure, radiation intensity, temperature, and velocity, on every time step:

1. Click *Output Control* .

2. Click the **Trn Results** tab.

3. In the **Transient Results** list box, click *Add new item* , set **Name** to `Transient Results 1`, and click **OK**.

4. Configure the following setting(s) of `Transient Results 1`:

Setting	Value
Option	Selected Variables
Output Variables List	Pressure, Radiation Intensity, Temperature, Velocity [a]

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
Output Variable Operators	(Selected)
Output Variable Operators > **Output Var. Operators**	 All [b]
Output Frequency > **Option**	 Every Timestep

a. Use the **Ctrl** key to select more than one variable.

b. This causes the gradients of the selected variables to be written to the transient results files, along with other information.

To create the thermostat thermometer, set up a monitor point at the thermometer location. Also set up monitors to track the expressions for the temperature at the inlet, the temperature at the outlet, and the on/off status of the air conditioner.

1. Configure the following setting(s):

Tab	Setting	Value
Monitor	**Monitor Objects**	(Selected)

2. Create a new **Monitor Points and Expressions** item named `Thermometer`.

3. Configure the following setting(s) of `Thermometer`:

Setting	Value
Output Variables List	Temperature
Cartesian Coordinates	2.95, 1.5, 1.25

4. Create a new **Monitor Points and Expressions** item named `Temp at Inlet`.

5. Configure the following setting(s) of `Temp at Inlet`:

Setting	Value
Option	Expression
Expression Value	TIn

6. Create a new **Monitor Points and Expressions** item named `Temp at VentOut`.

7. Configure the following setting(s) of `Temp at VentOut`:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
Option	Expression
Expression Value	TVentOut

8. Create a new **Monitor Points and Expressions** item named `ACOnStatus`.
9. Configure the following setting(s) of `ACOnStatus`:

Setting	Value
Option	Expression
Expression Value	ACOn

10. Click **OK**.

19.4.15. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .
2. Configure the following setting(s):

Setting	Value
File name	HVAC.def

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, **Solver Input File** is set.

4. If using stand-alone mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

19.5. Obtaining the Solution Using CFX-Solver Manager

When CFX-Pre has shut down and CFX-Solver Manager has started, start the solver and view the monitor points as the solution progresses:

1. Click **Start Run**.

   After a few minutes, a **User Points** tab will appear.

   On that tab, plots will appear showing the values of the monitor points:

   - `ACOnStatus`
   - `Temp at Inlet`
   - `Temp at VentOut`

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- `Thermometer (Temperature)`

2. Click one of the plot lines to see the value of the plot variable at that point.
3. It is difficult to see the plot values because all of the monitor points are plotted on the same scale. To see the plots in more detail, try displaying subsets of them as follows:
   a. Right-click in the plot area and select **Monitor Properties** from the shortcut menu.
   b. In the **Monitor Properties** dialog box, on the **Plot Lines** tab, expand the `USER POINT` branch in the tree.
   c. Clear the check boxes beside all of the monitor points except `ACOnStatus`, then click **Apply**.
   d. Observe the plot for `ACOnStatus`.

      You might have to move the dialog box out of the way to see the plot.
   e. In the **Monitor Properties** dialog box, toggle each of the check boxes beside the monitor points, so that all of the monitor points are selected except for `ACOnStatus`, then click **Apply**.
   f. Observe the plots for `Temp at Inlet`, `Temp at VentOut`, and `Thermometer (Temperature)`.
   g. Click **OK** to close the **Monitor Properties** dialog box.
4. Select the check box next to **Post-Process Results** when the completion message appears at the end of the run.
5. If using stand-alone mode, select the check box next to **Shut down CFX-Solver Manager**.
6. Click **OK**.

19.6. Viewing the Results Using CFD-Post

You will first create some graphic objects to visualize the temperature distribution and the thermometer location. You will then create an animation to show how the temperature distribution changes.

19.6.1. Creating Graphics Objects

In this section, you will create two planes and an isosurface of constant temperature, all colored by temperature. You will also create a color legend and a text label that reports the thermometer temperature.

19.6.1.1. Creating Planes

In order to show the key features of the temperature distribution, create two planes colored by temperature as follows:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Load the res file (HVAC_001.res) if you did not elect to load the results directly from CFX-Solver Manager.
2. Right-click a blank area in the viewer, select **Predefined Camera** > **Isometric View (Z up)**.
3. Create a ZX-Plane named Plane 1 with **Y**=2.25 [m]. Color it by Temperature using a user specified range from 19 [C] to 23 [C]. Turn off lighting (on the **Render** tab) so that the colors are accurate and can be interpreted properly using the legend.
4. Create an XY plane named Plane 2 with **Z**=0.35 [m]. Color it using the same settings as for the first plane, and turn off lighting.

19.6.1.2. Creating an Isosurface

In order to show the plumes of cool air from the inlet vent, create a surface of constant temperature as follows:

1. Click *Timestep Selector* . The **Timestep Selector** dialog box appears.
2. Double-click the value: 69 s.

   The time step is set to 69 s so that the cold air plume is visible.
3. Create an isosurface named Cold Plume as a surface of Temperature = 19 °C.
4. Color the isosurface by Temperature (select Use Plot Variable) and use the same color range as for the planes. Although the color of the isosurface will not show variation (by definition), it will be consistent with the coloration of the planes.
5. On the **Render** tab for the isosurface, set **Transparency** to 0.5. Leave lighting turned on to help show the 3D shape of the isosurface.
6. Click **Apply**.

> **Note:**
>
> The isosurface will not be visible in some time steps, but you will be able to see it when playing the animation (a step carried out later).

19.6.1.3. Adjusting the Legend

The legend title should not name the locator of any particular object since all objects are colored by the same variable and use the same range. Remove the locator name from the title and, in preparation for making an MPEG video later in this tutorial, increase the text size:

1. Edit Default Legend View 1.
2. On the **Definition** tab, change **Title Mode** to Variable.

   This will remove the locator name from the legend.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click the **Appearance** tab, then:

   a. Change **Precision** to 2, Fixed.

   b. Change **Text Height** to 0.03.

4. Click **Apply**.

19.6.1.4. Creating a Point for the Thermometer

In the next section, you will create a text label that displays the value of the expression TSensor, which represents the thermometer temperature. During the solver run, this expression was evaluated using a monitor point named Thermometer. Although this monitor point data is stored in the results file, it cannot be accessed. In order to support the expression for TSensor, create a point called Thermometer at the same location:

1. From the main menu, select **Insert** > **Location** > **Point**.
2. Set **Name** to Thermometer.
3. Set **Point** to (2.95,1.5,1.25).
4. Click the **Color** tab, then change **Color** to blue.
5. Click **Apply**.

   A marker appears at the thermometer location in the viewer. Note that you may have to rotate the results data to see the point.

19.6.1.5. Creating a Text Label

Create a text label that shows the currently-selected time step and thermometer temperature.

1. Click *Text* .
2. Accept the default name and click **OK**.
3. Configure the following setting(s):

Tab	Setting	Value
Definition	**Text String**	Time Elapsed:
	Embed Auto Annotation	(Selected) [a]
	Type	Time Value

a. The full text string should now be Time Elapsed: <aa>. The <aa> string represents the location where text is to be substituted.

4. Click **More** to add a second line of text to the text object.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Configure the following setting(s):

Tab	Setting	Value
Definition	**Text String** (the second one)	Sensor Temperature:
	Embed Auto Annotation	(Selected)
	Type	Expression
	Expression	TSensor
Appearance	**Height**	0.03

6. Click **Apply**.
7. Ensure that the visibility for `Text 1` is turned on.

The text label appears in the viewer, near the top.

19.6.2. Creating an Animation

1. Ensure that the view is set to **Isometric View (Z up)**.
2. Click *Timestep Selector* .
3. Double-click the first time value (0 s).
4. Click *Animation* .

   The **Animation** dialog box appears.
5. Set **Type** to **Keyframe Animation**.
6. Click *New* to create `KeyframeNo1`.
7. Select `KeyframeNo1`, then set **# of Frames** to `200`, then press **Enter** while in the **# of Frames** box.

   Tip:

   Be sure to press **Enter** and confirm that the new number appears in the list before continuing.

   This will place 200 intermediate frames between the first and (yet to be created) second key frames, for a total of 202 frames. This will produce an animation lasting about 8.4 s since the frame rate will be 24 frames per second. Since there are 76 unique frames, each frame will be shown at least once.
8. Use the **Timestep Selector** to load the last time value (225 s).

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9. In the **Animation** dialog box, click *New* to create `KeyframeNo2`.

   The **# of Frames** parameter has no effect for the last keyframe, so leave it at the default value.

10. Click *More Animation Options* to expand the **Animation** dialog box.

11. Select **Save Movie**.

12. Set **Format** to `MPEG1`.

13. Specify a filename for the movie.

14. Click the **Options** button.

15. Change **Image Size** to `720 x 480` (or a similar resolution).

16. Click the **Advanced** tab, and note the **Quality** setting. If your movie player cannot play the resulting MPEG, you can try using the **Low** or **Custom** quality settings.

17. Click **OK**.

18. Click *To Beginning* to rewind the active key frame to `KeyframeNo1`.

19. Click *Save animation state* and save the animation to a file. This will enable you to quickly restore the animation in case you want to make changes. Animations are not restored by loading ordinary state files (those with the .cst extension).

20. Click *Play the animation* .

21. If prompted to overwrite an existing movie, click **Overwrite**.

    The animation plays and builds an `mpg` file.

22. When you have finished, quit CFD-Post.

19.7. Further Discussion

- This tutorial uses an aggressive flow rate of air, a coarse mesh, large time steps, and a low cap on the maximum number of coefficient loops per time step. Running this tutorial with a flow rate of air that is closer to 5 changes of air per hour (0.03 $m^3 s^{-1}$), a finer mesh, smaller time steps, and a larger cap on the maximum number of coefficient loops, will produce more accurate results.
- Running this simulation for a longer total time will allow you to see more on/off cycles of the air conditioner.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 20: Combustion and Radiation in a Can Combustor

This tutorial includes:

20.1. Tutorial Features

20.2. Overview of the Problem to Solve

20.3. Preparing the Working Directory

20.4. Defining the Case Using CFX-Pre

20.5. Obtaining the Solution Using CFX-Solver Manager

20.6. Viewing the Results Using CFD-Post

20.1. Tutorial Features

In this tutorial you will learn about:

- Setting Up a Combustion Model in CFX-Pre.
- Using a Reacting Mixture.
- Using the Eddy Dissipation combustion model.
- Using the P1 radiation model.
- Creating thin surfaces for the inlet vanes.
- Using chemistry postprocessing.
- Changing object color maps in CFD-Post to prepare a grayscale image.
- Using the function calculator in CFD-Post.
- Creating a vector plot in CFD-Post.

Component	Feature	Details
CFX-Pre	User Mode	General mode
	Analysis Type	Steady State
	Fluid Type	Reacting Mixture
	Domain Type	Single Domain
	Turbulence Model	k-Epsilon
	Heat Transfer	Thermal Energy
	Combustion	

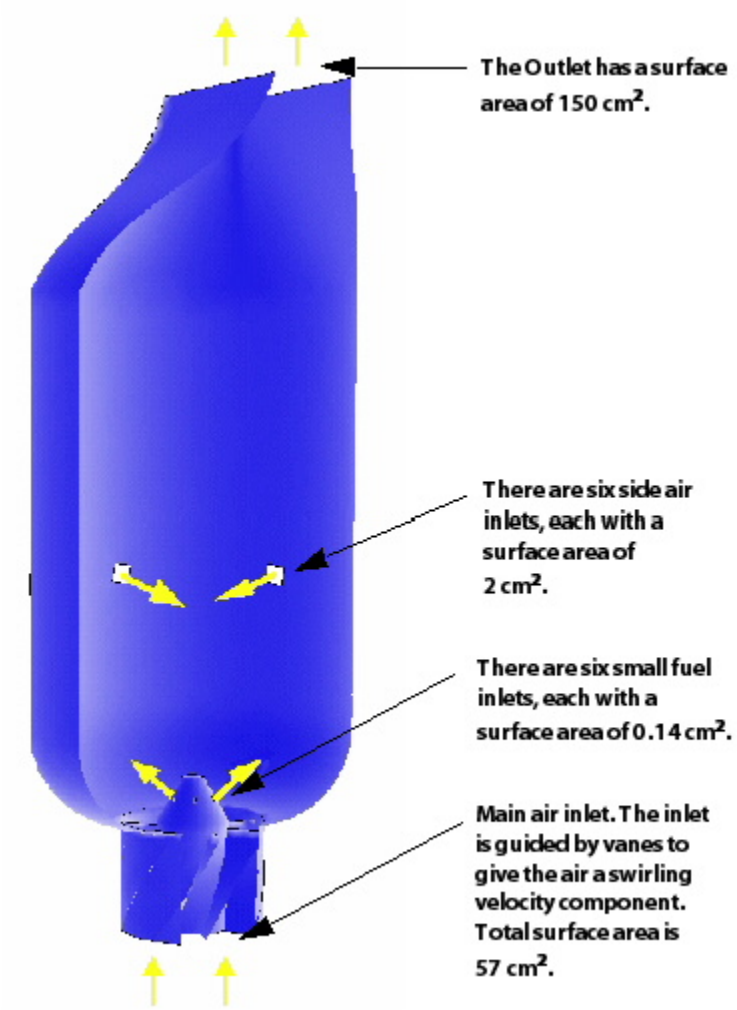
Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Component	Feature	Details
	Radiation	
	Boundary Conditions	Inlet (Subsonic)
		Outlet (Subsonic)
		Wall: No-Slip
		Wall: Adiabatic
		Wall: Thin Surface
	Timestep	Physical Time Scale
CFD-Post	Plots	Outline Plot (Wireframe)
		Sampling Plane
		Slice Plane
		Vector
	Other	Changing the Color Range
		Color map
		Legend
		Quantitative Calculation

20.2. Overview of the Problem to Solve

The can combustor is a feature of the gas turbine engine. Arranged around a central annulus, can combustors are designed to minimize emissions, burn very efficiently and keep wall temperatures as low as possible. This tutorial is designed to give a qualitative impression of the flow and temperature distributions inside a can combustor that burns methane in air. The basic geometry is shown below with a section of the outer wall cut away.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The simulation in this tutorial uses the Eddy Dissipation combustion model and the P1 radiation model.

Due to the fact that the fuel (methane) and oxidizer (air) undergo "fast" combustion (whereby the combustion rate is dominated by the rate of mixing of the materials), the Finite Rate Chemistry model is not a suitable combustion model for the combustor in this tutorial. The Combined EDM/FRC model capability is a superset of the Eddy Dissipation model capability, and has no benefit over the Eddy Dissipation model in this case. In fact, the convergence behavior of the Combined EDM/FRC model may be worse than that of the Eddy Dissipation model.

The Eddy Dissipation model is suitable for modeling "fast" combustion.

The Eddy Dissipation model tracks each individual chemical species (except for the constraint material) with its own transport equation. This model is flexible in that you can readily add new materials, such as additional fuels, to the simulation without complications. A limitation of this model is that radical or intermediate species, such as CO, cannot be calculated with adequate accuracy. This may lead to over-prediction of flame temperature, in particular in fuel-rich regions.

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- Running Ansys CFX Tutorials Using Ansys Workbench (p. 34)

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Changing the Display Colors (p. 37)

20.3. Preparing the Working Directory

1. Create a working directory.

   Ansys CFX uses a working directory as the default location for loading and saving files for a particular session or project.

2. Download the `combustor.zip` file here.

3. Unzip `combustor.zip` to your working directory.

   Ensure that the following tutorial input file is in your working directory:

   - `CombustorMesh.gtm`

4. Set the working directory and start CFX-Pre.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

20.4. Defining the Case Using CFX-Pre

You will first define a domain that includes a variable composition mixture. These mixtures are used to model combusting and reacting flows in CFX.

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `CombustorEDM`.
5. Click **Save**.
6. If prompted, click **Overwrite**.

20.4.1. Importing the Mesh

1. Right-click `Mesh` and select **Import Mesh** > **CFX Mesh**.

   The **Import Mesh** dialog box appears.

2. Configure the following setting(s):

Setting	Value
File name	CombustorMesh.gtm

3. Click **Open**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

20.4.2. Creating a Reacting Mixture

To enable combustion modeling, you must create a variable composition mixture.

20.4.2.1. To create the variable composition mixture

1. In the **Outline** tree, right-click **Materials** and select **Insert** > **Material**.
2. Set the name to `Methane Air Mixture` and click **OK**.
3. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Option**	Reacting Mixture
	Material Group	Gas Phase Combustion
	Reactions List	Methane Air WD1 NO PDF [a]
Mixture Properties	**Mixture Properties**	(Selected)
	Mixture Properties > Radiation Properties > **Refractive Index**	(Selected) [b]
	Mixture Properties > Radiation Properties > **Absorption Coefficient**	(Selected)
	Mixture Properties > Radiation Properties > **Scattering Coefficient**	(Selected)

a. The `Methane Air WD1 NO PD`Freaction specifies complete combustion of the fuel into its products in a single-step reaction. The formation of NO is also modeled and occurs in an additional reaction step. Click *Multi-select from extended list* to display the **Reactions List** dialog box, then click *Import Library Data* and select the appropriate reaction to import.

b. Setting the radiation properties explicitly will significantly shorten the solution time because the CFX-Solver will not have to calculate radiation mixture properties.

4. Click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

20.4.3. Creating the Domain

If `Default Domain` does not currently appear under **Flow Analysis 1** in the **Outline** tree:

Edit `Case Options` > `General` in the **Outline** tree view and ensure that **Automatic Default Domain** is turned on.

You now need to edit `Default Domain` so that it is representative of the Eddy Dissipation combustion and P 1 radiation models.

1. Edit `Default Domain` and configure the following setting(s):

Tab	Setting	Value
Basic Settings	Location and Type > **Location**	B152, B153, B154, B155, B156
	Fluid and Particle Definitions	Fluid 1
	Fluid and Particle Definitions > Fluid 1 > **Material**	Methane Air Mixture
	Domain Models > Pressure > **Reference Pressure**	1 [atm] [a]
Fluid Models	Heat Transfer > **Option**	Thermal Energy
	Combustion > **Option**	Eddy Dissipation
	Combustion > **Eddy Dissipation Model Coefficient B**	(Selected)
	Combustion > Eddy Dissipation Model Coefficient B > **EDM Coeff. B**	0.5 [b]
	Thermal Radiation > **Option**	P 1

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Component Models > Component > **N2**	(Selected)
	Component Models > Component > N2 > **Option**	Constraint

a. It is important to set a realistic reference pressure in this tutorial because the components of `Methane Air Mixture` are ideal gases.

b. This includes a simple model for partial premixing effects by turning on the Product Limiter. When it is selected, nonzero initial values are required for the products. The products limiter is not recommended for multi-step eddy dissipation reactions, and so is set for this single step reaction only.

2. Click **OK**.

20.4.4. Creating the Boundaries

20.4.4.1. Fuel Inlet Boundary

1. Create a new boundary by clicking *Boundary* and set the name to `fuelin`.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Inlet
	Location	fuelin
Boundary Details	Mass and Momentum > **Normal Speed**	40 [m s^-1]
	Heat Transfer > **Static Temperature**	300 [K]
	Component Details > **CH4**	(Selected)
	Component Details > CH4	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Mass Fraction**	1.0

3. Click **OK**.

20.4.4.2. Bottom Air Inlet Boundary

Two separate boundary conditions will be applied for the incoming air. The first is at the base of the can combustor. The can combustor employs vanes downstream of the bottom air inlet to give the incoming air a swirling velocity.

1. Create a new boundary named `airin`.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Inlet
	Location	airin
Boundary Details	Mass and Momentum > **Normal Speed**	10 [m s^-1]
	Heat Transfer > **Static Temperature**	300 [K]
	Component Details > **O2**	(Selected)
	Component Details > O2 > **Mass Fraction**	0.232 [a]
a. The remaining mass fraction at the inlet will be made up from the constraint component, `N2`.		

3. Click **OK**.

20.4.4.3. Side Air Inlet Boundary

The secondary air inlets are located on the side of the vessel and introduce extra air to aid combustion.

1. Create a new boundary named `secairin`.

2. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Basic Settings	**Boundary Type**	Inlet
	Location	secairin
Boundary Details	Mass and Momentum > **Normal Speed**	6 [m s^-1]
	Heat Transfer > **Static Temperature**	300 [K]
	Component Details > **O2**	(Selected)
	Component Details > O2 > **Mass Fraction**	0.232 [a]
a. The remaining mass fraction at the inlet will be made up from the constraint component, N2.		

3. Click **OK**.

20.4.4.4. Outlet Boundary

1. Create a new boundary named `out`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Outlet
	Location	out
Boundary Details	Mass and Momentum > **Option**	Average Static Pressure
	Mass and Momentum > **Relative Pressure**	0 [Pa]

3. Click **OK**.

20.4.4.5. Vanes Boundary

The vanes above the main air inlet are to be modeled as thin surfaces. To create a vane as a thin surface in CFX-Pre, you must specify a wall boundary on each side of the vanes.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You will first create a new region that contains one side of each of the eight vanes.

1. Create a new composite region by selecting **Insert** > **Regions** > **Composite Region**.
2. Set the name of the composite region to `Vane Surfaces`.
3. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Dimension (Filter)**	2D [a]
	Region List	F129.152, F132.152, F136.152, F138.152, F141.152, F145.152, F147.152, F150.152 [b]

a. This will filter out the 3D regions, leaving only 2D regions

b. Click *Multi-select from extended list* to open the **Selection Dialog** dialog box, then select F129.152, then hold the **Ctrl** key while selecting the remaining items. Click **OK**.

4. Click **OK**.
5. Create another composite region named `Vane Surfaces Other Side`.
6. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Dimension (Filter)**	2D
	Region List	F129.153, F132.153, F136.154, F138.154, F141.155, F145.155, F147.156, F150.156

7. Click **OK**.
8. Create a new boundary named `vanes`.
9. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Wall
	Location	`Vane Surfaces`, `Vane Surfaces Other Side`

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10. Click **OK**.

20.4.4.6. Default Wall Boundary

The default boundary for any undefined surface in CFX-Pre is a no-slip, smooth, adiabatic wall.

- For radiation purposes, the wall is assumed to be a perfectly absorbing and emitting surface (emissivity = 1).
- The wall is non-catalytic, that is, it does not take part in the reaction.

Since this tutorial serves as a basic model, heat transfer through the wall is neglected. As a result, no further boundary conditions need to be defined.

20.4.5. Setting Initial Values

1. Click *Global Initialization* .
2. Configure the following setting(s):

Tab	Setting	Value
Global Settings	Initial Conditions > Cartesian Velocity Components > **Option**	Automatic with Value
	Initial Conditions > Cartesian Velocity Components > **U**	0 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > **V**	0 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > **W**	5 [m s^-1]
	Initial Conditions > Component Details > **O2**	(Selected)
	Initial Conditions > Component Details	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> O2 > **Option**	 Automatic with Value
	Initial Conditions > Component Details > O2 > **Mass Fraction**	 0.232 [a]
	Initial Conditions > Component Details > **CO2**	 (Selected)
	Initial Conditions > Component Details > CO2 > **Option**	 Automatic with Value
	Initial Conditions > Component Details > CO2 > **Mass Fraction**	 0.01
	Initial Conditions > Component Details > **H2O**	 (Selected)
	Initial Conditions > Component Details > H2O > **Option**	 Automatic with Value
	Initial Conditions > Component Details > H2O	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Mass Fraction**	0.01

a. The initial conditions assume the domain consists mainly of air and the fraction of oxygen in air is 0.232. A small mass fraction of reaction products (CO_2 and H_2O) is needed for the EDM model to initiate combustion.

3. Click **OK**.

20.4.6. Setting Solver Control

1. Click *Solver Control* .

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Convergence Control > **Max. Iterations**	100
	Convergence Control > Fluid Timescale Control > **Timescale Control**	Physical Timescale
	Convergence Control > Fluid Timescale Control > **Physical Timescale**	0.025 [s]

3. Click **OK**.

20.4.7. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .

2. Configure the following setting(s):

Setting	Value
File name	CombustorEDM.def

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, **Solver Input File** is set.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. If using stand-alone mode, quit CFX-Pre, saving the simulation (`.cfx`) file.

20.5. Obtaining the Solution Using CFX-Solver Manager

The CFX-Solver Manager will be launched after CFX-Pre saves the CFX-Solver input file. You will be able to obtain a solution to the CFD problem by following the instructions below.

Note:

If a fine mesh is used for a formal quantitative analysis of the flow in the combustor, the solution time will be significantly longer than for the coarse mesh. You can run the simulation in parallel to reduce the solution time. For details, see Obtaining a Solution in Parallel (p. 196).

1. Ensure that the **Define Run** dialog box is displayed.

   Solver Input File should be set to `CombustorEDM.def`.

2. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. At the end of the run, a dialog box is displayed stating that the simulation has ended.

3. Select **Post-Process Results**.

4. If using stand-alone mode, select **Shut down CFX-Solver Manager**.

5. Click **OK**.

20.6. Viewing the Results Using CFD-Post

When CFD-Post opens, experiment with the **Edge Angle** setting for the `Wireframe` object and the various rotation and zoom features in order to place the geometry in a sensible position. A setting of about `8.25` should result in a detailed enough geometry for this exercise.

20.6.1. Temperature Within the Domain

1. Right-click a blank area in the viewer and select **Predefined Camera** > **View From +Y**.

2. Create a new plane named `Plane 1`.

3. Configure the following setting(s):

Tab	Setting	Value
Geometry	Definition > **Method**	 ZX Plane
Color	**Mode**	Variable
	Variable	Temperature

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Click **Apply**.

The large area of high temperature through most of the vessel is due to forced convection.

20.6.2. The NO Concentration in the Combustor

In the next step you will color `Plane 1` by the mass fraction of NO to view the distribution of NO within the domain. The NO concentration is highest in the high temperature region close to the outlet of the domain.

1. Modify the plane named `Plane 1`.
2. Configure the following setting(s):

Tab	Setting	Value
Color	**Variable**	NO.Mass Fraction

3. Click **Apply**.

20.6.3. Printing a Greyscale Graphic

Here you will change the color map (for `Plane 1`) to a grayscale map. The result will be a plot with different levels of gray representing different mass fractions of NO. This technique is especially useful for printing, to a black and white printer, any image that contains a color map. Conversion to grayscale by conventional means (that is, using graphics software, or letting the printer do the conversion) will generally cause color legends to change to a nonlinear distribution of levels of gray.

1. Modify the plane named `Plane 1`.
2. Configure the following setting(s):

Tab	Setting	Value
Color	**Color Map**	Inverse Greyscale

3. Click **Apply**.

20.6.4. Calculating NO Mass Fraction at the Outlet

In the next step, you will calculate the mass fraction of NO in the outlet stream.

1. Select **Tools** > **Function Calculator** or click the **Calculators** tab and select **Function Calculator**.
2. Configure the following setting(s):

Tab	Setting	Value
Function Calculator	**Function**	massFlowAve
	Location	out
	Variable	NO.Mass Fraction

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click **Calculate**.

A small amount of NO is released from the outlet of the combustor. This amount is lower than can normally be expected, and is mainly due to the coarse mesh and the short residence times in the combustor.

20.6.5. Viewing Flow Field

You will next look at the velocity vectors to show the flow field. You may notice a small recirculation in the center of the combustor. Running the problem with a finer mesh would show this region to be a larger recirculation zone. The coarseness of the mesh in this tutorial means that this region of flow is not accurately resolved.

1. On the **Outline** tab, under **User Locations and Plots**, clear the check box for **Plane 1**.

   `Plane 1` is no longer visible.

2. Create a new vector named `Vector 1`.

3. Configure the following setting(s):

Tab	Setting	Value
Geometry	Definition > **Locations**	 Plane 1
Symbol	**Symbol Size**	2

4. Click **Apply**.

5. Create a new plane named `Plane 2`.

6. Configure the following setting(s):

Tab	Setting	Value
Geometry	Definition > **Method**	 XY Plane
	Definition > **Z**	 0.03 [m]
	Plane Bounds > **Type**	 Rectangular
	Plane Bounds > **X Size**	 0.5 [m]
	Plane Bounds > **Y Size**	 0.5 [m]

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Plane Type > **Sample**	 (Selected)
	Plane Type > **X Samples**	 30
	Plane Type > **Y Samples**	 30
Render	**Show Faces**	(Cleared)

7. Click **Apply**.
8. Modify `Vector 1`.
9. Configure the following setting(s):

Tab	Setting	Value
Geometry	Definition > **Locations**	 Plane 2

10. Click **Apply**.

To view the swirling velocity field, right-click in the viewer and select **Predefined Camera** > **View From -Z**

You may also want to turn off the wireframe visibility. In the region near the fuel and air inlets, the swirl component of momentum (theta direction) results in increased mixing with the surrounding fluid in this region. As a result, more fuel is burned.

20.6.6. Viewing Radiation

Try examining the distribution of `Incident Radiation` and `Radiation Intensity` throughout the domain.

When you are finished, quit CFD-Post.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 21: Cavitation Around a Hydrofoil

This tutorial includes:

21.1. Tutorial Features

21.2. Overview of the Problem to Solve

21.3. Preparing the Working Directory

21.4. Simulating the Hydrofoil without Cavitation

21.5. Simulating the Hydrofoil with Cavitation

21.1. Tutorial Features

In this tutorial you will learn about:

- Modeling flow with cavitation.
- Using vector reduction in CFD-Post to clarify a vector plot with many arrows.
- Importing and exporting data along a polyline.
- Plotting computed and experimental results.

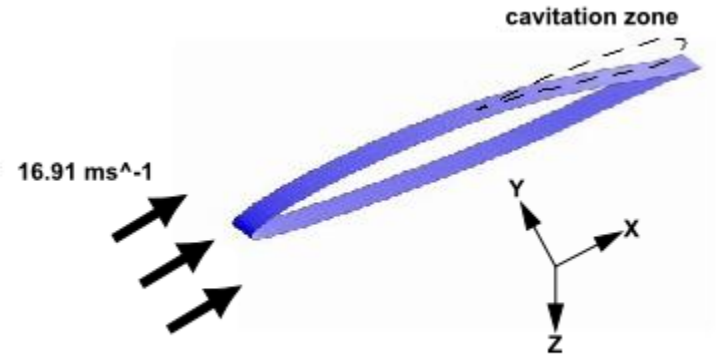
Component	Feature	Details
CFX-Pre	User Mode	General mode
	Analysis Type	Steady State
	Fluid Type	General Fluid
	Domain Type	Single Domain
	Turbulence Model	k-Epsilon
	Heat Transfer	Isothermal
	Multiphase	
	Boundary Conditions	Inlet (Subsonic)
		Outlet (Subsonic)
		Symmetry Plane
		Wall: No-Slip
		Wall: Free-Slip
	Timestep	Physical Time Scale
CFX-Solver Manager	Restart	
CFD-Post	Plots	Contour
		Line Locator

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Component	Feature	Details
		Polyline
		Slice Plane
		Streamline
		Vector
	Other	Chart Creation
		Data Export
		Printing
		Title/Text
		Variable Details View

21.2. Overview of the Problem to Solve

This example demonstrates cavitation in the flow of water around a hydrofoil. A two-dimensional solution is obtained by modeling a thin slice of the hydrofoil and using two symmetry boundary conditions.



In this tutorial, an initial solution with no cavitation is generated to provide an accurate initial guess for a full cavitation solution, which is generated afterwards.

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- Running Ansys CFX Tutorials Using Ansys Workbench (p. 34)
- Changing the Display Colors (p. 37)

21.3. Preparing the Working Directory

1. Create a working directory.

   Ansys CFX uses a working directory as the default location for loading and saving files for a particular session or project.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Download the `hydrofoil.zip` file here.
3. Unzip `hydrofoil.zip` to your working directory.

   Ensure that the following tutorial input files are in your working directory:

   - `HydrofoilExperimentalCp.csv`
   - `HydrofoilGrid.def`

4. Set the working directory and start CFX-Pre.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

21.4. Simulating the Hydrofoil without Cavitation

This section describes the step-by-step definition of the flow physics in CFX-Pre.

21.4.1. Defining the Case Using CFX-Pre

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `HydrofoilIni`.
5. Click **Save**.

21.4.1.1. Importing the Mesh

1. Right-click `Mesh` and select **Import Mesh** > **CFX-Solver Input**.

   The **Import Mesh** dialog box appears.

2. Configure the following setting(s):

Setting	Value
File name	HydrofoilGrid.def

3. Click **Open**.
4. Right-click a blank area in the viewer and select **Predefined Camera** > **View From -Z**.

21.4.1.2. Loading Materials

Since this tutorial uses `Water Vapour at 25 C` and `Water at 25 C`, you need to load these materials.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the **Outline** tree view, right-click `Materials` and select **Import Library Data**.

   The **Select Library Data to Import** dialog box is displayed.

2. Expand **Water Data**.

3. Select both `Water Vapour at 25 C` and `Water at 25 C` by holding **Ctrl** when selecting.

4. Click **OK**.

21.4.1.3. Creating the Domain

The fluid domain used for this simulation contains liquid water and water vapor. The volume fractions are initially set so that the domain is filled entirely with liquid.

1. Edit `Case Options` > `General` in the **Outline** tree view and ensure that **Automatic Default Domain** is turned on.

   A domain named `Default Domain` should now appear under the `Simulation` > `Flow Analysis 1` branch.

2. Double-click `Default Domain`.

3. Under **Fluid and Particle Definitions**, delete `Fluid 1` and create a new fluid definition called `Liquid Water`.

4. Use the button to create another fluid named `Water Vapor`.

5. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Fluid and Particle Definitions**	Liquid Water
	Fluid and Particle Definitions > Liquid Water > **Material**	Water at 25 C [a]
	Fluid and Particle Definitions	Water Vapor
	Fluid and Particle Definitions > Water Vapor > **Material**	Water Vapour at 25 C [a]
	Domain Models > Pressure > **Reference Pressure**	0 [atm]

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Fluid Models	Multiphase > **Homogeneous Model**	(Selected)
	Heat Transfer > **Option**	Isothermal
	Heat Transfer > **Fluid Temperature**	300 [K]
	Turbulence > **Option**	k-Epsilon
a. These two fluids have consistent reference enthalpies.		

6. Click **OK**.

21.4.1.4. Creating the Boundaries

The simulation requires inlet, outlet, wall and symmetry plane boundaries. The regions for these boundaries were imported with the grid file.

21.4.1.4.1. Inlet Boundary

1. Create a new boundary named `Inlet`.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Inlet
	Location	IN
Boundary Details	Mass And Momentum > **Normal Speed**	16.91 [m s^-1]
	Turbulence > **Option**	Intensity and Length Scale
	Turbulence > **Fractional Intensity**	0.03
	Turbulence > **Eddy Length Scale**	0.0076 [m]
Fluid Values	**Boundary Conditions**	Liquid Water
	Boundary Conditions	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Liquid Water > Volume Fraction > **Volume Fraction**	1
	Boundary Conditions	Water Vapor
	Boundary Conditions > Water Vapor > Volume Fraction > **Volume Fraction**	0

3. Click **OK**.

21.4.1.4.2. Outlet Boundary

1. Create a new boundary named `Outlet`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Outlet
	Location	OUT
Boundary Details	Mass And Momentum > **Option**	Static Pressure
	Mass And Momentum > **Relative Pressure**	51957 [Pa]

3. Click **OK**.

21.4.1.4.3. Free Slip Wall Boundary

1. Create a new boundary named `SlipWalls`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Wall
	Location	BOT, TOP
Boundary Details	Mass And Momentum > **Option**	Free Slip Wall

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click **OK**.

21.4.1.4.4. Symmetry Plane Boundaries

1. Create a new boundary named Sym1.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Symmetry
	Location	SYM1

3. Click **OK**.

1. Create a new boundary named Sym2.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Symmetry
	Location	SYM2

3. Click **OK**.

21.4.1.5. Setting Initial Values

1. Click *Global Initialization* .
2. Configure the following setting(s):

Tab	Setting	Value
Global Settings	Initial Conditions > Cartesian Velocity Components > **Option**	Automatic with Value
	Initial Conditions > Cartesian Velocity Components > **U**	16.91 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > **V**	0 [m s^-1]
	Initial Conditions	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Cartesian Velocity Components > **W**	 0 [m s^-1]
Fluid Settings	**Fluid Specific Initialization**	Liquid Water
	Fluid Specific Initialization > Liquid Water > Initial Conditions > Volume Fraction > **Option**	 Automatic with Value
	Fluid Specific Initialization > Liquid Water > Initial Conditions > Volume Fraction > **Volume Fraction**	 1
	Fluid Specific Initialization	Water Vapor
	Fluid Specific Initialization > Water Vapor > Initial Conditions > Volume Fraction > **Option**	 Automatic with Value
	Fluid Specific Initialization > Water Vapor > Initial Conditions > Volume Fraction > **Volume Fraction**	 0

3. Click **OK**.

21.4.1.6. Setting Solver Control

1. Click *Solver Control* .

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Convergence Control > **Max. Iterations**	100
	Convergence Control > Fluid Timescale Control > **Timescale Control**	Physical Timescale
	Convergence Control > Fluid Timescale Control > **Physical Timescale**	0.01 [s]

Note:

For the **Convergence Criteria**, an RMS value of at least 1e-05 is usually required for adequate convergence, but the default value is sufficient for demonstration purposes.

3. Click **OK**.

21.4.1.7. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .

2. Configure the following setting(s):

Setting	Value
File name	HydrofoilIni.def

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, **Solver Input File** is set.

4. Quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

21.4.2. Obtaining the Solution using CFX-Solver Manager

While the calculations proceed, you can see residual output for various equations in both the text area and the plot area. Use the tabs to switch between different plots (for example, **Momentum and Mass**, **Turbulence Quantities**, and so on) in the plot area. You can view residual plots for the fluid and solid domains separately by editing the workspace properties.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Ensure that the **Define Run** dialog box is displayed.
2. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. At the end of the run, a dialog box is displayed stating that the simulation has ended.
3. Select **Post-Process Results**.
4. If using stand-alone mode, select **Shut down CFX-Solver Manager**.
5. Click **OK**.

21.4.3. Viewing the Results Using CFD-Post

The following topics will be discussed:

21.4.3.1. Plotting Pressure Distribution Data
21.4.3.2. Exporting Pressure Distribution Data
21.4.3.3. Saving the Postprocessing State

21.4.3.1. Plotting Pressure Distribution Data

In this section, you will create a plot of the pressure coefficient distribution around the hydrofoil. The data will then be exported to a file for later comparison with data from the cavitating flow case, which will be run later in this tutorial.

1. Right-click a blank area in the viewer and select **Predefined Camera** > **View From -Z**.
2. Insert a new plane named `Slice`.
3. Configure the following setting(s):

Tab	Setting	Value
Geometry	Definition > **Method**	XY Plane
	Definition > **Z**	5e-5 [m]
Render	**Show Faces**	(Cleared)

4. Click **Apply**.
5. Create a new polyline named `Foil` by selecting **Insert** > **Location** > **Polyline** from the main menu.
6. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Geometry	**Method**	Boundary Intersection
	Boundary List	Default Domain Default
	Intersect With	Slice

7. Click **Apply**.

   Zoom in on the center of the hydrofoil (near the cavity) to confirm the polyline wraps around the hydrofoil.

8. Define the following expressions, remembering to click **Apply** after entering each definition:

Name	Definition
PCoef	(Pressure-51957[Pa])/(0.5*996.2[kg m^-3]*16.91[m s^-1]^2)
FoilChord	(X-minVal(X)@Foil)/(maxVal(X)@Foil-minVal(X)@Foil) [a]

a. This creates a normalized chord, measured in the X direction, ranging from 0 at the leading edge to 1 at the trailing edge of the hydrofoil.

9. Create a new variable named `Pressure Coefficient`.

10. Configure the following setting(s):

Setting	Value
Method	Expression
Scalar	(Selected)
Expression	PCoef

11. Click **Apply**.

12. Create a new variable named `Chord`.

13. Configure the following setting(s):

Setting	Value
Method	Expression
Scalar	(Selected)
Expression	FoilChord

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14. Click **Apply**.

> **Note:**
>
> Although the variables that were just created are only needed at points along the polyline, they exist throughout the domain.

Now that the variables `Chord` and `Pressure Coefficient` exist, they can be associated with the previously defined polyline (the locator) to form a chart line. This chart line will be added to the chart object, which is created next.

1. Select **Insert** > **Chart** from the main menu.
2. Set the name to `Pressure Coefficient Distribution`.
3. Configure the following setting(s):

Tab	Setting	Value
General	**Title**	Pressure Coefficient Distribution
Data Series	**Name**	Solver Cp
	Location	Foil
X Axis	Data Selection > **Variable**	Chord
	Axis Range > **Determine ranges automatically**	(Cleared)
	Axis Range > **Min**	0
	Axis Range > **Max**	1
	Axis Labels > **Use data for axis labels**	(Cleared)
	Axis Labels > **Custom Label**	Normalized Chord Position
Y Axis	Data Selection > **Variable**	Pressure Coefficient
	Axis Range	(Cleared)

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Determine Ranges Automatically**	
	Axis Range > **Min**	-0.5
	Axis Range > **Max**	0.4
	Axis Range > **Invert Axis**	(Selected)
	Axis Labels > **Use data for axis labels**	(Cleared)
	Axis Labels > **Custom Label**	Pressure Coefficient

4. Click **Apply**.
5. The chart appears in the **Chart Viewer**.

21.4.3.2. Exporting Pressure Distribution Data

You will now export the chord and pressure coefficient data along the polyline. This data will be imported and used in a chart later in this tutorial for comparison with the results for when cavitation is present.

1. Select **File** > **Export** > **Export**.

   The **Export** dialog box appears

2. Configure the following setting(s):

Tab	Setting	Value
Options	**File**	NoCavCpData.csv
	Locations	Foil
	Export Geometry Information	(Selected) [a]
	Select Variables	Chord, Pressure Coefficient

a. This causes X, Y, Z data to be included in the export file.

3. Click **Save**.

   The file `NoCavCpData.csv` will be written in the working directory.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

21.4.3.3. Saving the Postprocessing State

If you are running CFD-Post in stand-alone mode, you will need to save the postprocessing state for use later in this tutorial, as follows:

1. Select **File > Save State As**.
2. Under **File name** type `Cp_plot`, then click **Save**.

In the next part of this tutorial, the solver will be run with cavitation turned on. Similar postprocessing follows, and the effect of cavitation on the pressure distribution around the hydrofoil will be illustrated in a chart.

21.5. Simulating the Hydrofoil with Cavitation

Earlier in this tutorial, you ran a simulation without cavitation. The solution from that simulation will serve as a starting point for the next simulation, which involves cavitation.

21.5.1. Defining the Case Using CFX-Pre

1. Ensure that the following tutorial input file is in your working directory:
   - `HydrofoilIni_001.res`
2. Set the working directory and start CFX-Pre if it is not already running.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).
3. Select **File** > **Open Case**.
4. Select `HydrofoilIni_001.res` and click **Open**.
5. Save the case as `Hydrofoil.cfx`.

21.5.1.1. Adding Cavitation

1. Double-click `Default Domain` in the **Outline** tree view.
2. Configure the following setting(s):

Tab	Setting	Value
Fluid Pair Models	Fluid Pairs > Liquid Water | Water Vapor > Mass Transfer > **Option**	 Cavitation
	Fluid Pairs	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Liquid Water | Water Vapor > Mass Transfer > Cavitation > **Saturation Pressure**	(Selected)
	Fluid Pairs > Liquid Water | Water Vapor > Mass Transfer > Cavitation > Saturation Pressure > **Saturation Pressure**	3574 [Pa] [a]
a. Saturation pressure can also be set by creating homogeneous binary mixtures, which are not used in this tutorial.		

3. Click **OK**.

21.5.1.2. Modifying Solver Control

1. Click *Solver Control* .
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Convergence Control > **Max. Iterations**	150 [a]
a. This allows up to 150 further iterations, when run as a restart.		

3. Click **OK**.

21.5.1.3. Modifying Execution Control

1. Click *Execution Control* .
2. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Run Definition	Input File Settings > **Solver Input File**	 Hydrofoil.def [a]
a. You do not need to set the full path unless you are saving the solver file somewhere other than the working directory.		

3. Confirm that the rest of the execution control settings are set appropriately.
4. Click **OK**.

21.5.1.4. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, **Solver Input File** and the execution control settings are set.

2. If using stand-alone mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

21.5.2. Obtaining the Solution using CFX-Solver Manager

1. Ensure the **Define Run** dialog box is displayed.

   Solver Input File should be set to `Hydrofoil.def`.

2. Configure the following setting(s) for the initial values file:

Tab	Setting	Value
Initial Values	**Initial Values Specification**	Selected
	Initial Values Specification > **Initial Values**	 Initial Values 1
	Initial Values Specification > Initial Values > Initial Values 1 Settings > **File Name** [a]	 `Hydrofoil-Ini_001.res`
a. Click *Browse* and select the file from the working directory.		

   This is the solution from the starting-point run.

3. Click **Start Run**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CFX-Solver runs and attempts to obtain a solution. At the end of the run, a dialog box is displayed stating that the simulation has ended.

4. Select **Post-Process Results**.
5. If using stand-alone mode, select **Shut down CFX-Solver Manager**.
6. Click **OK**.

21.5.3. Viewing the Results Using CFD-Post

You will restore the state file saved earlier in this tutorial while preventing the first solution (which has no cavitation) from loading. This will cause the plot of pressure distribution to use data from the currently loaded solution (which has cavitation). Data from the first solution will be added to the chart object by importing `NoCavCpData.csv` (the file that was exported earlier). A file containing experimental data will also be imported and added to the plot. The resulting chart will show all three sets of data (solver data with cavitation, solver data without cavitation, and experimental data).

Note:

The experimental data is provided in `HydrofoilExperimentalCp.csv` which must be in your working directory before proceeding with this part of the tutorial.

Note:

If using Ansys Workbench, CFD-Post will already be in the state in which you left it in the first part of this tutorial. In this case, proceed to step 5 below.

1. Select **File** > **Load State**.
2. Clear **Load results**.
3. Select `Cp_plot.cst`.
4. Click **Open**.
5. Click the **Chart Viewer** tab.
6. Edit `Report` > `Pressure Coefficient Distribution`.
7. Click the **Data Series** tab.
8. Configure the following setting(s):

Tab	Setting	Value
Data Series	**Name**	Solver Cp - with cavitation

This reflects the fact that the user-defined variable `Pressure Coefficient` is now based on the current results.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9. Click **Apply**.

   You will now add the chart line from the first simulation.

10. Create a new polyline named `NoCavCpPolyline`.

11. Configure the following setting(s):

Tab	Setting	Value
Geometry	**File**	NoCavCpData.csv

12. Click **Apply**.

    The data in the file is used to create a polyline with values of `Pressure Coefficient` and `Chord` stored at each point on it.

13. Edit `Report` > `Pressure Coefficient Distribution`.

14. Click the **Data Series** tab.

15. Click **New** .

16. Select `Series 2` from the list box.

17. Configure the following setting(s):

Tab	Setting	Value
Data Series	**Name**	Solver Cp - no cavitation
	Location	NoCavCpPolyline
	Custom Data Selection	(Selected)
	X Axis > **Variable**	 Chord on NoCavCpPolyline
	Y Axis > **Variable**	 Pressure Coefficient on NoCavCpPolyline

18. Click **Apply**.

    The chart line (containing data from the first solution) is created, added to the chart object, and displayed in the **Chart Viewer**.

    You will now add a chart line to show experimental results.

19. Click **New** .

20. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Data Series	**Name**	Experimental Cp - with cavitation
	Data Source > **File**	(Selected)
	Data Source > **File**	HydrofoilExperimentalCp.csv
Line Display	Line Display > **Line Style**	Automatic
	Line Display > **Symbols**	Rectangle

21. Click **Apply**.

    The chart line (containing experimental data) is created, added to the chart object, and displayed in the **Chart Viewer**.

22. If you want to save an image of the chart, select **File** > **Save Picture** from the main menu while the **Chart Viewer** is displayed. This will enable you to save the chart to an image file.

23. When you are finished, close CFD-Post.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 22: Modeling a Ball Check Valve using Mesh Deformation and the CFX Rigid Body Solver

This tutorial includes:

22.1. Tutorial Features

22.2. Overview of the Problem to Solve

22.3. Preparing the Working Directory

22.4. Defining the Case Using CFX-Pre

22.5. Obtaining the Solution Using CFX-Solver Manager

22.6. Viewing the Results Using CFD-Post

22.1. Tutorial Features

In this tutorial you will learn about:

- Mesh motion and deformation.
- Rigid body simulation.
- Fluid structure interaction (without modeling solid deformation).
- Animation creation.

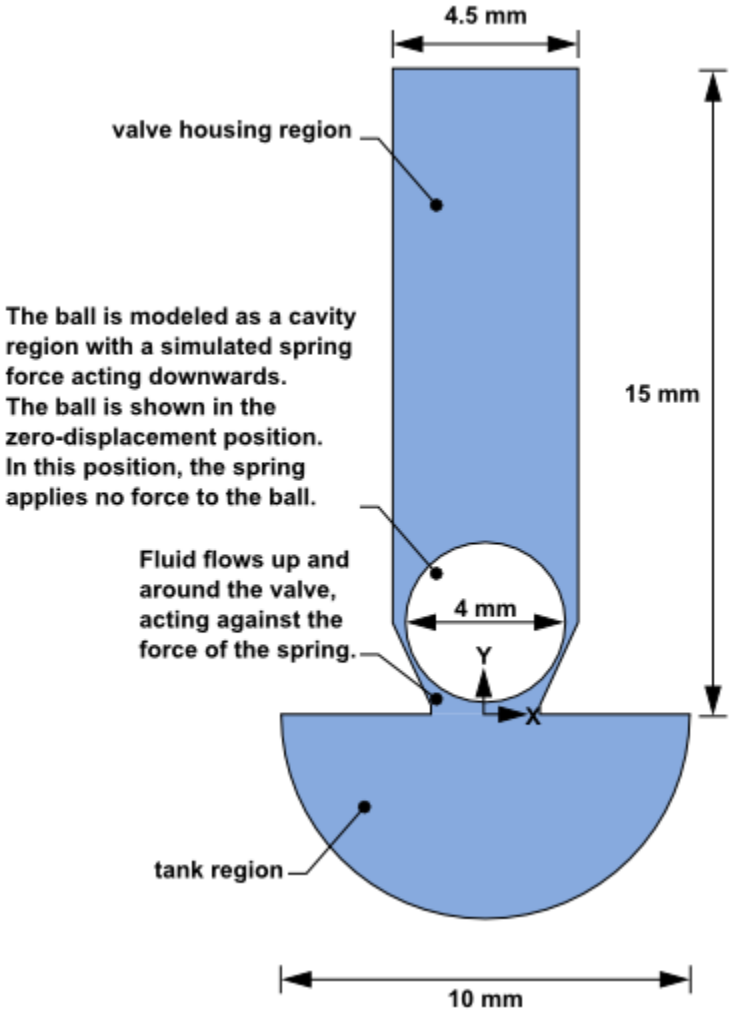
Component	Feature	Details
CFX-Pre	User Mode	General mode
	Analysis Type	Transient
	Fluid Type	General Fluid
	Domain Type	Single Domain
	Turbulence Model	k-Epsilon
	Heat Transfer	Isothermal
	Boundary Conditions	Opening
		Symmetry
		Wall
	Rigid Body	1 Degree of Freedom
	Mesh Motion	Unspecified
		Stationary
		Rigid Body Solution
CFD-Post	Plots	Slice Plane

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Component	Feature	Details
		Point
		Vector
		Animation

22.2. Overview of the Problem to Solve

This tutorial uses an example of a ball check valve to demonstrate two-way Fluid-Structure Interaction (FSI) between a ball and a fluid, as well as mesh deformation capabilities using Ansys CFX. A sketch of the geometry, modeled in this tutorial as a 2D slice (0.1 mm thick), is shown below.



Check valves are commonly used to enforce unidirectional flow of liquids and act as pressure-relieving devices. The check valve for this tutorial contains a ball connected to a spring with a stiffness constant of 300 N/m. The ball is made of steel with a density of 7800 kg/m^3 and is represented as a cavity region in the mesh with a diameter of 4 mm. Initially the center of mass of the ball is located at the coordinate point (0, 0.0023, 5e-05); this point is the spring origin, and all forces that interact with the ball are assumed to pass through this point. The tank region, located below the valve housing, is filled with Methanol (CH4O) at 25°C. High pressure from the liquid at the tank opening (6 atm relative pressure) causes the ball to move up, therefore enabling the fluid to escape through the valve to the atmosphere

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

at an absolute pressure of 1 atm. The forces on the ball are: the force due to the spring (not shown in the figure) and the force due to fluid flow. Gravity is neglected here for simplicity. The spring pushes the ball downward to oppose the force of the pressure when the ball is raised above its initial position. The pressure variation causes the ball to oscillate along the Y axis as a result of a dynamic imbalance in the forces. The ball eventually stops oscillating when the forces acting on it are in equilibrium.

In this tutorial the deformation of the ball itself is not modeled; mesh deformation is employed to modify the mesh as the ball moves. A rigid body simulation is used to predict the motion of the ball, and will be based on the forces that act on it.

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- Running Ansys CFX Tutorials Using Ansys Workbench (p. 34)
- Changing the Display Colors (p. 37)

22.3. Preparing the Working Directory

1. Create a working directory.

   Ansys CFX uses a working directory as the default location for loading and saving files for a particular session or project.

2. Download the `valve_fsi.zip` file here.

3. Unzip `valve_fsi.zip` to your working directory.

   Ensure that the following tutorial input file is in your working directory:

   - `ValveFSI.out`

4. Set the working directory and start CFX-Pre.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

22.4. Defining the Case Using CFX-Pre

This section describes the step-by-step definition of the flow physics in CFX-Pre.

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `ValveFSI`.
5. Click **Save**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

22.4.1. Importing the Mesh

1. Right-click `Mesh` and select **Import Mesh** > **Other**.

   The **Import Mesh** dialog box appears.

2. Configure the following setting(s):

Setting	Value
Files of type	PATRAN Neutral (*out *neu)
File name	ValveFSI.out
Options > **Mesh Units**	 mm [a]

a. This mesh was created using units of millimeters; however the units are not stored with this type of mesh. Set **Mesh Units** to `mm` when importing the mesh into CFX-Pre so that the mesh remains the intended size.

3. Click **Open**.

22.4.2. Defining a Transient Simulation

1. Right-click `Analysis Type` in the **Outline** tree view and select **Edit**.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Analysis Type > **Option**	 Transient
	Analysis Type > Time Duration > **Option**	 Total Time
	Analysis Type > Time Duration > **Total Time**	 7.5e-3 [s]
	Analysis Type > Time Steps > **Option**	 Timesteps
	Analysis Type > Time Steps	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Timesteps**	5.0e-5 [s]
	Analysis Type > Initial Time > **Option**	 Automatic with Value
	Analysis Type > Initial Time > **Time**	 0 [s]

3. Click **OK**.

Note:

You may ignore the physics validation message regarding the lack of definition of transient results files. You will set up the transient results files later.

22.4.3. Editing the Domain

In this section you will create the fluid domain, define the fluid and enable mesh motion.

1. If `Default Domain` does not currently appear under **Flow Analysis 1** in the **Outline** tree,

   edit `Case Options` > `General` in the **Outline** tree view and ensure that **Automatic Default Domain** is turned on and click **OK**.

2. In the tree view, right-click `Default Domain` and select **Edit**.

3. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Location and Type > **Location**	 CV3D REGION, CV3D SUB [a]
	Location and Type > **Domain Type**	 Fluid Domain
	Fluid and Particle Definitions	Fluid 1
	Fluid and Particle Definitions > Fluid 1 > **Material**	 Methanol CH4O [b]

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Domain Models > Pressure > **Reference Pressure**	1 [atm]
	Domain Models > Mesh Deformation > **Option**	Regions of Motion Specified [c]
	Domain Models > Mesh Deformation > **Displacement Rel. To**	Previous Mesh
	Domain Models > Mesh Deformation > Mesh Motion Model > **Option**	Displacement Diffusion [d] [e]
	Domain Models > Mesh Deformation > Mesh Motion Model > Mesh Stiffness > **Option**	Increase near Small Volumes
	Domain Models > Mesh Deformation > Mesh Motion Model > Mesh Stiffness > **Model Exponent**	1 [f]
	Domain Models > Mesh Deformation > Mesh Motion Model > Mesh Stiffness	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Reference Volume > **Option**	 Mean Control Volume
Fluid Models	Heat Transfer > **Option**	 Isothermal
	Heat Transfer > **Fluid Temperature**	 25 [C]

a. Click the *Multi-select from extended list* icon to open the **Selection Dialog** dialog box, then hold the **Ctrl** key while selecting both `CV3D REGION` and `CV3D SUB` from this list. Click **OK**.

b. To make Methanol an available option:

   i. Click the *Select from extended list* icon to open the **Material** dialog box.

   ii. Click the *Import Library Data* icon to open the **Select Library Data to Import** dialog box.

   iii. In that dialog box, expand `Constant Property Liquids` in the tree, select `Methanol CH4O` and click **OK**.

   iv. Select `Methanol CH4O` in the **Material** dialog box and click **OK**.

c. The `Regions of Motion Specified` option permits boundaries and subdomains to move, and makes mesh motion settings available.

d. To see the additional mesh motion settings, you may need to click *Roll Down* located beside **Mesh Motion Model**.

e. The Displacement Diffusion model for mesh motion attempts to preserve the relative mesh distribution of the initial mesh.

f. An exponent value of 1 is chosen in order to limit the distortion of the coarse mesh. If a finer mesh were used, the default value of 2 would most likely be appropriate.

4. Click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

22.4.4. Creating a Coordinate Frame

In this section, a secondary coordinate system will be created to define the center of mass of the ball. This secondary coordinate system will be used to define certain parameters of the rigid body in the next section.

1. In the **Outline** tree view, right-click **Coordinate Frames** and select **Insert** > **Coordinate Frame**.
2. Set the name to `Coord 1` and click **OK**.
3. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Option**	Axis Points
	Origin	(0, 0.0023, 5e-05)
	Z Axis Point	(0, 0.0023, 1)
	X-Z Plane Pt	(1, 0.0023, 0)

4. Select **OK**.

22.4.5. Creating a Rigid Body

A rigid body is a non-deformable object described by physical parameters: mass, center of mass, moment of inertia, initial velocities and accelerations, and orientation. The rigid body solver uses the interacting forces between the fluid and the rigid body and calculates the motion of the rigid body based upon the defined physical parameters. The rigid body may have up to six degrees of freedom (three translational and three rotational). You may also specify external forces and torques acting on the rigid body.

In this section, you will define a rigid body with 1 degree of freedom, translation in the Y direction. The rigid body definition will be applied to the wall boundary of the ball to define its motion. Further, you will specify an external spring force by defining a spring constant and the initial origin of the spring; in this simulation the origin is the center of mass of the ball. The force caused by the tank pressure will cause an upward translation and the defined external spring force will resist this translation.

1. In the **Outline** tree view, right-click **Flow Analysis 1** and select **Insert** > **Rigid Body**.
2. Set the name to `rigidBall` and click **OK**.
3. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Mass**	9.802e-6 [kg]
	Location	BALL
	Coordinate Frame	Coord 1
	Mass Moment of Inertia	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **XX Component**	0 [kg m^2] [a]
	Mass Moment of Inertia > **YY Component**	0 [kg m^2]
	Mass Moment of Inertia > **ZZ Component**	0 [kg m^2]
	Mass Moment of Inertia > **XY Component**	0 [kg m^2]
	Mass Moment of Inertia > **XZ Component**	0 [kg m^2]
	Mass Moment of Inertia > **YZ Component**	0 [kg m^2]
Dynamics	**External Force Definitions**	Create a new external force named `Spring Force` [b].
	External Force Definitions > Spring Force > **Option**	Spring
	External Force Definitions > Spring Force > Equilibrium Position > **X Component**	0 [m]
	External Force Definitions > Spring Force > Equilibrium Position > **Y Component**	0 [m]
	External Force Definitions > Spring Force > Equilibrium Position > **Z Component**	0 [m]
	External Force Definitions	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Spring Force > Linear Spring Constant > **X Component**	0 [N m^-1]
	External Force Definitions > Spring Force > Linear Spring Constant > **Y Component**	300 [N m^-1]
	External Force Definitions > Spring Force > Linear Spring Constant > **Z Component**	0 [N m^-1]
	Degrees of Freedom > Translational Degrees of Freedom > **Option**	Y axis
	Degrees of Freedom > Rotational Degrees of Freedom > **Option**	None
a. The **Mass Moment of Inertia** settings can have any values; they have no effect on the simulation because the rigid body has only a singular, translational, degree of freedom. b. To create a new item, you must first click the *Add new item* icon, then enter the name as required and click **OK**.		

4. Click **OK**.

22.4.6. Creating the Subdomain

1. Select **Insert** > **Subdomain** from the main menu or click *Subdomain* .
2. Set the subdomain name to Tank and click **OK**.
3. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Basic Settings	**Location**	CV3D SUB
Mesh Motion	Mesh Motion > **Option**	Stationary [a]

a. The stationary option for the tank volume (subdomain) ensures that the mesh does not fold at the sharp corners that exist where the valve joins the tank.

4. Click **OK**.

22.4.7. Creating the Boundaries

In the following subsections, you will create the required boundary conditions, specifying the appropriate mesh motion option for each.

In this tutorial, mesh motion specifications are applied to two and three dimensional regions of the domain. For example, the Ball boundary specifies the mesh motion in the form of the rigid body solution. However, mesh motion specifications are also used in this tutorial to help ensure that the mesh does not fold, as set for the Tank subdomain earlier in the tutorial, and the TankOpen boundary below. Two regions, VALVE HIGHX and VALVE LOWX, remain at the default boundary condition: smooth, no slip walls and no mesh motion (stationary).

22.4.7.1. Ball Boundary

1. Create a new boundary named Ball.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Wall
	Location	BALL
Boundary Details	Mesh Motion > **Option**	Rigid Body Solution
	Mesh Motion > **Rigid Body**	rigidBall
	Mass And Momentum > **Option**	No Slip Wall
	Mass And Momentum > **Wall Vel. Rel. To**	Mesh Motion

3. Click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

22.4.7.2. Symmetry Boundary

Because a 2D representation of the flow field is being modeled (using a 3D mesh, one element thick in the Z direction), you must create symmetry boundaries on the low and high Z 2D regions of the mesh.

1. Create a new boundary named Sym.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Symmetry
	Location	SYMP1, SYMP2 [a]
Boundary Details	Mesh Motion > **Option**	Unspecified

a. Hold the **Ctrl** key while selecting both SYMP1 and SYMP2 from the list.

3. Click **OK**.

22.4.7.3. Vertical Valve Wall Boundary

1. Create a new boundary named ValveVertWalls.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Wall
	Location	VPIPE HIGHX, VPIPE LOWX [a]
Boundary Details	Mesh Motion > **Option**	Unspecified [b]
	Mass And Momentum > **Option**	No Slip Wall
	Mass And Momentum > **Wall Vel. Rel. To**	Boundary Frame

a. Hold the **Ctrl** key while selecting both VPIPE HIGHX and VPIPE LOWX from the list.

b. The Unspecified setting allows the mesh nodes to move freely. The motion of the mesh points on this boundary will be strongly influenced by the motion of the ball. Because the ball moves vertically, the surrounding mesh nodes should also move vertically, at a similar rate to the ball. This mesh motion

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
specification helps to preserve the quality of the mesh on the upper surface of the ball.		

3. Click **OK**.

22.4.7.4. Tank Opening Boundary

1. Create a new boundary named `TankOpen`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Opening
	Location	BOTTOM
Boundary Details	Mesh Motion > **Option**	Stationary [a]
	Mass And Momentum > **Option**	Entrainment
	Mass And Momentum > **Relative Pressure**	6 [atm] [b]
	Turbulence > **Option**	Zero Gradient

a. The stationary option for the tank opening prevents the mesh nodes on this boundary from moving. If the tank opening had unspecified mesh motion, these mesh nodes would move vertically and separate from the non-vertical parts of the boundary.

b. As defined in the problem description. Note the units for this setting.

3. Click **OK**.

22.4.7.5. Valve Opening Boundary

1. Create a new boundary named `ValveOpen`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Opening

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Location	TOP
Boundary Details	Mesh Motion > **Option**	Stationary [a]
	Mass And Momentum > **Option**	Entrainment
	Mass And Momentum > **Relative Pressure**	0 [atm] [b]
	Turbulence > **Option**	Zero Gradient

a. The stationary option for the valve opening prevents the mesh nodes from moving.

b. This pressure value is relative to the fluid domain's reference pressure of 1 [atm].

3. Click **OK**.

Note:

Opening boundary types are used to enable the flow to leave and reenter the domain. This behavior is expected due to the oscillatory motion of the ball and due to the potentially large region of flow recirculation that may occur downstream from the ball.

22.4.8. Setting Initial Values

Because a transient simulation is being modeled, initial values are required for all variables.

1. Click *Global Initialization* .

2. Configure the following setting(s):

Tab	Setting	Value
Global Settings	Initial Conditions > Cartesian Velocity Components > **U**	0 [m s^-1]
	Initial Conditions	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Cartesian Velocity Components > **V**	 0.1 [m s^-1] [a]
	Initial Conditions > Cartesian Velocity Components > **W**	 0 [m s^-1]
	Initial Conditions > Static Pressure > **Relative Pressure**	 0 [Pa]
	Initial Conditions > Turbulence > **Option**	 Medium (Intensity = 5%)

a. This is an initial velocity to start a unidirectional fluid flow in the positive Y direction and to prevent initial backflow in the check-valve, improving solution convergence. Better values of velocity could be derived from the steady-state analysis (not considered for this tutorial).

3. Click **OK**.

22.4.9. Setting Solver Control

In this section you will edit the solver control settings to promote a quicker solution time and to enable the frequency of when the rigid body solver is run.

1. Click *Solver Control* .

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Transient Scheme > **Option**	 Second Order Backward Euler
	Convergence Control > **Max. Coeff. Loops**	 5

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Rigid Body Control	**Rigid Body Control**	(Selected)
	Rigid Body Control > Rigid Body Solver Coupling Control > **Update Frequency**	 Every Coefficient Loop [a]
a. By setting the **Update Frequency** to `Every Coefficient Loop` you are telling CFX-Solver to call the rigid body solver during every coefficient loop within each time step.		

3. Click **OK**.

22.4.10. Setting Output Control

This step sets up transient results files to be written at set intervals.

1. Click *Output Control* .

2. Click the **Trn Results** tab.

3. In the **Transient Results** tree view, click *Add new item* , set **Name** to `Transient Results 1`, and click **OK**.

4. Configure the following setting(s) of `Transient Results 1`:

Setting	Value
Option	Selected Variables
Output Variables List	Pressure, Velocity
Output Variable Operators	(Selected)
Output Variable Operators > **Output Var. Operators**	 All [a]
Output Frequency > **Option**	 Time Interval
Output Frequency > **Time Interval**	 5.0e-5 [s]
a. This causes the gradients of the selected variables to be written to the transient results files.	

5. Click the **Monitor** tab.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Select **Monitor Objects**.
7. Under **Monitor Points and Expressions**:
   a. Click *Add new item* .
   b. Set **Name** to `Ball Displacement` and click **OK**.
   c. Set **Option** to `Expression`.
   d. Set **Expression Value** to `rbstate(Position Y)@rigidBall`.
8. Click **OK**.

22.4.11. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .
2. Configure the following setting(s):

Setting	Value
File name	ValveFSI.def

3. Click **Save**.
4. CFX-Solver Manager starts automatically and, on the **Define Run** dialog box, **Solver Input File** is set.
5. Quit CFX-Pre, saving the simulation (`.cfx`) file.

22.5. Obtaining the Solution Using CFX-Solver Manager

When CFX-Pre has shut down and the CFX-Solver Manager has started, obtain a solution to the CFD problem by following the instructions below.

1. Ensure that the **Define Run** dialog box is displayed.

   Solver Input File should be set to `ValveFSI.def`.
2. Click **Start Run**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CFX-Solver runs and attempts to obtain a solution. This can take a long time depending on your system.

Note:

You can ignore the warning message indicating a negative sector volume in your mesh. This occurs as a result of large deformations in the mesh and will not significantly affect the results for this case.

3. While CFX-Solver Manager is running, you can check the progress of the monitor point you created in CFX-Pre by clicking the `User Points` tab in CFX-Solver Manager. The graph shows the Y position of the center of mass of the ball (in the global coordinate frame). Notice that the ball has a sinusoidal motion that diminishes in amplitude over time and that the maximum displacement of the ball occurs at around time step 17.

4. Select **Monitors** > **Rigid Body** > **Rigid Body Position** from the main menu. The position of the rigid body will be shown in the X, Y and Z directions relative to the global coordinate frame.

Note:

This graph is identical to the graph obtained from under the `User Points` tab (although the scale may be different). Normally, creating the monitor point for position is redundant since the rigid body positions are calculated automatically — the monitor point was created in this tutorial to demonstrate the `rbstate` function.

5. When a dialog box is displayed at the end of the run, select **Post-Process Results**.

6. Click **OK**.

22.6. Viewing the Results Using CFD-Post

In the following subsections, you will create a user location, point and vector plots, and an animation in CFD-Post. You will create an XY plane that lies midway between the two symmetry planes. The plane will be used to show the mesh motion; it will also serve as the location for a vector plot that will be used in the animation.

22.6.1. Creating a Slice Plane

22.6.2. Creating Points and a Vector Plot

22.6.3. Creating an Animation

22.6.1. Creating a Slice Plane

1. Right-click a blank area in the viewer and select **Predefined Camera** > **View From +Z**.

2. Select **Insert** > **Location** > **Plane** from the main menu. Accept the default name and click **OK**.

3. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Geometry	Definition > **Method**	XY Plane
	Definition > **Z**	5e-05 [m]
Render	**Show Faces**	(Cleared)
	Show Mesh Lines	(Selected)

4. Click **Apply**.

22.6.2. Creating Points and a Vector Plot

1. Select **Insert** > **Location** > **Point** from the main menu. Accept the default name and click **OK**.
2. Configure the following setting(s):

Tab	Setting	Value
Geometry	Definition > **Method**	XYZ
	Point	(0, 0.0003, 0)
Symbol	**Symbol**	Crosshair
	Symbol Size	5

3. Click **Apply** to create the point.

   This is a reference point for the minimum Y value of the ball at time step 0. However the final time step is currently selected. This will be corrected in the proceeding steps.

4. Select **Insert** > **Location** > **Point** from the main menu. Accept the default name and click **OK**.
5. Configure the following setting(s):

Tab	Setting	Value
Geometry	Definition > **Method**	XYZ
	Point	(0, 0.001266, 0)
Symbol	**Symbol**	Crosshair
	Symbol Size	5

6. Click **Apply** to create the point.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This is a reference point for the minimum Y value of the ball in the positive Y direction at the time of maximum displacement.

7. Click *Timestep Selector* and load the results for a few different time steps, selecting one entry at a time.

   For example, double-click rows with the step values of 0, 10, 20, 50, and 90 to see the ball in different positions. The mesh deformation will also be visible.

8. Create a new vector named `Vector 1`.

9. Configure the following setting(s):

Tab	Setting	Value
Geometry	**Locations**	Plane 1
	Variable	Velocity

10. Click **Apply** to show the vector plot in the **3D Viewer**.

22.6.3. Creating an Animation

You will create an animation showing the velocity in the domain as the ball moves.

1. Turn off the visibility of `Plane 1` to better see the vector plot.

2. Click the *Timestep Selector* and load the 1st time step.

3. Click *Animate timesteps* in the *Timestep Selector* dialog box.

   The **Animation** dialog box appears.

4. Set **Type** to **Keyframe Animation**.

5. Click *New* to create `KeyframeNo1`.

6. Select `KeyframeNo1`, then set **# of Frames** to `149`, then press **Enter** while the cursor is in the **# of Frames** box.

   Tip:

   Be sure to press **Enter** and confirm that the new number appears in the list before continuing.

7. Use the **Timestep Selector** to load the last time step.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. In the **Animation** dialog box, click *New* to create `KeyframeNo2`.

   Tip:

   The **# of Frames** parameter has no effect on the last keyframe, so leave it at the default value.

9. Ensure that *More Animation Options* is pushed down to show more animation settings.
10. Select **Loop**.
11. Ensure that *Repeat forever* (next to the **Repeat** setting) is not selected (not pushed down).
12. Click the **Options** button to open the **Animation Options** dialog box.
13. Configure the following setting(s):

Tab	Setting	Value
Options	Print Options > **Image Size**	720 x 480 (NTSC)
Advanced	MPEG Options > **Quality**	Custom
	MPEG Options > **Variable Bit Rate**	(Cleared)
	MPEG Options > **Bit Rate**	3000000 [a]

a. This limits the bit rate so that the movie will be playable in most players. You can lower this value if your player cannot process at this bit rate.

14. Click **OK**.
15. Select **Save Movie**.
16. Set **Format** to `MPEG1`.
17. Click *Browse* (next to **Save Movie**).
18. Set **File name** to `ValveFSI.mpg`.

    If required, set the path to a different directory.
19. Click **Save**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The movie file name (including the path) has been set, but the animation has not yet been produced.

20. Click *To Beginning* .

    This ensures that the animation will begin at the first keyframe.

21. After the first keyframe has been loaded, click *Play the animation* .

    - The MPEG will be created as the animation proceeds.
    - This will be slow, since results for each time step will be loaded and objects will be created.
    - To view the movie file, you need to use a viewer that supports the MPEG format.

    Note:

    To explore additional animation options, click the **Options** button. On the **Advanced** tab of the **Animation Options** dialog box, there is a **Save Frames As Image Files** check box. By selecting this check box, the JPEG or PPM files used to encode each frame of the movie will persist after movie creation; otherwise, they will be deleted.

22. Close the **Animation** dialog box when the animation is complete.
23. When you have finished, close the **Timestep Selector** dialog box and quit CFD-Post.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 23: Aerodynamic and Structural Performance of a Centrifugal Compressor

This tutorial includes:

23.1. Tutorial Features

23.2. Overview of the Problem to Solve

23.3. Preparing the Working Directory

23.4. Setting Up the Project

23.5. Defining the Geometry Using Ansys BladeGen

23.6. Defining the Mesh

23.7. Defining the Case Using CFX-Pre

23.8. Obtaining the Solution Using CFX-Solver Manager

23.9. Viewing the Results Using CFD-Post

23.10. Simulating the Structural Performance Using Static Structural

Note:

Because this tutorial makes use of Ansys BladeGen, it must be run on a Windows-based machine.

23.1. Tutorial Features

This tutorial addresses the following features:

Component	Feature	Details
Ansys BladeGen	Geometry	Transfer of geometry to Ansys TurboGrid and Mechanical Model
Ansys TurboGrid	Mesh	Shroud Tip defined by Profile
		ATM Optimized
Mechanical Model	Mesh	Virtual Topology
		Edge Sizing
		Face Meshing
		Sweep Method
CFX-Pre	User Mode	Turbo mode

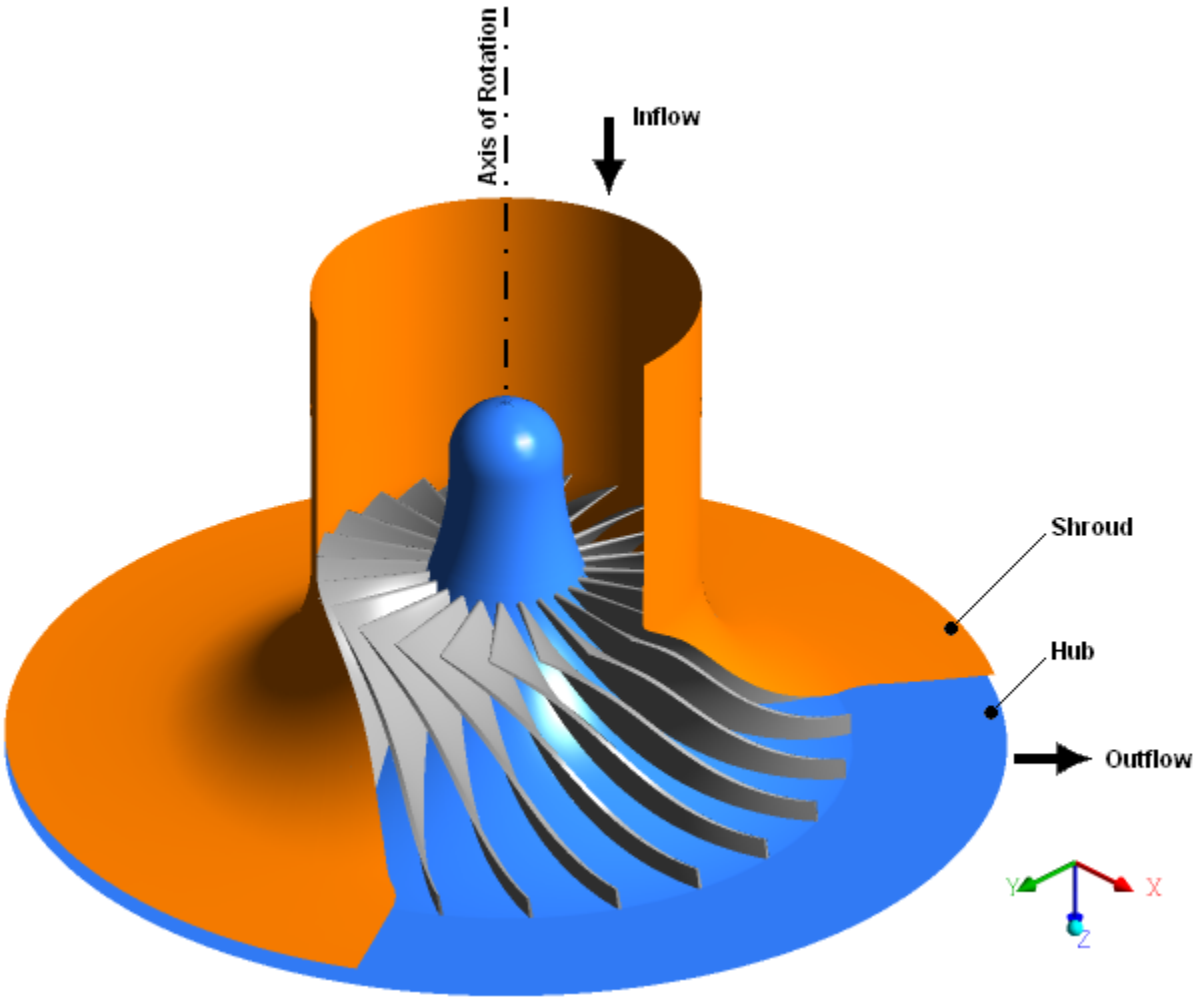
Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Component	Feature	Details
		General mode
	Machine Type	Centrifugal Compressor
	Component Type	Rotating
	Analysis Type	Steady State
	Domain Type	Fluid Domain
		Solid Domain
	Fluid Type	Air Ideal Gas
	Boundary Template	P-Total Inlet Mass Flow Outlet
	Flow Direction	Cylindrical Components
	Solid Type	Steel
	Heat Transfer	Thermal Energy
	Domain Interface	Fluid Fluid
		Fluid Solid
	Timestep	Physical Timescale
CFD-Post	Report	Computed Results Table
		Blade Loading Span 50
		Streamwise Plot of Pt and P
		Velocity Streamlines Stream Blade TE
Static Structural	Static Structural Analysis	Importing CFX pressure data
		Importing CFX temperature data
		Fixed Support
		Rotationally-induced inertial effect
	Static Structural Solutions	Equivalent Stress (von-Mises)
		Total Deformation

23.2. Overview of the Problem to Solve

This tutorial makes use of several Ansys software components to simulate the aerodynamic and structural performance of a centrifugal compressor.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The compressor has 24 blades that revolve about the Z axis at 22360 RPM. A clearance gap exists between the blades and the shroud of the compressor. The outer diameter of the blade row is approximately 40 cm.

To begin analysis of the aerodynamic performance, a mesh will be created in both Ansys TurboGrid and the Mechanical application using an existing design that is to be reviewed beforehand in Ansys BladeGen. Once the meshes have been created, initial parameters defining the aerodynamic simulation will be set in CFX-Pre and then solved in CFX-Solver. The aerodynamic solution from the solver will then be processed and displayed in CFD-Post.

You will then use the Mechanical application to simulate structural stresses and deformation on the blade due to pressure and temperature loads from the aerodynamic analysis and rotationally-induced inertial effects. You will view an animation that shows the resulting blade distortion.

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- Running Ansys CFX Tutorials Using Ansys Workbench (p. 34)
- Changing the Display Colors (p. 37)

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

23.3. Preparing the Working Directory

1. Create a working directory.

   Ansys CFX uses a working directory as the default location for loading and saving files for a particular session or project.

2. Download the `centrifugal_compressor.zip` file here.

3. Unzip `centrifugal_compressor.zip` to your working directory.

   Ensure that the following tutorial input file is in your working directory:

   - `Centrifugal_Compressor.bgd`

23.4. Setting Up the Project

1. Start Ansys Workbench by clicking the **Start** menu then selecting **All Programs > ANSYS 2021 R2 > Workbench 2021 R2**.

2. From the main menu, select **File** > **Save As**.

3. In the **Save As** dialog box, browse to the working directory and set **File name** to `Compressor`.

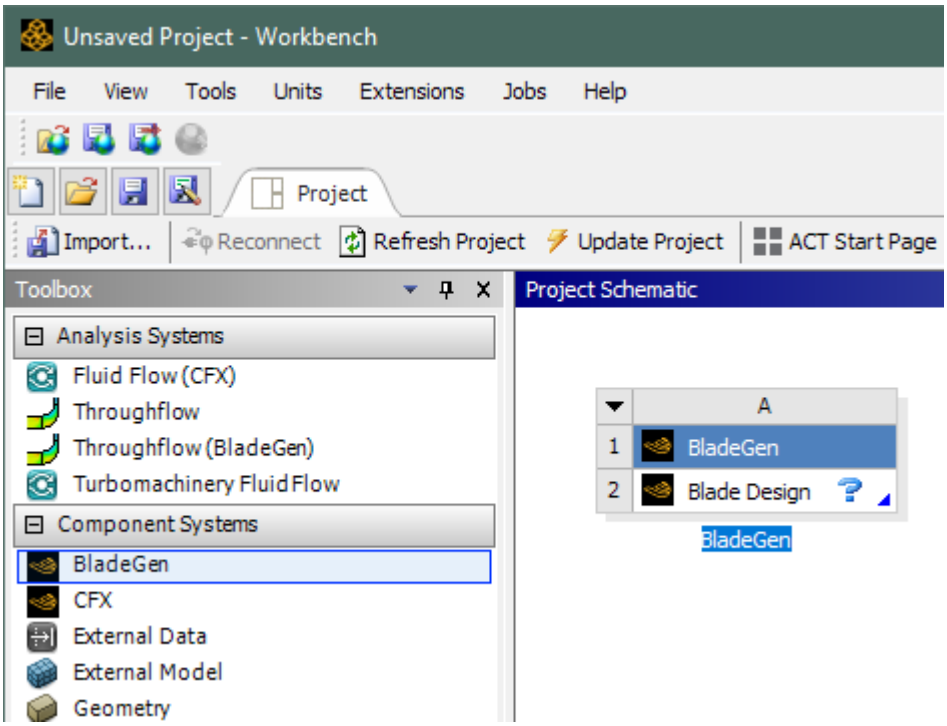
4. Click **Save**.

23.5. Defining the Geometry Using Ansys BladeGen

1. In the **Toolbox** view, expand **Component Systems** and double-click **BladeGen**.

   In the **Project Schematic** view, a **BladeGen** system opens and is ready to be given a name.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. In the name field, type `Compressor` and then either press **Enter** or click outside the name field to end the rename operation.

   If you need to begin a new rename operation, right-click the **BladeGen** cell (A1) and select **Rename**.

   Now that renaming systems has been demonstrated, most of the other systems involved in this tutorial will simply use default names.

23.5.1. Changing the Blade Design Properties

Change the **Blade Design** cell properties as follows:

1. In the **Project Schematic** view, in the **BladeGen** system, right-click the **Blade Design** cell and select **Properties**.

   The properties view shows properties that control how the geometry is exported to downstream systems.

2. In the properties view, clear the **Merge Blade Topology** and **Create Fluid Zone** check boxes.

   By clearing the **Create Fluid Zone** check box, you are specifying that only the blade geometry, and not the volume around the blade, should be sent to downstream cells.

3. Set **Shroud Clearance** to `Relative Layer`.

   This will make sure the tip clearance layer is not part of the geometry when it is transferred to the Mechanical application.

The properties should appear as follows:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Properties of Schematic A2: Blade Design

	A	B	D
1	Property	Value	P
2	General		
3	Component ID	Blade Design	
4	Directory Name	BG	
5	Notes		
6	Notes		
7	Used Licenses		
8	Last Update Used Licenses		
9	Import Options		
10	Create Hub	☐	
11	Create All Blades	☐	
12	Merge Blade Topology	☐	
13	Blade Loft Direction	Streamwise	
14	Shroud Clearance	Relative Layer	
15	Layer Number	1	☐
16	Create Fluid Zone	☐	
17	Blade Extension (%)	2	☐

You now have a **BladeGen** system that contains a **Blade Design** cell; the latter is presently in an unfulfilled state, as indicated by the question mark. In the next section, you will fulfill the cell requirements by loading the provided geometry file. In general, you could also fulfill the cell requirements by creating a new geometry.

23.5.2. Reviewing the Geometry

In this section, you will load the geometry from the provided `.bgd` file, and then review the blade design using Ansys BladeGen. BladeGen is a geometry-creation tool specifically designed for turbomachinery blades.

1. In the **Project Schematic** view, in the **BladeGen** system, right-click the **Blade Design** cell and select **Edit** or double-click the cell.

   BladeGen opens.

2. In BladeGen, in the main menu, select **File** > **Open**.

3. In the **Open** dialog box, browse to the working directory and select the file `Centrifugal_Compressor.bgd`.

4. Click **Open**.

5. Observe the blade design shown in BladeGen.

6. Quit BladeGen.

   To do this, in the main menu, select **File** > **Exit**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. In the **Project Schematic** view, the **Blade Design** cell now displays a green check mark to indicate that the cell is up-to-date. This means that you now have a geometry for the centrifugal compressor that is ready to be used for meshing purposes.

In the next section, you will create a CFD-compatible mesh based on this geometry.

23.6. Defining the Mesh

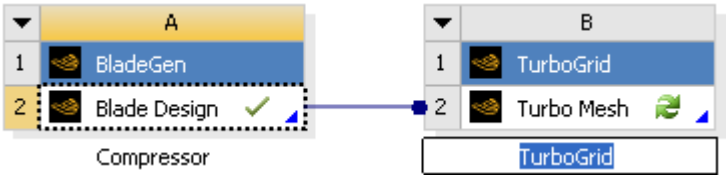
You will create a mesh in both Ansys TurboGrid and the Mechanical application. TurboGrid will create a CFD mesh that will be part of the fluid domain. The Mechanical application will generate the solid blade mesh that is required for solving for the volumetric temperature in the blade.

23.6.1. Defining the CFD Mesh Using Ansys TurboGrid

TurboGrid is a mesh creation tool specifically designed for turbomachinery blades. In this section, you will use TurboGrid to produce a CFD-compatible mesh based on the centrifugal compressor geometry.

1. In the **BladeGen** system, right-click the **Blade Design** cell and select **Transfer Data To New** > **TurboGrid**.

   A **TurboGrid** system opens and is ready to be given a name.



2. Press **Enter** to accept the default name.

   The **Turbo Mesh** cell displays a pair of green curved arrows to indicate that the cell has not received the latest upstream data. Normally, you would right-click such a cell and select **Refresh** to transfer data from the upstream cell. However, for a newly-added **Turbo Mesh** cell and whenever TurboGrid is not open, this action is not necessary because TurboGrid always reads the upstream cell data upon starting up.

   If you were to refresh the **Turbo Mesh** cell, it would show a question mark to indicate that, although the cell's inputs are current, further attention is required in order to bring the cell to an up-to-date status; that further action would typically be to run TurboGrid and produce a mesh with it.

3. In the **TurboGrid** system, right-click the **Turbo Mesh** cell and select **Edit**.

   TurboGrid opens.

The next several sections guide you through the steps to create the CFD mesh.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

23.6.1.1. Defining the Shroud Tip

For this compressor, the shroud is stationary and requires a clearance gap between the blade and shroud. Define the tip of the blade using the second blade profile in the blade geometry.

1. In TurboGrid, in the object selector, right-click `Geometry` > `Blade Set` > `Shroud Tip` and select **Edit**.
2. In the object editor, configure the following setting(s):

Tab	Setting	Value
Shroud Tip	**Override Upstream Geometry Options**	(Selected)
	Tip Option	Profile Number
	Tip Profile	2

3. Click **Apply**.

   This defines the shroud tip of the blade (the surface of the blade that is nearest to the shroud).

23.6.1.2. Creating the Topology and Mesh

Now that the geometry is defined, create the topology and mesh:

- Right-click `Topology Set` and turn off **Suspend Object Updates**.

  The topology and 3D mesh are generated.

23.6.1.3. Changing the Mesh Data Settings

The `Mesh Data` settings control the number and distribution of mesh elements.

1. In the object selector, right-click `Mesh Data` and select **Edit**.

   Alternatively, in the toolbar, click *Edit Mesh Data* .
2. In the object editor, configure the following setting(s):

Tab	Setting	Value
Mesh Size	**Method**	Target Passage Mesh Size
	Node Count	Medium (100000)
	Inlet Domain	(Selected)
	Outlet Domain	(Selected)
Passage	Spanwise Blade Distribution Parameters	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Method**	Element Count and Size
	Spanwise Blade Distribution Parameters > **# of Elements**	25
	Spanwise Blade Distribution Parameters > **Const Elements**	11
Inlet	Inlet Domain > **Define By**	Target Number of Elements
	Inlet Domain > **# of Elements**	50
Outlet	Outlet Domain > **Define By**	Target Number of Elements
	Outlet Domain > **# of Elements**	25

3. Click **Apply**.
4. Select **File** > **Close TurboGrid**.
5. Return to the **Project Schematic** view.

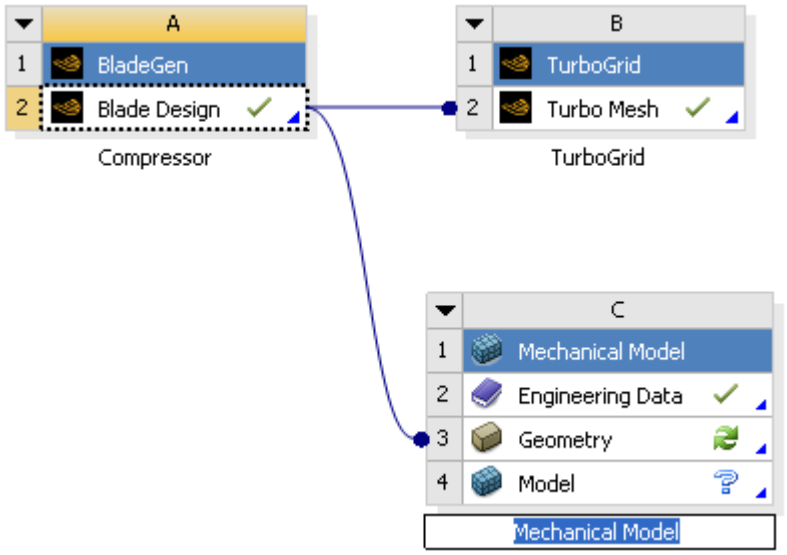
23.6.2. Defining the Structural Mesh Using Mechanical Model

A mesh is required for the solid blade geometry for CFX-Solver to solve for the volumetric temperature. This volumetric temperature will later be exported to Static Structural to calculate the stresses on the blade. In this section, you will use the Mechanical application to produce a solid blade mesh.

1. In the **BladeGen** system, right-click the **Blade Design** cell and select **Transfer Data To New** > **Mechanical Model**.

   A **Mechanical Model** system opens and is ready to be given a name.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Press **Enter** to accept the default name.
3. In the **Mechanical Model** system, right-click the **Geometry** cell and select **Update**.

   Ansys DesignModeler runs in the background, and imports the geometry from the **BladeGen** system.
4. Right-click the **Model** cell and select **Edit**.

   The Mechanical application opens.

The next several sections guide you through the steps to create the mesh.

23.6.2.1. Specifying the Global Mesh Controls

1. In the Mechanical application, in the **Outline** tree view, select `Project` > `Model (C4)` > `Mesh`.
2. In the details view, configure the following setting(s):

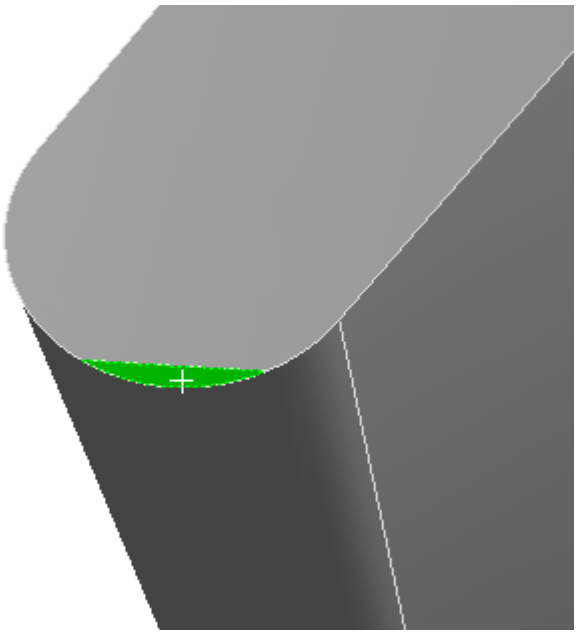
Group	Control	Value
Defaults	**Physics Preference**	Mechanical

23.6.2.2. Defining the Virtual Topology

The geometry has unnecessary small faces and edges, as shown in Figure 23.1: Small Face at the Leading Edge of the Blade (p. 541), which can influence the mapping of the mesh. Creating a virtual topology of virtual faces and edges will fix the problem by letting the Mechanical application ignore these geometry flaws.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.1: Small Face at the Leading Edge of the Blade

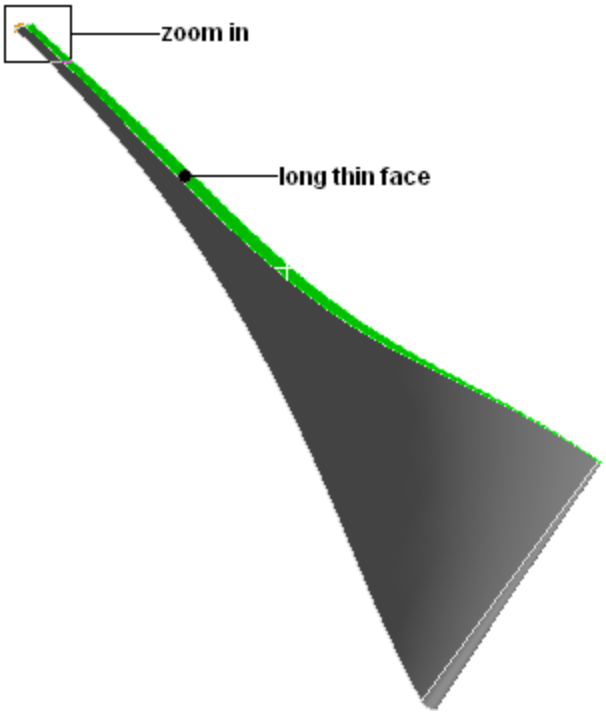


1. In the **Outline** tree view, right-click `Project` > `Model (C4)` and select **Insert** > **Virtual Topology**.
2. Right-click `Project` > `Model (C4)` > `Virtual Topology` and select **Generate Virtual Cells**.

   This will automatically create most of the virtual cells for you. You will have to make the rest of the virtual cells manually.
3. In the viewer, right-click and select **View** > **Back**.
4. Select the long thin face of the blade that is shown highlighted in green in Figure 23.2: Back View of the Blade (p. 542).
5. Zoom in on the trailing edge as shown in Figure 23.2: Back View of the Blade (p. 542).

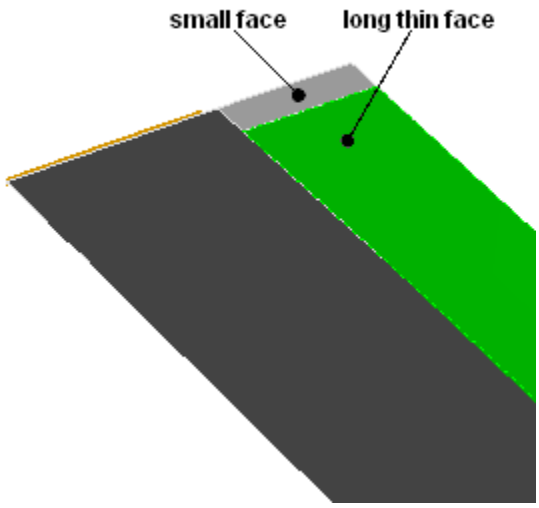
Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.2: Back View of the Blade



6. Hold **Ctrl** and select the small face at the trailing edge beside the long thin face shown in Figure 23.3: Small Face at the Trailing Edge of the Blade (p. 542).

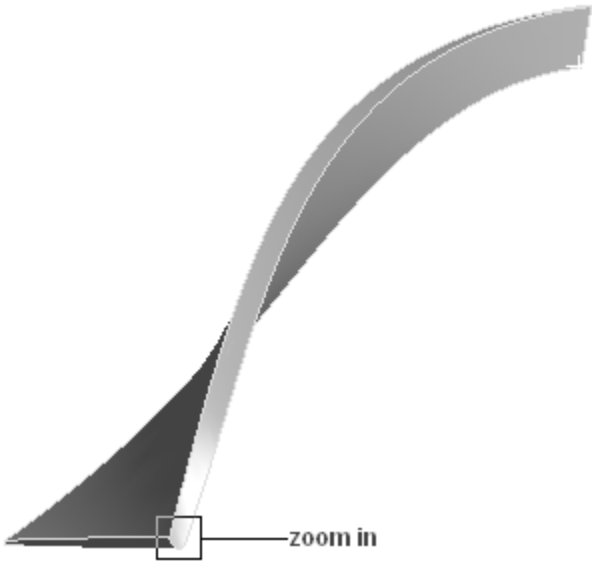
Figure 23.3: Small Face at the Trailing Edge of the Blade



7. In the viewer, right-click and select **Insert** > **Virtual Cell**.
8. In the viewer, right-click and select **View** > **Bottom**.
9. Zoom in on the leading edge as shown in Figure 23.4: Bottom View of the Blade (p. 543).

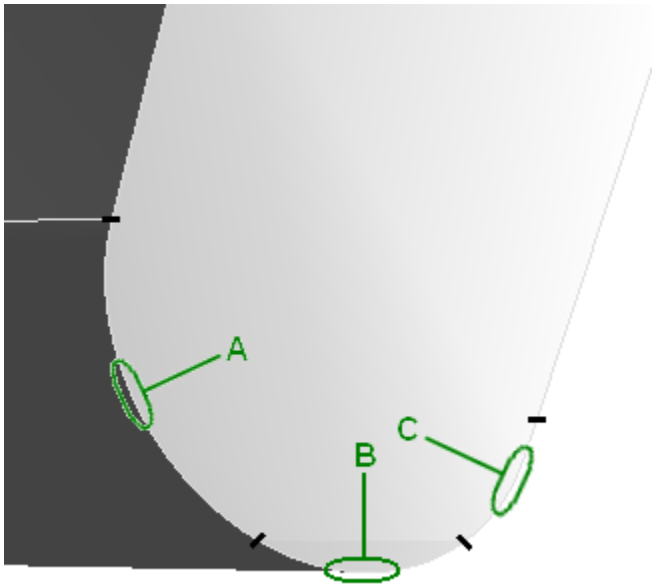
Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.4: Bottom View of the Blade



10. Hold **Ctrl** and select the three edges marked "A", "B" and "C" in Figure 23.5: Three Edges at the Leading Edge of the Blade (p. 543).

Figure 23.5: Three Edges at the Leading Edge of the Blade



11. In the viewer, right-click and select **Insert** > **Virtual Cell**.

23.6.2.3. Specifying the Sizing Controls

You will now apply local mesh sizing on four edges at the blade tip.

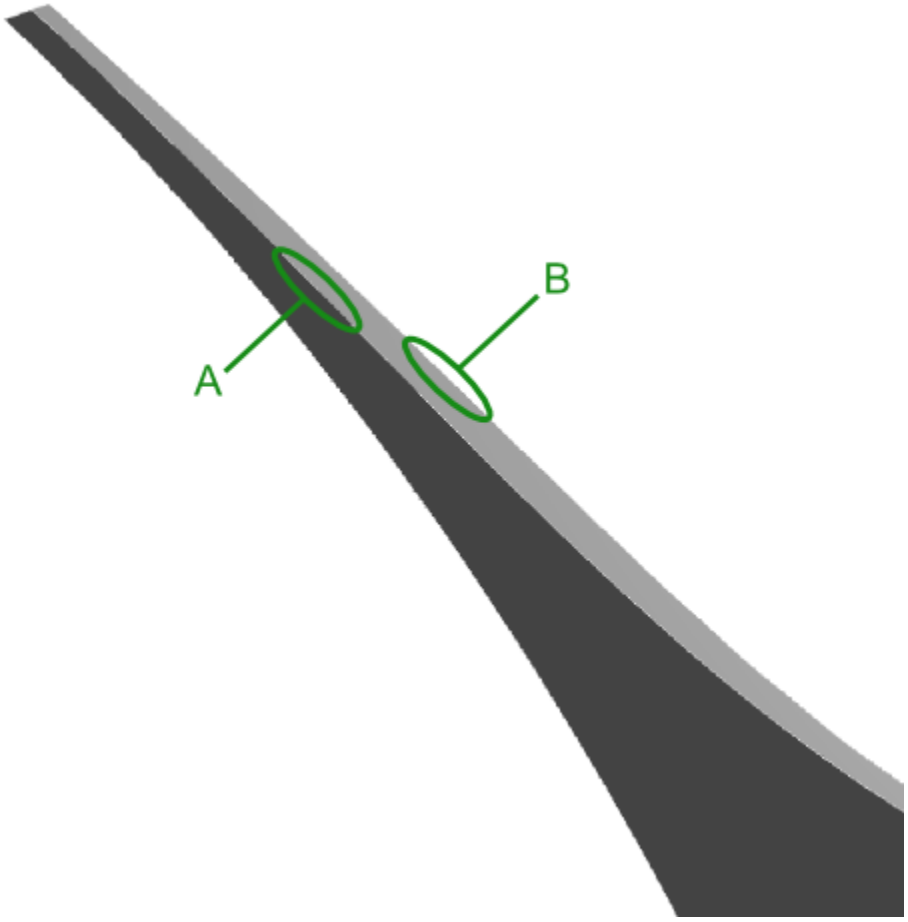
Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the **Outline** tree view, right-click `Project` > `Model (C4)` > `Mesh` and select **Insert** > **Sizing**.

2. In the viewer, right-click and select **View** > **Back** so that the model is oriented similar to the one shown in Figure 23.2: Back View of the Blade (p. 542).

3. Hold **Ctrl** and select the two edges marked "A" and "B" in Figure 23.6: Two Edges on the Back of the Blade (p. 544).

   Figure 23.6: Two Edges on the Back of the Blade



4. In the details view, click **Apply** in the field beside **Scope** > **Geometry**.

   The field beside **Scope** > **Geometry** should now display `2 Edges`. You have now chosen the two edges on which to apply the Sizing control.

5. Configure the following setting(s):

Setting	Value
Definition > **Type**	 Number of Divisions
Definition	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
> **Number of Divisions**	55
Definition > **Behavior**	Hard [a]
Definition > **Bias Type** [b]	_ __ ____ __ _
Definition > **Bias Factor** [c]	10

a. Choosing the **Hard** option ensures that the number of divisions and bias are fixed on the edge and cannot be changed by the meshing algorithm.

b. **Bias Type** adjusts the spacing ratio of nodes on an edge.

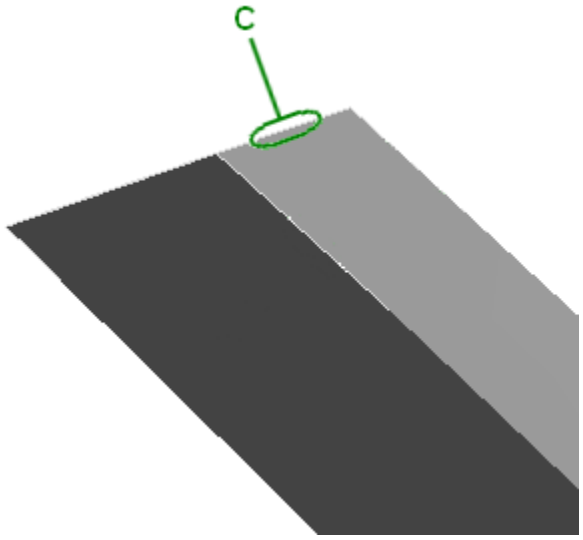
c. **Bias Factor** adjusts the ratio of the largest edge division to the smallest edge division.

6. In the **Outline** tree view, right-click `Project` > `Model (C4)` > `Mesh` and select **Insert** > **Sizing**.

7. Zoom in on the trailing edge as indicated in Figure 23.2: Back View of the Blade (p. 542).

8. Select the edge marked "C" in Figure 23.7: Edge on the Trailing Edge of the Blade (p. 545).

Figure 23.7: Edge on the Trailing Edge of the Blade



9. In the details view, click **Apply** in the field beside **Scope** > **Geometry**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

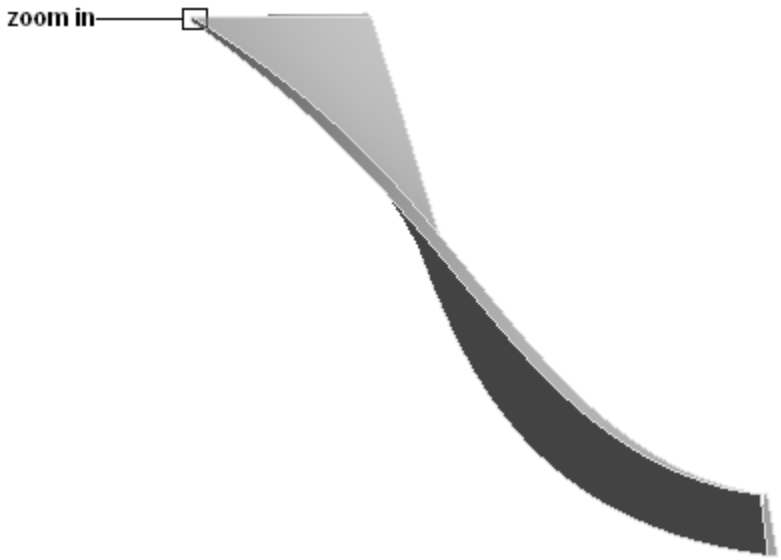
The field beside **Scope** > **Geometry** should now display `1 Edge`.

10. Configure the following setting(s):

Setting	Value
Definition > **Type**	Number of Divisions
Definition > **Number of Divisions**	3
Definition > **Behavior**	Hard
Definition > **Bias Type**	No Bias

11. In the **Outline** tree view, right-click `Project` > `Model (C4)` > `Mesh` and select **Insert** > **Sizing**.
12. In the viewer, right-click and select **View** > **Top**.
13. Zoom in on the leading edge as shown in Figure 23.8: Top View of the Blade (p. 546).

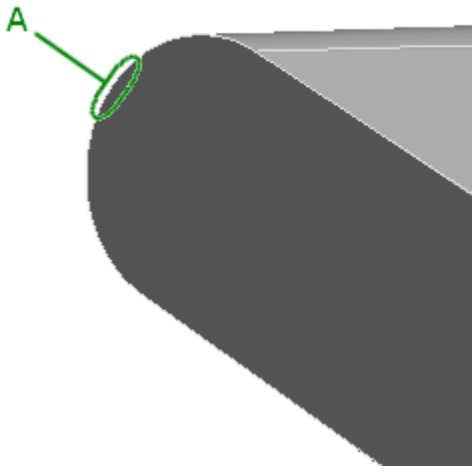
Figure 23.8: Top View of the Blade



14. Select the edge marked "A" in Figure 23.9: Leading Edge of the Blade (p. 547).

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.9: Leading Edge of the Blade



15. In the details view, click **Apply** in the field beside **Scope** > **Geometry**.

    The field beside **Scope** > **Geometry** should now display `1 Edge`.

16. Configure the following setting(s):

Setting	Value
Definition > **Type**	Number of Divisions
Definition > **Number of Divisions**	3
Definition > **Behavior**	Hard
Definition > **Bias Type**	_____ __ _ __ _____
Definition > **Bias Factor**	3

23.6.2.4. Specifying the Face Meshing Controls

You will now introduce a Face Meshing control on the blade tip that will generate a structured mesh based on the Sizing controls that you just defined.

1. In the **Outline** tree view, right-click `Project` > `Model (C4)` > `Mesh` and select **Insert** > **Face Meshing**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Select the virtual face at the top of the blade where you just defined the Sizing controls (the face that is at the forefront of the displayed geometry in Figure 23.9: Leading Edge of the Blade (p. 547)).

3. In the details view, click **Apply** in the field beside **Scope** > **Geometry**.

   The field beside **Scope** > **Geometry** should now display `1 Face`. This is the face at the blade tip.

23.6.2.5. Specifying the Method Controls

1. In the **Outline** tree view, right-click `Project` > `Model (C4)` > `Mesh` and select **Show** > **Sweepable Bodies**.

   This will select the sweepable bodies in the viewer; in this case, it is the whole blade geometry.

2. Right-click `Project` > `Model (C4)` > `Mesh` and select **Insert** > **Method**.

3. In the details view, set **Definition** > **Method** to `Sweep`.

4. Set **Definition** > **Src/Trg Selection** to `Manual Source and Target`.

5. In the details view, click the field beside **Definition** > **Source** where it says `No Selection`.

   Two buttons appear in the field: **Apply** and **Cancel**.

6. Select the virtual face at the top of the blade where you just defined the Face Meshing control (the face that is at the forefront of the displayed geometry in Figure 23.9: Leading Edge of the Blade (p. 547)).

   In the details view, click **Apply** in the field beside **Definition** > **Source**.

   The face has been selected as the source for the sweep method.

7. Follow a similar procedure for **Definition** > **Target** to set the bottom virtual face of the blade as the target for the sweep method (the face that is attached to the hub and opposite to the virtual face at the top of the blade).

8. Configure the following setting(s):

Setting	Value
Definition > **Type**	Number of Divisions
Definition > **Sweep Num Divs**	15
Definition > **Sweep Bias Type**	_ __ ____ __ _
Definition	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
> **Sweep Bias** [a]	10
a. This is the same as **Bias Factor**.	

23.6.2.6. Generating the Mesh

1. In the **Outline** tree view, right-click `Project` > `Model (C4)` > `Mesh` and select **Update**.

   After a few moments, the mesh is generated. You will now observe the mesh statistics.

2. Click `Project` > `Model (C4)` > `Mesh`.

3. In the details view, expand **Statistics**.

   Note the node and element count.

   Note:

   To get better results, the mesh would require more number of elements especially through the thickness of the blade; however, the current mesh is sufficient for the purpose of the tutorial.

4. Set **Mesh Metric** to any option and observe the mesh metric bar graph that appears.

5. Quit the Mechanical application.

   To do this, select the **File** tab and select **Close Mechanical**.

6. Return to the **Project Schematic** view.

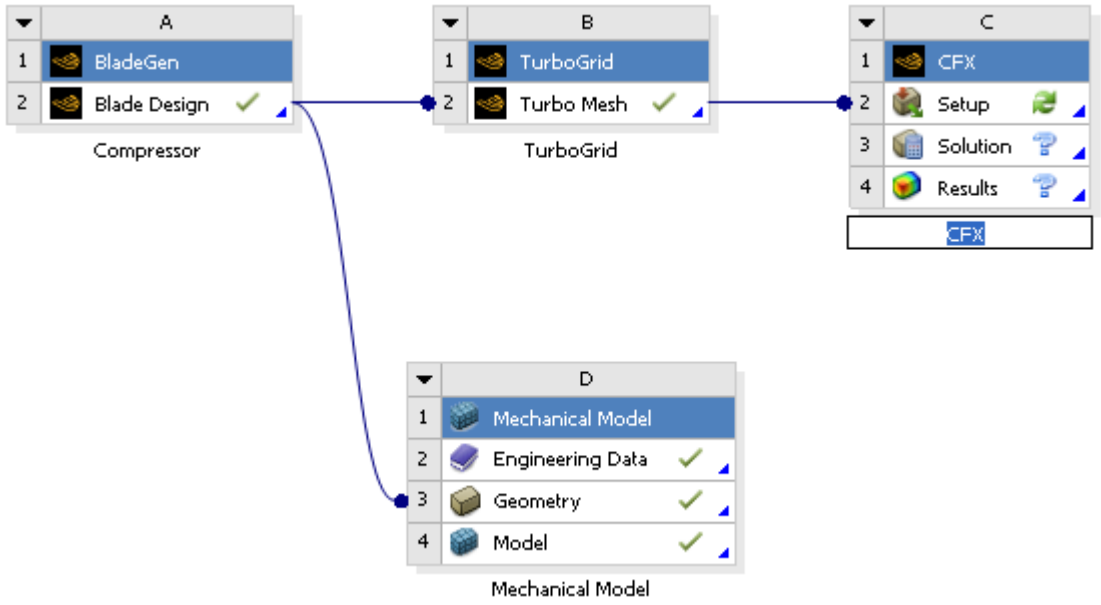
23.7. Defining the Case Using CFX-Pre

This section involves using CFX-Pre in Ansys Workbench. CFX-Pre is a CFD physics preprocessor that has a Turbo mode facility for setting up turbomachinery CFD simulations. In this section, you will use CFX-Pre in Turbo mode to define a CFD simulation based on the centrifugal compressor mesh that you created earlier. Later in General mode, you will create a solid domain for the blade and make a fluid-solid interface.

1. In the **TurboGrid** system, right-click the **Turbo Mesh** cell and select **Transfer Data To New** > **CFX**.

   A **CFX** system opens and is ready to be given a name.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Press **Enter** to accept the default name.
3. Drag the **Model** cell of the **Mechanical Model** system to the **Setup** cell of the **CFX** system.
4. In the **Mechanical Model** system, right-click the **Model** cell and select **Update**.

   The **Mesh** cell now displays a green check mark to indicate that the cell is up to date. This means that the blade mesh is available for use in the CFD simulation. The **Setup** cell displays a pair of green curved arrows to indicate that the cell has not received the latest upstream data. Since you have not yet opened CFX-Pre, you can disregard the cell status; CFX-Pre will automatically read the upstream cell data upon starting up from this cell for the first time.
5. In the **CFX** system, right-click the **Setup** cell and select **Edit**.

   CFX-Pre opens.

The next several sections guide you through the steps to create the CFD simulation.

23.7.1. Defining the Fluid Region Using Turbo Mode

You will define the fluid domain, boundaries and interfaces in Turbo mode.

In CFX-Pre, in the main menu, select **Tools** > **Turbo Mode**.

23.7.1.1. Configuring the Basic Settings

1. In the **Basic Settings** panel, configure the following setting(s):

Setting	Value
Machine Type	Centrifugal Compressor
Axes	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
> **Rotation Axis**	Z
Analysis Type > **Type**	 Steady State

Leave the other settings at their default values.

2. Click **Next**.

23.7.1.2. Defining the Components

1. In the **Component Definition** panel, right-click `Components` and select **Add Component**.
2. In the **New Component** dialog box, set **Name** to `R1` and **Type** to `Rotating`.

> **Note:**
>
> If the **Automatic Default Domain** option in CFX-Pre is selected, a domain with the name R1 will already be present under the **Component Definition** panel. In this case, you do not need to create a new domain, and you can continue by editing the existing R1 component

3. Click **OK**.
4. Select `Components` > `R1` and configure the following setting(s):

Setting	Value
Component Type > **Type**	 Rotating
Component Type > **Value**	 22360 [rev min^-1]
Mesh > Available Volumes > **Volumes**	 Inlet, Outlet, Passage Main
Wall Configuration	(Selected)
Wall Configuration > Tip Clearance at Shroud > **Yes**	 (Selected)

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Leave the other settings at their default values.

Note:

When a component is defined, Turbo mode will automatically select a list of regions that correspond to certain boundary types. This information should be reviewed under **Region Information** to ensure that all is correct.

5. Click **Next**.

23.7.1.3. Defining the Physics

You will now set the properties of the fluid domain and some solver parameters.

1. In the **Physics Definition** panel, configure the following setting(s):

Setting	Value
Fluid	Air Ideal Gas
Model Data > **Reference Pressure**	 1 [atm]
Inflow/Outflow Boundary Templates > **P-Total Inlet Mass Flow Outlet**	 (Selected)
Inflow/Outflow Boundary Templates > Inflow > **P-Total**	 0 [atm]
Inflow/Outflow Boundary Templates > Inflow > **T-Total**	 20 [C]
Inflow/Outflow Boundary Templates > Inflow > **Flow Direction**	 Cylindrical Components
Inflow/Outflow Boundary Templates > Inflow > **Inflow Direction (a,r,t)**	 1, 0, 0

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
Inflow/Outflow Boundary Templates > Outflow > **Mass Flow**	 Per Component
Inflow/Outflow Boundary Templates > Outflow > **Mass Flow Rate**	 0.167 [kg s^-1]
Solver Parameters	(Selected)
Solver Parameters > **Convergence Control**	 Physical Timescale
Solver Parameters > **Physical Timescale**	 0.0002 [s]

Leave the other settings at their default values.

2. Click **Next**.

23.7.1.4. Specifying the Domain Interfaces

CFX-Pre will try to create appropriate interfaces using the region names presented previously under **Region Information** in the **Component Definition** panel.

1. In the **Interface Definition** panel, verify that each interface is set correctly; select an interface listed in the tree view and then examine the associated settings (shown in the lower portion of the panel) and highlighted regions in the viewer.

   If no regions appear highlighted in the viewer, ensure that highlighting is turned on in the viewer toolbar.

2. Click **Next**.

23.7.1.5. Specifying the Boundaries

CFX-Pre will try to create appropriate boundary conditions using the region names presented previously under **Region Information** in the **Component Definition** panel.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the **Boundary Definition** panel, verify that each boundary is set correctly; select a boundary listed in the tree view and then examine the associated settings and highlighted regions.

   Note:

   Later, you will delete `R1 Blade` boundary in General mode after creating a fluid-solid interface at the blade.

2. Click **Next**.

23.7.1.6. Setting the Final Operations

1. In the **Final Operations** panel, set **Operation** to `Enter General Mode`.
2. Click **Finish**.

23.7.2. Defining the Solid Region Using General Mode

You will need to create a solid domain for the blade followed by making a fluid-solid interface in General mode.

23.7.2.1. Specifying the Domains

1. In the **Outline** tree view, right-click `Simulation` > `Flow Analysis 1` and select **Insert** > **Domain**.
2. In the **Insert Domain** dialog box, set **Name** to `Solid Blade`.
3. Click **OK**.
4. Configure the following setting(s) of `Solid Blade`:

Tab	Setting	Value
Basic Settings	Location and Type > **Location**	B24 [a]
	Location and Type > **Domain Type**	Solid Domain
	Solid Definitions	Solid 1
	Solid Definitions > Solid 1 > **Material**	Steel
	Domain Models	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Domain Motion > **Option**	 Rotating
	Domain Models > Domain Motion > **Angular Velocity**	 22360 [rev min^-1]
	Domain Models > Domain Motion > Axis Definition > **Option**	 Coordinate Axis
	Domain Models > Domain Motion > Axis Definition > **Rotation Axis**	 Global Z
Solid Models	Heat Transfer > **Option**	 Thermal Energy
a. This is the mesh region that you generated using the Mechanical application.		

5. Click **OK**.

23.7.2.2. Specifying the Boundaries

You do not need to create any boundaries for the solid domain because the interfaces will make the boundaries for you. The only surface that must be specified is the bottom of the blade at the hub. You will review the default boundary because it will later represent the bottom of the blade after all the interfaces have been specified.

1. In the **Outline** tree view, right-click `Simulation` > `Flow Analysis 1` > `Solid Blade` > `Solid Blade Default` and select **Edit**.

2. Configure the following setting(s) of `Solid Blade Default`:

Tab	Setting	Value
Basic Settings	**Boundary Type**	Wall

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Boundary Details	Heat Transfer > **Option**	 Adiabatic

3. Click **OK**.

23.7.2.3. Specifying the Domain Interfaces

An interface is required between the two meshes at the blade surface.

1. In the **Outline** tree view, right-click `Simulation` > `Flow Analysis 1` and select **Insert** > **Domain Interface**.
2. In the **Insert Domain Interface** dialog box, set **Name** to `R1 to Solid Blade`.
3. Click **OK**.
4. Configure the following setting(s) of `R1 to Solid Blade`:

Tab	Setting	Value
Basic Settings	**Interface Type**	Fluid Solid
	Interface Side 1 > **Domain (Filter)**	 R1
	Interface Side 1 > **Region List**	 BLADE
	Interface Side 2 > **Domain (Filter)**	 Solid Blade
	Interface Side 2 > **Region List**	 F26.24,F28.24,F30.24, F31.24,F49.24 [a]
	Interface Models > **Option**	 General Connection [b]
Mesh Connection	Mesh Connection Method > Mesh Connection > **Option**	 Automatic

a. The only region not selected is `F47.24`, which is the blade surface attached to the hub.

b. As long as the connection is general, the interface will enable heat transfer.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Click **OK**.

   A warning message appears notifying that the region for `R1 Blade` has already been specified as a boundary.

6. In the **Outline** tree view, right-click `Simulation` > `Flow Analysis 1` > `R1` > `R1 Blade` and select **Delete**.

   The warning message disappears.

7. Quit CFX-Pre.

   To do this, in the main menu, select **File** > **Close CFX-Pre**.

8. Return to the **Project Schematic** view.

23.8. Obtaining the Solution Using CFX-Solver Manager

Generate a solution for the CFD simulation that you just prepared:

1. In the **CFX** system, right-click the **Solution** cell and select **Update**.

   After some time, a CFD solution will be generated. If the progress indicator is not visible, you can display it by clicking

or, to see detailed output, right-click the **Solution** cell and select **Display Monitors**.

2. After the solution has been generated, return to the **Project Schematic** view.

23.9. Viewing the Results Using CFD-Post

CFD-Post enables you to view the results in various ways, including tables, charts, and figures. You can present the results in the form of a report that can be viewed in CFD-Post, or exported for viewing in another application.

Create a report and examine some of the results as follows:

1. In the **CFX** system, right-click the **Results** cell and select **Edit**.

   CFD-Post opens.

2. In CFD-Post, select **File** > **Report** > **Report Templates**.

3. In the **Report Templates** dialog box, select **Centrifugal Compressor Rotor Report** and click **Load**.

4. In the **Outline** tree view, right-click `Report` > `Compressor Performance Results Table` and select **Edit**.

   This table presents measures of aerodynamic performance including required power and efficiencies.

5. Right-click `Report` > `Blade Loading Span 50` and select **Edit**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

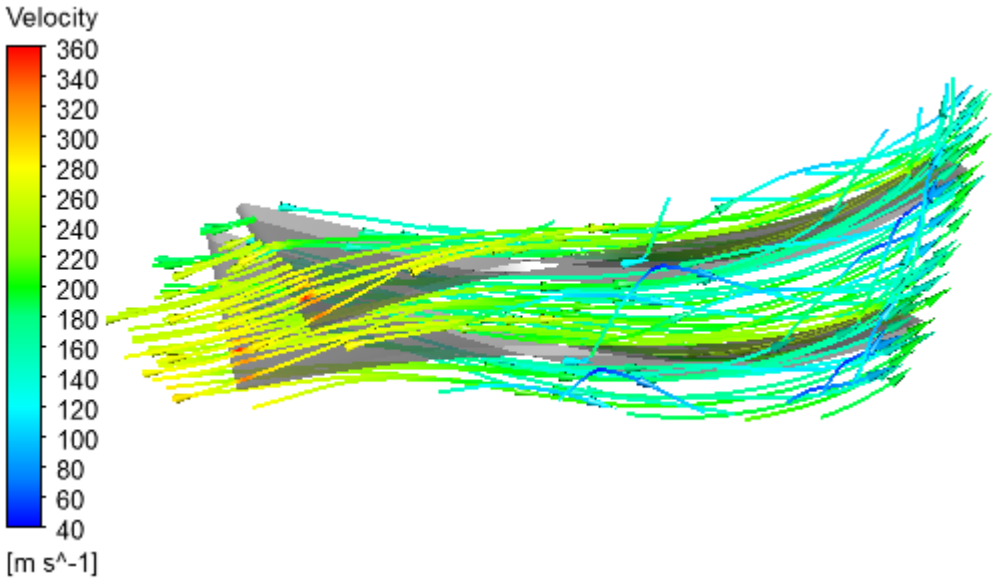
This is a plot of the pressure vs. streamwise distance along both the pressure and suction sides of the blade at mid-span.

6. Right-click `Report` > `Streamwise Plot of Pt and P` and select **Edit**.

   This is a plot of the streamwise variation of pressure and total pressure.

7. Right-click `Report` > `Velocity Streamlines Stream Blade TE View` and select **Edit**.

   This is a trailing-edge view of the streamlines that start upstream of the blade.



8. To view a full report, click the **Report Viewer** tab found near the bottom right of the window.

   A report will be generated that includes all figures available under `Report` in the tree view. This report can be viewed in CFD-Post or published to be viewed externally as an `.html` or `.txt` file.

   Note that if you have visited the **Report Viewer** before loading the template, or have otherwise made any changes to the report definition after first viewing the report, you need to click

   in the **Report Viewer** to update the report as displayed.

9. Quit CFD-Post.

   To do this, in the main menu, select **File** > **Close CFD-Post**.

10. Return to the **Project Schematic** view.

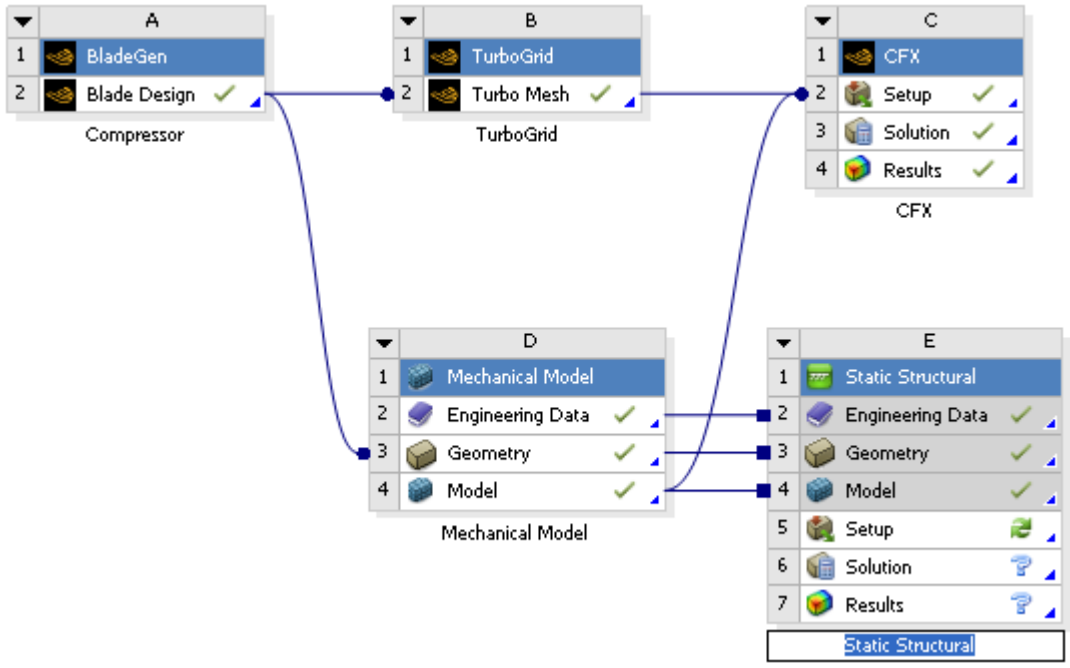
23.10. Simulating the Structural Performance Using Static Structural

This section describes the steps required to simulate structural stresses and deformation on the blade due to pressure and temperature loads from the aerodynamic analysis.

1. Expand **Analysis Systems** in the **Toolbox** view and drag **Static Structural** to the **Model** cell of the **Mechanical Model** system in the **Project Schematic** view.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the **Project Schematic** view, a **Static Structural** system opens and is ready to be given a name.



2. Press **Enter** to accept the default name.
3. Drag the **Solution** cell of the **CFX** system to the **Setup** cell of the **Static Structural** system.

   This makes the CFD solution data available for use in the structural analysis.
4. In the **Static Structural** system, right-click the **Setup** cell and select **Edit**.

   The Mechanical application opens.

The next several sections guide you through the steps to create a structural simulation with and without rotationally-induced inertial effects.

23.10.1. Simulating the Structural Performance without Rotational Velocity

This **Static Structural** system will be used to create the simulation without the inertial loads.

23.10.1.1. Importing the Loads

1. In the Mechanical application, in the **Outline** tree view, right-click `Project` > `Model (D4, E4)` > `Static Structural (D5)` > `Imported Load (C3)` and select **Insert** > **Pressure**.

   After a short time, `Project` > `Model (D4, E4)` > `Static Structural (D5)` > `Imported Load (C3)` > `Imported Pressure` will appear and will be selected.
2. In the details view, click the field beside **Scope** > **Geometry** where it says `No Selection`.
3. In the viewer, right-click and select **Select All**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Alternatively, in the main menu, select **Edit** > **Select All**.

4. In the viewer, right-click and select **View** > **Bottom**.

5. Hold **Ctrl** and select the long thin face of the blade that is at the forefront of the displayed geometry.

   This is the surface that is attached to the hub.

6. In the details view, click **Apply** in the field beside **Scope** > **Geometry**.

   The field beside **Scope** > **Geometry** should now display `5 Faces`. You have now chosen the solid model surface onto which the CFD pressure data will be applied.

7. Set **Transfer Definition** > **CFD Surface** to `R1 to Solid Blade Side 1`.

   You have now chosen the CFD boundary from which to get the CFD pressure data. You will do the same for the body temperature.

8. In the **Outline** tree view, right-click `Project` > `Model (D4, E4)` > `Static Structural (D5)` > `Imported Load (C3)` and select **Insert** > **Body Temperature**.

9. In the details view, click the field beside **Scope** > **Geometry** where it says `No Selection`.

10. In the viewer, right-click and select **Select All**.

11. In the details view, click **Apply** in the field beside **Scope** > **Geometry**.

    The field beside **Scope** > **Geometry** should now display `1 Body`. You have now chosen the solid model onto which the CFD temperature data will be applied.

12. Set **Transfer Definition** > **CFD Domain** to `Solid Blade`.

    You have now chosen the CFD domain from which to get the CFD temperature data.

13. In the **Outline** tree view, right-click `Project` > `Model (D4, E4)` > `Static Structural (D5)` > `Imported Load (C3)` and select **Import Load**.

    Wait for the Mechanical application to map the loads.

14. To verify that the pressure and temperature data were applied correctly to the blade, inspect `Imported Pressure` > `Imported Load Transfer Summary` and `Imported Body Temperature` > `Imported Load Transfer Summary` under `Project` > `Model (D4, E4)` > `Static Structural (D5)` > `Imported Load (C3)`.

    The slight discrepancy shown in the load transfer summary for the imported pressure is due to a difference in the TurboGrid and Mechanical mesh. In general, accurate load mapping requires that the surface mesh elements match.

23.10.1.2. Specifying the Supports

1. In the **Outline** tree view, right-click `Project` > `Model (D4, E4)` > `Static Structural (D5)` and select **Insert** > **Fixed Support**.

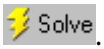
Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. In the viewer, right-click and select **View** > **Bottom**.
3. Select the long thin face of the blade that is at the forefront of the displayed geometry.
4. In the details view, click **Apply** in the field beside **Scope** > **Geometry**.

   The face that you selected is now connected to the hub.

23.10.1.3. Obtaining the Solution

1. In the **Outline** tree view, right-click `Project` > `Model (D4, E4)` > `Static Structural (D5)` > `Solution (D6)` and select **Insert** > **Stress** > **Equivalent (von-Mises)**.
2. Right-click `Project` > `Model (D4, E4)` > `Static Structural (D5)` > `Solution (D6)` and select **Insert** > **Deformation** > **Total**.
3. Right-click `Project` > `Model (D4, E4)` > `Static Structural (D5)` > `Solution (D6)` and select **Solve**.

   Alternatively, in the toolbar, click 

.

   Wait for the solver to finish.
4. Select `Project` > `Model (D4, E4)` > `Static Structural (D5)` > `Solution (D6)` > `Equivalent Stress` to prepare to examine the von-Mises stress results.
5. In the **Graph** view, click *Play* to animate the physical deformation of the blade along with the associated von-Mises stress results.
6. In the **Outline** tree view, select `Project` > `Model (D4, E4)` > `Static Structural (D5)` > `Solution (D6)` > `Total Deformation` to prepare to examine the total deformation solution.
7. In the **Graph** view, click *Play* to animate the physical deformation.
8. Quit the Mechanical application.

   To do this, select the **File** tab and select **Close Mechanical**.
9. Return to the **Project Schematic** view.

23.10.2. Simulating the Structural Performance with Rotational Velocity

This section shows how to add inertial effects due to rotation. You will duplicate the existing **Static Structural** system and use the new system to solve for the simulation with rotationally-induced inertial effects.

1. Click the upper-left corner of the **Static Structural** system and select **Duplicate**.

   A dialog box asks if the upstream connections should be kept for the duplicate system.
2. In the **Ansys Workbench** dialog box, click **Yes**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The connections enable the new (duplicate) system to receive updates from upstream systems, for example, after changing the geometry in BladeGen.

3. A second **Static Structural** system appears.
4. Rename the newly created system from `Copy of Static Structural` to `With Rotation`.
5. In the new system, right-click the **Setup** cell and select **Edit**.

   The Mechanical application opens.

23.10.2.1. Specifying the Loads

You will now specify the rotational velocity as the inertial load.

1. In the Mechanical application, in the **Outline** tree view, right-click `Project` > `Model (D4, E4, F4)` > `Static Structural 2 (E5)` and select **Insert** > **Rotational Velocity**.
2. In the main menu, select **Units** > **RPM**.
3. In the details view, configure the following setting(s):

Setting	Value
Definition > **Define By**	Components
Definition > **Z Component**	22360 [a]
a. The value will become `22360 RPM (ramped)` when you finish entering the number.	

23.10.2.2. Obtaining the Solution

1. In the **Outline** tree view, right-click `Project` > `Model (D4, E4, F4)` > `Static Structural 2 (E5)` > `Solution (E6)` and select **Solve**.

   Wait for the solver to finish.
2. To animate the total deformation or equivalent stress of the blade, select the corresponding object (either `Total Deformation` or `Equivalent Stress`, respectively) under `Project` > `Model (D4, E4, F4)` > `Static Structural 2 (E5)` > `Solution (E6)` and click *Play* in the **Graph** view.
3. Quit the Mechanical application.

   To do this, select the **File** tab and select **Close Mechanical**.
4. Return to the **Project Schematic** view.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. In the main menu, select **File** > **Save**.

   Alternatively, in the toolbar, click *Save Project*

.

6. Quit Ansys Workbench.

   To do this, in the main menu, select **File** > **Exit**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 24: Axial Turbine Equilibrium and Non-Equilibrium Steam Predictions

This tutorial includes:

24.1. Tutorial Features

24.2. Overview of the Problem to Solve

24.3. Preparing the Working Directory

24.4. Simulating the Equilibrium Phase Change Case

24.5. Simulating the Non-equilibrium Phase Change Case

24.1. Tutorial Features

In this tutorial you will learn about:

- Selection of material properties from the International Association for the Properties of Water and Steam (IAPWS) database.
- Setting property table ranges.
- Setting up an equilibrium steam calculation.
- Reviewing solution variables particular to the equilibrium solution.
- Setting up a non-equilibrium steam calculation.
- Postprocessing features special to the non-equilibrium solution.
- Reviewing solution variables particular to the non-equilibrium solution.

Component	Feature	Details
CFX-Pre	User Mode	Turbo Wizard
	Machine Type	Axial Turbine
	Fluid Type	Homogeneous Binary Mixture (equilibrium solution)
		Multi-Fluid Model (non-equilibrium solution)
	Domain Type	Multiple Domain

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

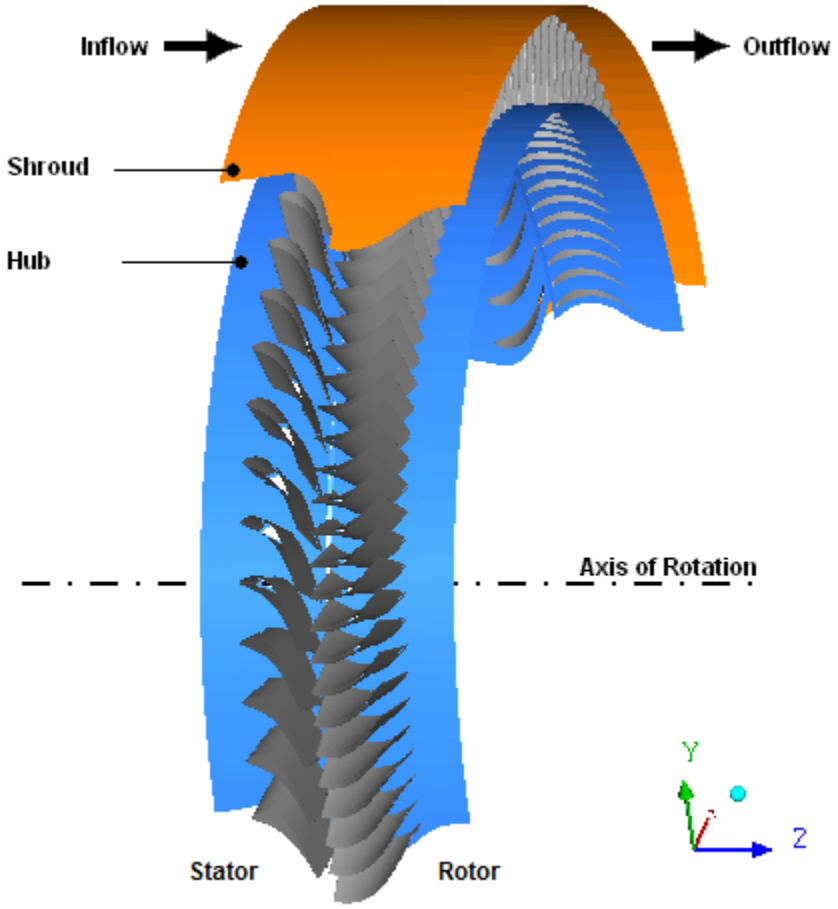
Component	Feature	Details
	Turbulence Model	k-Epsilon
	Heat Transfer	Total Energy (equilibrium solution)
		Fluid Dependent (non-equilibrium solution)
	Boundary Conditions	Inlet (subsonic): Total Pressure/Temperature and Mass Fraction(s) (equilibrium solution)
		Inlet (subsonic): Total Pressure/Temperature, Volume Fractions and Droplet Number (non-equilibrium solution)
		Outlet: Static Pressure
	Domain Interfaces	Frozen Rotor
		Periodic
	Timestep	Physical Time Scale
	Material Properties	IAPWS Water Database
	Fluid Pair Models	Small droplet heat transfer, small droplet phase change (non-equilibrium solution)
	Fluid Models	Nucleation, small droplet temperature, droplets with phase change (non-equilibrium solution)
CFD-Post	Location	Turbo Surface
	Plots	Contour

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.2. Overview of the Problem to Solve

The following tutorial uses an axial turbine to demonstrate setting up and executing equilibrium and non-equilibrium steam calculations using the IAPWS water database for properties.

The full geometry contains 60 stator blades and 113 rotor blades. The following figure shows approximately half of the full geometry. The inflow average velocity is in the order of 100 m/s.



The geometry to be modeled consists of a single stator blade passage and two rotor blade passages. This is an approximation to the full geometry since the ratio of rotor blades to stator blades is close to, but not exactly, 2:1. In the stator blade passage a 6° section is being modeled (360°/60 blades), while in the rotor blade passage, a 6.372° section is being modeled (2*360°/113 blades). This produces a pitch ratio at the interface between the stator and rotor of 0.942. As the flow crosses the interface, it is scaled to enable this type of geometry to be modeled. This results in an approximation of the inflow to the rotor passage. Furthermore, the flow across the interface will not appear continuous due to the scaling applied.

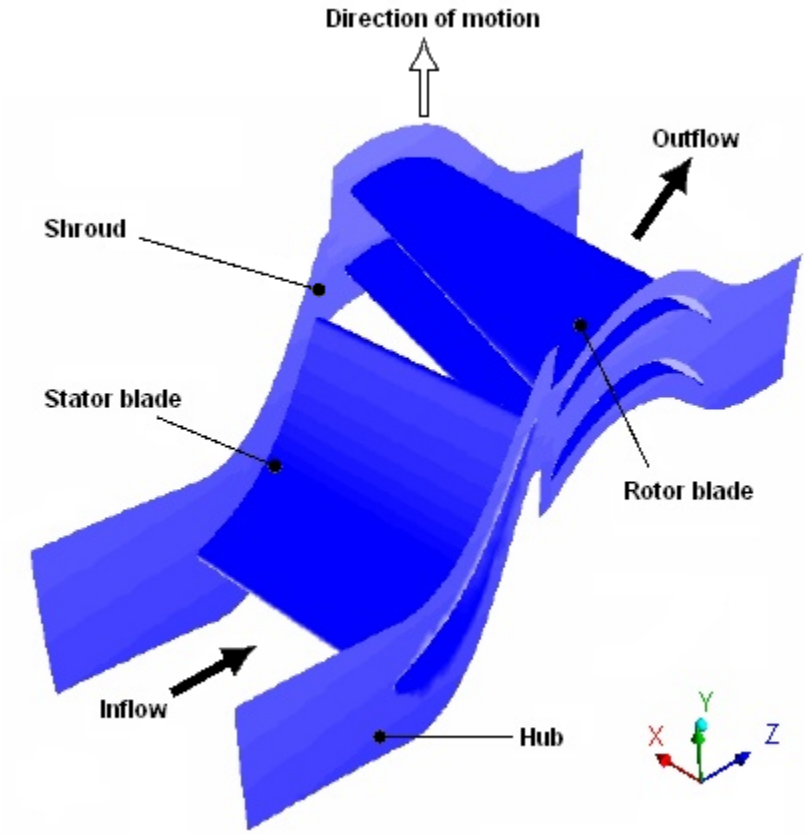
In this example, the rotor rotates at 523.6 rad/s about the Z axis while the stator is stationary. Periodic boundaries are used to enable only a small section of the full geometry to be modeled.

The important problem parameters are:

- Total inlet pressure = 0.265 bar

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Static outlet pressure = 0.0662 bar
- Total inlet temperature = 328.5 K



In this tutorial, you will generate two steady-state solutions: one using a multicomponent fluid consisting of a homogeneous binary mixture of liquid water and water vapor, the other using two separate phases to represent liquid water and water vapor in a non-equilibrium simulation. The solution variables particular to the equilibrium and non-equilibrium solutions will be processed in order to understand the differences between the two solutions.

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- Running Ansys CFX Tutorials Using Ansys Workbench (p. 34)
- Changing the Display Colors (p. 37)

24.3. Preparing the Working Directory

1. Create a working directory.

   Ansys CFX uses a working directory as the default location for loading and saving files for a particular session or project.

2. Download the `water_vapor.zip` file here.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Unzip `water_vapor.zip` to your working directory.

   Ensure that the following tutorial input files are in your working directory:

   - `stator.gtm`
   - `rotor.grd`

4. Set the working directory and start CFX-Pre.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

24.4. Simulating the Equilibrium Phase Change Case

In this section, you will simulate the equilibrium case where the fluid consists of a homogeneous binary mixture of liquid water and water vapor.

24.4.1. Defining the Case Using CFX-Pre

The following sections describe the equilibrium simulation setup in CFX-Pre.

This tutorial uses the Turbomachinery wizard in CFX-Pre. This preprocessing mode is designed to simplify the setup of turbomachinery simulations.

1. In CFX-Pre, select **File** > **New Case**.
2. Select **Turbomachinery** and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `WaterVaporEq`.
5. If you are notified the file already exists, click **Overwrite**.
6. Click **Save**.

24.4.1.1. Basic Settings

1. In the **Basic Settings** panel, configure the following setting(s):

Setting	Value
Machine Type	Axial Turbine
Analysis Type > **Type**	 Steady State

   Leave the other settings at their default values.

2. Click **Next**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.4.1.2. Component Definition

Create the stator and rotor components as follows:

1. Right-click a blank area near the **Component Definition** tree and select **Add Component** from the shortcut menu.
2. Create a new component of type `Stationary`, named `S1`.
3. Beside **Mesh** > **File**, click *Browse* .

   The **Import Mesh** dialog box appears.
4. Configure the following setting(s):

Setting	Value
File name	stator.gtm [a]
a. You may have to select the `CFX Mesh (*gtm *cfx)` option under **Files of type** in order to see the file.	

5. Click **Open**.

   Note that when a component is defined, Turbo Mode attempts to configure the **Region Information** settings, which you can review and manually adjust as necessary.
6. Create a new component of type `Rotating`, named `R1`.
7. Set **Component Type** > **Value** to 523.6 [radian s^-1].
8. Beside **Mesh** > **File**, click *Browse* .

   The **Import Mesh** dialog box appears.
9. Configure the following setting(s):

Setting	Value
File name	rotor.grd [a]
Options > **Mesh Units**	 m
a. You may have to select the `CFX-TASCflow (*grd)` option under **Files of type** in order to see the file.	

10. Click **Open**.
11. Click **Passages and Alignment** > **Edit**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12. Configure the following setting(s):

Setting	Value
Passage and Alignment > Passages/Mesh > **Passages per Mesh**	2
Passage and Alignment > **Passages to Model**	2
Passage and Alignment > **Passages in 360**	113

13. Click **Passages and Alignment** > **Done**.

14. Click **Next**.

Note:

The components must be ordered as above (stator then rotor) in order for the interface to be created correctly. The order of the two components can be changed, if necessary, by right-clicking S1 and selecting **Move Component Up**.

24.4.1.3. Physics Definition

In this section, you will set properties of the fluid domain and some solver parameters. Note that initially you will choose the fluid to be `Water Ideal Gas`, but later you will create a new fluid based on the IAPWS database for water and override this initial setting with it.

1. In the **Physics Definition** panel, configure the following setting(s):

Setting	Value
Fluid	Water Ideal Gas
Model Data > **Reference Pressure**	0 [atm]
Model Data > **Heat Transfer**	Total Energy
Model Data > **Turbulence**	k-Epsilon
Inflow/Outflow Boundary Templates > **P-Total Inlet P-Static Outlet**	(Selected)
Inflow/Outflow Boundary Templates	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
> Inflow > **P-Total**	 0.265 [bar]
Inflow/Outflow Boundary Templates > Inflow > **T-Total**	 328.5 [K] [a]
Inflow/Outflow Boundary Templates > Inflow > **Flow Direction**	 Normal to Boundary
Inflow/Outflow Boundary Templates > Outflow > **P-Static**	 0.0662 [bar]
Interface > **Default Type**	 Frozen Rotor
Solver Parameters	(Selected)
Solver Parameters > **Convergence Control**	 Physical Timescale
Solver Parameters > **Physical Timescale**	 0.0005 [s] [b]
a. From the problem description. b. The physical timescale that will be set up is derived from the rotational speed of the 113 rotor blades.	

2. Click **Next**.

24.4.1.4. Interface Definition

CFX-Pre will try to create appropriate interfaces using the region names viewed previously in the **Region Information** section (in the **Component Definition** setup screen). In this case, you should see that a periodic interface has been generated for both the rotor and the stator. The generated periodic interface can be edited or deleted. Interfaces are required when modeling a small section of the true geometry. An interface is also needed to connect the two components together across the frame change.

1. Review the various interfaces but do not change them.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click **Next**.

24.4.1.5. Boundary Definition

CFX-Pre will try to create appropriate boundary conditions using the region names presented previously in the **Region Information** section. In this case, you should see a list of generated boundary conditions. They can be edited or deleted in the same way as the interface connections that were set up earlier.

1. Review the various boundary definitions but do not change them.

2. Click **Next**.

24.4.1.6. Final Operations

1. Set **Operation** to `Enter General Mode`.

2. Click **Finish**.

   After you click **Finish**, a dialog box appears stating that a Turbo report will not be included in the solver file because you are entering General mode.

3. Click **Yes** to continue.

24.4.1.7. Defining the Properties of Water

Earlier in the physics definition portion of the Turbomachinery wizard, you specified `Water Ideal Gas` as the fluid in the domain. Here, you will specify a homogeneous binary mixture to replace it. To create the mixture, you will take two pure fluids from the IAPWS database for water and combine them. The pure fluids that will be combined are `H2Og`, representing water vapor, and `H2Ol`, representing liquid water. The mixture will be named `H2Olg`.

The present simulations use the published IAPWS-IF97 (International Association for the Properties of Water and Steam - Industrial Formulation 1997) water tables for properties. The published IAPWS-IF97 equations have been implemented in Ansys CFX, enabling you to directly select them for use in your simulations. The present example uses these properties in a tabular format requiring you to specify the range of the properties (such as min/max pressure and temperature bounds) and the number of data points in each table. Note that the IAPWS-IF97 properties have been tested for extrapolation into metastable regions, a fact that will be used for the non-equilibrium calculations that require this kind of state information.

1. Click *Material*

2. Name the new material `H2Og`.

3. Enter the following settings for `H2Og`:

Tab	Setting	Value
Basic Settings	**Material Group**	IAPWS IF97

Tab	Setting	Value
	Thermodynamic State	(Selected)
	Thermodynamic State > **Thermodynamic State**	Gas
Material Properties	Thermodynamic Properties > **Table Generation**	(Selected)
	Thermodynamic Properties > Table Generation > **Minimum Temperature**	(Selected)
	Thermodynamic Properties > Table Generation > Minimum Temperature > **Min. Temperature**	250 [K] [a]
	Thermodynamic Properties > Table Generation > **Maximum Temperature**	(Selected)
	Thermodynamic Properties > Table Generation > Maximum Temperature > **Max. Temperature**	400 [K]
	Thermodynamic Properties > Table Generation > **Minimum Absolute Pressure**	(Selected)
	Thermodynamic Properties > Table Generation > Minimum Absolute Pressure > **Min. Absolute Pres.**	0.01 [bar]
	Thermodynamic Properties > Table Generation > **Maximum Absolute Pressure**	(Selected)

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Thermodynamic Properties > Table Generation > Maximum Absolute Pressure > **Max. Absolute Pres.**	0.6 [bar]
	Thermodynamic Properties > Table Generation > **Maximum Points**	(Selected)
	Thermodynamic Properties > Table Generation > Maximum Points > **Maximum Points**	100
a. The H2Og minimum temperature is set to 250 K because the vapor might possibly be supercooled (in the NES calculations) to temperatures lower than the triple point temperature.		

4. Click **OK**.
5. In the **Outline** tree, under `Materials`, right-click `H2Og` and select **Duplicate**.
6. Rename `Copy of H2Og` to `H2Ol` (using the letter "l" as in "liquid").
7. Open `H2Ol` for editing.
8. On the **Basic Settings** tab, change **Thermodynamic State** > **Thermodynamic State** from `Gas` to `Liquid`.
9. Click **OK**
10. Create a new material named `H2Olg`.
11. Enter the following settings for `H2Olg`:

Tab	Setting	Value
Basic Settings	**Options**	Homogeneous Binary Mixture
	Material Group	IAPWS IF97
	Material 1	H2Og
	Material 2	H2Ol
Saturation Properties	**Option**	IAPWS Library

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Table Generation	(Selected)
	Table Generation > **Minimum Temperature**	(Selected)
	Table Generation > Minimum Temperature > **Min. Temperature**	273.15 [K] [a]
	Table Generation > **Maximum Temperature**	(Selected)
	Table Generation > Maximum Temperature > **Max. Temperature**	400 [K]
	Table Generation > **Minimum Absolute Pressure**	(Selected)
	Table Generation > Minimum Absolute Pressure > **Min. Absolute Pres.**	0.01 [bar]
	Table Generation > **Maximum Absolute Pressure**	(Selected)
	Table Generation > Maximum Absolute Pressure > **Max. Absolute Pres.**	0.6 [bar]
	Table Generation > **Maximum Points**	(Selected)
	Thermodynamic Properties > Table Generation > Maximum Points > **Maximum Points**	100

a. The minimum temperature is set to 273.15 K due to the fact that the saturation properties implied by H2Ogl are not likely to be used below the triple point temperature.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12. Click **OK**.

24.4.1.8. Modifications to Domain and Boundary Conditions

You now need to update the initial setting for the domain fluid (initially set while in the Turbomachinery wizard) with the new homogeneous binary mixture fluid that you have just created. This mixture (H2Olg) acts as a container fluid identifying two child materials, H2Ol and H2Og, each representing the liquid and vapor properties in the pure fluid system. The equilibrium solution uses the binary mixture fluid, H2Olg, and assumes that equilibrium conditions relate H2Ol and H2Og at all times. In the non-equilibrium solution (in the second part of this tutorial), H2Ol and H2Og are each used separately to define the fluids that are active in the domain; this is a requirement since the equilibrium constraint is no longer applicable in that case.

1. Open domain `R1` for editing.
2. On the **Basic Settings** tab under **Fluid and Particle Definitions**, delete any existing items by selecting them and clicking *Remove selected item* .
3. Click *Add new item* .
4. Set **Name** to `H2Olg` and click **OK**.
5. Set **Fluid and Particle Definitions** > **H2Olg** > **Material** to `H2Olg` and click **Apply**.
6. On the **Fluid Models** tab set **Component Models** > **Component** > **H2Og** > **Option** to `Equilibrium Fraction`.
7. Click **OK**.
8. Open `Simulation` > `Flow Analysis 1` > `S1` > `S1 Inlet` for editing.
9. On the **Boundary Details** tab, set **Component Details** > **H2Og** > **Mass Fraction** to `1.0`.
10. Click **OK**.

24.4.1.9. Setting Initial Values

1. Click *Global Initialization* .
2. Enter the following settings:

Tab	Setting	Value
Global Settings	Initial Conditions > Cartesian Velocity Components > **Option**	Automatic with Value
	Initial Conditions	

Tab	Setting	Value
	> Cartesian Velocity Components > **U**	0 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > **V**	0 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > **W**	100 [m s^-1] [a]
	Initial Conditions > Static Pressure > **Option**	Automatic with Value
	Initial Conditions > Static Pressure > **Relative Pressure**	0.2 [bar]
	Initial Conditions > Temperature > **Option**	Automatic with Value
	Initial Conditions > Temperature > **Temperature**	328.5 [K] [a]
	Initial Conditions > Component Details > H2Og > **Option**	Automatic with Value
	Initial Conditions > Component Details > H2Og > **Mass Fraction**	1.0

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
a. From the problem description.		

3. Click **OK**.

24.4.1.10. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .

2. Configure the following setting(s):

Setting	Value
File name	WaterVaporEq.def

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, **Solver Input File** is set.

4. Quit CFX-Pre, saving the simulation (`.cfx`) file.

24.4.2. Obtaining the Solution Using CFX-Solver Manager

When CFX-Pre has shut down, and CFX-Solver Manager has started, obtain a solution to the CFD problem by following the instructions below.

1. In CFX-Solver Manager, click **Start Run**.

2. At the end of the run, on the completion message that appears, select **Post-Process Results**.

3. If using stand-alone mode, select **Shut down CFX-Solver Manager**.

4. Click **OK**.

24.4.3. Viewing the Results Using CFD-Post

The equilibrium case produces solution variables unique to these model settings. The most important ones are static pressure, mass fraction and temperature, which will be briefly described here.

1. When CFD-Post starts, the **Domain Selector** dialog box might appear. If it does, ensure that both the R1 and S1 domains are selected, then click **OK** to load the results from these domains.

2. Click the **Turbo** tab.

3. The **Turbo Initialization** dialog box is displayed, and asks you whether you want to auto-initialize all components. Click **Yes**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Turbo** tree view shows the two components in domains R1 and S1. In this case, the initialization works without problems. If there were any problems initializing a component, this would be indicated in the tree view.

Note:

If you do not see the **Turbo Initialization** dialog box, or as an alternative to using that dialog box, you can initialize all components by clicking the **Initialize All Components** button, which is visible initially by default, or after double-clicking the **Initialization** object in the **Turbo** tree view.

24.4.3.1. Specifying Locators for Plots

Make a 2D surface at 50% span to be used as a locator for plots:

1. On the main menu select **Insert** > **Location** > **Turbo Surface** and name it `Turbo Surface 1`.
2. On the **Geometry** tab, set **Method** to `Constant Span` and **Value** to `0.5`.
3. Click **Apply**.
4. Turn off the visibility of `Turbo Surface 1`.

24.4.3.2. Static Pressure and Mass Fraction Contour Plots

In the equilibrium solution, phase transition occurs the moment saturation conditions are reached in the flow. The amount of moisture ultimately created from phase transition is determined in the following manner.

The static pressure solution field yields the saturation enthalpy through the function $h_{\text{sat}}(P)$, available from the IAPWS database. During the solution, if the predicted mixture static enthalpy, h, falls below $h_{\text{sat}}(P)$ at any point, the mass fraction of the condensed phase (also called *wetness*) can be directly calculated also based on the IAPWS properties. The degree to which wetness is generated depends on the amount by which h is less than $h_{\text{sat}}(P)$.

Note that the mass fraction is predicted at each iteration in the solution, and is used to update other two-phase mixture properties required at each step in the solution. The final results therefore include all of the two-phase influences, but assuming equilibrium conditions. The equilibrium solution mass fraction contour plot will be compared to the non-equilibrium solution one in Viewing the Results Using CFD-Post (p. 588).

In the next step, you are asked to create the pressure and mass fraction contour plots.

1. Create a static pressure contour plot on `Turbo Surface 1`:
   a. Create a new contour plot named `Static Pressure`.
   b. In the details view on the **Geometry** tab, set **Locations** to `Turbo Surface 1` and **Variable** to `Pressure`, then click **Apply**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Turn off the visibility of `Static Pressure` when you have finished observing the results.
3. Create a contour plot on `Turbo Surface 1` that shows the mass fraction of the liquid phase:
   a. Create a new contour plot named `Mass Fraction of Liquid Phase`.
   b. In the details view on the **Geometry** tab, set **Locations** to `Turbo Surface 1` and **Variable** to `H2Ol.Mass Fraction`, then click **Apply**.
4. Turn off the visibility of **Mass Fraction of Liquid Phase** when you have finished observing the results.

24.4.3.3. Static Temperature Contour Plots

In the equilibrium solution, the condensed and gas phases share the same temperature and, as a result, predictions of thermodynamic losses are not possible.

In the following step you are asked to view the temperature field in the solution, and note that, due to the equilibrium constraint, it represents conditions for the mixture.

1. Create a contour plot on `Turbo Surface 1` that shows the static temperature.
   a. Create a new contour plot named `Static Temperature`.
   b. In the details view on the **Geometry** tab, set **Locations** to `Turbo Surface 1` and **Variable** to `Temperature`, then click **Apply**.
2. Once you have observed the results save the state and exit CFD-Post.

24.5. Simulating the Non-equilibrium Phase Change Case

The non-equilibrium calculation introduces a number of additional transport equations to the equilibrium solution, namely volume fractions for each phase and droplet number for all condensing phases. In addition, energy equations need to be specified for each of the phases in the solution. The setup of these equations is automated based on model selections to be described subsequently. Important to the predictions are interphase heat and mass transfer between the vapor and condensed phases due to small droplets created by homogeneous nucleation. Selection of the required phase pair conditions is made easier by provision of special small droplet models (where small droplet implies droplet sizes generally below one μm). Phase transition is initiated based on predicted metastable state (measured by supercooling level) conditions in the flow in conjunction with a classical homogeneous nucleation model.

The non-equilibrium solution is therefore closely dependent on the evolving conditions along the flow path leading to phase transition and subsequent strong interaction (by heat/mass transfer) between phases.

24.5.1. Defining the Case Using CFX-Pre

1. Ensure that the following tutorial input file is in your working directory:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- `WaterVaporEq.cfx`

2. Set the working directory and start CFX-Pre if it is not already running.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

3. Select **File** > **Open Case**.
4. Select the simulation file `WaterVaporEq.cfx`.
5. Click **Open**.
6. Select **File** > **Save Case As**.
7. Change the name to `WaterVaporNonEq.cfx`.
8. Click **Save**.

24.5.1.1. Modifying the Domains

The non-equilibrium case is different from the equilibrium case in that the creation of the second phase (that is, the liquid water) is based on vapor supercooling in conjunction with fluid expansion rate and a nucleation model. The location where the phase transition happens is not specified, but evolves as part of the solution.

In this section, you will specify a nucleation model for the phase that is considered condensable.

You will also set the droplet number and volume fraction of the condensable phase to zero at the inlet. The condensable phase will appear within the domain by homogeneous nucleation. If this case were to involve wetness at the inlet, you would have a choice of specifying either droplet number or droplet diameter as a boundary along with the volume fraction.

1. Open domain `R1`.
2. On the **Basic Settings** tab under **Fluid and Particle Definitions**, delete any existing items by selecting them and clicking *Remove selected item* .
3. Create two new materials named `H2Og` and `H2Ol` by using the *Add new item* icon.
4. Configure the following setting(s) of domain `R1`:

Tab	Setting	Value
Basic Settings	**Fluid and Particle Definitions**	H2Og
	Fluid and Particle Definitions > H2Og > **Material**	 H2Og
	Fluid and Particle Definitions	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> H2Og > Morphology > **Option**	Continuous Fluid
	Fluid and Particle Definitions	H2Ol
	Fluid and Particle Definitions > H2Ol > **Material**	H2Ol
	Fluid and Particle Definitions > H2Ol > Morphology > **Option**	Droplets (Phase Change)
Fluid Models	Multiphase > **Homogeneous Model**	(Selected)
	Heat Transfer > **Homogeneous Model**	(Cleared)
	Heat Transfer > **Option**	Fluid Dependent
Fluid Specific Models	**Fluid**	H2Og
	Fluid > H2Og > Heat Transfer Model > **Option**	Total Energy
	Fluid	H2Ol
	Fluid > H2Ol > Heat Transfer Model > **Option**	Small Droplet Temperature
	Fluid > H2Ol	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Nucleation Model**	(Selected)
	Fluid > H2Ol > Nucleation Model > **Option**	Homogeneous
	Fluid > H2Ol > Nucleation Model > **Nucleation Bulk Tension Factor** [a]	(Selected)
	Fluid > H2Ol > Nucleation Model > Nucleation Bulk Tension Factor > **Nucleation Bulk Tension**	1.0
Fluid Pair Models	**Fluid Pair**	H2Og | H2Ol
	Fluid Pair > H2Og | H2Ol > Interphase Transfer > **Option**	Particle Model
	Fluid Pair > H2Og | H2Ol > Mass Transfer > **Option**	Phase Change
	Fluid Pair > H2Og | H2Ol > Mass Transfer > Phase Change Model > **Option**	Small Droplets
	Fluid Pair	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> H2Og | H2Ol > Heat Transfer > **Option**	 Small Droplets

a. The **Nucleation Bulk Tension Factor** scales the bulk surface tension values used in the nucleation model. Classical nucleation models are very sensitive to the bulk surface tension, and only slight adjustments will modify the nucleation rate quite significantly. It is common practice in CFD simulations to alter the bulk surface tension values slightly in order to bring results in-line with experiment. Studies suggest that with the IAPWS database and conditions less than 1 bar, a **Nucleation Bulk Tension Factor** of 1.0 is the best first setting.

Note:

The small droplet setting for H2Ol heat transfer implies that the temperature is algebraically determined as a function of the droplet diameter, which in turn is calculated from other solution variables such as H2Ol volume fraction and droplet number.

5. Click **OK**.

6. Apply the same settings to domain S1.

   Most of the settings will have been already copied from domain R1 to domain S1, however the nucleation model settings must be set explicitly.

7. Open `S1 Inlet` and enter the following settings:

Tab	Setting	Value
Boundary Details	Heat Transfer > **Option**	 Fluid Dependent
Fluid Values	**Boundary Conditions**	H2Og
	Boundary Conditions > H2Og > Heat Transfer > **Option**	 Total Temperature
	Boundary Conditions > H2Og > Heat Transfer > **Total Temperature**	 328.5 [K]

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Boundary Conditions > H2Og > Volume Fraction > **Option**	Value
	Boundary Conditions > H2Og > Volume Fraction > **Volume Fraction**	1.0
	Boundary Conditions	H2Ol
	Boundary Conditions > H2Ol > Volume Fraction > **Option**	Value
	Boundary Conditions > H2Ol > Volume Fraction > **Volume Fraction**	0
	Boundary Conditions > H2Ol > Droplet Number > **Option**	Specified Number [a]
	Boundary Conditions > H2Ol > Droplet Number > **Droplet Number**	0 [m^-3]
a. Not `Specified Diameter`.		

8. Click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9. Click *Solver Control* .

10. On the **Basic Settings** tab, set **Convergence Control** > **Fluid Timescale Control** > **Physical Timescale** to `5e-005 [s]` and click **OK**.

    Because the non-equilibrium simulation involves vapor and therefore tends to be unstable, it is recommended that you set the physical timescale to a relatively small value. The value set here was found to be suitable for this simulation by trial and error.

24.5.1.2. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .

2. Configure the following setting(s):

Setting	Value
File name	WaterVaporNonEq.def

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, **Solver Input File** is set.

4. Quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

24.5.2. Obtaining the Solution Using CFX-Solver Manager

The **Define Run** dialog box will be displayed when CFX-Solver Manager launches. **CFX-Solver Input File** will already be set to the name of the CFX-Solver input file just written.

1. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. At the end of the run, a dialog box is displayed stating that the simulation has ended.

   Note:

   You may notice messages in the solver output regarding problems with evaluating material properties. This is a result of the absolute pressure reaching values outside the range of internal material property tables. In this case, the messages are temporary and stop appearing well before convergence. If you encounter a simulation where the messages persist, or you otherwise suspect that the results might be adversely affected, you can change the ranges of internal material property tables by editing the relevant materials in CFX-Pre.

2. Select **Post-Process Results**.

3. If using stand-alone mode, select **Shut down CFX-Solver Manager**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Click **OK**.

24.5.3. Viewing the Results Using CFD-Post

The non-equilibrium calculation creates a number of solution variables. The most important ones, which are briefly described in the following section, are supercooling, nucleation rate, droplet number, mass fraction, and particle diameter.

1. When CFD-Post starts, the **Domain Selector** dialog box might appear. If it does, ensure that both the R1 and S1 domains are selected, then click **OK** to load the results from these domains.

2. Click the **Turbo** tab.

3. The **Turbo Initialization** dialog box is displayed, and asks you whether you want to auto-initialize all components. Click **Yes**.

   The **Turbo** tree view shows the two components in domains R1 and S1. In this case, the initialization works without problems. If there were any problems initializing a component, this would be indicated in the tree view.

   Note:

   If you do not see the **Turbo Initialization** dialog box, or as an alternative to using that dialog box, you can initialize all components by clicking the **Initialize All Components** button, which is visible initially by default, or after double-clicking the **Initialization** object in the **Turbo** tree view.

24.5.3.1. Specifying Locators for Plots

Make a 2D surface at 50% span to be used as a locator for plots:

1. On the main menu select **Insert** > **Location** > **Turbo Surface** and name it `Turbo Surface 1`.

2. On the **Geometry** tab, set **Method** to `Constant Span` and **Value** to `0.5`.

3. Click **Apply**.

4. Turn off the visibility of `Turbo Surface 1`.

24.5.3.2. Supercooling Contour Plot

The non-equilibrium solution provides considerable detail on conditions related to phase transition. In particular, the solution tracks the evolution of metastable conditions in the flow through a supercooling variable obtained on the basis of local pressure and gas phase temperature. The supercooling level is the primary variable influencing the nucleation model. The nucleation model provides an estimate of the rate of production of critically sized nuclei that are stable enough to grow in a supercooled vapor flow.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the next step, you will plot the degree of supercooling, where the supercooling represents the difference between the saturation temperature, set by the local pressure in the flow, and the associated local gas phase temperature. At the inlet, supercooling is often shown as negative, indicating superheated conditions. At equilibrium conditions, no supercooling is allowed since liquid and vapor phases always share the same temperature. Also, critical homogeneous nucleation at pressures below one atmosphere generally involves supercooling levels of up to 35 to 40 K, depending on rate of expansion in the flow.

1. Create a supercooling contour plot on `Turbo Surface 1`.

   This will display the degree of non-equilibrium conditions in the gas phase prior to homogeneous phase transition.

   a. Create a new contour plot named `Degree of Supercooling`.

   b. In the details view on the **Geometry** tab, set **Locations** to `Turbo Surface 1` and **Variable** to `H2Og.Supercooling` then click **Apply**.

2. Turn off the visibility of `Degree of Supercooling` when you have finished observing the results.

24.5.3.3. Nucleation Rate and Droplet Number Contour Plots

In the next step, you will plot the nucleation rate to show the level of nucleation attained, along with the droplet number concentration that is present in the flow following nucleation. Note that the nucleation rates reach very high levels, with peak values remaining in the flow for only a short time (that is, as long as supercooled conditions remain). The supercooled droplets released at nucleation grow rapidly, taking mass from the vapor phase and releasing thermal energy, which acts to rapidly reduce vapor supercooling to near zero. This fact removes further significant nucleation and is why the phase transition process is limited to a narrow region in the flow in most cases.

1. Create a nucleation rate contour plot on `Turbo Surface 1`.

   This will display the nucleation front at the point of maximum supercooling.

   a. Create a new contour plot named `Nucleation Rate`.

   b. In the details view on the **Geometry** tab, set **Locations** to `Turbo Surface 1` and **Variable** to `H2Ol.Nucleation Rate`.

   c. Set **Range** to `Local` and then click **Apply**.

2. Turn off the visibility of `Nucleation Rate` when you have finished observing the results.

3. Create a droplet number contour plot on `Turbo Surface 1`.

   This will display the predicted droplet concentration resulting from phase transition.

   a. Create a new contour plot named `Droplet Number`.

   b. In the details view on the **Geometry** tab, set **Locations** to `Turbo Surface 1` and **Variable** to `H2Ol.Droplet Number`.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. Click **Apply**.

4. Turn off the visibility of **Droplet Number** when you have finished observing the results.

24.5.3.4. Mass Fraction and Particle Diameter Contour Plots

In this section, you will plot the mass fraction (or wetness) of the condensed phase. From the mass fraction and droplet number, it is possible to derive a particle diameter, which you will also plot in this section. Notice that the particle diameters appear in the flow at very small sizes, in the range of 5.0E-9 m, but grow rapidly so that, when leaving the nucleation zone, they are approximately an order of magnitude larger. Since the pressure is dropping through the turbine, droplet sizes continue to increase along with the wetness. To emphasize the particle diameters proceeding out of the nucleation zone, you will set the contour range for viewing. In flow regions near walls, where expansion rate is reduced, small amounts of liquid may have droplet diameters that grow to large sizes relative to those coming from the nucleation zone. Without setting the range, these particle diameters near the walls will be emphasized.

1. Create a contour plot showing the mass fraction of the condensed phase on `Turbo Surface 1`.

   Condensed droplets grow in size and accumulate mass at the expense of the gas phase.

   a. Create a new contour plot named `Mass Fraction of Condensed Phase H2Ol`.

   b. In the details view on the **Geometry** tab, set **Locations** to `Turbo Surface 1` and **Variable** to `H2Ol.Mass Fraction`.

   c. Click **Apply**.

      Comparing the mass fraction in the non-equilibrium case to the equilibrium solution previously viewed, you can see that phase transition is considerably delayed such that it occurs in the blade passages of the rotor rather than the stator. This is a typical consequence of non-equilibrium flow, and reflects real flow situations.

2. Turn off the visibility of **Mass Fraction of Condensed Phase H2Ol** when you have finished observing the results.

3. Create a particle diameter contour plot on `Turbo Surface 1`.

   The size of the condensed droplets is calculated from the droplet number and volume fraction of the condensed phase.

   a. Create a new contour plot named `Particle Diameter`.

   b. In the details view on the **Geometry** tab, set **Locations** to `Turbo Surface 1` and **Variable** to `H2Ol.Particle Diameter`.

   c. Set **Range** to `User Specified` then set **Min** to `0 [m]` and **Max** to `1e-07 [m]`.

   d. Click **Apply**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Turn off the visibility of `Particle Diameter` when you have finished observing the results.

24.5.3.5. Gas and Condensed Phase Static Temperature Contour Plots

In a non-equilibrium prediction, the gas phase temperature is different from the condensed phase temperature. The former is determined from a transport equation and the latter from an algebraic relationship relating droplet temperature to its diameter (through a small droplet model already described).

In this section, you will plot the different temperature fields. It should be noted that, for the case of the condensed phase temperature field, before droplets are actually formed, there is no meaningful droplet temperature. At the point droplets are formed by nucleation, their temperature is at the gas phase. Once the droplets have grown in size, their temperature is very close to the saturation temperature. Because the non-equilibrium solution considers the condensed phase temperatures separate from the gas phase, the influence of thermodynamic losses are included in the predictions. This is because it becomes possible to account for heat flow between the vapor and condensed phases as they pass through the domain. Due to this, non-equilibrium efficiency predictions are more accurate than ones obtained using an equilibrium model.

1. Create a gas phase static temperature contour plot on `Turbo Surface 1`:

   a. Create a new contour plot named `Gas Phase Static Temperature`.

   b. In the details view on the **Geometry** tab, set **Locations** to `Turbo Surface 1` and **Variable** to `H2Og.Temperature`.

   c. Click **Apply**.

2. Turn off the visibility of **Gas Phase Static Temperature** when you have finished observing the results.

3. Create a condensed phase static temperature contour plot on `Turbo Surface 1`.

   a. Create a new contour plot named `Condensed Phase Static Temperature`.

   b. In the details view on the **Geometry** tab, set **Locations** to `Turbo Surface 1` and **Variable** to `H2Ol.Temperature`.

   c. Click **Apply**.

4. Once you have observed the results, save the state and exit CFD-Post.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 25: Modeling a Gear Pump using an Immersed Solid

This tutorial includes:

25.1. Tutorial Features

25.2. Overview of the Problem to Solve

25.3. Preparing the Working Directory

25.4. Defining the Case Using CFX-Pre

25.5. Obtaining the Solution Using CFX-Solver Manager

25.6. Viewing the Results Using CFD-Post

25.1. Tutorial Features

In this tutorial you will learn about:

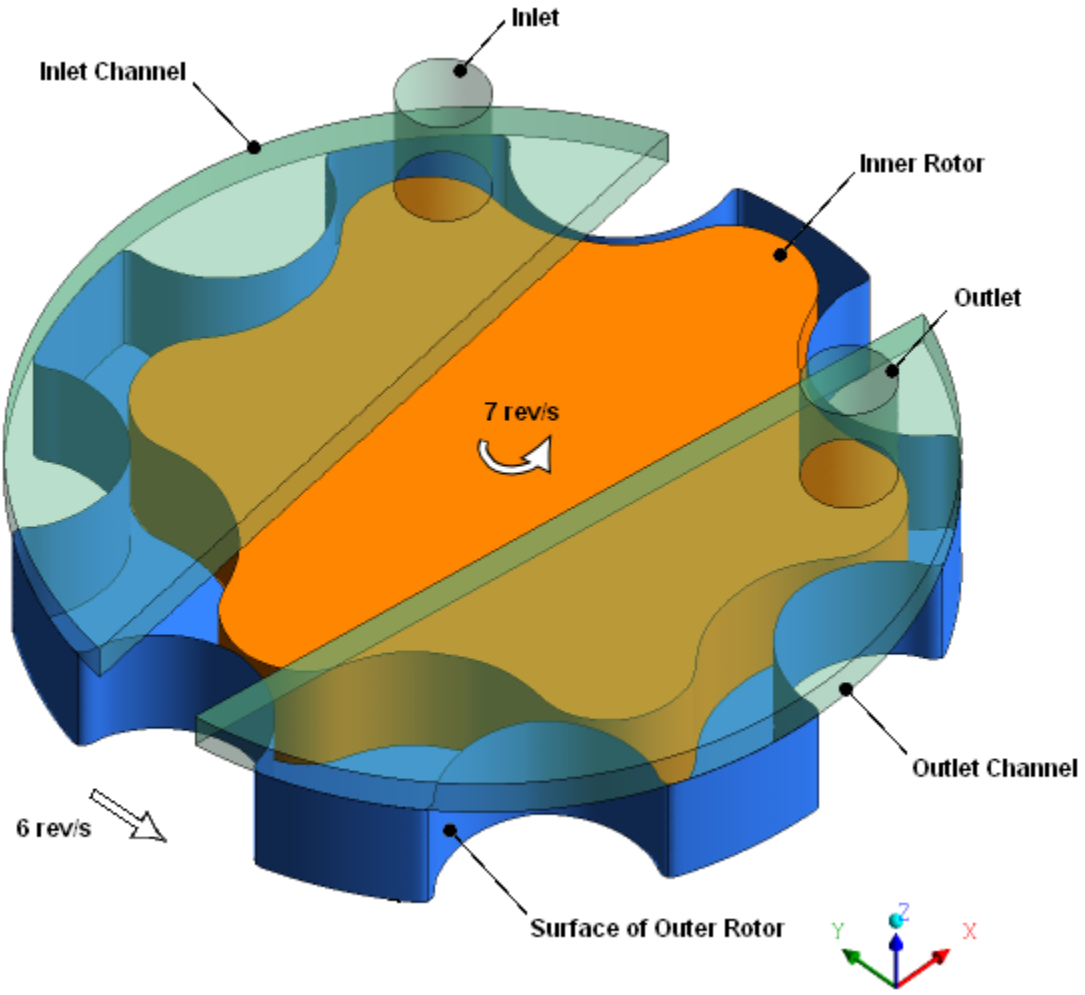
- Setting up an immersed solids domain.
- Applying a counter-rotating wall boundary.
- Monitoring an expression during a solver run.
- Creating an XY-transient chart in CFD-Post.
- Creating a keyframe animation.

Component	Feature	Details
CFX-Pre	User Mode	General mode
	Domain Type	Immersed Solid
		Fluid Domain
	Analysis Type	Transient
	Fluid Type	Continuous Fluid
	Boundary Conditions	Inlet Boundary
		Outlet Boundary
	Domain Interface	Fluid Fluid
CFD-Post	Chart	Mass Flow Rate
	Animation	Keyframe

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

25.2. Overview of the Problem to Solve

In this tutorial, you will simulate a gear pump that drives a flow of water. This tutorial makes use of the Immersed Solids capability of Ansys CFX in order to model a solid that moves through a fluid.



The outlet has an average relative static pressure of 1 psi; the relative total pressure at the inlet is 0 psi. The inner rotor (gear) rotates at a rate of 7 revolutions per second; the outer rotor rotates at 6 revolutions per second. The diameter of the fluid region between the rotors is approximately 7.3 cm.

You will use an immersed solid domain to model the inner rotor, a rotating fluid domain to model the water immediately surrounding the inner rotor, and a stationary fluid domain to model the water in the inlet and outlet channels. To model the stationary pump housing (not shown in the figure), you will apply a counter-rotating wall condition to the top (high Z) surface of the rotating fluid domain, on the non-overlap portion (which lies between the inlet and outlet channels). To model the upper surfaces of the teeth of the outer rotor, you will apply a rotating wall condition on the non-overlap portions of the lower (low Z) surfaces of the inlet and outlet chambers.

The following conditions will be met to promote the establishment of a cyclic flow pattern:

- The mesh of the rotating domain should be rotationally periodic so that it looks the same after each (outer) rotor tooth passes.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The mesh on the outer boundary of the immersed solid domain should be rotationally periodic so that it looks the same after each (inner) rotor tooth passes. (The mesh inside the immersed solid domain has no effect in this tutorial.)
- An integer number of time steps should pass as one rotor tooth passes.

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- Running Ansys CFX Tutorials Using Ansys Workbench (p. 34)
- Changing the Display Colors (p. 37)

25.3. Preparing the Working Directory

1. Create a working directory.

   Ansys CFX uses a working directory as the default location for loading and saving files for a particular session or project.

2. Download the `immersed_solid.zip` file here.

3. Unzip `immersed_solid.zip` to your working directory.

   Ensure that the following tutorial input file is in your working directory:

   - `ImmersedSolid.gtm`

4. Set the working directory and start CFX-Pre.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

25.4. Defining the Case Using CFX-Pre

This section describes the step-by-step definition of the flow physics in CFX-Pre for a steady-state simulation.

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `ImmersedSolid`.
5. Click **Save**.

25.4.1. Importing the Mesh

1. Right-click `Mesh` and select **Import Mesh** > **CFX Mesh**.

The **Import Mesh** dialog box appears.

2. Configure the following setting(s):

Setting	Value
File name	ImmersedSolid.gtm

3. Click **Open**.

25.4.2. Creating Expressions for Time Step and Total Time

Next, you will create an expression defining the time step size for this transient analysis. One tooth of the inner (or outer) rotor passes every 1/42 s. Choose a time step that resolves this motion in 30 intervals.

1. From the main menu, select **Insert** > **Expressions, Functions and Variables** > **Expression**.
2. In the **Insert Expression** dialog box, type `dt`.
3. Click **OK**.
4. Set **Definition**, to `(1/42)[s]/30`.
5. Click **Apply** to create the expression.

Next, you will create an expression defining the total simulation time. Make the simulation run long enough for 3 rotor teeth to pass: 3/42 s. This will give the solution time to establish a periodic nature.

1. Create an expression called `total time`.
2. Set **Definition** to `(3/42)[s]`.
3. Click **Apply**.

25.4.3. Setting the Analysis Type

Define the simulation as transient, using the expressions you created earlier.

1. Under the **Outline** tab, edit *Analysis Type* .
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Analysis Type > **Option**	Transient
	Analysis Type > Time Duration	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Option**	Total Time
	Analysis Type > Time Duration > **Total Time**	total time [a]
	Analysis Type > Time Steps > **Option**	Timesteps
	Analysis Type > Time Steps > **Timesteps**	dt
	Analysis Type > Initial Time > **Option**	Automatic with Value
	Analysis Type > Initial Time > **Time**	0 [s]
a. You first need to click the *Enter Expression* icon beside the field.		

3. Click **OK**.

25.4.4. Creating the Domains

This simulation requires three domains: two fluid domains and one immersed solid domain. First you will create an immersed solid domain.

25.4.4.1. Creating an Immersed Solid Domain

Create the immersed solid domain as follows:

1. Select **Insert** > **Domain** from the main menu, or click *Domain* .
2. In the **Insert Domain** dialog box, set the name to `ImmersedSolid` and click **OK**.
3. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Basic Settings	Location and Type > **Location**	Inner Rotor
	Location and Type > **Domain Type**	Immersed Solid
	Location and Type > **Coordinate Frame**	Coord 0
	Domain Models > Domain Motion > **Option**	Rotating
	Domain Models > Domain Motion > **Angular Velocity**	7 [rev s^-1]
	Domain Models > Domain Motion > Axis Definition > **Option**	Two Points
	Domain Models > Domain Motion > Axis Definition > **Rotation Axis From**	0.00383, 0, 0
	Domain Models > Domain Motion > Axis Definition > **Rotation Axis To**	0.00383, 0, 1

4. Click **OK**.

25.4.4.2. Creating the Stationary Fluid Domain

Create the stationary fluid domain according to the problem description:

1. Create a new domain named `StationaryFluid`.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Location and Type > **Location**	Channels
	Location and Type > **Domain Type**	Fluid Domain
	Location and Type > **Coordinate Frame**	Coord 0
	Fluid and Particle Definitions	Fluid 1
	Fluid and Particle Definitions > Fluid 1 > **Option**	Material Library
	Fluid and Particle Definitions > Fluid 1 > **Material**	Water
	Fluid and Particle Definitions > Fluid 1 > Morphology > **Option**	Continuous Fluid
	Domain Models > Pressure > **Reference Pressure**	0 [psi]
	Domain Models > Buoyancy Models > **Option**	Non Buoyant
	Domain Models > Domain Motion > **Option**	Stationary
	Domain Models	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Mesh Deformation > **Option**	 None
Fluid Models	Heat Transfer > **Option**	 None
	Turbulence > **Option**	 k-Epsilon
	Turbulence > **Wall Function**	 Scalable
	Combustion > **Option**	 None
	Thermal Radiation > **Option**	 None
Initialization	**Domain Initialization**	(Selected)
	Domain Initialization > Initial Conditions > **Velocity Type**	 Cartesian
	Domain Initialization > Initial Conditions > Cartesian Velocity Components > **Option**	 Automatic with Value
	Domain Initialization > Initial Conditions > Cartesian Velocity Components > **U**	 0 [m s^-1]
	Domain Initialization > Initial Conditions > Cartesian Velocity Components > **V**	 0 [m s^-1]
	Domain Initialization	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Initial Conditions > Cartesian Velocity Components > **W**	0 [m s^-1]
	Domain Initialization > Initial Conditions > Static Pressure > **Option**	Automatic with Value
	Domain Initialization > Initial Conditions > Static Pressure > **Relative Pressure**	1 [psi] * step(-y/1 [cm]) [a]
	Domain Initialization > Initial Conditions > Turbulence > **Option**	Medium (Intensity = 5%)
a. This initial condition improves numerical stability by avoiding an adverse pressure gradient at the outlet.		

3. Click **OK**.

25.4.4.3. Creating the Rotating Fluid Domain

Create the rotating fluid domain according to the problem description:

1. In the **Outline** tree view, right-click `Simulation` > `Flow Analysis 1` > `Stationary Fluid` and select **Duplicate**.
2. Right-click `Simulation` > `Flow Analysis 1` > `Copy of StationaryFluid` and select **Rename**.
3. Rename the domain to `RotatingFluid`.
4. Edit `RotatingFluid`.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Location and Type > **Location**	Gear Chamber
	Domain Models > Domain Motion > **Option**	Rotating
	Domain Models > Domain Motion > **Angular Velocity**	6 [rev s^-1]
	Domain Models > Domain Motion > Axis Definition > **Option**	Coordinate Axis
	Domain Models > Domain Motion > Axis Definition > **Rotation Axis**	Global Z

6. Click **OK**.

25.4.5. Creating the Domain Interface

Add a domain interface that connects the `StationaryFluid` and `RotatingFluid` domains:

1. Click **Insert** > **Domain Interface** from the main menu or click *Domain Interface* .
2. Accept the default domain interface name and click **OK**.
3. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Interface Type**	Fluid Fluid
	Interface Side 1 > **Domain (Filter)**	StationaryFluid
	Interface Side 1	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Region List**	Channel Side
	Interface Side 2 > **Domain (Filter)**	 RotatingFluid
	Interface Side 2 > **Region List**	 Chamber Side
	Interface Models > **Option**	 General Connection
	Interface Models > Frame Change/Mixing Model > **Option**	 Transient Rotor Stator
	Interface Models > Pitch Change > **Option**[i]	 None
Additional Interface Models	Mass and Momentum > **Option**	 Conservative Interface Flux
	Mass and Momentum > Interface Model > **Option**	 None
Mesh Connection	Mesh Connection Method > Mesh Connection > **Option**	 GGI

i. Setting this option to `None` will generate a global warning in the message window below the viewer. In this case, the warning can be ignored because the full 360° are being modeled on both sides of the interface.

4. Click **OK**.

Apply a counter-rotating no-slip wall condition to the non-overlap portion of the domain interface on the rotating domain side, because this surface represents part of the stationary housing of the pump.

1. Edit `RotatingFluid` > `Domain Interface 1 Side 2`.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If this object does not appear in the tree view, then edit `Case Options` > `General`, select **Show Interface Boundaries in Outline Tree**, and click **OK**.

2. Configure the following setting(s):

Tab	Setting	Value
Nonoverlap Conditions	**Nonoverlap Conditions**	(Selected)
	Nonoverlap Conditions > Mass and Momentum > **Option**	No Slip Wall
	Nonoverlap Conditions > Mass and Momentum > **Wall Velocity**	(Selected)
	Nonoverlap Conditions > Mass and Momentum > Wall Velocity > **Option**	Counter Rotating Wall

3. Click **OK**.

Apply a rotating no-slip wall condition to the non-overlap portions of the domain interface on the stationary domain side, because these surfaces represent faces of the rotor teeth of the outer rotor, and the latter rotates at 6 rev/s about the Z axis.

1. Edit `StationaryFluid` > `Domain Interface 1 Side 1`.

2. Configure the following setting(s):

Tab	Setting	Value
Nonoverlap Conditions	**Nonoverlap Conditions**	(Selected)
	Nonoverlap Conditions > Mass and Momentum > **Option**	No Slip Wall
	Nonoverlap Conditions > Mass and Momentum > **Wall Velocity**	(Selected)
	Nonoverlap Conditions	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Mass and Momentum > Wall Velocity > **Option**	Rotating Wall
	Nonoverlap Conditions > Mass and Momentum > Wall Velocity > **Angular Velocity**	6 [rev s^-1]
	Nonoverlap Conditions > Mass and Momentum > Wall Velocity > Axis Definition > **Option**	Coordinate Axis
	Nonoverlap Conditions > Mass and Momentum > Wall Velocity > Axis Definition > **Rotation Axis**	Global Z

3. Click **OK**.

25.4.6. Creating Boundary Conditions

This section outlines the steps to create the inlet and outlet boundary conditions, as specified in the problem description.

25.4.6.1. Inlet Boundary

Create a total pressure inlet at a relative pressure of 0 psi:

1. In the **Outline** tree view, right-click `StationaryFluid` and select **Insert** > **Boundary**.
2. Set **Name** to `in` and click **OK**.
3. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Basic Settings	**Boundary Type**	Inlet
	Location	Inlet
Boundary Details	Mass And Momentum > **Option**	Total Pressure (stable)
	Mass And Momentum > **Relative Pressure**	0 [psi]
	Flow Direction > **Option**	Normal to Boundary Condition
	Turbulence > **Option**	Medium (Intensity = 5%)

4. Click **OK**.

25.4.6.2. Outlet Boundary

Create an outlet with a relative average static pressure of 1 psi:

1. Create a boundary named `out` in the `StationaryFluid` domain.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Outlet
	Location	Outlet
Boundary Details	Mass And Momentum > **Option**	Average Static Pressure
	Mass And Momentum > **Relative Pressure**	1 [psi]
	Mass And Momentum > **Pres. Profile Blend**	0.05
	Pressure Averaging > **Option**	Average Over Whole Outlet

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click **OK**.

25.4.7. Setting Solver Control

1. Click *Solver Control* .
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Advection Scheme > **Option**	 High Resolution
	Transient Scheme > **Option**	 Second Order Backward Euler
	Transient Scheme > Timestep Initialization > **Option**	 Automatic
	Turbulence Numerics > **Option**	 First Order
	Convergence Control > **Min. Coeff. Loops**	 1
	Convergence Control > **Max. Coeff. Loops**	 10
	Convergence Control > Fluid Timescale Control > **Timescale Control**	 Coefficient Loops
	Convergence Criteria > **Residual Type**	 RMS
	Convergence Criteria > **Residual Target**	 1.0 E −4

3. Click **OK**.

25.4.8. Setting Output Control

Set up the solver to output transient results files that record pressure, velocity, and velocity in the stationary frame, on every time step:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click *Output Control* .
2. Click the **Trn Results** tab.
3. In the **Transient Results** list box, click *Add new item* , set **Name** to `Transient Results 1`, and click **OK**.
4. Configure the following setting(s) of `Transient Results 1`:

Setting	Value
Option	Selected Variables
File Compression	Default
Output Variables List	Pressure, Velocity, Velocity in Stn Frame
Output Boundary Flows	(Selected)
Output Boundary Flows > **Boundary Flows**	 All
Output Frequency > **Option**	 Every Timestep

5. Click the **Monitor** tab.
6. Select **Monitor Objects**.
7. Under **Monitor Points and Expressions**:
   a. Click *Add new item* .
   b. Accept the default name and click **OK**.
   c. Set **Option** to `Expression`.
   d. Set **Expression Value** to `massFlow()@in`.
8. Click **OK**.

25.4.9. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .
2. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
File name	ImmersedSolid.def

3. Click **Save**.

   Note:

   A warning message will appear due to the global warning that was mentioned earlier in Creating the Domain Interface (p. 602). Click **Yes**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, **Solver Input File** is set.

4. If using stand-alone mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

25.5. Obtaining the Solution Using CFX-Solver Manager

When CFX-Pre has shut down and the CFX-Solver Manager has started, obtain a solution to the CFD problem by following the instructions below:

1. In CFX-Solver Manager, ensure that the **Define Run** dialog box is displayed.

   If CFX-Solver Manager is launched from CFX-Pre, the information required to perform a solver run is entered automatically in the **Define Run** dialog box.

2. Click **Start Run**.

   The solver run begins and the progress is displayed in a split screen.

3. Click the **User Points** tab (which appears after the first time step has been computed) and monitor the value of `Monitor Point 1` as the solution proceeds.

4. Rescale the monitor plot so that you can readily see the time-periodic oscillations in mass flow that occur after the initial transient phase:

   a. Right-click anywhere in the **User Points** plot and select **Monitor Properties**.

   b. In the **Monitor Properties: User Points** dialog box, on the **Range Settings** tab, select **Set Manual Scale (Linear)**.

   c. Set the lower and upper bounds to 0.015 and 0.055 respectively.

   d. Click **OK**.

5. Select the check box next to **Post-Process Results** when the completion message appears at the end of the run.

6. If using stand-alone mode, select the check box next to **Shut down CFX-Solver Manager**.

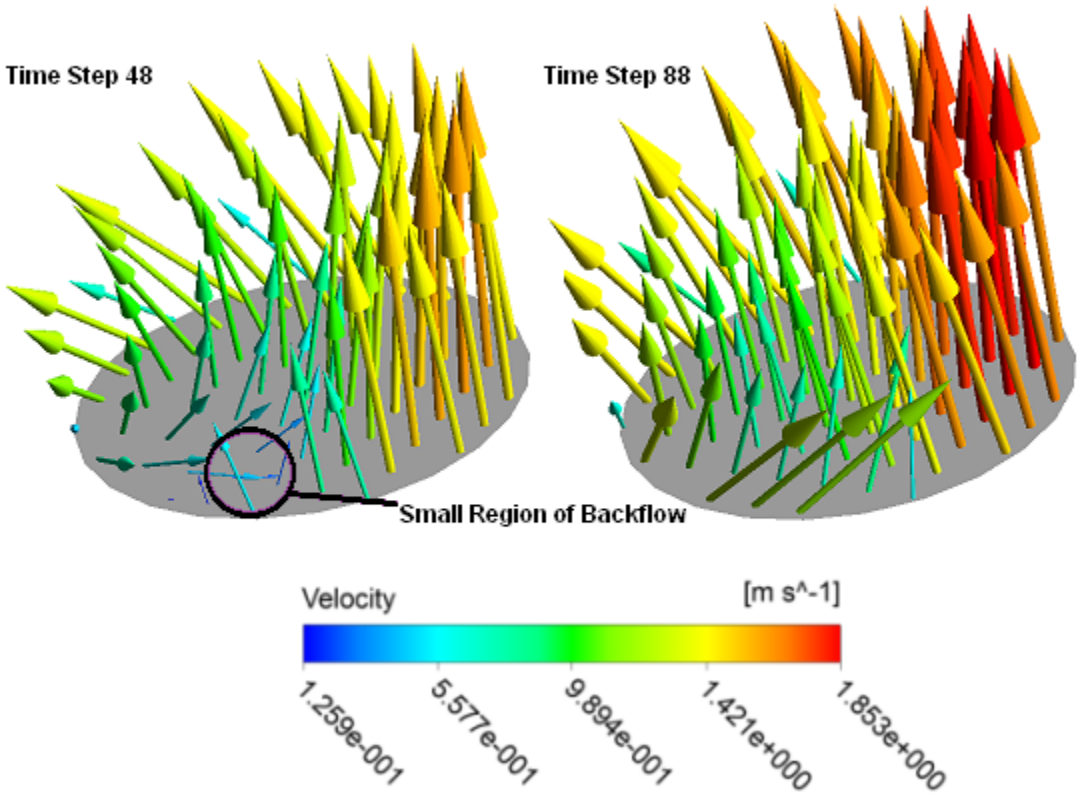
Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Click **OK**.

Note:

During the Solver Manager run, you may observe a notice at the 47th and 48th time steps warning you that "A wall has been placed at portion(s) of an OUTLET boundary condition ... to prevent fluid from flowing into the domain." The mass flow at the inlet drops to its lowest level throughout the cycle at this point, causing a reduction in the velocity at the outlet. Because there is turbulence at the outlet, this reduced velocity allows a tiny vortex to produce a small, virtually negligible, amount of backflow at the outlet. Figure 25.1: Velocity Vectors on the Outlet (p. 610) shows velocity vectors at the outlet when the mass flow at the inlet is lowest (48th time step) and when the mass flow is greatest (88th time step). In the figure, you can see where this slight backflow occurs for the 48th time step.

Figure 25.1: Velocity Vectors on the Outlet



25.6. Viewing the Results Using CFD-Post

In this section, you will generate a chart to show the mass flow rate through the machine as a function of time. You will also prepare an animation of the machine in operation, complete with velocity vectors.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

25.6.1. Creating a Chart of Mass Flow versus Time

During the solver run, you observed a monitor plot that showed mass flow versus time step. Here, you will make a similar plot of mass flow versus time. As you did in the monitor plot, adjust the vertical axis range to focus on the time-periodic oscillations in mass flow that occur after the initial transient phase.

1. When CFD-Post starts, the **Domain Selector** dialog box might appear. If it does, ensure that the `ImmersedSolid`, `RotatingFluid` and `StationaryFluid` domains are selected, then click **OK** to load the results from these domains.
2. A dialog box will notify you that the case contains immersed solid domain. Click **OK** to continue.
3. Create a new chart named `Mass Flow Rate`.

   The **Chart Viewer** appears.
4. Configure the following setting(s):

Tab	Setting	Value
General	**Type**	XY-Transient or Sequence
	Title	Mass Flow Rate at the Inlet over Time

5. Click the **Data Series** tab.
6. If the **Data Series** list box is empty, right-click in it and select **New**, or click *New* .
7. Configure the following setting(s):

Tab	Setting	Value
Data Series	**Series 1**	(Selected)
	Name	Inlet Mass Flow
	Data Source > **Expression**	(Selected)
	Data Source > **Expression**	massFlow()@in [a]
Y Axis	Axis Range > **Determine ranges automatically**	(Cleared)
	Axis Range > **Min**	0.015
	Axis Range	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Max**	0.055
a. You will have to either type it manually or right-click and choose **Functions** > **CFD-Post** > **massFlow()@** from the shortcut menu, then type `in`.		

8. Click **Apply**.

The mass flow rate settles into a repeating pattern with a period of 1/42 s, which is the time it takes a rotor tooth to pass.

25.6.2. Creating a Velocity Vector Plot

Create a slice plane and then make a vector plot on the slice plane as follows:

1. Click the **3D Viewer** tab.
2. Create a new plane named `Plane 1`.
3. Configure the following setting(s):

Tab	Setting	Value
Geometry	**Domains**	RotatingFluid
	Definition > **Method**	XY Plane
	Definition > **Z**	0.003 [m]

4. Click **Apply**.
5. Turn off the visibility of `Plane 1`.
6. Create a new vector plot named `Vector 1`.
7. Configure the following setting(s):

Tab	Setting	Value
Geometry	**Domains**	All Domains
	Definition > **Locations**	Plane 1
	Definition > **Sampling**	Rectangular Grid
	Definition	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Spacing**	0.03
	Definition > **Variable**	 Velocity
Color	**Mode**	Use Plot Variable
	Range	User Specified
	Min	0 [m s^-1]
	Max	0.8 [m s^-1]
Symbol	**Symbol**	Arrow3D
	Symbol Size	15
	Normalized Symbols	(Cleared)

8. Click **Apply**.

25.6.3. Changing the Appearance in Preparation for an Animation

Make the inlet and outlet visible as follows:

1. Edit `StationaryFluid` > `in`.
2. Configure the following setting(s):

Tab	Setting	Value
Render	**Show Faces**	(Cleared)
	Show Mesh Lines	(Selected)
	Show Mesh Lines > **Edge Angle**	 105 [degree]
	Show Mesh Lines > **Line Width**	 2
	Show Mesh Lines > **Color Mode**	 Default

3. Click **Apply**.
4. Apply the same settings to `StationaryFluid` > `out`.

Make the inlet and outlet channels visible as follows:

1. Edit `StationaryFluid` > `StationaryFluid Default`.
2. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Render	**Show Faces**	(Selected)
	Show Faces > **Transparency**	0.8
	Show Mesh Lines	(Cleared)

3. Click **Apply**.

Make the walls of the rotating fluid domain visible as follows:

1. Edit `RotatingFluid` > `RotatingFluid Default`.
2. Configure the following setting(s):

Tab	Setting	Value
Color	**Mode**	Constant
	Color	(White)
Render	**Show Faces**	(Selected)
	Show Faces > **Transparency**	0.0
	Show Mesh Lines	(Cleared)

3. Click **Apply**.

Make the walls of the immersed solid domain visible as follows:

1. Edit `ImmersedSolid` > `ImmersedSolid Default`.
2. Configure the following setting(s):

Tab	Setting	Value
Color	**Mode**	Constant
	Color	(Blue)
Render	**Show Faces**	(Selected)
	Show Faces > **Transparency**	0.0
	Show Mesh Lines	(Cleared)

3. Click **Apply**.

Make the following other changes in preparation for the animation that you will create in the next section:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Right-click a blank area in the viewer and select **Predefined Camera** > **View From +Z**.
2. Rotate the view a few degrees so that you can see the 3D nature of the geometry.
3. Turn off the visibility of `User Locations and Plots` > `Wireframe`.

25.6.4. Creating a Keyframe Animation

In this section, you will generate an animation that shows the changing velocity field on `Plane 1`. To take advantage of the periodic nature of the solution, you will record a short animation that can be played in a repeating loop in an MPEG player. Start the animation at the 61st time step (a time at which the flow has settled into a repeating pattern) and end it at the 90th time step. The 60th time step corresponds with 2/42 s, and the 90th corresponds with 3/42 s; the 1/42 s interval is the period over which the solution repeats. Because the 60th and 90th time steps look the same, the 60th time step is omitted to avoid having a pair of adjacent identical frames in the animation when the latter is played in a repeating loop.

1. Click *Timestep Selector* and load the 61st time step.
2. Click *Animation*.

   The **Animation** dialog box appears.
3. Set **Type** to **Keyframe Animation**.
4. Click *New* to create `KeyframeNo1`.
5. Select `KeyframeNo1`, then set **# of Frames** to 28, then press **Enter** while the cursor is in the **# of Frames** box.

   Tip:

   Be sure to press **Enter** and confirm that the new number appears in the list before continuing.
6. Use the **Timestep Selector** to load the 90th time step.
7. In the **Animation** dialog box, click *New* to create `KeyframeNo2`.
8. Ensure that *More Animation Options* is pushed down to show more animation settings.
9. Select **Loop**.
10. Ensure that the *Repeat forever* button next to **Repeat** is not selected (not pushed down).
11. Select **Save Movie**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12. Set **Format** to `MPEG1`.
13. Click *Browse* next to **Save Movie**.
14. Set **File name** to `ImmersedSolid.mpg`.
15. If required, set the path location to a different directory.
16. Click **Save**.

    The movie file name (including path) has been set, but the animation has not yet been produced.
17. Click *To Beginning* .

    This ensures that the animation will begin at the first keyframe.
18. After the first keyframe has been loaded, click *Play the animation* .
    - The MPEG will be created as the animation proceeds.
    - This will be slow, since results for each time step will be loaded and objects will be created.
    - To view the movie file, you need to use a viewer that supports the MPEG format.

    Note:

    To explore additional animation options, click the **Options** button. On the **Advanced** tab of the **Animation Options** dialog box, there is a **Save Frames As Image Files** check box. By selecting this check box, the JPEG or PPM files used to encode each frame of the movie will persist after movie creation; otherwise, they will be deleted.
19. When you have finished, quit CFD-Post.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 26: Drop Curve for Cavitating Flow in a Pump

This tutorial includes:

26.1. Tutorial Features

26.2. Overview of the Problem to Solve

26.3. Preparing the Working Directory

26.4. Simulating the Pump with High Inlet Pressure

26.5. Simulating the Pump with Cavitation and High Inlet Pressure

26.6. Simulating the Pump with Cavitation and a Range of Inlet Pressures

26.1. Tutorial Features

In this tutorial you will learn about:

- Preparing and running a series of related simulations to generate cavitation performance data for a pump.
- Creating a drop curve chart in CFD-Post.
- Using isosurfaces in CFD-Post to visualize regions of cavitation.

Component	Feature	Details
CFX-Pre	User Mode	General mode
	Analysis Type	Steady State
	Fluid Type	Water at 25 C
		Water Vapour at 25 C
	Fluid Models	Homogeneous Model
	Domain Type	Single Domain
	Turbulence Model	k-Epsilon
	Heat Transfer	Isothermal
	Boundary Conditions	Inlet (Subsonic)
		Outlet (Subsonic)
		Wall (Counter Rotating)
	Timestep	Physical Time Scale
CFD-Post	Plots	Contour

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

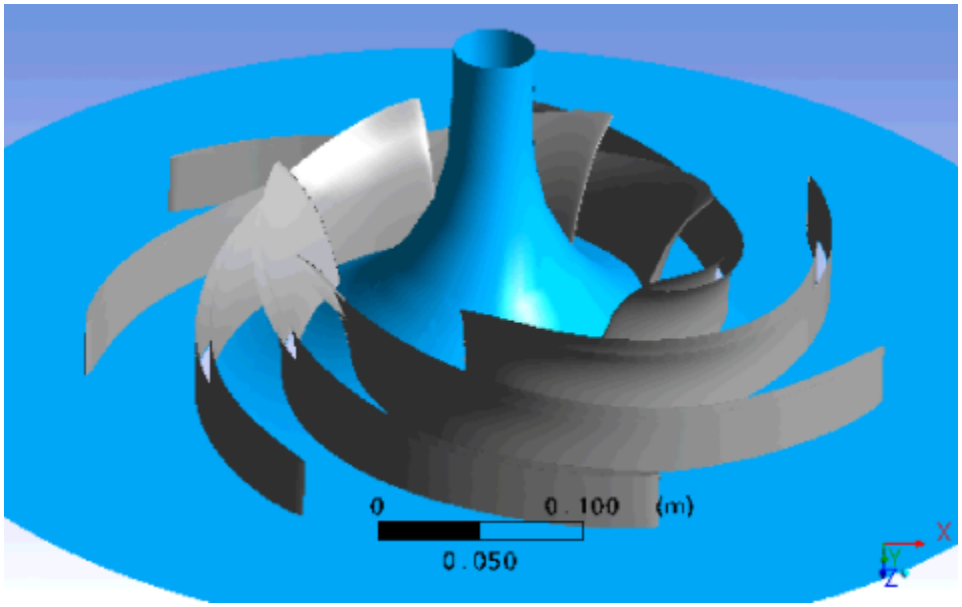
26.2. Overview of the Problem to Solve

This tutorial uses a simple pump to illustrate the basic concepts of setting up, running, and postprocessing a cavitation problem in Ansys CFX.

When liquid is suddenly accelerated in order to move around an obstruction, a decrease in the local pressure is present. Sometimes, this pressure decrease is substantial enough that the pressure falls below the saturation pressure determined by the temperature of the liquid. In such cases, the fluid begins to vaporize in a process called cavitation. Cavitation involves a very rapid increase in the volume occupied by a given mass of fluid, and when significant, can influence the flow distribution and operating performance of the device. In addition, the vaporization of the liquid, and the subsequent collapse of the vapor bubbles as the local pressure recovers, can cause damage to solid surfaces. For these reasons (among others), it is desirable, in the design and operation of devices required to move liquid, to be able to determine if cavitation is present, including where and the extent of the cavitation. Furthermore it is also useful to examine this behavior for a range of conditions.

The model conditions for this example are turbulent and incompressible. The speed and direction of rotation of the pump is 132 rad/sec about the Z axis (positive rotation following the right hand rule). The relevant problem parameters are:

- Inflow total pressure = 100000 Pa
- Outflow mass flow = 16 kg/s
- Inlet turbulence intensity = 0.03
- Inlet length scale = 0.03 m



The SHF (Societe Hydraulique Francaise) pump has seven impeller blades. Due to the periodic nature of the geometry, only a single blade passage of the original pump must be modeled, therefore minimizing the computer resources required to obtain a solution.

The objective of this tutorial is to show pump cavitation performance in the form of a drop curve. The drop curve is a chart of Head versus Net Positive Suction Head (NPSH). This tutorial provides the data

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

for the drop curve, but also has instructions for optionally generating the data by running a series of simulations with progressively lower inlet pressures. Each simulation is initialized with the results of the previous simulation.

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- Running Ansys CFX Tutorials Using Ansys Workbench (p. 34)
- Changing the Display Colors (p. 37)

26.3. Preparing the Working Directory

1. Create a working directory.

   Ansys CFX uses a working directory as the default location for loading and saving files for a particular session or project.

2. Download the `drop_curve.zip` file here.

3. Unzip `drop_curve.zip` to your working directory.

   Ensure that the following tutorial input file is in your working directory:

   - `Cavitation.gtm`

4. Set the working directory and start CFX-Pre.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

26.4. Simulating the Pump with High Inlet Pressure

A steady-state high-pressure (inlet pressure of 100,000 Pa) simulation of the pump without cavitation (that is, simulation of the pump without water vapor) will first be set up to be used as an initial values file for the cavitation simulation later on in the tutorial.

26.4.1. Defining the Case Using CFX-Pre

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `CavitationIni`.
5. Click **Save**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.4.1.1. Importing the Mesh

1. Right-click Mesh and select **Import Mesh** > **CFX Mesh**.

   The **Import Mesh** dialog box appears.

2. Configure the following setting(s):

Setting	Value
File name	Cavitation.gtm

3. Click **Open**.

26.4.1.2. Loading Materials

Because this tutorial uses water at 25 °C and water vapor at 25 °C, you need to load these materials. Note that you will use only the liquid water for the first part of the tutorial. The vapor is being loaded now in anticipation of using it for the cavitation model later in the tutorial.

1. In the **Outline** tree view, right-click Simulation > Materials and select **Import Library Data**.

   The **Select Library Data to Import** dialog box is displayed.

2. Expand Water Data.

3. While holding down the **Ctrl** key, select both Water Vapour at 25 C and Water at 25 C.

4. Click **OK**.

26.4.1.3. Creating the Domain

1. Edit Case Options > General in the **Outline** tree view and ensure that **Automatic Default Domain** is turned on.

   A domain named Default Domain should appear under the Simulation branch.

2. Rename Default Domain to Pump.

3. Edit Pump.

4. Under **Fluid and Particle Definitions**, delete Fluid 1 and create a new fluid definition named Liquid Water.

5. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Fluid and Particle Definitions**	Liquid Water

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Fluid and Particle Definitions > Liquid Water > **Material**	Water at 25 C [a]
	Domain Models > Pressure > **Reference Pressure**	0 [atm]
	Domain Models > Domain Motion > **Option**	Rotating
	Domain Models > Domain Motion > **Angular Velocity**	132 [radian s^-1]

a. Click the *Ellipsis* icon ... to open the **Material** dialog box.

6. Click **OK**.

26.4.1.4. Creating the Boundaries

26.4.1.4.1. Inlet Boundary

1. Create a new boundary named `Inlet`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Inlet
	Location	INBlock INFLOW
Boundary Details	Mass And Momentum > **Option**	Stat. Frame Tot. Press
	Mass And Momentum > **Relative Pressure**	100000 [Pa]
	Flow Direction > **Option**	Cartesian Components

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Flow Direction > **X Component**	0
	Flow Direction > **Y Component**	0
	Flow Direction > **Z Component**	1
	Turbulence > **Option**	Intensity and Length Scale
	Turbulence > Option > **Fractional Intensity**	0.03
	Turbulence > Option > **Eddy Length Scale**	0.03 [m]

3. Click **OK**.

26.4.1.4.2. Outlet Boundary

1. Create a new boundary named `Outlet`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Outlet
	Location	OUTBlock OUTFLOW
Boundary Details	Mass and Momentum > **Option**	Mass Flow Rate
	Mass and Momentum > **Mass Flow Rate**	16 [kg/s]

3. Click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.4.1.4.3. Wall Boundaries

Set up the hub and shroud to be a stationary (non-rotating) wall.

1. Create a new boundary named `Stationary Wall`.
2. Configure the following setting(s) of `Stationary Wall`:

Tab	Setting	Value
Basic Settings	**Boundary Type**	Wall
	Location	OUTBlock HUB, OUTBlock SHROUD [a]
Boundary Details	Mass and Momentum > **Wall Velocity**	(Selected)
	Mass and Momentum > Wall Velocity > **Option**	Counter Rotating Wall

a. Click the *Ellipsis* icon to open the **Selection Dialog** dialog box. In that dialog box, select multiple items while holding down the **Ctrl** key. Click **OK**.

3. Click **OK**.

26.4.1.5. Creating Domain Interfaces

1. Click **Insert** > **Domain Interface** and, in the dialog box that appears, set **Name** to `Periodic Interface` and click **OK**.
2. Configure the following setting(s) of `Periodic Interface`:

Tab	Setting	Value
Basic Settings	Interface Side 1 > **Region List**	INBlock PER1, OUTBlock PER1, Passage PER1
	Interface Side 2 > **Region List**	INBlock PER2, OUTBlock PER2, Passage PER2
	Interface Models	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Option**	Rotational Periodicity

3. Click **OK**.

26.4.1.5.1. Inblock to Passage Interface

1. Select **Insert** > **Domain Interface** and in the dialog box that appears, set **Name** to `Inblock to Passage Interface` and click **OK**.

2. Configure the following settings of `Inblock to Passage Interface`:

Tab	Setting	Value
Basic Settings	Interface Side 1 > **Region List**	 OUTFLOW INBlock
	Interface Side 2 > **Region List**	 INFLOW Passage
Mesh Connection	Mesh Connection Method > Mesh Connection > **Option**	 1:1

3. Click **OK**.

26.4.1.5.2. Passage to Outblock Interface

1. Select **Insert** > **Domain Interface** and in the dialog box that appears, set the Name to `Passage to Outblock Interface` and click **OK**.

2. Configure the following settings of `Passage to Outblock Interface`:

Tab	Setting	Value
Basic Settings	Interface Side 1 > **Region List**	 OUTFLOW Passage
	Interface Side 2 > **Region List**	 INFLOW OUTBlock
Mesh Connection	Mesh Connection Method > Mesh Connection > **Option**	 1:1

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click **OK**.

With the boundary conditions and domain interfaces defined above, the default boundary of a rotating wall is applied to the blade and the upstream portions of the hub and shroud.

26.4.1.6. Setting Initial Values

The initial values that will be set up are consistent with the inlet boundary conditions settings.

1. Click *Global Initialization* .

2. Configure the following setting(s):

Tab	Setting	Value
Global Settings	Initial Conditions > Cartesian Velocity Components > **Option**	 Automatic with Value
	Initial Conditions > Cartesian Velocity Components > **U**	 0 [m/s]
	Initial Conditions > Cartesian Velocity Components > **V**	 0 [m/s]
	Initial Conditions > Cartesian Velocity Components > **W**	 1 [m/s]
	Initial Conditions > Static Pressure > **Option**	 Automatic with Value
	Initial Conditions > Static Pressure > **Relative Pressure**	 100000 [Pa]
	Initial Conditions	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Turbulence > **Option**	 Intensity and Length Scale
	Initial Conditions > Turbulence > Fractional Intensity > **Option**	 Automatic with Value
	Initial Conditions > Turbulence > Fractional Intensity > **Value**	 0.03
	Initial Conditions > Turbulence > Eddy Length Scale > **Option**	 Automatic with Value
	Initial Conditions > Turbulence > Eddy Length Scale > **Value**	 0.03 [m]

3. Click **OK**.

26.4.1.7. Setting Solver Controls

1. Click *Solver Control* .

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Convergence Control > **Max Iterations**	 500
	Convergence Control	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Fluid Timescale Control > **Timescale Control**	 Physical Timescale
	Convergence Control > Fluid Timescale Control > **Physical Timescale**	 1e-3[s] [a]
	Convergence Criteria > **Residual Target**	 7.5e-6
a. The physical timescale that will be set up is derived from the rotational speed of the blades and the fact that there are 7 blades in the full machine.		

3. Click **OK**.

26.4.1.8. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .
2. Configure the following setting(s):

Setting	Value
File name	CavitationIni.def

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, **Solver Input File** is set.

4. Save the simulation.

26.4.2. Obtaining the Solution Using CFX-Solver Manager

CFX-Solver Manager should be running. You will be able to obtain a solution to the CFD problem by following the instructions below.

1. Ensure that the **Define Run** dialog box is displayed.
2. Click **Start Run**.

   You may see a notice about an artificial wall at the inlet. This notice indicates that the flow is trying to exit at the inlet. This can be ignored because the amount of reverse flow is very low.

   CFX-Solver runs and attempts to obtain a solution. At the end of the run, a dialog box is displayed stating that the simulation has ended.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Select **Post-Process Results**.
4. If using stand-alone mode, select **Shut down CFX-Solver Manager**.
5. Click **OK**.

26.4.3. Viewing the Results Using CFD-Post

CFD-Post should be running.

This case is run with temperatures around 300 K. The vapor pressure of water at this temperature is around 3574 Pa. To confirm that water vapor or cavitation is not likely for this operating condition of the pump, an isosurface of pressure at 3574 Pa will be created.

Create an isosurface of pressure at 3574 [Pa]:

1. Select **Insert** > **Location** > **Isosurface** and accept the default name.
2. Configure the following setting(s) in the details view:

Tab	Setting	Value
Geometry	Definition > **Variable**	 Pressure
	Definition > **Value**	 3574 [Pa]

3. Click **Apply**.

   Notice that the isosurface does not appear. There is no place in the blade passage where the pressure is equal to 3574 Pa, which implies that there is no water vapor.
4. Quit CFD-Post, saving the state at your discretion.

26.5. Simulating the Pump with Cavitation and High Inlet Pressure

The simulation will be modified in CFX-Pre to include water vapor and enable the cavitation model. Monitor points will also be defined to observe the Net Positive Suction Head (NSPH) and pressure head values.

26.5.1. Defining the Case Using CFX-Pre

The following topics are discussed:

26.5.1.1. Modifying the Domain and Boundary Conditions
26.5.1.2. Creating Expressions
26.5.1.3. Adding Monitor Points
26.5.1.4. Writing the CFX-Solver Input (.def) File

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.5.1.1. Modifying the Domain and Boundary Conditions

1. Ensure that the following tutorial input files are in your working directory:

   - `CavitationIni.cfx`
   - `CavitationIni_001.res`

2. Set the working directory and start CFX-Pre if it is not already running.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

3. Open `CavitationIni.cfx` and save it as `Cavitation_100000.cfx`.

   "100000" indicates the inlet pressure of the simulation.

4. Open `Pump` for editing.

5. In the **Fluid and Particle Definitions** section, click *Add new item* and name it `Water Vapor`.

6. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Fluid and Particle Definitions**	Liquid Water
	Fluid and Particle Definitions > Liquid Water > **Material**	Water at 25 C [a]
	Fluid and Particle Definitions	Water Vapor
	Fluid and Particle Definitions > Water Vapor > **Material**	Water Vapour at 25 C
Fluid Models	Multiphase > **Homogeneous Model**	(Selected) [b]
Fluid Pair Models	Fluid Pair > Liquid Water | Water Vapor > Mass Transfer > **Option**	Cavitation

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Fluid Pair > Liquid Water | Water Vapor > Mass Transfer > Cavitation > **Option**	Rayleigh Plesset
	Fluid Pair > Liquid Water | Water Vapor > Mass Transfer > Cavitation > **Mean Diameter**	2e-6 [m]
	Fluid Pair > Liquid Water | Water Vapor > Mass Transfer > Cavitation > **Saturation Pressure**	(Selected)
	Fluid Pair > Liquid Water | Water Vapor > Mass Transfer > Cavitation > Saturation Pressure > **Saturation Pressure**	3574 [Pa] [c]

a. Click the *Ellipsis* icon to open the **Material** dialog box, then click the *Import Library Data* icon to open the **Select Library Data to Import** dialog box. In that dialog box, expand `Water Data` in the tree, then multi-select `Water at 25 C` and `Water Vapour at 25 C` and click **OK**.

b. The homogeneous model will be selected because the interphase transfer rate is very large in the pump. This results in all fluids sharing a common flow field and turbulence.

c. The pressure for a corresponding saturation temperature at which the water in the pump will boil into its vapor phase is 3574 Pa.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Click **OK**.

   Error messages appear, but you will correct those problems in the next steps.

8. Open `Inlet` for editing and configure the following setting(s):

   Note that you are setting the inlet up to be 100% liquid water, hence a volume fraction of 1. Consequently, the volume fraction of the vapor is set to 0.

Tab	Setting	Value
Fluid Values	**Boundary Conditions**	Water Vapor
	Boundary Conditions > Water Vapor > Volume Fraction > **Volume Fraction**	0
	Boundary Conditions	Liquid Water
	Boundary Conditions > Liquid Water > Volume Fraction > **Volume Fraction**	1

9. Click **OK**.

10. Open `Outlet` for editing.

11. On the **Boundary Details** tab, set **Mass and Momentum > Option** to `Bulk Mass Flow Rate` and **Mass Flow Rate** to `16 [kg s^-1]`.

12. Click **OK**.

    The problems have been resolved and the error messages have disappeared.

26.5.1.2. Creating Expressions

Expressions defining the Net Positive Suction Head (NPSH) and Head are created in order to monitor their values as the inlet pressure is decreased. By monitoring these values a drop curve can be produced.

Create the following expressions.

Name	Definition
Ptin	massFlowAve(Total Pressure in Stn Frame)@Inlet

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition
Ptout	massFlowAve(Total Pressure in Stn Frame)@Outlet
Wden	996.82 [kg m^-3]
Head	(Ptout-Ptin)/(Wden*g)
NPSH	(Ptin- Pvap)/(Wden*g)
Pvap	3574 [Pa]

26.5.1.3. Adding Monitor Points

Two monitor points will be added to track the NPSH and head using the expressions created in the previous step.

1. Click *Output Control* .

2. On the **Monitor** tab, select **Monitor Objects** and click *Add new item* .

3. Enter `NPSH Point` as the name of the monitor point then enter the following settings:

Tab	Setting	Value
Monitor	Monitor Objects > Monitor Points and Expressions > NPSH Point > **Option**	 Expression
	Monitor Objects > Monitor Points and Expressions > NPSH Point > **Expression Value**	 NPSH

4. Create a second monitor point named `Head Point` with the same parameters as the first, with the exception that **Expression Value** is set to `Head`.

5. Click **OK**.

26.5.1.4. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .

2. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
File name	Cavitation_100000.def

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, **Solver Input File** is set.

4. Save the simulation.

26.5.2. Obtaining the Solution using CFX-Solver Manager

CFX-Solver Manager should be running. Obtain a solution to the CFD problem by following these instructions:

1. Ensure that the **Define Run** dialog box is displayed.
2. Under the **Initial Values** tab, select **Initial Values Specification**.
3. Select `CavitationIni_001.res` for the initial values file using the *Browse* tool.
4. Click **Start Run**.

   You may see a notice about an artificial wall at the inlet. This notice indicates that the flow is trying to exit at the inlet. This can be ignored because the amount of reverse flow is very low.

   CFX-Solver runs and attempts to obtain a solution. A dialog box is displayed stating that the simulation has ended.

5. Select **Post-Process Results**.
6. If using stand-alone mode, select **Shut down CFX-Solver Manager**.
7. Click **OK**.

26.5.3. Viewing the Results Using CFD-Post

You will create an isosurface to observe the volume fraction of water vapor at 25 °C. Note that the pressure below the threshold is the same as found earlier in Simulating the Pump with High Inlet Pressure (p. 619).

Create an isosurface for the volume fraction of water vapor at 25 °C, at 0.1:

1. CFD-Post should be running; start it if necessary.
2. Click **Insert** > **Location** > **Isosurface** and accept the default name.
3. Configure the following setting(s) in the details view:

Tab	Setting	Value
Geometry	Definition	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Variable**	Water Vapor.Volume Fraction
	Definition > **Value**	 0.1

4. Click **Apply**.

   Notice that the isosurface is clear. There is no water vapor at 25 °C in the blade passage for the simulation with cavitation because at an inlet total pressure of 100000 Pa, the minimum static pressure in the model is above the vapor pressure.

5. Quit CFD-Post saving the state at your discretion.

26.6. Simulating the Pump with Cavitation and a Range of Inlet Pressures

In order to construct a drop curve for this cavitation case, the inlet pressure must be decremented from its initial value of 100000 Pa to 17500 Pa, and the Head and NPSH values must be recorded for each simulation. The results are provided in Table 26.1: Pump Performance Data (p. 634).

Table 26.1: Pump Performance Data

Inlet Pressure Pa	NPSH m	Head m
100000	9.859e+00	3.537e+01
80000	7.813e+00	3.535e+01
60000	5.767e+00	3.535e+01
40000	3.721e+00	3.536e+01
30000	2.698e+00	3.538e+01
20000	1.675e+00	3.534e+01
18000	1.470e+00	3.528e+01
17500	1.419e+00	3.184e+01

Optionally, if you want to generate the data shown in Table 26.1: Pump Performance Data (p. 634), then follow the instructions in the following two sections (Writing CFX-Solver Input (.def) Files for Lower Inlet Pressures (p. 635) and Obtaining the Solutions using CFX-Solver Manager (p. 635)). Those instructions involve running several simulations in order to obtain a set of results files. As a benefit to doing this, you will have the results files required to complete an optional postprocessing exercise at the end of this tutorial. This optional postprocessing exercise involves using isosurfaces to visualize the regions of cavitation, and visually comparing these isosurfaces between different results files.

If you want to use the provided table data to produce a drop curve, proceed to Generating a Drop Curve (p. 636).

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.6.1. Writing CFX-Solver Input (.def) Files for Lower Inlet Pressures

Produce a set of definition (`.def`) files for the simulation, with each definition file specifying a progressively lower value for the inlet pressure:

1. CFX-Pre should be running; start it if necessary.
2. Open `Cavitation_100000.cfx`.
3. Open `Simulation` > `Flow Analysis 1` > `Pump` > `Inlet`.
4. On the **Boundary Details** tab change **Mass and Momentum** > **Relative Pressure** to `80000 [Pa]`.
5. Click **OK**.
6. Right-click `Simulation` and select **Write Solver Input File**.
7. Set **File name** to `Cavitation_80000.def`.
8. Click **Save**.
9. Change the inlet pressure and save a corresponding CFX-Solver input file for each of the 6 other pressures: 60000 Pa, 40000 Pa, 30000 Pa, 20000 Pa, 18000 Pa, and 17500 Pa.

Note:

There are other techniques for defining a set of related simulations. For example, you could use configuration control, as demonstrated in Flow from a Circular Vent (p. 163).

26.6.2. Obtaining the Solutions using CFX-Solver Manager

Run each of the CFX-Solver input files that you created in the previous step:

1. Start CFX-Solver Manager if it is not already running.
2. Ensure that the **Define Run** dialog box is displayed.
3. Under **Solver Input File**, click *Browse* and select `Cavitation_80000.def`.
4. Under the **Initial Values** tab, select **Initial Values Specification**.
5. Select `Cavitation_100000_001.res` for the initial values file using the *Browse* tool.
6. Click **Start Run**.

   You may see a notice about an artificial wall at the inlet. This notice indicates that the flow is trying to exit at the inlet. This can be ignored because the amount of reverse flow is very low.

   CFX-Solver runs and attempts to obtain a solution. At the end of the run, a dialog box is displayed stating that the simulation has ended.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Clear **Post-Process Results**.

8. Click **OK**.

9. Repeat this process until you have run all the CFX-Solver input files for all 6 other inlet pressures: 60000 Pa, 40000 Pa, 30000 Pa, 20000 Pa, 18000 Pa, and 17500 Pa. The pump simulation with cavitation model at an inlet pressure of 17500 Pa will converge poorly because the performance of the pump is decreasing considerably around that pressure. Note that the initial values should be taken from the previously generated results (`.res`) file.

26.6.3. Viewing the Results Using CFD-Post

To see the pump performance, you will generate a drop curve to show the pump performance over a range of inlet pressures. After generating the drop curve, there is an optional exercise for visualizing the cavitation regions using isosurfaces.

The optional exercise of visualizing the cavitation regions requires the results files from the 60000 Pa, 40000 Pa, 20000 Pa, and 17500 Pa simulations. If you have not generated those results files and want to complete the optional exercise, then generate the results files by following the instructions in:

- Writing CFX-Solver Input (.def) Files for Lower Inlet Pressures (p. 635)
- Obtaining the Solutions using CFX-Solver Manager (p. 635)

26.6.3.1. Generating a Drop Curve

To generate a drop curve, you will need the values for Net Positive Suction Head (NPSH) and head as the inlet pressure decreases. This data is provided in Table 26.1: Pump Performance Data (p. 634). If you want to use that data, proceed to Creating a Table of the Head and NPSH Values (p. 636). If you have chosen to run all of the simulations and have obtained all of the results files, you can obtain the drop curve data yourself by following the instructions in the Creating a Head-versus-NPSH Chart (Optional Exercise) (p. 639) section.

26.6.3.1.1. Creating a Table of the Head and NPSH Values

1. Start CFD-Post.

2. Click **Insert** > **Table** and set the name to `Drop Curve Values`.

3. Enter the values from Table 26.1: Pump Performance Data (p. 634) for the 8 inlet pressures in the table.

   Enter the NPSH values in the left column and the head values in the right column.

4. Click *Save Table* , and configure the following setting(s):

Setting	Value
File name	Drop Curve Values
Files of type	Comma Separated Values — Excel Readable (*.csv)

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Click **Save**.

26.6.3.1.2. Creating a Head-versus-NPSH Chart

1. Click **Insert** > **Chart**.
2. Set the name to `Drop Curve` and click **OK**.
3. Configure the following setting(s):

Tab	Setting	Value
General	**Title**	Drop Curve
Data Series	Data Source > **File**	(Selected)
	Data Source > File > **Browse**	Drop Curve Values.csv [a]
X Axis	Axis Range > **Determine ranges automatically**	(Cleared)
	Axis Range > **Min**	0
	Axis Range > **Max**	10
	Axis Labels > **Use data for axis labels**	(Cleared)
	Axis Labels > **Custom Label**	NPSH [m]
Y Axis	Axis Range > **Determine ranges automatically**	(Cleared)
	Axis Range > **Min**	0
	Axis Range > **Max**	45
	Axis Labels > **Use data for axis labels**	(Cleared)

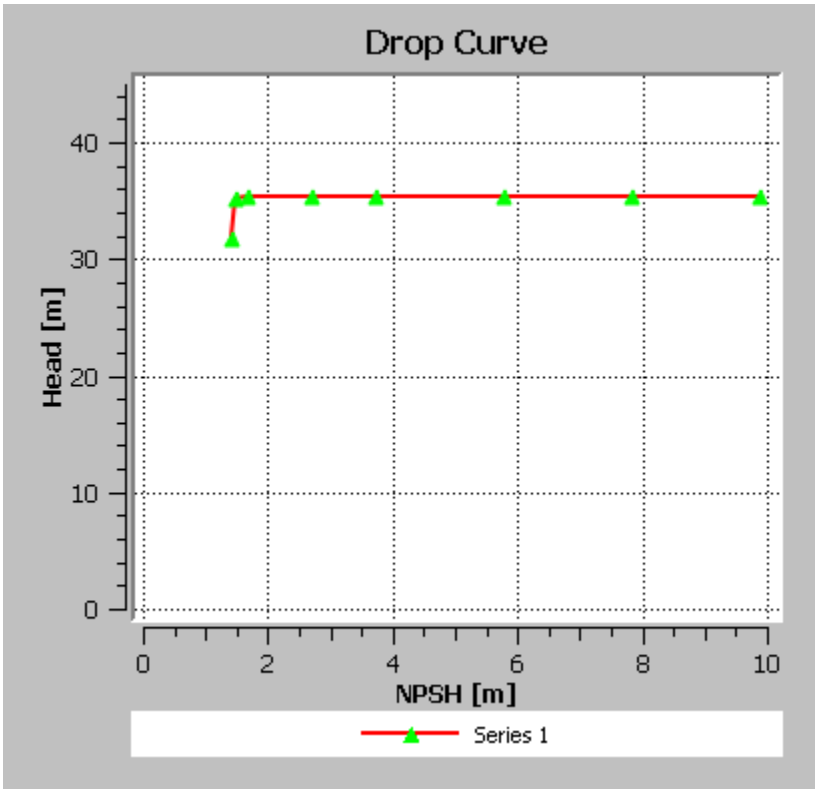
Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Axis Labels > **Custom Label**	 Head [m]
Line Display	Line Display > **Symbols**	 Triangle
a. Created in previous steps.		

4. Click **Apply**.

26.6.3.1.3. Viewing the Drop Curve

Here is what the drop curve created in the earlier steps should look like:



You can see here that there is not significant degradation in the performance curve as the inlet total pressure is dropped. This is due to the fact that, for a part of the test, the inlet total pressure is sufficiently high to prevent cavitation, which implies that the normalized pressure rise across the pump is constant. Also, although you may start at a high inlet pressure where there is no cavitation, as you drop the inlet pressure, cavitation will appear but will have no significant impact on performance (incipient cavitation) until the blade passage has sufficient blockage due to vapor. At that point, performance degrades (rapidly in this case).

When the inlet total pressure reaches a sufficiently low value, cavitation occurs. The performance curve then starts to drop at a pressure of 18000 Pa as you can see on the chart. What is called

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the point of cavitation is often marked by the NPSH at which the pressure rise has fallen by a few percent, which is around 17500 Pa in this case.

If you want to complete an optional exercise on visualizing the cavitation regions, proceed to Visualizing the Cavitation Regions (Optional Exercise) (p. 641). Otherwise, quit CFD-Post, saving the state at your discretion.

26.6.3.1.4. Creating a Head-versus-NPSH Chart (Optional Exercise)

1. Start CFD-Post.

2. To load the results file, select **File** > **Load Results** or click *Load Results* .

3. On the right side of the **Load Results File** dialog box, note down the current setting under **CFX run history and multi-configuration options**. Set this option to **Load complete history as:** > **A single case**, unless already set.

   Important:

   This setting, under **CFX run history and multi-configuration options**, persists when you close CFD-Post. Ensure that you set this back to the original setting noted above, as instructed to do so at the end of the tutorial. Not doing so could lead to undesirable results when postprocessing other cases.

4. In the **Load Results File** dialog box, select `Cavitation_17500_001.res`.

5. Click **Open**.

   When you started the CFX-Solver run using initial values, by default the **Continue History From** option was on. This enables the results file to retain a reference to the initial value results file. When the final results file is loaded into CFD-Post using the **Load complete history as: A single case**, it includes results from all the initial values files as well as the final results. Each of the previous initial values files is available as a time step (in this case a sequence) through the **Timestep Selector**.

6. Click **OK** when prompted with a **Process Multiple Results as a Sequence** message.

7. Click **Insert** > **Chart** or click *Chart* .

8. Set the name to `Drop Curve` and click **OK**.

9. Configure the following setting(s):

Tab	Setting	Value
General	**Type**	XY - Transient or Sequence
	Title	Drop Curve
Data Series	Data Source > **Expression**	 (Selected)

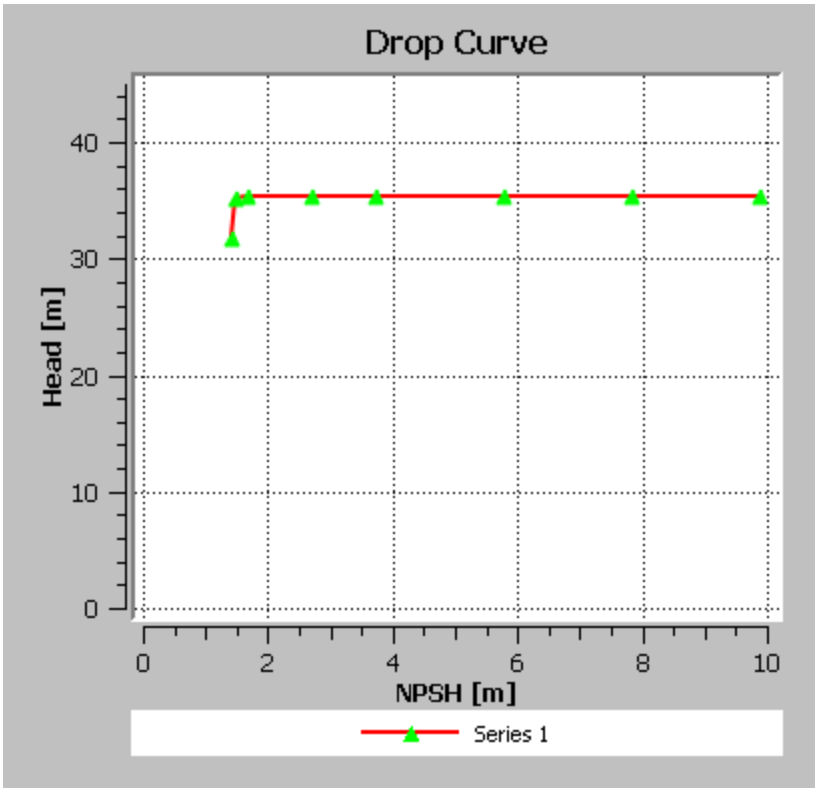
Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Data Source > **Expression**	 Head
X Axis	Data Selection > **Expression**	 NPSH
Y Axis	Axis Range > **Determine ranges automatically**	 (Cleared)
	Axis Range > **Min**	 0
	Axis Range > **Max**	 45
Line Display	Line Display > **Symbols**	 Triangle

10. Click **Apply**.

26.6.3.1.5. Viewing the Drop Curve

Here is what the drop curve created in the earlier steps should look like:



Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can see here that there is not significant degradation in the performance curve as the inlet total pressure is dropped. This is due to the fact that, for a part of the test, the inlet total pressure is sufficiently high to prevent cavitation, which implies that the normalized pressure rise across the pump is constant. Also, although you may start at a high inlet pressure where there is no cavitation, as you drop the inlet pressure, cavitation will appear but will have no significant impact on performance (incipient cavitation) until the blade passage has sufficient blockage due to vapor. At that point, performance degrades (rapidly in this case).

When the inlet total pressure reaches a sufficiently low value, cavitation occurs. The performance curve then starts to drop at a pressure of 18000 Pa as you can see on the chart. What is called the point of cavitation is often marked by the NPSH at which the pressure rise has fallen by a few percent, which is around 17500 Pa in this case.

Important:

If you want to complete an optional exercise on visualizing the cavitation regions, proceed to Visualizing the Cavitation Regions (Optional Exercise) (p. 641). Otherwise, proceed to Restoring CFX run history and multi-configuration options (p. 642).

26.6.3.2. Visualizing the Cavitation Regions (Optional Exercise)

This is an optional part of the tutorial that requires the results files from the 60000 Pa, 40000 Pa, 20000 Pa, and 17500 Pa simulations. If you have not generated those results files and want to complete this optional exercise, then generate the results files by following the instructions in:

- Writing CFX-Solver Input (.def) Files for Lower Inlet Pressures (p. 635)
- Obtaining the Solutions using CFX-Solver Manager (p. 635)

Cavitation does not occur for the 100000 Pa and 80000 Pa simulations. Create an isosurface for 10% water vapor (by volume fraction), for the 60000 Pa, 40000 Pa, 20000 Pa, and 17500 Pa simulations:

1. Use `Cavitation_60000_001.res` to create an isosurface:
   a. Launch CFD-Post and load `Cavitation_60000_001.res`.
   b. Select **Insert** > **Location** > **Isosurface** and accept the default name.
   c. Configure the following setting(s) in the details view:

Tab	Setting	Value
Geometry	Definition > **Variable**	Water Vapor.Volume Fraction
	Definition > **Value**	0.1

   d. Click **Apply**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Add `Cavitation_40000_001.res` to the current results:

   a. Select **File > Load Results**.

   b. Under **Case options**, select both **Keep current cases loaded** and **Open in new view**.

   c. Select `Cavitation_40000_001.res`.

   d. Click **Open**.

   e. Click a blank area inside the viewport named **View 2** (which contains the results that you just loaded) to make that viewport active, then turn on visibility for the isosurface in the **Outline** tree view.

3. In a similar way, load `Cavitation_20000_001.res` and `Cavitation_17500_001.res` and make the isosurface visible on these results.

4. Click *Synchronize camera in displayed views* so that all viewports maintain the same camera position.

5. Rotate the view (from any viewport) to inspect the results.

   Observe that the amount of water vapor increases as the inlet pressure decreases.

Important:

If you created the drop curve by setting the **CFX run history and multi-configuration options**, proceed to Restoring CFX run history and multi-configuration options (p. 642). Otherwise, quit CFD-Post, saving the state at your discretion.

26.6.3.3. Restoring CFX run history and multi-configuration options

As mentioned above the setting under **CFX run history and multi-configuration options** persists when you close CFD-Post. This section outlines the steps to restore **CFX run history and multi-configuration options** to its original setting.

1. Select **File** > **Close** to close the current file.

2. Click **Close** if prompted to save.

3. Load a results file by selecting **File** > **Load Results** or click *Load Results* .

4. On the right side of the **Load Results File** dialog box, restore the original settings under **CFX run history and multi-configuration options**.

5. In the **Load Results File** dialog box, select `Cavitation_17500_001.res`.

6. Click **Open**.

7. Quit CFD-Post, by selecting **File** > **Quit**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 27: Spray Dryer

This tutorial includes:

27.1. Tutorial Features

27.2. Overview of the Problem to Solve

27.3. Preparing the Working Directory

27.4. Defining the Case Using CFX-Pre

27.5. Obtaining the Solution Using CFX-Solver Manager

27.6. Viewing the Results Using CFD-Post

27.1. Tutorial Features

In this tutorial you will learn about:

- Importing a CCL file in CFX-Pre.
- Editing and creating boundary conditions in CFX-Pre.
- Adding particles that evaporate.
- Creating a domain interface in CFX-Pre.
- Creating contour plots and inserting particle tracking in CFD-Post.

Component	Feature	Details
CFX-Pre	User Mode	General mode
	Analysis Type	Steady State
	Fluid Type	General Fluid
	Domain Type	Single Domain
	Boundary Conditions	Water Nozzle
		Air Inlet
		Outlet
		Domain 1 Default
	Domain Interface	Fluid Fluid
	Timescale	Physical Timescale
	Particle Coupling Control	Selected
	Extra Output Variables List	Selected

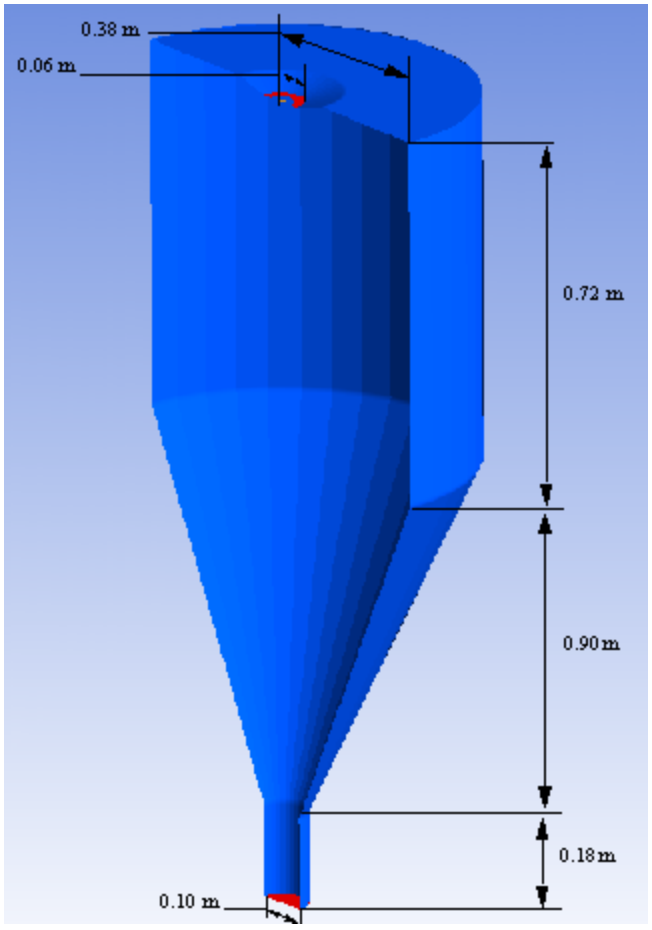
Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Component	Feature	Details
CFD-Post	Plots	Contour Plots
		Particle Tracking

27.2. Overview of the Problem to Solve

In this example, a spray dryer is modeled in which water drops are evaporated by a hot air flow. The goal of this tutorial is to observe the variation of gas temperature and mass fraction of water vapor, and of averaged values of mean droplet diameter and droplet temperature in the spray dryer, as well as the temperature and size of individual water drops as they travel through the spray dryer.

The following figure shows approximately half of the full geometry. The spray dryer has two inlets named `Water Nozzle` and `Air Inlet`, and one outlet named `Outlet`. The `Water Nozzle` is where the liquid water enters in a primary air flow at a mass flow rate of 1.33e-4 kg/s. The `Air Inlet` is for the swirling, drying air flow. The `Water Nozzle` inlet is located in the middle of the circular `Air Inlet`. When the spray dryer is operating, the inlets are located at the top of the vessel and the outlet at the bottom.



Periodic boundaries are used to enable only a small section of the full geometry to be modeled. The geometry to be modeled consists of a 9 degree section of the axisymmetric dryer shape. The relevant parameters of this problem are:

- Static temperature at `Water Nozzle` = 300 K

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Size distribution for the drops being created by the `Water Nozzle` is prescribed using discrete diameter values and associated fractions of the droplet mass flow rate.
- `Air Inlet` mass and momentum axial component = 30 m/s (downwards along the axis of the spray dryer), `Air Inlet` mass and momentum radial component = 0 m/s, `Air Inlet` mass and momentum theta component = 10 m/s
- Static temperature at `Air Inlet` = 423 K
- Relative pressure at `Outlet` = 0 Pa
- Normal speed of Water = 10 m/s

The approach for solving this problem is to first import a CCL file with the fluid properties, domain and boundary conditions in CFX-Pre. Minor changes will be made to the information imported from the CCL file. Boundary conditions and a domain interface will also be added. In CFD-Post, contour plots will be created to see the variation of temperature, mass fraction of water, average mean particle diameter of liquid water, and averaged temperature of liquid water in the spray dryer. Finally, particle tracking will be used for plotting the temperature of liquid water.

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- Running Ansys CFX Tutorials Using Ansys Workbench (p. 34)
- Changing the Display Colors (p. 37)

27.3. Preparing the Working Directory

1. Create a working directory.

   Ansys CFX uses a working directory as the default location for loading and saving files for a particular session or project.

2. Download the `spray_dryer.zip` file here.

3. Unzip `spray_dryer.zip` to your working directory.

   Ensure that the following tutorial input file is in your working directory:

   - `spraydryer9.gtm`

4. Set the working directory and start CFX-Pre.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

27.4. Defining the Case Using CFX-Pre

This section describes the step-by-step definition of the flow physics in CFX-Pre for a steady-state simulation.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** and click **OK**.
3. Edit `Case Options` > `General` in the **Outline** tree view and ensure that **Automatic Default Domain** is turned off.
4. Click **OK**.
5. Select **File** > **Save Case As**.
6. Under **File name**, type `SprayDryer`.
7. Click **Save**.

Later in this tutorial, you will import a template that sets up, among other things, a domain.

27.4.1. Importing the Mesh

1. Right-click `Mesh` and select **Import Mesh** > **CFX Mesh**.

   The **Import Mesh** dialog box appears.

2. Configure the following setting(s):

Setting	Value
File name	spraydryer9.gtm

3. Click **Open**.

27.4.2. Importing the Evaporating CCL Drops Model Template

Ansys CFX Command Language (CCL) consists of commands used to carry out actions in CFX-Pre, CFX-Solver Manager and CFD-Post. The physics for this simulation such as materials, domain, and domain properties will be imported from a CCL file. Model library templates contain CCL physics definitions for complex simulations and are located in the <*CFXROOT*>`/etc/model-templates/` directory. You will first analyze the `evaporating_drops.ccl` model template and then import it into the simulation.

Note:

The physics for a simulation can be saved to a CCL (CFX Command Language) file at any time by selecting **File** > **Export** > **CCL**.

---

1. Browse to <*CFXROOT*>`/etc/model-templates/`, open `evaporating_drops.ccl` with a text editor, and take the time to look at the information it contains.

   The template sets up the materials water vapor H2O with a thermal conductivity of 193e-04 W/mK and water liquid H2Ol, which enters from the Water Nozzle. Note that the water data could also have been imported from the library in CFX-Pre. The template also creates a continuous gas

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

phase named `Gas mixture` containing H2O and `Air Ideal Gas` and a binary mixture of H2O and H2Ol, which determines the rate of evaporation of the water. A domain named `Domain 1` that includes the `Gas mixture` and H2Ol as a fluid pair as well as an inlet boundary is also specified in the CCL file. The inlet boundary is set up with a default static temperature of 573 K.

Note:

When viewing a template file in a text editor, be careful to avoid modifying the original file.

2. Select **File** > **Import** > **CCL**

   The **Import CCL** dialog box appears.

3. Under **Import Method**, select **Append**. This will start with the existing CCL already generated and append the imported CCL.

Note:

Replace is useful if you have defined physics and want to update or replace them with newly-imported physics.

4. Browse to *<CFXROOT>*`/etc/model-templates/` and select `evaporating_drops.ccl`.

5. Click **Open**.

Note:

An error message related to the parameter `Location` will appear in the message window. This error occurs as the CCL contains a location placeholder that is not part of the mesh. Ignore this error message as the issue will be addressed when `Domain 1` is being edited.

27.4.3. Editing the Domain

The fluid domain imported in the CCL file will be edited in this section.

1. In the tree view, right-click `Domain 1`, then click **Edit**.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Location and Type > **Location**	 B34
	Domain Models > Buoyancy Model	

Tab	Setting	Value
	> **Option**	Buoyant
	Domain Models > Buoyancy Model > **Gravity X Dirn.**	0.0 [m s^-2]
	Domain Models > Buoyancy Model > **Gravity Y Dirn.**	-9.81 [m s^-2]
	Domain Models > Buoyancy Model > **Gravity Z Dirn.**	0.0 [m s^-2]
	Domain Models > Buoyancy Model > **Buoy. Ref. Density**	1.2 [kg m^-3] [a]
Fluid Specific Models	**Fluid**	Gas mixture
	Fluid > Gas mixture > Fluid Buoyancy Model > **Option**	Non Buoyant [b]
	Fluid	H2Ol
	Fluid > H2Ol > Fluid Buoyancy Model > **Option**	Density Difference

a. The buoyancy reference density is set to 1.2 as representative of air.

b. Because any natural convection in the gas can be neglected, you can set the fluid to non buoyant.

3. Click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

27.4.4. Creating and Editing the Boundary Conditions

In this section, the `Inlet` and `Domain 1 Default` boundary conditions that were imported in the CCL file will be edited. Two boundary conditions, `Air Inlet` and `Outlet` will also be created for the spray dryer simulation.

27.4.4.1. Water Nozzle Boundary

The inlet boundary where the water enters in a primary air flow will be renamed and edited with the particle mass flow rate set consistent with the problem description. The particle diameter distribution will be set to `Discrete Diameter Distribution`, which will enable us to have particles of more than one specified diameter. Diameter values will be listed as specified in the problem description. A mass fraction as well as a number fraction will be specified for each of the diameter entries. The total of mass fractions and the total of number fractions will sum to unity.

1. In the tree view, under `Domain 1`, right-click `inlet`, then click **Rename**. Set the new name to `Water Nozzle`.
2. In the tree view, right-click `Water Nozzle`, then click **Edit**.
3. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Inlet
	Location	two fluid nozzle
Boundary Details	Mass and Momentum > **Option**	Normal Speed
	Mass and Momentum > **Normal Speed**	10.0 [m s^-1]
	Heat Transfer > **Option**	Static Temperature
	Heat Transfer > **Static Temperature**	300.0 K
Fluid Values	H2Ol > Mass and Momentum > **Option**	Normal Speed
	H2Ol > Mass and Momentum > **Normal Speed**	10.0 [m s^-1]
	H2Ol > Particle Position	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Number of Positions > **Number**	500 [a]
	H2Ol > Particle Mass Flow > **Mass Flow Rate**	3.32e-6 [kg s^-1] [b]
	H2Ol > Particle Diameter Distribution > **Option**	Discrete Diameter Distribution
	H2Ol > Particle Diameter Distribution > **Diameter List**	5.9e-6, 1.25e-5, 1.39e-5, 1.54e-5, 1.7e-5, 1.88e-5, 2.09e-5, 2.27e-5, 2.48e-5, 3.11e-5 [m]
	H2Ol > Particle Diameter Distribution > **Mass Fraction List**	10*0.1
	H2Ol > Particle Diameter Distribution > **Number Fraction List**	10*0.1
	H2Ol > Heat Transfer > **Option**	Static Temperature
	H2Ol > Heat Transfer > **Static Temperature**	300.0 K

a. The number of representative drops was chosen to be 500 through experience of particle transport calculations.

b. Note that this mass flow is only 1/40th of the total mass flow rate of water because only a 9 degree sector is modeled.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Click **OK**.

27.4.4.2. Air Inlet Boundary

A second inlet in which the swirling, drying air flow will enter will be created with temperature component, mass and momentum axial, radial and theta components set consistent with the problem description.

1. Select **Insert** > **Boundary** from the main menu or click *Boundary* .
2. Under **Name**, type `Air Inlet`.
3. Click **OK**.
4. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Inlet
	Location	air inlet
Boundary Details	Mass and Momentum > **Option**	Cyl. Vel. Components
	Mass and Momentum > **Axial Component**	-30.0 [m s^-1]
	Mass and Momentum > **Radial Component**	0.0 [m s^-1]
	Mass and Momentum > **Theta Component**	10.0 [m s^-1]
	Axis Definition > **Option**	Coordinate Axis
	Axis Definition > **Rotation Axis**	Global Y
	Heat Transfer > **Option**	Static Temperature
	Heat Transfer > **Static Temperature**	423.0 K

5. Click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

27.4.4.3. Outlet Boundary

The outlet boundary will be created as an opening with pressure as specified in the problem description.

1. Select **Insert** > **Boundary** from the main menu or click *Boundary* .
2. Under **Name**, type `Outlet`.
3. Click **OK**.
4. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Outlet
	Location	outlet
Boundary Details	Mass and Momentum > **Option**	Average Static Pressure
	Mass and Momentum > **Relative Pressure**	0.0 [Pa]

5. Click **OK**.

27.4.4.4. Domain 1 Default

The `Domain 1 Default` boundary will be edited to use a heat transfer coefficient of 3.0 [W m^-2 K^-1] and an outside temperature of 300 K.

1. In the tree view, right-click `Domain 1 Default`, then click **Edit**.
2. Configure the following setting(s):

Tab	Setting	Value
Boundary Details	Heat Transfer > **Option**	Heat Transfer Coefficient
	Heat Transfer > **Heat Trans. Coeff.**	3.0 [W m^-2 K^-1]
	Heat Transfer > **Outside Temperature**	300.0 [K]

3. Click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

27.4.5. Creating a Domain Interface

A domain interface will be created to connect the `Domain 1`, `periodic1` and `periodic 2` regions. The two sides of the periodic interface, `periodic1` and `periodic 2`, will be mapped by a single rotational transformation about an axis.

1. Select **Insert** > **Domain Interface**. Accept the default name.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Interface Type**	Fluid Fluid
	Interface Side 1 > **Domain (filter)**	Domain 1
	Interface Side 1 > **Region List**	periodic1
	Interface Side 2 > **Domain (filter)**	Domain 1
	Interface Side 2 > **Region List**	periodic 2
	Interface Models > **Option**	Rotational Periodicity
	Interface Models > Axis Definition > **Rotation Axis**	Global Y
Mesh Connection	Mesh Connection Method > Mesh Connection > **Option**	Automatic

3. Click **OK**.

27.4.6. Setting Solver Control

1. Click *Solver Control* .

2. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Basic Settings	Convergence Control > **Max. Iterations**	100
	Convergence Control > Fluid Timescale Control > **Timescale Control**	Physical Timescale
	Convergence Control > Fluid Timescale Control > **Physical Timescale**	0.05 [s] [a]
	Convergence Criteria > **Residual Type**	RMS
	Convergence Criteria > **Residual Target**	1.E-4
a. Based on the air inlet speed and the size of the dryer.		

3. Click **OK**.

27.4.7. Setting Output Control

In this section, two additional variables, H2O1.Averaged Mean Particle Diameter and H2O1.Averaged Temperature will be specified. These variables will be used when viewing the results in CFD-Post to understand the flow behavior.

1. Click *Output Control* .

2. Configure the following setting(s):

Tab	Setting	Value
Results	**Extra Output Variables List**	Selected
	Extra Output Variables List > **Extra Output Var. List**	H2Ol.Averaged Mean Particle Diameter, H2Ol.Averaged Temperature [a]
a. Click the *Ellipsis* icon to open the **Extra Output Variable List** dialog box, then select `H2Ol.Averaged Mean Particle Diameter` and `H2Ol.Averaged Temperature`, holding the **Ctrl** key. Click **OK**.		

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click **OK**.

27.4.8. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .

2. Configure the following setting(s):

Setting	Value
File name	SprayDryer.def

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, **Solver Input File** is set.

4. Quit CFX-Pre, saving the simulation (`.cfx`) file.

27.5. Obtaining the Solution Using CFX-Solver Manager

When CFX-Pre has shut down and the CFX-Solver Manager has started, obtain a solution to the CFD problem by following the instructions below.

1. Ensure that the **Define Run** dialog box is displayed.

   Solver Input File should be set to `SprayDryer.def`.

2. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. At the end of the run, a dialog box is displayed stating that the simulation has ended.

3. Select **Post-Process Results**.

4. If using stand-alone mode, select **Shut down CFX-Solver Manager**.

5. Click **OK**.

27.6. Viewing the Results Using CFD-Post

In this section, contour plots located on one of the periodic regions of the spray dryer will be created to illustrate the variation of temperature, water mass fraction, liquid water average mean particle diameter and liquid water averaged temperature. Finally, particle tracking will be used for plotting the temperature of liquid water. Particle tracking will trace the mean flow behavior in and around the complex geometry of the spray dryer.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

27.6.1. Displaying the Temperature Using a Contour Plot

A contour plot located at the `Domain Interface 1 Side 1` region will first be created, and used to show the temperature variation through the spray dryer.

1. Right-click a blank area in the viewer and select **Predefined Camera** > **View From -Z**.

   This ensures that the view is set to a position that is best suited to display the results.

2. From the main menu, select **Insert** > **Contour**.

3. Set the name to `Temperature Contour`. Click **OK**.

4. Configure the following setting(s):

Tab	Setting	Value
Geometry	**Location**	Domain Interface 1 Side 1
	Variable	Temperature

5. Click **Apply**.

6. When you have finished, right-click the contour you just created in the tree view and select **Hide**.

27.6.2. Displaying the Water Mass Fraction Using a Contour Plot

A contour plot located at the `Domain Interface 1 Side 1` region will be created and used to show the `H2O.Mass Fraction` variation through the spray dryer.

- Repeat steps 1-6 in the Displaying the Temperature Using a Contour Plot (p. 656) section. In step 3, change the contour name to `H2O Mass Fraction Contour`. In step 4, change the variable to `H2O.Mass Fraction`.

27.6.3. Displaying the Liquid Water Averaged Mean Particle Diameter Using a Contour Plot

A contour plot located at the `Domain Interface 1 Side 1` region will be created and used to show the `H2Ol.Averaged Mean Particle Diameter` variation through the spray dryer.

- Repeat steps 1-6 in the Displaying the Temperature Using a Contour Plot (p. 656) section. In step 3, change the contour name to `H2Ol Averaged Mean Particle Diameter Contour`. In step 4, change the variable to `H2Ol.Averaged Mean Particle Diameter`. Click the *Ellipsis* icon to open the **Variable Selector** dialog box in order to see the entire variable list.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

27.6.4. Displaying the Liquid Water Averaged Temperature Using a Contour Plot

A contour plot located at the `Domain Interface 1 Side 1` region will be created and used to show the `H2Ol.Averaged Temperature` variation through the spray dryer.

- Repeat steps 1-6 in the Displaying the Temperature Using a Contour Plot (p. 656) section. In step 3, change the contour name to `H2Ol Averaged Temperature Contour`. In step 4, change the variable to `H2Ol.Averaged Temperature`.

27.6.5. Displaying the Liquid Water Temperature Using Particle Tracking

This section outlines the steps for using particle tracking to trace the variation of the water temperature.

1. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)**.

   This ensures that the view is set to a position that is best suited to display the results.

2. From the main menu, select **Insert** > **Particle Track**.

3. Set the name to `H2Ol Temperature`. Click **OK**.

4. Configure the following setting(s):

Tab	Setting	Value
Color	**Mode**	Variable
	Variable	H2Ol.Temperature

5. Click **Apply**.

   From the contours and particle tracks, notice that the water droplets entering the spray dryer through the `Water Nozzle` recirculates in the region between the two inlets before merging in with the stream of hot air entering the spray dryer through the `Air Inlet`. Based on this flow of the water drop, the temperature of the hot gas coming from the `Air Inlet` decreases as the process takes place. During the spray drying cycle, the air transfers its thermal energy to the liquid water drops, leading to evaporation. As the air carries the thermal energy by convection, liquid water droplets that are close to the `Air Inlet` see their temperature increase, which leads to evaporation, resulting in a decrease in droplet diameter and an increase in the amount of water vapor.

27.6.6. Displaying the Diameter of a Water Drop Using Particle Tracking

This section outlines the steps for using particle tracking to trace the variation of water droplet diameter.

1. Repeat steps 2-5 in the Displaying the Liquid Water Temperature Using Particle Tracking (p. 657) section. In step 3, change the name to `H2Ol Mean Particle Diameter`. In step 4, change the variable name to `H2Ol.Mean Particle Diameter`

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

From the water drop diameter particle track, you can see that as the air from the `Air Inlet` transfers its thermal energy to the liquid water, the diameter of water drops decreases as they evaporate. So when the water drop move away from the `Water Nozzle`, its diameter decreases as a function of the temperature increase.

2. Quit CFD-Post, saving the state (`.cst`) file at your discretion.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 28: Coal Combustion

This tutorial includes:

28.1. Tutorial Features

28.2. Overview of the Problem to Solve

28.3. Preparing the Working Directory

28.4. Simulating the Coal Combustion without Swirl and without Nitrogen Oxide

28.5. Simulating the Coal Combustion with Swirl and without Nitrogen Oxide

28.6. Simulating the Coal Combustion with Swirl and with Nitrogen Oxide

28.1. Tutorial Features

In this tutorial you will learn about:

- Importing a CCL file in CFX-Pre.
- Setting up and using Proximate/Ultimate analysis for hydrocarbon fuels in CFX-Pre.
- Viewing the results for nitrogen oxide in CFD-Post.

Component	Feature	Details
CFX-Pre	User Mode	General mode
	Analysis Type	Steady State
	Fluid Type	Reacting Mixture, Hydrocarbon Fuel
	CCL File	Import
	Domain Type	Single Domain
	Boundaries	Coal Inlet
		Air Inlet
		Outlet
		No-slip Wall
		Periodic
		Symmetry
CFD-Post	Plots	Particle Tracking

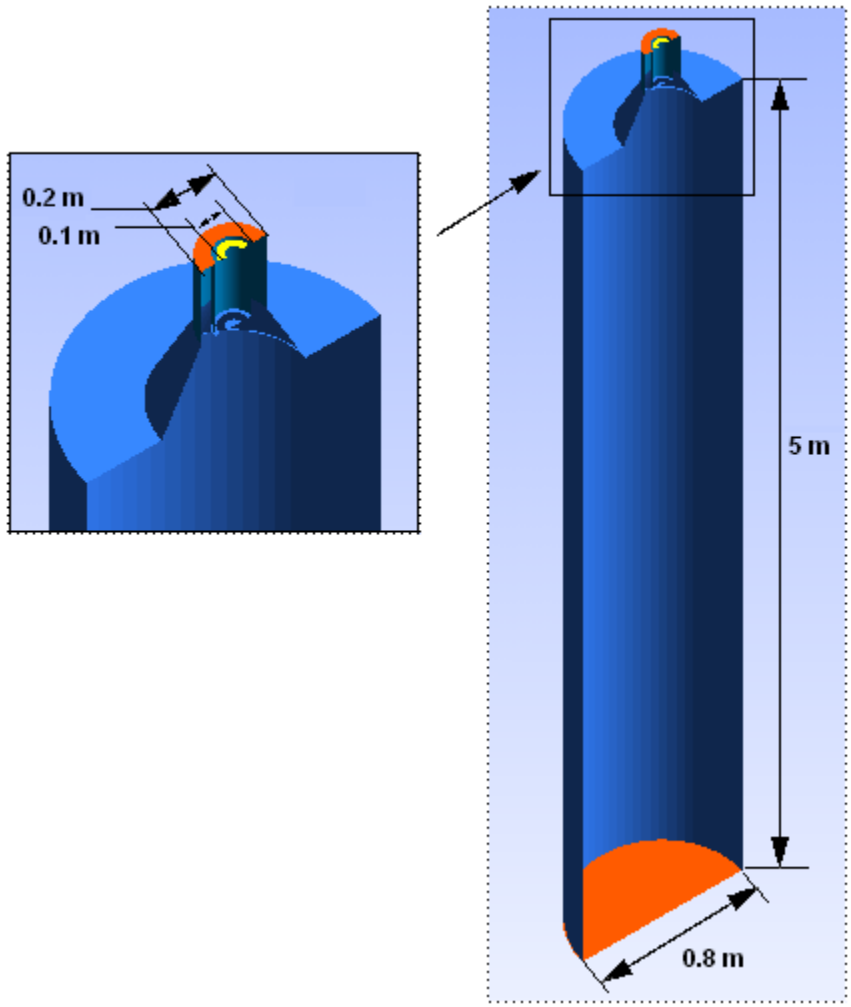
Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.2. Overview of the Problem to Solve

In this tutorial, you will model coal combustion and radiation in a furnace. Three different coal combustion simulations will be set up:

- Coal Combustion with no-swirl burners where there is no release of nitrogen oxide during the burning process.
- Coal Combustion with swirl burners where there is no release of nitrogen oxide during the burning process.
- Coal Combustion with swirl burners where there is release of nitrogen oxide during the burning process.

The following figure shows half of the full geometry. The coal furnace has two inlets: `Coal Inlet` and `Air Inlet`, and one outlet. The `Coal Inlet` (see the inner yellow annulus shown in the figure inset) has air entering at a mass flow rate of 1.624e-3 kg/s and pulverized coal particles entering at a mass flow rate of 1.015e-3 kg/s. The `Air Inlet` (see the outer orange annulus shown in the figure inset) is where heated air enters the coal furnace at a mass flow rate of 1.035e-2 kg/s. The outlet is located at the opposite end of the furnace and is at a pressure of 1 atm.



Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The provided mesh occupies a 5 degree section of an axisymmetric coal furnace. Each simulation will make use of either symmetric or periodic boundaries to model the effects of the remainder of the furnace. In the case of non-swirling flow, a pair of symmetry boundaries is sufficient; in the case of flow with swirl, a periodic boundary with rotational periodicity is required.

The relevant parameters of this problem are:

- `Coal Inlet` static temperature = 343 K
- Size distribution for the drops being created by the `Coal Inlet` = 12, 38, 62, 88 μm
- `Air Inlet` static temperature = 573 K
- `Outlet` average static pressure = 0 Pa
- `Coal Gun` wall fixed temperature = 800 K
- `Coal Inlet` wall fixed temperature = 343 K
- `Air Inlet` wall fixed temperature = 573 K
- `Furnace wall` fixed temperature = 1400 K
- O2 mass fraction = 0.232
- Proximate/ultimate analysis data for the coal. Note that proximate/ultimate analysis data is used to characterize the properties of the coal including the content of moisture, volatile, free carbon, and ash, as well as the mass fractions of carbon, hydrogen and oxygen (the major components), sulfur and nitrogen.

The approach for solving this problem is to first import, into CFX-Pre, a CCL file with the proximate/ultimate analysis data for the coal and the required materials and reactions. The first simulation will be without nitrogen oxide or swirl. Only small changes to the boundary conditions will be made to create the second simulation, which has swirl in the flow. After each of the first two simulations, you will use CFD-Post to see the variation of temperature, water mass fraction and radiation intensity. You will examine particle tracks colored by temperature and by ash mass fraction. The last simulation has swirl and also involves the release of nitrogen oxide. Finally, you will use CFD-Post to see the distribution of nitrogen oxide in the third simulation.

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- Running Ansys CFX Tutorials Using Ansys Workbench (p. 34)
- Changing the Display Colors (p. 37)

28.3. Preparing the Working Directory

1. Create a working directory.

   Ansys CFX uses a working directory as the default location for loading and saving files for a particular session or project.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Download the `coal_combustion.zip` file here.
3. Unzip `coal_combustion.zip` to your working directory.

   Ensure that the following tutorial input files are in your working directory:

   - `CoalCombustion.gtm`
   - `CoalCombustion_Reactions_Materials.ccl`

4. Set the working directory and start CFX-Pre.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

28.4. Simulating the Coal Combustion without Swirl and without Nitrogen Oxide

You will first create a simulation where there is no release of nitrogen oxide, a hazardous chemical, during the process. Swirl burners will not be used in this simulation.

28.4.1. Defining the Case Using CFX-Pre

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** and click **OK**.
3. Select **File** > **Save Case As**.
4. Set **File name** to `CoalCombustion_nonox.cfx`.
5. Click **Save**.

28.4.1.1. Importing the Mesh

1. Right-click `Mesh` and select **Import Mesh** > **CFX Mesh**.

   The **Import Mesh** dialog box appears.

2. Configure the following setting(s):

Setting	Value
File name	CoalCombustion.gtm

3. Click **Open**.
4. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)** from the shortcut menu.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.4.1.2. Importing the Coal Combustion Materials CCL File

CFX Command Language (CCL) consists of commands used to carry out actions in CFX-Pre, CFX-Solver Manager, and CFD-Post. The proximate/ultimate analysis data for the coal as well as the materials and reactions required for the combustion simulation will be imported from the CCL file. You will review, then import, the contents of the `CoalCombustion_Reactions_Materials.ccl` file.

Note:

The physics for a simulation can be saved to a CCL (CFX Command Language) file at any time by selecting **File** > **Export** > **CCL**.

---

1. Open `CoalCombustion_Reactions_Materials.ccl` with a text editor and take the time to look at the information it contains.

   The CCL sets up the following reactions:

   - `Fuel Gas Oxygen`
   - `HC Fuel Char Field`
   - `HC Fuel Devolat`
   - `Prompt NO Fuel Gas PDF`
   - `Thermal NO PDF`.

   The CCL also sets up the following materials:

   - `Ash`
   - `Char`
   - `Fuel Gas`
   - `Gas mixture`
   - `HC Fuel`
   - `HC Fuel Gas Binary Mixture`
   - `Raw Combustible`

   The reactions `Prompt NO Fuel Gas PDF` and `Thermal NO PDF` are used only in the third simulation. Other pure substances required for the simulation will be loaded from the standard CFX-Pre materials library.

2. In CFX-Pre, select **File** > **Import** > **CCL**.

   The **Import CCL** dialog box appears.

3. Under **Import Method**, select **Replace**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This will replace the materials list in the current simulation with the ones in the newly imported CCL.

4. Under **Import Method**, select **Auto-load materials**.

   This will load pure materials such as CO2, H2O, N2, O2, and NO — the materials referenced by the imported mixtures and reactions — from the CFX-Pre materials library.

5. Select `CoalCombustion_Reactions_Materials.ccl` (the file you reviewed earlier).

6. Click **Open**.

7. Expand the `Materials` and `Reactions` branches under `Simulation` to make sure that all the materials and reactions described above are present.

28.4.1.3. Creating the Domain

Create a new domain named `Furnace` as follows:

1. Right-click `Simulation` > `Flow Analysis 1` in the **Outline** tree view and click **Insert** > **Domain**.

2. Set **Name** to `Furnace`.

3. Click **OK**

4. On the **Basic Settings**, tab under **Fluid and Particle Definitions**, delete `Fluid 1` and create a new fluid definition named `Gas Mixture`.

5. Click *Add new item* and create a new fluid definition named `HC Fuel`.

6. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Location and Type > **Location**	 B40
	Fluid and Particle Definitions	Gas Mixture
	Fluid and Particle Definitions > Gas Mixture > **Material**	 Gas Mixture [a]
	Fluid and Particle Definitions > Gas Mixture > Morphology	 Continuous Fluid

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Option**	
	Fluid and Particle Definitions	HC Fuel
	Fluid and Particle Definitions > HC Fuel > **Material**	 HC Fuel [**b**]
	Fluid and Particle Definitions > HC Fuel > Morphology > **Option**	 Particle Transport Solid
	Fluid and Particle Definitions > HC Fuel > Morphology > **Particle Diameter Change**	 (Selected)
	Fluid and Particle Definitions > HC Fuel > Morphology > Particle Diameter Change > **Option**	 Mass Equivalent[**c**]
Fluid Models	Multiphase > **Multiphase Reactions**	 (Selected)
	Multiphase > Multiphase Reactions > **Reactions List**	 HC Fuel Char Field, HC Fuel Devolat
	Heat Transfer > **Option**	 Fluid Dependent
	Combustion	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Option**	Fluid Dependent
	Thermal Radiation > **Option**	 Fluid Dependent
Fluid Specific Models	**Fluid**	Gas Mixture
	Fluid > Gas Mixture > Heat Transfer > Heat Transfer > **Option**	 Thermal Energy
	Fluid > Gas Mixture > Thermal Radiation > **Option**	 Discrete Transfer
	Fluid > Gas Mixture > Thermal Radiation > **Number of Rays**	 (Selected)
	Fluid > Gas Mixture > Thermal Radiation > Number of Rays > **Number of Rays**	 32 [d]
	Fluid	HC Fuel
	Fluid > HC Fuel > Heat Transfer > Heat Transfer > **Option**	 Particle Temperature
Fluid Pair Models	**Fluid Pair**	Gas Mixture | HC Fuel

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Fluid Pair > Gas Mixture | HC Fuel > **Particle Coupling**	Fully Coupled
	Fluid Pair > Gas Mixture | HC Fuel > Momentum Transfer > Drag Force > **Option**	Schiller Naumann
	Fluid Pair > Gas Mixture | HC Fuel > Heat Transfer > **Option**	Ranz Marshall
	Fluid Pair > Gas Mixture | HC Fuel > Thermal Radiation Transfer > **Option**	Opaque
	Fluid Pair > Gas Mixture | HC Fuel > Thermal Radiation Transfer > **Emissivity**	1 [**e**]
	Fluid Pair > Gas Mixture | HC Fuel > Thermal Radiation Transfer > **Particle Coupling**	(Selected)
	Fluid Pair > Gas Mixture | HC Fuel > Thermal Radiation Transfer	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Particle Coupling > **Particle Coupling**	 Fully Coupled

a. Click the *Ellipsis* icon to open the **Material** dialog box, then select `Gas Mixture` under the `Gas Phase Combustion` branch. Click **OK**.

b. Click the *Ellipsis* icon to open the **Material** dialog box, then select `HC Fuel` under the `Particle Solids` branch. Click **OK**.

c. The use of the `Mass Equivalent` option for the particle diameter is used here for demonstration only. A physically more sensible setting for coal particles, which often stay the same size or get bigger during combustion, would be the use of the `Swelling Model` option with a **Swelling Factor** of `0.0` (the default) or larger.

d. Increasing the number of rays to 32 from the default 8, increases the number of rays leaving the bounding surfaces and increases the accuracy of the thermal radiation calculation.

e. With this setting, the particles are modeled as black bodies.

7. Click **OK**.

28.4.1.4. Creating the Boundary Conditions

In this section you will create boundary conditions for the coal inlet, the air inlet, the outlet, and multiple no-slip walls. You will also create two symmetry-plane boundary conditions for this no-swirl case.

28.4.1.4.1. Coal Inlet Boundary

You will create the coal inlet boundary with mass flow rate and static temperature set consistently with the problem description. The particle diameter distribution will be set to `Discrete Diameter Distribution` to model particles of more than one specified diameter. Diameter values will be listed as specified in the problem description. A mass fraction as well as a number fraction will be specified for each of the diameter entries. The total of mass fractions and the total of number fractions will sum to unity.

1. Create a boundary named `Coal Inlet`.

2. Configure the following setting(s) of `Coal Inlet`:

Tab	Setting	Value
Basic Settings	**Boundary Type**	Inlet
	Location	CoalInlet

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Boundary Details	Mass and Momentum > **Option**	Mass Flow Rate
	Mass and Momentum > **Mass Flow Rate**	0.001624 [kg s^-1]
	Flow Direction > **Option**	Normal to Boundary Condition
	Heat Transfer > **Option**	Static Temperature
	Heat Transfer > **Static Temperature**	343 [K]
	Component Details	O2
	Component Details > O2 > **Option**	Mass Fraction
	Component Details > O2 > **Mass Fraction**	0.232
Fluid Values	Boundary Conditions > HC Fuel > Particle Behavior > **Define Particle Behavior**	(Selected)
	Boundary Conditions > HC Fuel > Mass and Momentum > **Option**	Zero Slip Velocity
	Boundary Conditions > HC Fuel > Particle Position > **Option**	Uniform Injection
	Boundary Conditions	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> HC Fuel > Particle Position > **Particle Locations**	(Selected)
	Boundary Conditions > HC Fuel > Particle Position > Particle Locations > **Particle Locations**	Equally Spaced
	Boundary Conditions > HC Fuel > Particle Position > Number of Positions > **Option**	Direct Specification
	Boundary Conditions > HC Fuel > Particle Position > Number of Positions > **Number**	200
	Boundary Conditions > HC Fuel > Particle Mass Flow > **Mass Flow Rate**	0.001015 [kg s^-1]
	Boundary Conditions > HC Fuel > **Particle Diameter Distribution**	(Selected)
	Boundary Conditions > HC Fuel > Particle Diameter Distribution	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Option**	Discrete Diameter Distribution
	Boundary Conditions > HC Fuel > Particle Diameter Distribution > **Diameter List**	12, 38, 62, 88 [micron]
	Boundary Conditions > HC Fuel > Particle Diameter Distribution > **Mass Fraction List**	0.18, 0.25, 0.21, 0.36
	Boundary Conditions > HC Fuel > Particle Diameter Distribution > **Number Fraction List**	0.25, 0.25, 0.25, 0.25
	Boundary Conditions > HC Fuel > Heat Transfer > **Option**	Static Temperature
	Boundary Conditions > HC Fuel > Heat Transfer > **Static Temperature**	343 [K]

3. Click **OK**.

28.4.1.4.2. Air Inlet Boundary

Create the air inlet boundary with mass flow rate and static temperature set consistently with the problem description, as follows:

1. Create a new boundary named `Air Inlet`.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Inlet
	Location	AirInlet
Boundary Details	Mass and Momentum > **Option**	Mass Flow Rate
	Mass and Momentum > **Mass Flow Rate**	0.01035 [kg s^-1]
	Flow Direction > **Option**	Normal to Boundary Condition
	Heat Transfer > **Option**	Static Temperature
	Heat Transfer > **Static Temperature**	573 [K]
	Component Details	O2
	Component Details > O2 > **Option**	Mass Fraction
	Component Details > O2 > **Mass Fraction**	0.232

3. Click **OK**.

28.4.1.4.3. Outlet Boundary

Create the outlet boundary with pressure specified, as follows:

1. Create a new boundary named `Outlet`.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Outlet
	Location	Outlet
Boundary Details	Mass and Momentum	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Option**	Average Static Pressure
	Mass and Momentum > **Relative Pressure**	0[Pa]
	Mass and Momentum > **Pres. Profile Blend**	0.05

3. Click **OK**.

28.4.1.4.4. Coal Gun No-Slip Wall Boundary

Create the `Coal Gun Wall` boundary with a fixed temperature as specified in the problem description, as follows:

1. Create a new boundary named `Coal Gun Wall`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Wall
	Location	CoalGunWall
Boundary Details	Heat Transfer > **Option**	Temperature
	Heat Transfer > **Fixed Temperature**	800 [K]
	Thermal Radiation > **Option**	Opaque
	Thermal Radiation > **Emissivity**	0.6 [a]
	Thermal Radiation > **Diffuse Fraction**	1

a. The wall has an emissivity value of 0.6 since about half of the radiation can travel through the surface and half is reflected and/or absorbed at the surface.

3. Click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.4.1.4.5. Coal Inlet No-Slip Wall Boundary

Create the `Coal Inlet Wall` boundary with fixed temperature as specified in the problem description, as follows:

1. Create a new boundary named `Coal Inlet Wall`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Wall
	Location	CoalInletInnerWall, CoalInletOuterWall [**a**]
Boundary Details	Heat Transfer > **Option**	Temperature
	Heat Transfer > **Fixed Temperature**	343 [K]
	Thermal Radiation > **Option**	Opaque
	Thermal Radiation > **Emissivity**	0.6
	Thermal Radiation > **Diffuse Fraction**	1

a. Click the *Ellipsis* icon to open the **Selection Dialog** dialog box. Select multiple items by holding the **Ctrl** key. Click **OK**.

3. Click **OK**.

28.4.1.4.6. Air Inlet No-Slip Wall Boundary

Create the `Air Inlet Wall` boundary with fixed temperature as specified in the problem description, as follows:

1. Create a new boundary named `Air Inlet Wall`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Wall
	Location	AirInletInnerWall, AirInletOuterWall

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Boundary Details	Heat Transfer > **Option**	Temperature
	Heat Transfer > **Fixed Temperature**	573 [K]
	Thermal Radiation > **Option**	Opaque
	Thermal Radiation > **Emissivity**	0.6
	Thermal Radiation > **Diffuse Fraction**	1

3. Click **OK**.

28.4.1.4.7. Furnace No-Slip Wall Boundary

Create the `Furnace Wall` boundary with a fixed temperature as specified in the problem description, as follows:

1. Create a new boundary named `Furnace Wall`.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Wall
	Location	FurnaceFrontWall, FurnaceOuterWall
Boundary Details	Heat Transfer > **Option**	Temperature
	Heat Transfer > **Fixed Temperature**	1400 [K]
	Thermal Radiation > **Option**	Opaque
	Thermal Radiation > **Emissivity**	0.6
	Thermal Radiation > **Diffuse Fraction**	1

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click **OK**.

28.4.1.4.8. Quarl No-Slip Wall Boundary

1. Create a new boundary named `Quarl Wall`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Wall
	Location	QuarlWall
Boundary Details	Heat Transfer > **Option**	Temperature
	Heat Transfer > **Fixed Temperature**	1200 [K]
	Thermal Radiation > **Option**	Opaque
	Thermal Radiation > **Emissivity**	0.6
	Thermal Radiation > **Diffuse Fraction**	1

3. Click **OK**.

28.4.1.4.9. Symmetry Plane Boundaries

You will use symmetry plane boundaries on the front and back regions of the cavity.

1. Create a new boundary named `Symmetry Plane 1`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Symmetry
	Location	PeriodicSide1

3. Click **OK**.
4. Create a new boundary named `Symmetry Plane 2`.
5. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Basic Settings	**Boundary Type**	Symmetry
	Location	PeriodicSide2

6. Click **OK**.

28.4.1.5. Setting Solver Control

1. Click *Solver Control* .
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Convergence Control > **Max. Iterations**	600
	Convergence Control > Fluid Timescale Control > **Timescale Control**	Physical Timescale
	Convergence Control > Fluid Timescale Control > **Physical Timescale**	0.005 [s] [a]
Particle Control	Particle Coupling Control > **First Iteration for Particle Calculation**	(Selected)
	Particle Coupling Control > First Iteration for Particle Calculation > **First Iteration**	25 [b]
	Particle Coupling Control > **Iteration Frequency**	(Selected)
	Particle Coupling Control > Iteration Frequency > **Iteration Frequency**	10 [c]
	Particle Under Relaxation Factors	(Selected)

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Particle Under Relaxation Factors > **Vel. Under Relaxation**	0.75
	Particle Under Relaxation Factors > **Energy**	0.75
	Particle Under Relaxation Factors > **Mass**	0.75
	Particle Ignition	(Selected)
	Particle Ignition > **Ignition Temperature**	1000 [K]
	Particle Source Smoothing	(Selected)
	Particle Source Smoothing > **Option**	Smooth
Advanced Options	**Thermal Radiation Control**	(Selected)
	Thermal Radiation Control > **Coarsening Control**	(Selected)
	Thermal Radiation Control > Coarsening Control > **Target Coarsening Rate**	(Selected)
	Thermal Radiation Control > Coarsening Control > Target Coarsening Rate > **Rate**	16 [d]

a. Based on the air inlet speed and the size of the combustor.

b. The First Iteration parameter sets the coefficient-loop iteration number at which particles are first tracked; it allows the continuous-phase flow to develop before tracking droplets through the flow. Experience has shown that the value usually has to be increased to 25 from the default of 10.

c. The Iteration Frequency parameter is the frequency at which particles are injected into the flow after the First Iteration for Particle Calculation iteration number. The iteration frequency allows the continuous phase to settle down between injections because it is affected by sources of

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	momentum, heat, and mass from the droplet phase. Experience has shown that the value usually has to be increased to 10 from the default of 5. d. The Target Coarsening Rate parameter controls the size of the radiation element required for calculating Thermal Radiation. Decreasing the size of the element to 16, from the default 64, increases the accuracy of the solution obtained, while increasing the computing time required for the calculations.	

3. Click **OK**.

28.4.1.6. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .
2. Configure the following setting(s):

Setting	Value
File name	CoalCombustion_nonox.def

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, **Solver Input File** is set.

4. Quit CFX-Pre, saving the simulation (`.cfx`) file.

28.4.2. Obtaining the Solution using CFX-Solver Manager

When CFX-Pre has shut down and the CFX-Solver Manager has started, obtain a solution to the CFD problem by following the instructions below:

1. Ensure that the **Define Run** dialog box is displayed.

   Solver Input File should be set to `CoalCombustion_nonox.def`.

2. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. At the end of the run, a dialog box is displayed stating that the simulation has ended.

3. Select **Post-Process Results**.
4. If using stand-alone mode, select **Shut down CFX-Solver Manager**.
5. Click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.4.3. Viewing the Results Using CFD-Post

In this section, you will make plots showing the variation of temperature, water mass fraction, and radiation intensity on the `Symmetry Plane 1` boundary. You will also color the particle tracks, which were produced by the solver and included in the results file, by temperature and by ash mass fraction. The particle tracks help to illustrate the mean flow behavior in the coal furnace.

28.4.3.1. Displaying the Temperature on a Symmetry Plane

1. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)**.

   This orients the geometry with the inlets at the top, as shown at the beginning of this tutorial.

2. Edit `Cases` > `CoalCombustion_nonox_001` > `Furnace` > `Symmetry Plane 1`.

3. Configure the following setting(s):

Tab	Setting	Value
Color	**Mode**	Variable
	Variable	Temperature
Render	**Show Faces**	(Selected)
	Lighting	(Cleared) [a]

a. Turning off the lighting makes the colors accurate in the 3D view, but can make it more difficult to perceive depth. As an alternative to turning off the lighting, you can try rotating the view to a different position.

4. Click **Apply**.

   As expected for a non-swirling case, the flame appears a significant distance away from the burner. The flame is likely unstable, as evidenced by the rate of solver convergence; the next simulation in this tutorial involves swirl, which tends to stabilize the flame, and has much faster solver convergence.

28.4.3.2. Displaying the Water Mass Fraction

Change the variable used for coloring the plot to `H2O.Mass Fraction` and click **Apply**.

From the plot it can be seen that water is produced a significant distance away from the burner, as was the flame in the previous plot. As expected, the mass fraction of water is high where the temperature is high.

28.4.3.3. Displaying the Radiation Intensity

1. Change the variable used for coloring the plot to `Radiation Intensity` and click **Apply**.

   This plot is directly related to the temperature plot. This result is consistent with radiation being proportional to temperature to the fourth power.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. When you are finished, right-click `Symmetry Plane 1` in the **Outline** tree view and select **Hide**.

28.4.3.4. Displaying the Temperature of the Fuel Particles

Color the existing particle tracks for the solid particles by temperature:

1. Edit `Cases` > `CoalCombustion_nonox_001` > `Res PT for HC Fuel`.
2. Configure the following setting(s):

Tab	Setting	Value
Color	**Mode**	Variable
	Variable	HC Fuel.Temperature

3. Click **Apply**.

   Observing the particle tracks, you can see that coal enters the chamber at a temperature of around 343 K. The temperature of the coal, as it moves away from the inlet, rises as it reacts with the air entering from the inlet. The general location where the temperature of the coal increases rapidly is close to the location where the flame appears to be according to the plots created earlier. Downstream of this location, the temperature of the coal particles begins to drop.

28.4.3.5. Displaying the Ash Mass Fraction using Particle Tracking

1. Change the plot of the particle tracks so that they are colored by `HC Fuel.Ash.Mass Fraction`.

   The ashes form in the flame region, as expected.

2. Quit CFD-Post, saving the state (`.cst`) file at your discretion.

28.5. Simulating the Coal Combustion with Swirl and without Nitrogen Oxide

You will now create a simulation where swirl burners are used and where there is no release of nitrogen oxide during the process. Swirl burners inject a fuel axially into the combustion chamber surrounded by an annular flow of oxidant (normally air) that has, upon injection, some tangential momentum. This rotational component, together with the usually divergent geometry of the burner mouth, cause two important effects:

- They promote intense mixing between fuel and air, which is important for an efficient and stable combustion, and low emissions.
- They originate a recirculation region, just at the burner mouth, which traps hot combustion products and acts as a permanent ignition source, hence promoting the stability of the flame.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.5.1. Defining the Case Using CFX-Pre

1. Ensure that the following tutorial input file is in your working directory:

   - `CoalCombustion_nonox.cfx`

2. Set the working directory and start CFX-Pre if it is not already running.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

3. Select **File** > **Open Case**.

4. From your working directory, select `CoalCombustion_nonox.cfx` and click **Open**.

5. Select **File** > **Save Case As**.

6. Set **File name** to `CoalCombustion_nonox_swirl.cfx`.

7. Click **Save**.

28.5.1.1. Editing the Boundary Conditions

To add swirl to the flow, you will edit the `Air Inlet` boundary to change the flow direction specification from `Normal to Boundary Condition` to `Cylindrical Components`. You will also edit the `Outlet` boundary to change the `Pressure Profile Blend` setting from 0.05 to 0; the reason for this change is explained later. You will also delete the two symmetry plane boundary conditions and replace them with a periodic domain interface.

28.5.1.1.1. Air Inlet Boundary

1. Edit `Simulation` > `Flow Analysis 1` > `Furnace` > `Air Inlet`.

2. Configure the following setting(s):

Tab	Setting	Value
Boundary Details	Flow Direction > **Option**	Cylindrical Components
	Flow Direction > **Axial Component**	0.88
	Flow Direction > **Radial Component**	0
	Flow Direction > **Theta Component**	1
	Axis Definition > **Rotational Axis**	Global Z

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click **OK**.

28.5.1.1.2. Outlet Boundary

The average pressure boundary condition leaves the pressure profile unspecified while constraining the average pressure to the specified value. In some situations, leaving the profile fully unspecified is too weak and convergence difficulties may result. The 'Pressure Profile Blend' feature works around this by blending between an unspecified pressure profile and a fully specified pressure profile. By default, the pressure profile blend is 5%. For swirling flow, however, imposing any amount of a uniform pressure profile is inconsistent with the radial pressure profile which should naturally develop in response to the fluid rotation, and the pressure profile blend must be set to 0.

1. Edit `Simulation` > `Flow Analysis 1` > `Furnace` > `Outlet`.

2. Configure the following setting(s):

Tab	Setting	Value
Boundary Details	Mass and Momentum > **Option**	Average Static Pressure
	Mass and Momentum > **Pres. Profile Blend**	0

3. Click **OK**.

28.5.1.1.3. Deleting the Symmetry Plane Boundaries

1. In the **Outline** tree view, right-click `Simulation` > `Flow Analysis 1` > `Furnace` > `Symmetry Plane 1`, then select **Delete**.

2. Repeat step 1 to delete `Symmetry Plane 2`.

28.5.1.2. Creating a Domain Interface

You will insert a domain interface to connect the `Periodic Side 1` and `Periodic Side 2` regions.

1. Create a domain interface named `Periodic`.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Interface Type**	Fluid Fluid
	Interface Side 1 > **Region List**	PeriodicSide1
	Interface Side 2	

Tab	Setting	Value
	> **Region List**	PeriodicSide2
	Interface Models > **Option**	 Rotational Periodicity

3. Click **OK**.

28.5.1.3. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .
2. Configure the following setting(s):

Setting	Value
File name	CoalCombustion_nonox_swirl.def

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, **Solver Input File** is set.

4. Quit CFX-Pre, saving the simulation (`.cfx`) file.

28.5.2. Obtaining the Solution Using CFX-Solver Manager

When CFX-Pre has shut down and the CFX-Solver Manager has started, obtain a solution to the CFD problem by following the instructions below.

1. Ensure that the **Define Run** dialog box is displayed.

   Solver Input File should be set to `CoalCombustion_nonox_swirl.def`.

2. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. At the end of the run, a dialog box is displayed stating that the simulation has ended.

3. Select **Post-Process Results**.
4. If using stand-alone mode, select **Shut down CFX-Solver Manager**.
5. Click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.5.3. Viewing the Results Using CFD-Post

In this section, you will make plots showing the variation of temperature, water mass fraction, and radiation intensity on the `Periodic Side 1` boundary. You will also color the existing particle tracks by temperature and by ash mass fraction.

28.5.3.1. Displaying the Temperature on a Periodic Interface

1. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)**.
2. Edit `Cases` > `CoalCombustion_nonox_swirl_001` > `Furnace` > `Periodic Side 1`.
3. Configure the following setting(s):

Tab	Setting	Value
Color	**Mode**	Variable
	Variable	Temperature
Render	**Show Faces**	(Selected)
	Lighting	(Cleared)

4. Click **Apply**.

   As expected, the flame appears much closer to the burner than in the previous simulation, which had no swirl. This is due to the fact that the swirl component applied to the air from `Air Inlet` tends to entrain coal particles and keep them near the burner for longer, therefore helping them to burn.

28.5.3.2. Displaying the Water Mass Fraction

Change the variable used for coloring the plot to `H2O.Mass Fraction` and click **Apply**.

Similar to the no-swirl case, the mass fraction of water with swirl is directly proportional to the temperature of the furnace.

28.5.3.3. Displaying the Radiation Intensity

1. Change the variable used for coloring the plot to `Radiation Intensity` and click **Apply**.
2. When you are finished, right-click `Periodic Side 1` in the **Outline** tree view and select **Hide**.

28.5.3.4. Displaying the Temperature using Particle Tracking

1. Edit `Cases` > `CoalCombustion_nonox_swirl_001` > `Res PT for HC Fuel`.
2. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Color	**Mode**	Variable
	Variable	HC Fuel.Temperature

3. Click **Apply**.

28.5.3.5. Displaying the Ash Mass Fraction using Particle Tracking

1. Change the plot of the particle tracks so that they are colored by `HC Fuel.Ash.Mass Fraction`.
2. Quit CFD-Post, saving the state (`.cst`) file at your discretion.

28.6. Simulating the Coal Combustion with Swirl and with Nitrogen Oxide

You will now create a simulation that involves both swirl and the release of nitrogen oxide. The CCL file that was previously imported contains the nitrogen oxide material, `NO`, and reactions, `Prompt NO Fuel Gas PDF` and `Thermal NO PDF`, required for this combustion simulation. Nitrogen oxide is calculated as a postprocessing step in the solver.

28.6.1. Defining the Case Using CFX-Pre

1. Ensure that the following tutorial input files are in your working directory:
   - `CoalCombustion_nonox_swirl.cfx`
   - `CoalCombustion_nonox_swirl_001.res`
2. Set the working directory and start CFX-Pre if it is not already running.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).
3. Select **File** > **Open Case**.
4. From your working directory, select `CoalCombustion_nonox_swirl.cfx` and click **Open**.
5. Select **File** > **Save Case As**.
6. Set **File name** to `CoalCombustion_noxcpp_swirl.cfx`.
7. Click **Save**.

28.6.1.1. Editing the Domain

In this section, you will edit the `Furnace` domain by adding the new material `NO` to the materials list. CFX-Solver requires that you specify enough information for the mass fraction of `NO` at each of the system inlets. In this case, set the `NO` mass fraction at the air and coal inlets to zero.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Edit `Simulation` > `Flow Analysis 1` > `Furnace`.
2. Configure the following setting(s):

Tab	Setting	Value
Fluid Specific Models	**Fluid**	Gas Mixture
	Fluid > Gas Mixture > Combustion > **Chemistry Post Processing**	(Selected)
	Fluid > Gas Mixture > Combustion > Chemistry Post Processing > **Materials List**	NO
	Fluid > Gas Mixture > Combustion > Chemistry Post Processing > **Reactions List**	Prompt NO Fuel Gas PDF,Thermal NO PDF

These settings enable the combustion simulation with nitrogen oxide (NO) as a postprocessing step in the solver. The NO reactions are defined in the same way as any participating reaction but the simulation of the NO reactions is performed after the combustion simulation of the air and coal. With this one-way simulation, the NO will have no effect on the combustion simulation of the air and coal.

3. Click **OK**.
4. Edit `Simulation` > `Flow Analysis 1` > `Furnace` > `Air Inlet`.
5. Configure the following setting(s):

Tab	Setting	Value
Boundary Details	**Component Details**	NO
	Component Details > NO	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Mass Fraction**	0.0

6. Click **OK**.
7. Edit `Simulation` > `Flow Analysis 1` > `Furnace` > `Coal Inlet`.
8. Configure the following setting(s):

Tab	Setting	Value
Boundary Details	**Component Details**	NO
	Component Details > NO > **Mass Fraction**	 0.0

9. Click **OK**.

28.6.1.2. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .
2. Configure the following setting(s):

Setting	Value
File name	CoalCombustion_noxcpp_swirl.def

3. Click **Save**.
4. Quit CFX-Pre, saving the simulation as `CoalCombustion_noxcpp_swirl.cfx`.

28.6.2. Obtaining the Solution Using CFX-Solver Manager

When CFX-Pre has shut down and the CFX-Solver Manager has started, obtain a solution to the CFD problem by following the instructions below.

1. Ensure that the **Define Run** dialog box is displayed.

   Solver Input File should be set to `CoalCombustion_noxcpp_swirl.def`.
2. Under the **Initial Values** tab, select **Initial Values Specification**.
3. Select `CoalCombustion_nonox_swirl_001.res` for the initial values file using the *Browse* tool.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The fluid solution from the previous case has not changed for this simulation. Loading the results from the previous case as an initial guess eliminates the need for the solver to solve for the fluids solutions again.

4. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. At the end of the run, a dialog box is displayed stating that the simulation has ended.

5. Select **Post-Process Results**.

6. If using stand-alone mode, select **Shut down CFX-Solver Manager**.

7. Click **OK**.

28.6.3. Viewing the Results Using CFD-Post

In this section, you will make a plot on the `Periodic Side 1` region showing the variation of concentration of nitrogen oxide through the coal furnace.

1. Right-click a blank area in the viewer and select **Predefined Camera** > **Isometric View (Z up)**.

2. Edit `Cases` > `CoalCombustion_noxcpp_swirl_001` > `Furnace` > `Periodic Side 1`.

3. Configure the following setting(s):

Tab	Setting	Value
Color	**Mode**	Variable
	Variable	NO.Mass Fraction
Render	**Show Faces**	(Selected)
	Lighting	(Cleared)

4. Click **Apply**.

   You can see that NO is produced in the high-temperature region near the inlet. Further downstream, the mass fraction of NO is more uniform.

5. Quit CFD-Post, saving the state (`.cst`) file at your discretion.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 29: Steam Jet

This tutorial includes:

29.1. Tutorial Features

29.2. Overview of the Problem to Solve

29.3. Preparing the Working Directory

29.4. Defining the Case Using CFX-Pre

29.5. Obtaining the Solution Using CFX-Solver Manager

29.6. Viewing the Results Using CFD-Post

29.1. Tutorial Features

In this tutorial you will learn about:

- Importing a CCL file in CFX-Pre.
- High-speed multicomponent, multiphase flow with interphase mass transfer.
- Model customization using CEL.
- Handling mass sources based on species transfer.
- Source linearization.

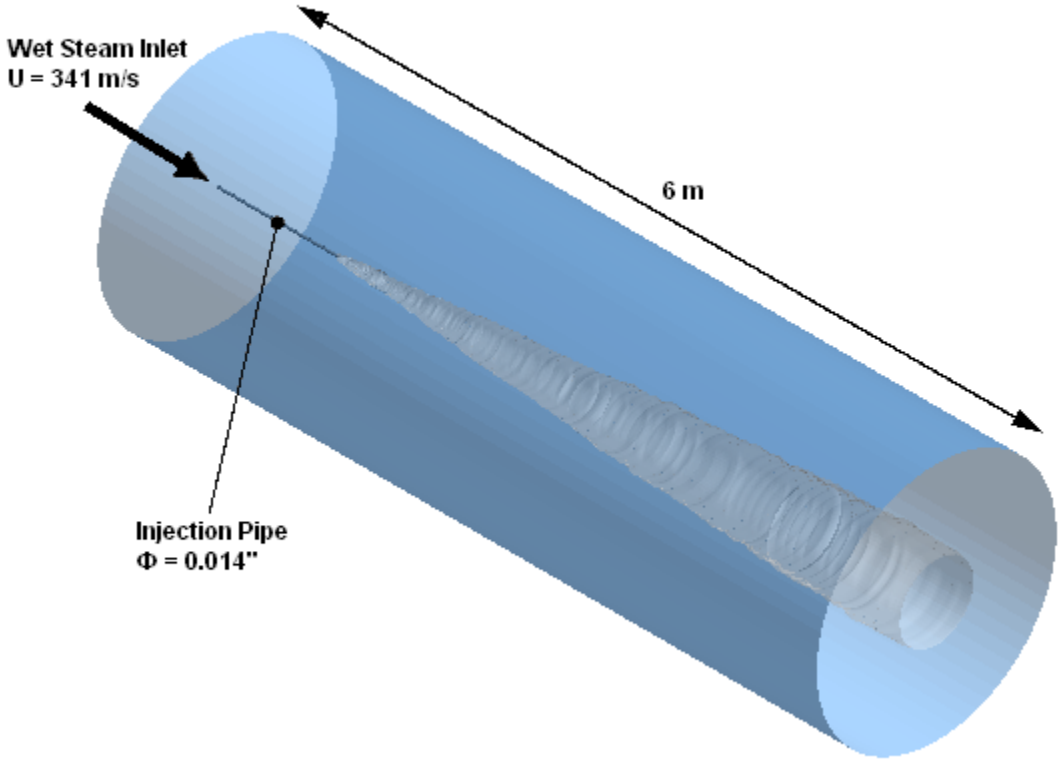
Component	Feature	Details
CFX-Pre	User Mode	General mode
	Domain Type	Single Domain
	Analysis Type	Steady State
	Fluid Type	Continuous Fluid
		Dispersed Fluid
	CCL File	Import
	Boundary Conditions	Inlet Boundary
		Opening Boundary
		Outlet Boundary
		Steam Jet Default
		Symmetry Boundary
	Timestep	Physical Timescale
CFD-Post	Plots	Default Locators
		Line Locator

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Component	Feature	Details
	Other	Chart Creation

29.2. Overview of the Problem to Solve

This tutorial investigates the simulation of a high-speed steam jet from a pipe. Steam at 373 K leaves the pipe at 341 m s^-1 and contains some liquid water. The surrounding air temperature is 25 °C.



Evaporation and condensation of water are modeled using mass sources and sinks applied to two subdomains; one is used for evaporation and the other is used for condensation. The rate of mass transfer is modeled using Sherwood number-based mass diffusion at the surface of liquid drops.

To take advantage of the symmetrical nature of the domain, a thin slice of the flow field is modeled and symmetry boundaries are used to represent the rest of the flow field. An opening boundary is used around the outside edges of the domain; the flow direction is restricted to be normal to this boundary in order to provide sufficient constraints on the flow solution.

It is strongly recommended that you complete the previous tutorials before trying this one. However, if this is the first tutorial you are working with, it is important to review the following topics before beginning:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Running Ansys CFX Tutorials Using Ansys Workbench (p. 34)
- Changing the Display Colors (p. 37)

29.3. Preparing the Working Directory

1. Create a working directory.

   Ansys CFX uses a working directory as the default location for loading and saving files for a particular session or project.

2. Download the `steam_jet.zip` file here.

3. Unzip `steam_jet.zip` to your working directory.

   Ensure that the following tutorial input files are in your working directory:

   - `steam_jet.gtm`
   - `steam_jet_expressions.ccl`
   - `steam_jet_additional_variables.ccl`

4. Set the working directory and start CFX-Pre.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

29.4. Defining the Case Using CFX-Pre

This section describes the step-by-step definition of the flow physics in CFX-Pre for a steady-state simulation.

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** and click **OK**.
3. Select **File** > **Save Case As**.
4. Set **File name** to `SteamJet.cfx`.
5. Click **Save**.

29.4.1. Importing the Mesh

1. Right-click `Mesh` and select **Import Mesh** > **CFX Mesh**.

   The **Import Mesh** dialog box appears.

2. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
File name	steam_jet.gtm

3. Click **Open**.

29.4.2. Importing the Steam Jet CCL

CFX Command Language (CCL) consists of commands used to carry out actions in CFX-Pre, CFX-Solver Manager, and CFD-Post. Expressions and Additional Variables required for the steam jet simulation will be imported from CCL files. This section outlines the steps to analyze `steam_jet_expressions.ccl` and `steam_jet_additional_variables.ccl`, and then import them into the simulation.

Note:

The physics for a case can be saved to a CCL (CFX Command Language) file at any time by selecting **File** > **Export** > **CCL**.

1. Select CCL files from your working directory, and open them one at a time with a text editor and take the time to look at the information they contain. The information contained in the CCL files is outlined below:

   - The CCL file `steam_jet_expressions.ccl` creates expressions required for setting up the following data:
     - Liquid/gas interface
     - Interphase diffusive transport coefficient
     - Heats of vaporization
     - Liquid-gas mass transfer rate for water [1]
     - Local false step linearization of the interphase mass transfer (IPMT).
   - The CCL file `steam_jet_additional_variables.ccl` creates the following Additional Variables:
     - Pressure linearization coefficient `PCoef`
     - Water IPMT flux liquid-to-gas `WaFluxLG`

[1] The expression to calculate the volumetric mass transfer rate of the phase on which the mass transfer is based (CMwa) is multiplied by the ratio between the molecular weight of the absorbing species and the molecular weight of the phase on which the mass transfer is based; that is, a two-component gas mixture and water, respectively. However, due to the variable composition of the gas mixture (according to Creating and Loading Materials (p. 696)), its molecular weight varies with time. This causes the ratio of the molecular weights between water and the gas mixture to vary also. Thus, an expression that is used to calculate the molecular weight of the gas mixture depends on the mole fractions of vapour and air in the gas mixture at any given time (as seen in `steam_jet_expressions.ccl`).

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– Water IPMT flux gas-to-liquid `WaFluxGL`

– Local IPMT false timestep `FalseDt`

– Saturation temperature `SatTemp`

– Saturation pressure `SatPres`

– Latent heat at saturation `SatLHeat`

These variables are recognizable by CFD-Post to enable plotting and further analysis during postprocessing.

2. Select **File** > **Import** > **CCL**

   The **Import CCL** dialog box appears.

3. Under **Import Method**, select **Append**. This option appends the changes to the existing case.

   Note:

   The **Replace** option is useful if you have defined the physics and want to update or replace the existing physics using the newly imported CCL.

4. From your working directory, select `steam_jet_expressions.ccl`.

5. Click **Open**. The CCL is now loaded as indicated by the status bar in the bottom right corner of the window. After a short pause, the CCL and the **Outline** tree view will be updated.

6. Repeat steps 2 to 5 to import `steam_jet_additional_variables.ccl`.

7. In the **Outline** tree view, expand the `Additional Variables` and `Expressions` branches under `Simulation` > `Expressions, Functions and Variables` to confirm that new objects have been added after importing the CCL files.

29.4.3. Creating a Steady-State Analysis

The characteristics of this case do not change as a function of time, and therefore a steady-state analysis is appropriate.

1. Right-click `Analysis Type` in the **Outline** tree view and select **Edit**.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Analysis Type > **Option**	 Steady State

3. Click **OK**.

29.4.4. Creating and Loading Materials

In addition to providing template fluids, CFX allows you to create custom fluids for use in all your CFX models. A custom fluid may be defined as a pure substance, but may also be defined as a mixture, consisting of a number of transported fluid components. This type of fluid model is useful for applications involving mixtures, reactions, and combustion.

In order to define custom fluids, CFX-Pre provides the **Material** details view. This tool allows you to define your own fluids as pure substances, fixed composition mixtures or variable composition mixtures using a range of template property sets defined for common materials.

The steam jet application requires two fluids: a variable-composition mixture containing the materials `Steam3v` (which is dry steam) and `Air Ideal Gas`; a fixed-composition mixture containing only the material `Steam3l` (which is liquid water). In a variable composition mixture, the proportion of each component material will change throughout the simulation; while in a fixed composition mixture, the proportion of each component material is fixed.

29.4.4.1. Loading the Steam3v, Steam3l, and Steam3vl Materials

You are first going to load some of the materials that take part in the process (`Steam3v`, `Steam3l`). The only other material that takes part in the process, `Air Ideal Gas`, is already loaded.

Load the materials `Steam3v` and `Steam3l` from the CFX-Pre Materials Library by loading `Steam3vl`.

1. In the **Outline** tree view, right-click `Simulation` > `Materials` and select **Import Library Data**.

   The **Select Library Data to Import** dialog box appears.

2. Click the browse button next to **File to Import**.

   The **Import CCL** dialog box appears. If the dialog box opens behind any existing windows, reposition it so that it is completely visible in order to complete the next step.

3. Under **Import Method**, select **Append**. This option appends the CCL changes to the existing case.

4. Select `MATERIALS-iapws.ccl` from the `etc/materials-extra` directory and click **Open**.

5. In the **Select Library Data to Import** dialog box, expand `Wet Steam` and select `Steam3vl`.

6. Click **OK**.

7. In the **Outline** tree view, expand `Simulation` > `Materials` to confirm that `Steam3v` and `Steam3l` have been added to the list.

In the next sections you will create the variable-composition mixture and the fixed-composition mixture.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

29.4.4.2. Creating the Gas Mixture Material

Create a new material named `Gas mixture` that will be composed of `Air Ideal Gas` and `Steam3v`. This material will be injected into the gas inlet during the steady-state simulation.

1. Create a new material named `Gas mixture`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Option**	Variable Composition Mixture
	Materials List	Air Ideal Gas, Steam3v [a]
a. Select multiple items from the drop-down list by holding the **Ctrl** key.		

3. Click **OK**.

29.4.4.3. Creating the Liquid Mixture Material

Create a new material named `Liquid mixture` that will be composed of `Steam3l`.

1. Create a new material named `Liquid mixture`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Option**	Fixed Composition Mixture
	Materials List	Steam3l
	Child Materials > Steam3l > **Mass Fraction**	 1.0

3. Click **OK**.

29.4.5. Creating the Domain

This section outlines the steps to create a new domain `Steam Jet`.

1. Select **Insert** > **Domain** from the menu bar, or click *Domain* .
2. In the **Insert Domain** dialog box, set the name to `Steam Jet` and click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. On the **Basic Settings** tab, configure the following setting(s) under **Location and Type**:

Setting	Value
Location	B26
Domain Type	Fluid Domain
Coordinate Frame	Coord 0

4. On the **Basic Settings** tab, delete any existing items under **Fluid and Particle Definitions** by selecting them and clicking *Remove selected item* .

5. Under **Fluid and Particle Definitions**, create two fluid definitions named `Gas` and `Liquid` by using the *Add new item* icon.

   The new fluids named `Gas` and `Liquid` appear under **Fluid and Particle Definitions**.

6. On the **Basic Settings** tab, configure the following setting(s) under **Fluid and Particle Definitions**:

Setting	Value
(List Box)	Gas
Gas > **Material**	Gas mixture
Gas > Morphology > **Option**	Continuous Fluid
(List Box)	Liquid
Liquid > **Material**	Liquid mixture
Liquid > Morphology > **Option**	Dispersed Fluid
Liquid > Morphology > **Mean Diameter**	liqLength [a]

a. Click the *Enter Expression* icon to specify the CEL expression.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. On the **Fluid Models** tab, configure the following setting(s):

Setting	Value
Heat Transfer > **Option**	Fluid Dependent
Turbulence > **Option**	Fluid Dependent
Combustion > **Option**	None
Thermal Radiation > **Option**	None

8. On the **Fluid Models** tab under **Additional Variable Models** > **Additional Variable**, select `FalseDt` and select the **FalseDt** check box.

9. Make sure that **Additional Variable Models** > **Additional Variable** > **FalseDt** > **Option** is set to `Fluid Dependent`.

10. Repeat the previous two steps for the rest of the Additional Variables (PCoef, SatLheat, SatPres, SatTemp, WaFluxGL, WaFluxLG).

11. On the **Fluid Specific Models** tab, select `Gas` in the list box, then configure the following setting(s):

Setting	Value
Heat Transfer Model > **Option**	Total Energy
Turbulence > **Option**	k-Epsilon
Turbulence > Wall Function > **Wall Function**	Scalable
Component Models > **Component**	Air Ideal Gas
Component Models > Component > Air Ideal Gas > **Option**	Constraint

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
Component Models > **Component**	Steam3v
Component Models > Component > Steam3v > **Option**	Transport Equation
Component Models > Component > Steam3v > **Kinematic Diffusivity**	(Selected)
Component Models > Component > Steam3v > Kinematic Diffusivity > **Kinematic Diffusivity**	KinDiff
Additional Variable Models	PCoef
Additional Variable Models > **PCoef**	(Selected)
Additional Variable Models > PCoef > **Add. Var. Value**	dFLUXwadp

12. On the **Fluid Specific Models** tab, select `Liquid` in the list box, then configure the following setting(s):

Setting	Value
Heat Transfer Model > **Option**	Total Energy
Turbulence > **Option**	Dispersed Phase Zero Equation

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13. Under **Additional Variable Models** (for Liquid), select `FalseDt` in the list box, then configure the following setting(s):

Setting	Value
FalseDt	(Selected)
FalseDt > **Add. Var. Value**	 DtFalseMf

14. Repeat the previous step for the rest of the Additional Variables (PCoef, SatLheat, SatPres, SatTemp, WaFluxGL, WaFluxLG) using the following values:

Additional Variable	Expression
PCoef	dFLUXwadp
SatLheat	HtVapwa
SatPres	VpWat
SatTemp	SatT
WaFluxGL	FLUXwa1
WaFluxLG	FLUXwa2

15. On the **Fluid Pair Models** tab, select `Gas | Liquid` in the list box, then configure the following setting(s):

Setting	Value
Surface Tension Coefficient	(Selected)
Surface Tension Coefficient > **Surf. Tension Coeff.**	 srfTenCoef
Interphase Transfer > **Option**	 Particle Model
Momentum Transfer > Drag Force > **Option**	 Schiller Naumann
Turbulence Transfer > **Option**	 None
Mass Transfer > **Option**	 None
Heat Transfer > **Option**	 Ranz Marshall

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16. Click **OK**.

29.4.6. Creating Subdomains

To model evaporation and condensation, mass sources and sinks are defined for the materials `Steam3v` and `Steam3l`, according to the expressions imported earlier. In the next two sections you will set up two subdomains; one is used for evaporation and the other is used for condensation.

29.4.6.1. Gas-to-Liquid Source Subdomain

This section outlines the steps to create the subdomain `GastoLiq`. This subdomain will be defined in a manner consistent with the process of condensation.

1. Create a subdomain named `GastoLiq`.

2. On the **Basic Settings** tab, configure the following setting(s):

Setting	Value
Location	B26
Coordinate Frame	Coord 0

3. On the **Fluid Sources** tab, select `Gas` in the list box, then select the **Gas** check box and configure the following setting(s):

Setting	Value
Equation Sources	Continuity
Equation Sources > **Continuity**	(Selected)
Equation Sources > Continuity > **Option**	Fluid Mass Source
Equation Sources > Continuity > **Source**	-Liquid.WaFluxGL
Equation Sources > Continuity > **MCF/Energy Sink Option**	(Selected)
Equation Sources > Continuity > MCF/Energy Sink Option	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
> **Sink Option**	Spec. Mass Frac. and Loc. Temp.
Equation Sources > Continuity > **Mass Source Volume Fraction Coefficient**	(Selected)
Equation Sources > Continuity > Mass Source Volume Fraction Coefficient > **Volume Frac. Coeff.**	-Gas.density/DtFalseMf
Equation Sources > Continuity > Variables > Steam3v.mf > **Option**	Value
Equation Sources > Continuity > Variables > Steam3v.mf > **Value**	1
Equation Sources > Continuity > Variables > Temperature > **Option**	Value
Equation Sources > Continuity > Variables > Temperature > **Value**	Gas.T
Equation Sources	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
> Continuity > Variables > Turbulence Eddy Dissipation > **Option**	Value
Equation Sources > Continuity > Variables > Turbulence Eddy Dissipation > **Value**	Gas.ed
Equation Sources > Continuity > Variables > Turbulence Kinetic Energy > **Option**	Value
Equation Sources > Continuity > Variables > Turbulence Kinetic Energy > **Value**	Gas.ke
Equation Sources > Continuity > Variables > Velocity > **Option**	Cartesian Vector Components
Equation Sources > Continuity > Variables > Velocity	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
> **U**	Gas.Velocity u
Equation Sources > Continuity > Variables > Velocity > **V**	Gas.Velocity v
Equation Sources > Continuity > Variables > Velocity > **W**	Gas.Velocity w
Equation Sources	Steam3v.mf
Equation Sources > **Steam3v.mf**	(Selected)
Equation Sources > Steam3v.mf > **Option**	Source
Equation Sources > Steam3v.mf > **Source**	0 [kg m^-3 s^-1]
Equation Sources > Steam3v.mf > **Source Coefficient**	(Selected)
Equation Sources > Steam3v.mf > Source Coefficient > **Source Coefficient**	dFLwadYG [a]

a. This source coefficient is required only for the mass transfer rates between gas and liquid phases. The source is set to 0 [kg m^3 s^-1] because there is no external source and therefore no additional mass is transferred into the system. The source is defined only to enable setting a linearization coefficient.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. On the **Fluid Sources** tab, select `Liquid` in the list box, then select the `Liquid` check box and configure the following setting(s):

Setting	Value
Equation Sources	Continuity
Equation Sources > **Continuity**	(Selected)
Equation Sources > Continuity > **Option**	Fluid Mass Source
Equation Sources > Continuity > **Source**	Liquid.WaFluxGL
Equation Sources > Continuity > **Mass Source Volume Fraction Coefficient**	(Selected)
Equation Sources > Continuity > Mass Source Volume Fraction Coefficient > **Volume Frac. Coeff.**	-Liquid.density/DtFalseMf
Equation Sources > Continuity > Variables > Temperature > **Option**	Value
Equation Sources > Continuity > Variables > Temperature > **Value**	Gas.T
Equation Sources > Continuity	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
> Variables > Velocity > **Option**	Cartesian Vector Components
Equation Sources > Continuity > Variables > Velocity > **U**	Gas.Velocity u
Equation Sources > Continuity > Variables > Velocity > **V**	Gas.Velocity v
Equation Sources > Continuity > Variables > Velocity > **W**	Gas.Velocity w
Equation Sources	Energy
Equation Sources > **Energy**	(Selected)
Equation Sources > Energy > **Option**	Source[a]
Equation Sources > Energy > **Source**	Liquid.WaFluxGL*HtVapwa
Equation Sources > Energy	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
> **Source Coefficient**	(Selected)
Equation Sources > Energy > Source Coefficient > **Source Coefficient**	 -Liquid.vf*Liquid.density*Liquid.Cp/DtFalseMf
a. The liquid energy source is defined to account for latent heat due to the phase change. The latent heat is assumed to be transferred to the liquid.	

5. Click **OK**.

29.4.6.2. Liquid-to-Gas Source Subdomain

This section outlines the steps to create the subdomain `LiqtoGas`. This subdomain will be defined in a manner consistent with the process of vaporization.

1. Create a new subdomain named `LiqtoGas`.

2. On the **Basic Settings** tab, configure the following setting(s):

Setting	Value
Location	B26
Coordinate Frame	Coord 0

3. On the **Fluid Sources** tab, select `Gas` in the list box, then select the **Gas** check box and configure the following setting(s):

Setting	Value
Equation Sources	Continuity
Equation Sources > **Continuity**	 (Selected)
Equation Sources > Continuity > **Option**	 Fluid Mass Source
Equation Sources > Continuity > **Source**	 Liquid.WaFluxLG
Equation Sources	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
> Continuity > Variables > Steam3v.mf > **Option**	Value
Equation Sources > Continuity > Variables > Steam3v.mf > **Value**	1
Equation Sources > Continuity > Variables > Temperature > **Option**	Value
Equation Sources > Continuity > Variables > Temperature > **Value**	SatT
Equation Sources > Continuity > Variables > Turbulence Eddy Dissipation > **Option**	Value
Equation Sources > Continuity > Variables > Turbulence Eddy Dissipation > **Value**	Gas.ed

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
Equation Sources > Continuity > Variables > Turbulence Kinetic Energy > **Option**	Value
Equation Sources > Continuity > Variables > Turbulence Kinetic Energy > **Value**	Gas.ke
Equation Sources > Continuity > Variables > Velocity > **Option**	Cartesian Vector Components
Equation Sources > Continuity > Variables > Velocity > **U**	Liquid.Velocity u
Equation Sources > Continuity > Variables > Velocity > **V**	Liquid.Velocity v
Equation Sources > Continuity > Variables	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
> Velocity > **W**	 Liquid.Velocity w

4. On the **Fluid Sources** tab, select `Liquid` in the list box, then select the **Liquid** check box and configure the following setting(s):

Setting	Value
Equation Sources	Continuity
Equation Sources > **Continuity**	 (Selected)
Equation Sources > Continuity > **Option**	 Fluid Mass Source
Equation Sources > Continuity > **Source**	 -Liquid.WaFluxLG
Equation Sources > Continuity > **MCF/Energy Sink Option**	 (Selected)
Equation Sources > Continuity > MCF/Energy Sink Option > **Sink Option**	 Spec. Mass Frac. and Temp.
Equation Sources > Continuity > Variables > Temperature > **Option**	 Value
Equation Sources > Continuity > Variables	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
> Temperature > **Value**	 SatT
Equation Sources > Continuity > Variables > Velocity > **Option**	 Cartesian Vector Components
Equation Sources > Continuity > Variables > Velocity > **U**	 0 [m s^-1]
Equation Sources > Continuity > Variables > Velocity > **V**	 0 [m s^-1]
Equation Sources > Continuity > Variables > Velocity > **W**	 0 [m s^-1]
Equation Sources	Energy
Equation Sources > **Energy**	 (Selected)
Equation Sources > Energy > **Option**	 Source[a]
Equation Sources	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
> Energy > **Source**	 -Liquid.WaFluxLG*HtVapwa
Equation Sources > Energy > **Source Coefficient**	 (Selected)
Equation Sources > Energy > Source Coefficient > **Source Coefficient**	 -Liquid.vf*Liquid.density*Liquid.Cp/DtFalseMf
a. The liquid energy source is defined to account for latent heat due to the phase change. The latent heat is assumed to come from the liquid.	

5. Click **OK**.

29.4.7. Creating Boundaries

This section outlines the steps to create the following boundaries: an inlet boundary, `Gas Inlet`, for the location where the steam is injected, an opening boundary, `Opening`, for the outer edges of the domain, and two symmetry boundaries. The wall of the injection pipe will assume the default boundary (a smooth, no-slip wall).

29.4.7.1. Inlet Boundary

At the gas inlet, create an inlet boundary that injects wet steam at a normal speed and static temperature set consistent with the problem description. The steam contains a liquid and vapor component whose sum of volume fractions is unity.

1. Create a new boundary named `Gas Inlet`.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Inlet
	Location	gas inlet
Boundary Details	Mass And Momentum > **Option**	 Normal Speed
	Mass And Momentum > **Normal Speed**	 341 [m s^-1]

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Turbulence > **Option**	Fluid Dependent
	Heat Transfer > **Option**	Static Temperature
	Heat Transfer > **Static Temperature**	373 K
Fluid Values	**Boundary Conditions**	Gas
	Boundary Conditions > Gas > Turbulence > **Option**	Low (Intensity = 1%)
	Boundary Conditions > Gas > Volume Fraction > **Option**	Value
	Boundary Conditions > Gas > Volume Fraction > **Volume Fraction**	1-0.45*0.4/1000 [a]
	Boundary Conditions > Gas > **Component Details**	Steam3v
	Boundary Conditions > Gas > Component Details > Steam3v > **Option**	Mass Fraction
	Boundary Conditions > Gas	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Component Details > Steam3v > **Mass Fraction**	 1
	Boundary Conditions	Liquid
	Boundary Conditions > Liquid > Volume Fraction > **Option**	 Value
	Boundary Conditions > Liquid > Volume Fraction > **Volume Fraction**	 0.45*0.4/1000 [a]
a. Click the *Enter Expression* icon to specify the CEL expression.		

3. Click **OK**.

29.4.7.2. Opening Boundary for the Outside Edges

For the outer edges of the domain, where dry air may be entrained into the flow, specify an opening boundary with a fixed pressure and flow direction. The direction specification is necessary to sufficiently constrain the velocity. At this opening boundary you need to set the temperature of air that may enter through the boundary. Set the opening temperature to be consistent with the problem description.

1. Create a new boundary named `Opening`.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Opening
	Location	air inlet,outer edge,outlet [a]
Boundary Details	Mass And Momentum > **Option**	 Opening Pres. and Dirn
	Mass And Momentum	

Tab	Setting	Value
	> **Relative Pressure**	0 [Pa]
	Flow Direction > **Option**	Normal to Boundary Condition
	Turbulence > **Option**	Medium (Intensity = 5%)
	Heat Transfer > **Option**	Opening Temperature
	Heat Transfer > **Opening Temperature**	25 [C] [b]
Fluid Values	**Boundary Conditions**	Gas
	Boundary Conditions > Gas > Volume Fraction > **Option**	Value
	Boundary Conditions > Gas > Volume Fraction > **Volume Fraction**	1
	Boundary Conditions > Gas > **Component Details**	Steam3v
	Boundary Conditions > Gas > Component Details > Steam3v > **Option**	Mass Fraction
	Boundary Conditions > Gas	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Component Details > Steam3v > **Mass Fraction**	 0.0
	Boundary Conditions	Liquid
	Boundary Conditions > Liquid > Volume Fraction > **Option**	 Value
	Boundary Conditions > Liquid > Volume Fraction > **Volume Fraction**	 0
a. Click the *Ellipsis* icon ... to open the selection dialog box, then select multiple items by holding the **Ctrl** key. Click **OK**. b. Ensure that units are set to [C].		

3. Click **OK**.

29.4.7.3. Creating the Symmetry Plane Boundaries

1. Create a new boundary named SymP1.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Symmetry
	Location	F29.26

3. Click **OK**.
4. Create a new boundary named SymP2.
5. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Symmetry

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Location	F27.26

6. Click **OK**.

Note:

The gas-to-liquid and liquid-to-gas mass transfer model is not symmetric. Consequently, the boundary layer between the liquid and gas acts as an interface that is not explicitly simulated. For simplicity, the boundary layer is imagined to *belong* to the gas phase in which the mass transfer is driven by a departure from equilibrium. The equilibrium condition must therefore be known. For a pure liquid the equilibrium boundary layer vapor fraction is just the gas saturation value. For a multicomponent liquid the boundary layer vapor fraction is related to the liquid component mole fraction via Raoult's law, for example. For this case the gas saturation value of the partial pressure is used, which is the same as the ratio of the partial pressure to the gas pressure if it is in terms of mole fraction.

29.4.8. Creating a Time Step Expression

The conditions at each boundary determine the size of the time scale used in the time step. Generally, you can estimate an effective time step by dividing the displacement by the velocity at which a fluid is traveling. In this case, however, the velocity at the gas inlet approaches the speed of sound and the time step must be calculated by taking the height of the inlet and dividing it by the velocity at which the steam enters the system. The lower velocities at the outlet and opening boundaries enable the time step to be increased after the gas inlet properties have converged. Once all the values at the inlet, outlet, and openings have converged, a much larger time step is used to enable the overall solution to settle. In order to account for all these time step changes, an expression will be created.

Since the flow velocities are high at the jet inlet, you need to use a very small time step to capture the property variations at this location. The flow velocity decreases as you move away from the jet inlet, therefore the time step can be increased systematically for better efficiency. You will now create a time step control expression called `Dtstep` that ramps up the time scale in stages:

1. Right-click `Expressions` in the **Outline** tree view and select **Insert** > **Expression**.
2. Set the name to `Dtstep` and click **OK**.
3. In the **Definition** area, type or copy and paste the following expression:

```
if (aitern <= 20, 1.0E-5[s], 5.0E-3 [s])
```

4. Click **Apply**.

29.4.9. Setting Solver Control

1. Click *Solver Control* .
2. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Basic Settings	Advection Scheme > **Option**	High Resolution
	Convergence Control > **Max Iterations**	1500
	Convergence Control > Fluid Timescale Control > **Timescale Control**	Physical Timescale
	Convergence Control > Fluid Timescale Control > **Physical Timescale**	Dtstep
	Convergence Criteria > **Residual Type**	RMS
	Convergence Criteria > **Residual Target**	1.0E-4
	Convergence Criteria > **Conservation Target**	(Selected)
	Convergence Criteria > Conservation Target > **Value**	0.005
Advanced Options	Dynamic Model Control > **Global Dynamic Model Control**	(Selected)
	Multiphase Control	(Selected)
	Multiphase Control > **Volume Fraction Coupling**	(Selected)
	Multiphase Control > Volume Fraction Coupling > **Option**	Segregated

3. Click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

29.4.10. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .
2. In the **Write Solver Input File** dialog box, set **File name** to `SteamJet.def` and click **Save**.
3. If using stand-alone mode, quit CFX-Pre, saving the case (`.cfx`) file at your discretion.

29.5. Obtaining the Solution Using CFX-Solver Manager

When CFX-Pre has shut down and the CFX-Solver Manager has started, obtain a solution to the CFD problem by following the instructions below:

1. Ensure that the **Define Run** dialog box is displayed.

   Solver Input File should be set to `SteamJet.def`

2. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. At the end of the run, a dialog box is displayed stating that the simulation has ended.

3. Note the number of iterations required to obtain a solution.
4. Select **Post-Process Results**.
5. If using stand-alone mode, select **Shut down CFX-Solver Manager**.
6. Click **OK**.

29.6. Viewing the Results Using CFD-Post

In this section, the steam molar fraction in the gas fluid, the gas-to-liquid and liquid-to-gas mass transfer rates, and the false time step will be illustrated on various regions.

29.6.1. Displaying the Steam Molar Fraction

1. Right-click a blank area in the viewer and select **Predefined Camera** > **View From -Z**.

   This ensures that the view is set to a position that is best suited to display the results.

2. From the menu bar, select **Insert** > **Contour**.
3. Under **Name**, type `Steam Molar Fraction` and click **OK**.
4. Configure the following setting(s):

Tab	Setting	Value
Geometry	**Locations**	SymP2

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Variable	Gas.Steam3v.Molar Fraction

5. Click **Apply**.

This will result in SymP2 shown colored by the molar fraction of steam.

29.6.2. Displaying the Gas-to-Liquid Mass Transfer Rate

1. Turn off the visibility of `Steam Molar Fraction`.
2. From the menu bar, select **Insert** > **Contour**.
3. Under **Name**, type `Gas to Liquid Flux` and click **OK**.
4. Configure the following setting(s):

Tab	Setting	Value
Geometry	**Locations**	SymP2
	Variable	Liquid.WaFluxGL

5. Click **Apply**.

This will result in SymP2 shown colored by the gas-to-liquid mass transfer rate.

29.6.3. Displaying the Liquid-to-Gas Mass Transfer Rate

1. Turn off the visibility of `Gas to Liquid Flux`.
2. From the menu bar, select **Insert** > **Contour**.
3. Under **Name**, type `Liquid to Gas Flux` and click **OK**.
4. Configure the following setting(s):

Tab	Setting	Value
Geometry	**Locations**	SymP2
	Variable	Liquid.WaFluxLG

5. Click **Apply**.

This will result in SymP2 shown colored by the liquid-to-gas mass transfer rate.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

29.6.4. Displaying the Gas-to-Liquid and Liquid-to-Gas Phase Transfer Rates in Synchronous Views

1. In the viewer toolbar, open the viewport drop-down menu and click the option with two horizontal viewports.
2. In the viewer toolbar, disable *Synchronize visibility in displayed views* .
3. Click a blank area in `View 2` so that it becomes the active view.
4. In the tree view, select the check box beside `Gas to Liquid Flux`.

Note:

- You must disable *Synchronize visibility in displayed views* to enable separate contours to be displayed in each viewport
- Under `User Locations and Plots` in the tree view, ensure that only `Liquid to Gas Flux` is visible in `View 1`, and only `Gas to Liquid Flux` is visible in `View 2`.

---

5. In `View 2`, right-click a blank area in the viewer and select **Predefined Camera** > **View From -Z**.

   This ensures that the view is set to a position that is best suited to display the results.

You may want to zoom in near the gas inlet to view the differences between the gas-to-liquid and liquid-to-gas phase transfer rates.

29.6.5. Creating a Chart to Plot the False Time Step Along a Line

1. In the tree view, right-click `User Locations and Plots` and select **Insert** > **Location** > **Line**.
2. In the `Insert Line` dialog box, use the default name and click **OK**.
3. Configure the following setting(s):

Tab	Setting	Value
Geometry	Definition > **Method**	Two Points
	Definition > **Point 1**	0, 0.005, 0.0002
	Definition	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Point 2**	5, 0.005, 0.0002
	Line Type > **Cut**	 (Selected)

4. Click **Apply**.
5. From the menu bar, select **Insert** > **Chart**.
6. Name the chart `False Time Step` and click **OK**.
7. Configure the following setting(s):

Tab	Setting	Value
General	**Title**	False Time Step
Data Series	Data Source > **Location**	 Line 1
X Axis	Data Selection > **Variable**	 X
Y Axis	Data Selection > **Variable**	 Liquid.FalseDt
	Axis Range > **Logarithmic Scale**	 (Selected)

8. Click **Apply**.

The false time step peaks where the interphase mass transfer rate changes sign, and hence goes through zero. This is true because the false time step is inversely proportional to the absolute mass transfer rate.

When you have finished viewing the chart, quit CFD-Post.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 30: Modeling a Buoy using the CFX Rigid Body Solver

This tutorial includes:

30.1. Tutorial Features

30.2. Overview of the Problem to Solve

30.3. Preparing the Working Directory

30.4. Simulating the Buoy with Fully Coupled Mesh Motion

30.5. Simulating the Buoy with Decoupled Mesh Motion

30.6. Comparing the Two Cases Using CFD-Post

30.1. Tutorial Features

In this tutorial you will learn about:

- Modeling a multiphase simulation in CFX-Pre.
- Creating and editing a rigid body in CFX-Pre.
- Creating and editing a subdomain in CFX-Pre.
- Creating a keyframe animation in CFD-Post.

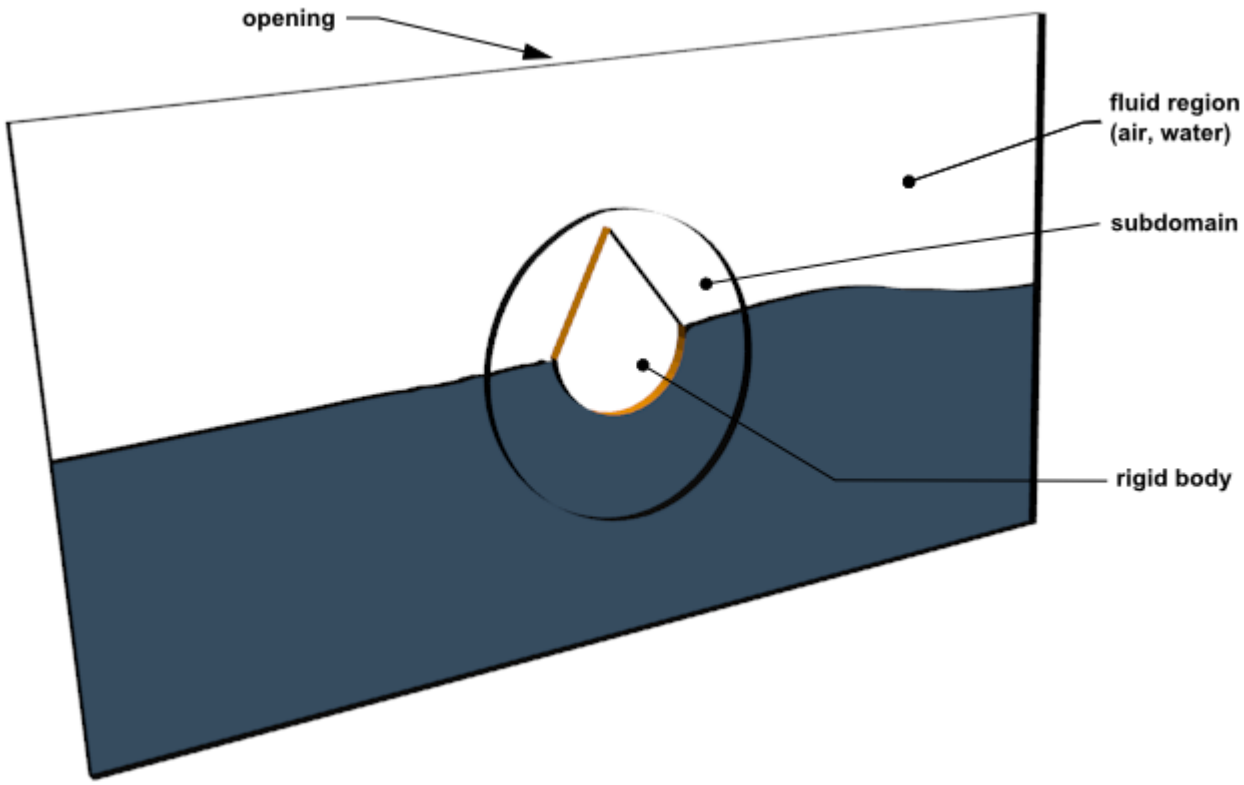
Component	Feature	Details
CFX-Pre	User Mode	General mode
	Analysis Type	Transient
	Fluid Type	General Fluid
	Domain Type	Single Domain
	Turbulence Model	Shear Stress Transport
	Heat Transfer	Isothermal
	Buoyant Flow	
	Multiphase	Homogeneous Model
	Rigid Body	3 degrees of freedom
	Boundary Conditions	Symmetry Plane
		Wall: No Slip
		Wall (Specified Displacement)
		Opening

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Component	Feature	Details
		Mesh Motion option of Rigid Body Solution
	Subdomain	Mesh Motion option of Rigid Body Solution
	Ansys CFX Command Language (CCL)	Importing Expressions
CFD-Post	Plots	Contour Plot
	Animations	Keyframe

30.2. Overview of the Problem to Solve

In this tutorial you will model the interaction between a rigid body (represented by a buoy) and two fluids (air and water) that make up the surrounding region, using a six degrees of freedom rigid-body solver. In this case, the fluid forces acting on the buoy cause motion that is constrained to three degrees of freedom: vertical and horizontal translation and rotation about an axis perpendicular to the translational directions. The motion of the floating buoy results from interactions between itself and the wave motion of the surrounding fluid created by an initial contraction of the domain in the X direction.



The rigid body is surrounded by a fluid volume that is part air and part water (both at a static temperature of 25 °C). Because the rigid body has a density of 500 kg m^-3 – less than that of water (997 kg m^-3) – it floats atop the water's surface. The right-side wall, highlighted yellow in the image above, is given an initial velocity in the negative X direction, thereby causing the fluid domain to shrink. This in turn causes waves in the water. An opening is required along the top face to enable air to move in and out of the fluid region while the volume fraction of air and water are in a state of flux. Because of this

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

contraction of the fluid region, you will also need to define the mesh motion of the domain, subdomain, and several of the boundary conditions.

A homogeneous, multiphase model will be used for this simulation because the air and water will maintain a well-defined interface. When setting up the initial conditions for the simulation, CCL-defined step functions will be used to determine the volume fractions of water and air defined by a function of height.

The relevant fluid parameters of this problem are:

- Density of water = `997 [kg m^-3]`
- Static temperature of water = `25 [C]`
- Density of air = `1.185 [kg m^-3]`
- Static temperature of air = `25 [C]`

The relevant physical parameters of the rigid body are:

- Mass = `39.39 [kg]`
- Density = `500 [kg m^-3]`
- Volume = `0.07878 [m^3]`
- Mass moment of inertia (XX, YY, ZZ, XY, XZ, YZ) = `(4.5, 2.1, 6.36, 0, 0, 0) [kg m^2]`
- Initial Center of Mass (X, Y, Z) = `(0, −0.1438, 0.05) [m]`

The first step in solving this problem is to import a pre-existing mesh file into CFX-Pre. A CCL file containing several mathematical expressions for this simulation will also be imported into CFX-Pre. The transient analysis conditions will then be defined and the default domain edited. A number of boundary conditions will also be created within CFX-Pre. Mesh motion within the domain and several of the boundary conditions will be specified because the domain will contract at the beginning of the simulation, and because the buoy will move freely due to wave motion causing motion of the fluids and hence the rigid body within the domain. In the first simulation, the motion of the mesh surrounding the rigid body will be fully coupled to the motion of the buoy, including the rotation of the buoy; the mesh will both rotate and translate with the buoy. In the second simulation the rotational and translational motion will be decoupled, enabling the inner cylindrical subdomain to rotate at the same rate as the buoy while the outer domain will deform solely with the translational motion of the buoy. In both simulations in CFD-Post, a contour plot will be created to visualize the air/water makeup of the fluid region and the mesh will be visible to observe the mesh when it deforms. In addition, one animation for each simulation will be produced in order to show the complex motion of the rigid body and mesh deformation.

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- Running Ansys CFX Tutorials Using Ansys Workbench (p. 34)
- Changing the Display Colors (p. 37)

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

30.3. Preparing the Working Directory

1. Create a working directory.

   Ansys CFX uses a working directory as the default location for loading and saving files for a particular session or project.

2. Download the `buoy.zip` file here.

3. Unzip `buoy.zip` to your working directory.

   Ensure that the following tutorial input files are in your working directory:

   - `Buoy.gtm`
   - `Buoy.ccl`

4. Set the working directory and start CFX-Pre.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

30.4. Simulating the Buoy with Fully Coupled Mesh Motion

In this simulation, the motion of the mesh surrounding the rigid body will be fully coupled to the motion of the buoy, including the rotation of the buoy; the mesh will both rotate and translate with the buoy.

30.4.1. Defining the Case Using CFX-Pre

This section describes the step-by-step definition of the flow physics in CFX-Pre.

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `Buoy`.
5. Click **Save**.

30.4.1.1. Importing the Mesh

1. Right-click `Mesh` and select **Import Mesh** > **CFX Mesh**.

   The **Import Mesh** dialog box appears.

2. Configure the following setting(s):

Setting	Value
File name	Buoy.gtm

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click **Open**.

30.4.1.2. Importing the Required Expressions From a CCL File

The mathematical expressions for this simulation will be imported from a CFX Command Language (CCL) file. These expressions will be used to set a monitor point and the physical parameters of the simulation: the fluid properties and the displacement of the walls and opening.

Note:

The expressions or physics for a simulation can be saved to a CCL file at any time by selecting **File** > **Export** > **CCL**.

1. Select **File** > **Import** > **CCL**.

   The **Import CCL** dialog box appears.

2. Under **Import Method**, select **Append**. This will start with the existing CCL already generated by CFX-Pre and append the imported CCL.

   Note:

   Replace is useful if you have defined physics and want to update or replace them with newly-imported physics.

3. Select `Buoy.ccl`.

4. Click **Open**.

5. Double-click the `Expressions` section in the **Outline** tree to see a list of the expressions that have been imported.

   All expressions required for this simulation are displayed. Take a moment to look over each expression. A brief description of each expression will be provided wherever it is implemented within this tutorial.

6. Close the `Expressions` section by clicking *Close* located at the top of the left workspace.

   Note:

   Note that you could have entered these expressions manually into CFX-Pre by inserting new expressions and defining each with an appropriate formula.

30.4.1.3. Defining a Transient Simulation

1. In the **Outline** tree view, right-click `Analysis Type` and select **Edit**.

2. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Basic Settings	Analysis Type > **Option**	Transient
	Analysis Type > Time Duration > **Option**	Total Time
	Analysis Type > Time Duration > **Total Time**	7.0 [s]
	Analysis Type > Time Steps > **Option**	Timesteps
	Analysis Type > Time Steps > **Timesteps**	0.025 [s] [a]
	Analysis Type > Initial Time > **Option**	Automatic with Value
	Analysis Type > Initial Time > **Time**	0 [s]

a. A total time of `7.0 [s]` is implemented so that you get an adequate overview of the rigid body motion during postprocessing. The `0.025 [s]` time step provides enough detail in the solution without requiring an excessive amount of computation time for CFX-Solver.

3. Click **OK**.

Note:

You may ignore the physics validation messages regarding the lack of definition of transient results files and the lack of initial condition values. You will set up the transient results files and set the initial conditions later.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

30.4.1.4. Editing the Domain

In this section you will create the fluid domain to reflect the multiphase, homogeneous region surrounding the buoy, define the fluids, and enable mesh motion.

1. Edit `Case Options` > `General` in the **Outline** tree view, ensure that **Automatic Default Domain** and **Automatic Default Interfaces** are both selected, and click **OK**.

2. In the tree view, right-click `Default Domain`, select **Rename**, and set the new name to `buoy`.

3. In the tree view, right-click the newly renamed domain and select **Edit**.

4. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Location and Type > **Location**	`Assembly` [a]
	Fluid and Particle Definitions	Delete `Fluid 1` [b]
	Fluid and Particle Definitions	Create a new fluid named `Air at 25 C` [c]
	Fluid and Particle Definitions	Create a new fluid named `Water at 25 C` [c]
	Fluid and Particle Definitions > Air at 25 C > **Material**	Air at 25 C
	Fluid and Particle Definitions > Water at 25 C > **Material**	Water at 25 C [d]
	Domain Models > Buoyancy Model > **Option**	Buoyant
	Domain Models > Buoyancy Model > **Gravity X Dirn.**	0 [m s^-2]

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Domain Models > Buoyancy Model > **Gravity Y Dirn.**	`-g` [e]
	Domain Models > Buoyancy Model > **Gravity Z Dirn.**	0 [m s^-2]
	Domain Models > Buoyancy Model > **Buoy. Ref. Density**	`denAir` [e][f]
	Domain Models > Mesh Deformation > **Option**	Regions of Motion Specified [g]
	Domain Models > Mesh Deformation > Mesh Motion Model > **Option**	Displacement Diffusion [h][i]
	Domain Models > Mesh Deformation > Mesh Motion Model > Mesh Stiffness > **Option**	Value
	Domain Models > Mesh Deformation > Mesh Motion Model > Mesh Stiffness > **Mesh Stiffness**	1.0 [m^5 s^-1] / volcvol [e][j]
Fluid Models	Multiphase	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Homogeneous Model**	(Selected) [k]
	Multiphase > Free Surface Model > **Option**	Standard
	Multiphase > Free Surface Model > **Interface Compression Level**	(Selected)
	Multiphase > Free Surface Model > Interface Compression Level > **Interface Compression**	2
	Heat Transfer > **Homogeneous Model**	(Selected)
	Turbulence > **Option**	Shear Stress Transport
Fluid Pair Models	Fluid Pair > Air at 25 C | Water at 25 C > Interphase Transfer > **Option**[l]	Mixture Model
	Fluid Pair > Air at 25 C | Water at 25 C > Interphase Transfer > **Interface Len. Scale**	1.0 [mm]

a. Click *Multi-select from extended list* to open the **Selection Dialog** box, then select `Assembly` from this list. Click **OK**.

b. Ensure that you have `Fluid 1` selected and click *Remove Selected Item* .

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
c. To create a new item, you must first click the *Add new item* icon, then enter the name as required and click **OK**. d. Click *Select from extended list* and then on *Import Library Data* in the upper right corner of the resulting `Material` dialog box. When the `Select Library Data to Import` box appears, click *Expand* located beside **Water Data**. Select `Water at 25 C` from the list. Click **OK**. e. In order to enter an expression, you must first click *Enter Expression* . f. The buoyancy reference density is set to 1.185 kg/m^3, which is representative of air. g. This mesh deformation option enables you to specify the motion of the boundary mesh nodes using user-defined expressions created in the CFX Expression Language (CEL). These expressions of mesh motion are included in the CCL file that was imported at the beginning of the tutorial. h. To see the additional mesh motion settings, you may need to click *Roll Down* located beside **Mesh Motion Model**. i. The Displacement Diffusion model for mesh motion preserves the relative mesh distribution of the initial mesh. j. The variable `volcvol` (volume of finite volumes) is a predefined variable related to the local mesh element volume. It is used here in the calculation of the mesh stiffness value. In this example, the mesh stiffness is set to be inversely proportional to `volcvol`, which results in higher stiffness in regions of smaller element size; these are the regions that are most likely to experience mesh folding. This is similar to the default setting of **Mesh Stiffness** > **Option** set to `Increase Near Small Volumes` with **Reference Volume** > **Option** set to `Mean Control Volume`. k. In a homogeneous, multiphase model, all fluids share a flow field, turbulence field, and so on. This is valid for models where the fluids have completely stratified; this is the case in this simulation. l. The interphase transfer model controls the calculation of interfacial area density, which is required by certain interfacial transfer processes. In this case, the homogeneous model is used and no other interfacial transfer processes are active so the actual setting does not matter.		

5. Click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

30.4.1.5. Creating a Rigid Body

In this section you will specify the properties of a rigid body with three degrees of freedom: translation in the X and Y directions and rotation about the Z axis. The rigid body definition will be applied to the wall boundary of the buoy to define the motion characteristics of the buoy. Further, you will specify the direction of gravity that acts upon the buoy's mass. Aside from gravity, no external forces are specified to act continuously on the buoy, however the motion of the buoy will be driven by fluid forces (from both the air and water) acting on the surface of the rigid body.

1. It is very important to give the rigid body a coordinate frame that is centered on its center of mass. Create a coordinate frame centered on the rigid body in its initial position, and oriented with its axes aligned with the rigid body axes/global coordinate axes as follows:

   a. Select **Insert** > **Coordinate Frame**.

   b. In the dialog box that appears, set **Name** to `RigidBodyCoordFrame` and click **OK**. The **Basic Settings** tab for the coordinate frame appears.

   c. Set **Option** to `Axis Points`.

   d. Set **Origin** to `0, –0.1438, 0.05`.

   e. Set **Z Axis Point** to `0, –0.1438, 1`.

   f. Set **X-Z Plane Pt** to `1, –0.1438, 0.05`.

   g. Click **OK**.

2. In the **Outline** tree view, right-click **Flow Analysis 1** and select **Insert** > **Rigid Body**.

3. Accept the default name, `Rigid Body 1`, by clicking **OK**.

4. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Mass**	39.39 [kg] [a]
	Location	BUOY
	Coordinate Frame	RigidBodyCoordFrame
	Mass Moment of Inertia > **XX Component**	4.5 [kg m^2]
	Mass Moment of Inertia > **YY Component**	2.1 [kg m^2]
	Mass Moment of Inertia > **ZZ Component**	6.36 [kg m^2]
	Mass Moment of Inertia > **XY Component**	0 [kg m^2]

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Mass Moment of Inertia > **XZ Component**	 0 [kg m^2]
	Mass Moment of Inertia > **YZ Component**	 0 [kg m^2]
Dynamics	Degrees of Freedom > Translational Degrees of Freedom > **Option**	 X and Y axes
	Degrees of Freedom > Rotational Degrees of Freedom > **Option**	 Z axis
	Gravity	(Selected)
	Gravity > **Option**	 Cartesian Components
	Gravity > **Gravity X Dirn.**	 0 [m s^-2]
	Gravity > **Gravity Y Dirn.**	 -g [b]
	Gravity > **Gravity Z Dirn.**	 0 [m s^-2]
Initial Conditions	**Center of Mass**	(Selected)
	Center of Mass > **Option**	 Automatic[c]
	Linear Velocity	(Selected)
	Linear Velocity > **Option**	 Automatic with Value
	Linear Velocity > **X Component**	 0 [m s^-1]
	Linear Velocity > **Y Component**	 0 [m s^-1]
	Linear Velocity	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Z Component**	0 [m s^-1]
	Angular Velocity	(Selected)
	Angular Velocity > **Option**	Automatic with Value
	Angular Velocity > **X Component**	0 [radians s^-1]
	Angular Velocity > **Y Component**	0 [radians s^-1]
	Angular Velocity > **Z Component**	0 [radians s^-1]

a. The values in this table are taken directly from the problem description found in the Overview of the Problem to Solve (p. 726) section.

b. In order to enter an expression, you must first click *Enter Expression* .

c. Setting this option to **Automatic** defaults the center of mass of the rigid body to the origin of the `RigidBodyCoordFrame`. In most cases, this will be the correct setting.

5. Click **OK**.

30.4.1.6. Creating the Boundary Conditions

In this section symmetry boundaries will be created for the front and back planes of the given geometry; this is required because a 2D representation of the flow field is being modeled. Wall boundaries will also be created for the bottom, stationary side, moving side, and buoy body sections of the fluid region. Because the right-side wall will be provided with an initial velocity in the negative X direction, you will define mesh motion along this direction for the moving wall boundaries. An opening boundary will also be created along the top of the fluid region to enable air to flow freely in and out of this region because of the interactions between the air and water (as a result of the moving wall).

30.4.1.6.1. Symmetry Boundaries

The front and back planes each require a symmetry boundary.

1. Create a new boundary named `back`.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Symmetry
	Location	BACK A, BACK B [a]
Boundary Details	Mesh Motion > **Option**	Unspecified [b]

a. Hold the **Ctrl** key while selecting both BACK A and BACK B from the list.

b. In the unspecified mesh motion option, no mesh motion constraints are applied directly to the nodes. Instead, mesh motion is governed by the constraints in other regions of the mesh.

3. Click **OK**.

4. Create a second boundary named `front`.

5. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Symmetry
	Location	FRONT A, FRONT B [a]
Boundary Details	Mesh Motion > **Option**	Unspecified[b]

a. Hold the **Ctrl** key while selecting both FRONT A and FRONT B from the list.

b. In the unspecified mesh motion option, no mesh motion constraints are applied directly to the nodes. Instead, mesh motion is governed by the constraints in other regions of the mesh.

6. Click **OK**.

30.4.1.6.2. Wall Boundaries

The top, bottom, and sides of the fluid region all require wall boundaries. In addition, the surface between the fluid region and the rigid body, `Rigid Body 1`, requires a wall boundary; this wall boundary will move according to the rigid body solution.

1. Create a new boundary named `Buoy Surface`.

2. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Basic Settings	**Boundary Type**	Wall
	Location	BUOY
Boundary Details	Mesh Motion > **Option**	Rigid Body Solution
	Mesh Motion > **Rigid Body**	Rigid Body 1
	Mass and Momentum > **Option**	No Slip Wall
	Wall Roughness > **Option**	Smooth Wall

3. Click **OK**.
4. Create a new boundary named `wall`.
5. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Wall
	Location	BOTTOM, S1, S2 [a]
Boundary Details	Mesh Motion > **Option**	Specified Displacement
	Mesh Motion > Displacement > **Option**	Cartesian Components
	Mesh Motion > Displacement > **X Component**	wallMeshMot [b][c]
	Mesh Motion > Displacement > **Y Component**	0.0 [m] [d]
	Mesh Motion	

Tab	Setting	Value
	> Displacement > **Z Component**	 0.0 [m]
	Mass and Momentum > **Option**	 No Slip Wall

a. Hold the **Ctrl** key while selecting `BOTTOM`, `S1`, and `S2` from the list.

b. In order to enter an expression, you must first click *Enter Expression* .

c. This displacement expression, defined in the CCL, will enable the mesh nodes to shift in the X direction because the fluid domain is compressed in this direction during the simulation. The X-coordinate runs from -4.0 m to 4.0 m, but this mesh motion along the bottom of the fluid region only occurs when the X-coordinate is between 0 m and 4.0 m. Thus, `wallMesh Mot` uses a simple logical expression to apply the mesh motion only when the X-coordinate is greater than or equal to 0 m.

   The second displacement expression that is present in the CCL, `wxdisp`, allows the mesh nodes to shift in the X direction, because the right-side wall will be moving and compressing the fluid region during the simulation. It is defined in the CCL as a logical expression that remains steady at `0.0 [m]` until it is turned "on" at time `tOn`, at which point a time-dependent, sinusoidally increasing displacement is applied. This displacement increases until 1.0 seconds into the simulation, when it plateaus at this final value. The expression `wallMeshMot` is dependent on `wxdisp`, as can be seen in the CCL, and will therefore take the motion of the moving side wall into account. This expression will have no effect on the stationary wall because this wall has an X-coordinate that is outside the part of the expression that specifies a displacement. Thus, `wallMeshMot` can be applied to all three of these walls; it is equally applicable to each.

d. The left-side of the fluid region maintains its position throughout the simulation and it is necessary to define mesh deformation of the bottom in the X direction only. Therefore, set the mesh displacement in the Y and Z directions to `0.0 [m]`.

6. Click **OK**.

30.4.1.6.3. Opening Boundary

1. Create a new boundary named `top`.

2. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Basic Settings	**Boundary Type**	Opening
	Location	TOP
Boundary Details	Mesh Motion > **Option**	Specified Displacement
	Mesh Motion > Displacement > **Option**	Cartesian Components
	Mesh Motion > Displacement > **X Component**	wallMeshMot [a]
	Mesh Motion > Displacement > **Y Component**	0.0 [m]
	Mesh Motion > Displacement > **Z Component**	0.0 [m]
	Mass and Momentum > **Option**	Opening Pres. and Dirn
	Mass and Momentum > **Relative Pressure**	0 [Pa]
Fluid Values	Boundary Conditions > Air at 25 C > Volume Fraction > **Volume Fraction**	1.0
	Boundary Conditions > Water at 25 C > Volume Fraction > **Volume Fraction**	0.0 [b]

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
a. The same mesh motion is provided for the `top` boundary and the `bottom` boundary. They will move in unison. b. The `top` boundary comes into contact only with air, and not with water. The volume fraction of the opening for air is set to 1.0 and that of water to 0.0, therefore enabling only air to pass through the opening.		

3. Click **OK**.

Note:

Opening boundary types are used to enable the flow to leave and re-enter the domain. This behavior is expected due to the motion of the water and the interaction between the air and water in the fluid region.

30.4.1.7. Setting Initial Values

Because a transient simulation is being modeled, initial values are required for all variables.

1. Click *Global Initialization* .

2. Configure the following setting(s):

Tab	Setting	Value
Global Settings	Initial Conditions > Cartesian Velocity Components > **U**	0 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > **V**	0 [m s^-1]
	Initial Conditions > Cartesian Velocity Components > **W**	0 [m s^-1]
	Initial Conditions > Static Pressure	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

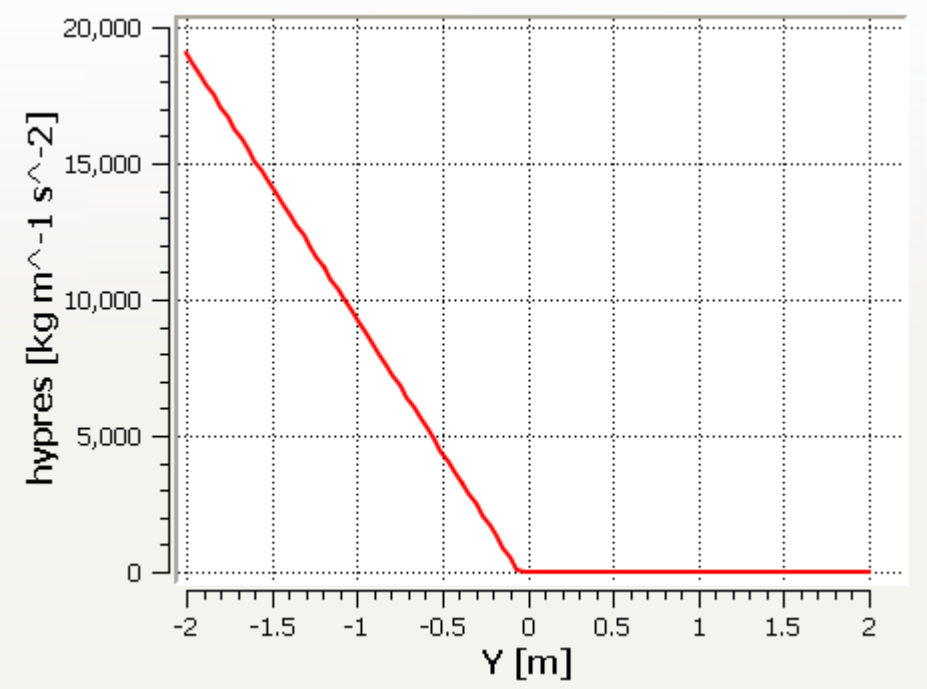
Tab	Setting	Value
	> **Option**	Automatic with Value
	Initial Conditions > Static Pressure > **Relative Pressure**	hypres [a][b]
	Initial Conditions > Turbulence > **Option**	Intensity and Eddy Viscosity Ratio
Fluid Settings	Fluid Specific Initialization > Air at 25 C > Initial Conditions > Volume Fraction > **Volume Fraction**	airvol [a]
	Fluid Specific Initialization > Water at 25 C > Initial Conditions > Volume Fraction > **Volume Fraction**	watvol [a][c]

a. In order to enter an expression, you must first click *Enter Expression* .

b. The expression `hypres` is defined in the CCL file and gives the relative pressure as a function of the volume of water in the fluid region and the vertical distance (Y direction value). The expression results in an initial pressure gradient that linearly decreases when the value of the Y-coordinate increases (or the water depth decreases). When the Y-coordinate reaches the interface between the air and water, the initial pressure plateaus at a value of zero (due to the reliance of the expression `hypres` on `watvol`). The initial pressure gradient produced by this expression can be observed in the image below.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value



To create this plot in CFX-Pre:

i. Double-click **Expressions** in the **Outline** tree view.

ii. Double-click the `hypres` expression.

iii. Select the **Plot** tab under `Details of` **hypres**.

iv. Select `Expression Variables: Y`.

v. Set the `Start of Range` to `-2.0 [m]` and `End of Range` to `2.0 [m]`.

vi. Click **Plot Expression**.

c. The `airvol` and `watvol` expressions were provided in the CCL file. The expression `watvol` is a step function that returns a value of `1.0` for all Y-coordinate values less than the initial water height (0.05 [m]), `0.0` for all Y-coordinate values greater than the initial water height, and `0.5` when the Y-coordinate value is equal to the initial water height.

3. Click **OK**.

30.4.1.8. Setting the Solver Control

In this section, you will adjust the solver control settings to promote a quicker solution time and to enable the frequency of when the rigid body solver is run.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click *Solver Control* .

2. Configure the following setting(s):

Tab	Setting	Value
Equation Class Settings	Equation Class > **Mesh Displacement**	(Selected)
	Equation Class > Mesh Displacement > **Convergence Control**	(Selected)
	Equation Class > Mesh Displacement > Convergence Control > **Max. Coeff. Loops**	4 [a]
	Equation Class > Mesh Displacement > Convergence Control > **Min. Coeff. Loops**	2
Rigid Body Control	**Rigid Body Control**	(Selected)
	Rigid Body Control > Rigid Body Solver Coupling Control > **Update Frequency**	Every Coefficient Loop [b]
	Rigid Body Control > **Angular Momentum Equation Control**	(Selected) [c]
Advanced Options	**Multiphase Control**	(Selected)
	Multiphase Control > **Initial Volume Fraction Smoothing**	(Selected)
	Multiphase Control > Initial Volume Fraction Smoothing > **Option**	Volume-Weighted [d]

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
a. The maximum number of coefficient loops is set to 4 and the minimum number of coefficient loops to 2 to ensure that the solver completes at least 2 loops per time step, and no more than 4. In this simulation it will ensure a relatively resolved and accurate solution within a short period of time. b. By setting the update frequency to every coefficient loop you are specifying that CFX-Solver will call the rigid body solver during every coefficient loop within each time step. This may increase total solution time, however the motion of the rigid body will be better resolved. c. This sets the integration scheme for the angular momentum equations to the second-order Simo Wong scheme, which is robust and energy-conserving. d. If the initial conditions for volume fraction have a discontinuity, startup robustness problems may occur. Choosing volume-weighted smoothing of these volume fractions may improve startup robustness.		

3. Click **OK**.

30.4.1.9. Setting the Output Control

In this section, you will set transient results for selected variables to be captured every three time steps. You will also create two monitor points so that you can track the progress in CFX-Solver Manager.

1. Click *Output Control* .

2. Click the **Trn Results** tab.

3. In the **Transient Results** editor, click *Add new item* , set **Name** to `Transient Results 1`, and click **OK**.

4. Configure the following setting(s) of `Transient Results 1`:

Setting	Value
Transient Results 1 > **Option**	Selected Variables
Transient Results 1 > **Output Variables List**	Pressure, Total Mesh Displacement, Velocity, Water at 25 C.Volume Fraction [a]
Transient Results 1 > Output Frequency	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
> **Option**	Time Interval
Transient Results 1 > Output Frequency > **Time Interval**	 tOn[b]
a. Click *Multi-select from extended list* beside the entry box, and make multiple selections in the **Output Variables List** by holding down the **Ctrl** key and clicking on the required variables. b. In order to enter an expression, you must first click *Enter Expression*.	

5. Click the **Monitor** tab and configure the following setting(s):

Monitor Objects	(Selected)
Monitor Objects > **Monitor Points and Expressions**	 Create a new Monitor Point and enter the name `Buoy Force` [a][b].
Monitor Objects > Monitor Points and Expressions > Buoy Force > **Option**	 Expression
Monitor Objects > Monitor Points and Expressions > Buoy Force > **Expression Value**	 force_y()@Buoy Surface
Monitor Objects > **Monitor Points and Expressions**	 Create a new Monitor Point and enter the name `Buoy Torq` [a][c].
Monitor Objects > Monitor Points and Expressions > Buoy Torq	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Option**	Expression
Monitor Objects > Monitor Points and Expressions > Buoy Torq > **Expression Value**	 torque_z()@Buoy Surface
a. To create a new item, you must first click the *Add new item* icon, then enter the name as required and click **OK**. b. This monitor point will track the force acting on the `rigid body` in the Y direction. c. This monitor point will track the torque of the `rigid body` relative to the Z axis.	

6. Click **OK**.

30.4.1.10. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .

2. Configure the following setting(s):

Setting	Value
File name	Buoy.def

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, **Solver Input File** is set.

4. Quit CFX-Pre, saving the simulation (`.cfx`) file.

30.4.2. Obtaining the Solution Using CFX-Solver Manager

When CFX-Pre has shut down and the CFX-Solver Manager has started, obtain a solution to the CFD problem by following the instructions below.

1. Ensure that the **Define Run** dialog box is displayed.

   Solver Input File should be set to `Buoy.def`.

2. Click **Start Run**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CFX-Solver runs and attempts to obtain a solution. At the end of the run, a dialog box is displayed stating that the simulation has ended.

Note:

After the CFX-Solver Manager has run for a short time, you can track the monitor points you created in CFX-Pre by clicking the `User Points` tab that appears at the top of the graphical interface of CFX-Solver Manager. The two monitor points — Buoy Force and Buoy Torq — are monitored in the global coordinate frame and not the coordinate frame attached to the buoy. You can also view the level of convergence of the rigid body solution through the `Rigid Body Convergence` tab. Finally, the rigid body position and Euler angles can be displayed by going to the main menu and selecting **Monitors** > **Rigid Body** > **Rigid Body Position** and **Monitors** > **Rigid Body** > **Rigid Body Euler Angles**, respectively.

Note:

New monitor points can be toggled within the current plot by right-clicking the plot and selecting **Monitor Properties**. A window will display available plot line variables. Select the box to the left of the property to display it — the plot will adjust the scale so that all properties appear.

3. Select **Post-Process Results**.
4. If using stand-alone mode, select **Shut down CFX-Solver Manager**.
5. Click **OK**.

30.4.3. Viewing the Results Using CFD-Post

In this section, you will create a contour plot for this case. An animation will then be created to show the movement of the rigid body in the fluid domain. Furthermore, the minimum face angle of the mesh will be calculated for comparison purposes between the simulations.

30.4.3.1. Creating a Contour Plot

1. Right-click a blank area in the viewer and select **Predefined Camera** > **View From +Z**.
2. Create a new **Plane** and accept the default name.
3. Configure the following setting(s):

Tab	Setting	Value
Geometry	Definition > **Method**	 XY Plane
	Definition	

Tab	Setting	Value
	> **Z**	0.05 [m]

4. Click **Apply**.
5. Create new contour and accept the default name.
6. Configure the following setting(s):

Tab	Setting	Value
Geometry	**Locations**	Plane 1
	Variable	Water at 25 C.Volume Fraction
	Color Map	White to Blue
	# of Contours	10

7. Click **Apply**.
8. Select **File** > **Save State** and choose the name `Buoy.cst`.
9. Click **Save**.

30.4.3.2. Creating a Keyframe Animation

A short animation of the rigid body and surrounding fluid region, starting from rest and given an initial velocity, will be created to show the complex motion of the rigid body and deformation of the mesh because of the waves created in the fluid region. You will record a short animation that can be played in an MPEG player.

1. Ensure that `Contour 1` is visible in the **3D Viewer** (make sure there is a check mark beside `Contour 1` in the **Outline** tree view).
2. Turn off the visibility of `Plane 1` and `Default Legend View 1` to better see the movement of the buoy.
3. Edit `Wireframe` and configure the following setting(s):

Tab	Setting	Value
Definition	**Show surface mesh**	(Selected)

4. Click **Apply**.
5. Click the *Timestep Selector* in the toolbar. Select the 1st time step and click **Apply**.
6. Click *Animation* in the *Timestep Selector* dialog box.

   The **Animation** dialog box appears.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Set **Type** to **Keyframe Animation**.

8. Click *New* to create `KeyframeNo1`.

9. Select `KeyframeNo1`, then set **# of Frames** to 93, then press **Enter** while the cursor is in the **# of Frames** box.

   Tip:

   Be sure to press **Enter** and confirm that the new number appears in the list before continuing.

10. In the **Timestep Selector**, select time step 280 and click **Apply**.

11. In the **Animation** dialog box, click *New* to create `KeyframeNo2`.

12. Ensure that *More Animation Options* is pushed down to show more animation settings.

13. Select **Loop**.

14. Ensure that *Repeat forever* (next to **Repeat**) is not selected (not pushed down).

15. Select **Save Movie**.

16. Set **Format** to `MPEG1`.

17. Prepare to save the movie file:

    a. If you want to save the file in your working directory, set **File name** to `Buoy.mpg` in the field beside **Save Movie**.

    b. If you want to save the file somewhere other than your working directory, click *Browse* (next to **Save Movie**) set the new path and movie filename. Click **Save**.

    The movie filename (including the path) has been set, but the animation has not yet been produced.

18. Click *To Beginning* .

    This ensures that the animation will begin at the first keyframe.

19. After the first keyframe has been loaded, click *Play the animation* .

    - The MPEG will be created as the animation proceeds.
    - This will be slow, since results will be loaded and objects will be created for each time step.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To view the movie file, you need to use a viewer that supports the MPEG format.

Note:

To explore additional animation options, click the **Options** button. On the **Advanced** tab of the **Animation Options** dialog box, there is a **Save Frames As Image Files** check box. By selecting this check box, the JPEG or PPM files used to encode each frame of the movie will persist after movie creation; otherwise, they will be deleted.

20. Close the **Animation** dialog box when the animation is complete.

30.4.3.3. Calculating the Minimum Mesh Face Angle

In this step, you will calculate the Minimum Face Angle of the mesh which is an indicator of the overall mesh quality during the deformation of the mesh. A Minimum Face Angle of greater than 15° is one indicator of a good quality mesh. However an angle of between 10° and 15° is also acceptable but may produce inaccuracies in that region of the mesh during the simulation.

1. Click the *Timestep Selector* , select the 162nd time step and click **Apply**.
2. Select **Tools** > **Mesh Calculator** or click the **Calculators** tab and select **Mesh Calculator**.
3. Configure the following setting(s):

Tab	Setting	Value
Mesh Calculator	**Function**	Minimum Face Angle

4. Click **Calculate**.
5. When you have finished, close the **Timestep Selector** dialog box and exit from CFD-Post.

The 162nd time step was chosen arbitrarily to contrast the mesh quality between this simulation and the following one. In this simulation the Minimum Face Angle during the 162nd time step is approximately 13°.

30.5. Simulating the Buoy with Decoupled Mesh Motion

In this section you will use a subdomain to decouple rotation (that is, to enable independent rotation of the mesh in each part of the domain). Furthermore, you will edit the domain interface boundaries to restrict mesh deformation on the outer part of the domain to translational only; the inner cylindrical part of the domain will rotate and translate at the same rate as the buoy.

30.5.1. Defining the Case Using CFX-Pre

1. Ensure that the following tutorial input file is in your working directory:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- `Buoy.cfx`

2. Set the working directory and start CFX-Pre if it is not already running.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

3. Select **File** > **Open Case**.
4. From your working directory, select `Buoy.cfx` and click **Open**.
5. Select **File** > **Save Case As**.
6. Set **File name** to `Buoy_decoupled.cfx`.
7. Click **Save**.

30.5.1.1. Creating a Subdomain

The subdomain, domain interfaces, and buoy must share common rigid body characteristics. This is necessary because the inner cylinder and the rigid body must translate and rotate at the same rate to properly isolate the motions. All relative movement between the inner cylinder and the buoy will be eliminated, causing zero mesh deformation within the inner cylinder. A subdomain is not strictly necessary to decouple rotational motions. However, the subdomain increases the robustness of the simulation by ensuring that the entire mesh within the inner cylinder has the same physical properties as the rigid body, not just at the buoy boundary and inner cylinder domain interface (this will be set up in the next step).

1. Select **Insert** > **Subdomain** from the main menu or click *Subdomain* .
2. Set the subdomain name to `rot_trans` and click **OK**.
3. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Location**	B86
Mesh Motion	Mesh Motion > **Option**	Rigid Body Solution
	Mesh Motion > **Rigid Body**	Rigid Body 1
	Mesh Motion > **Motion Constraints**	(Selected)
	Mesh Motion > Motion Constraints > **Motion Constraints**	None

4. Click **OK**.

30.5.1.2. Editing the Domain Interfaces

You will edit the domain interfaces to restrict the rotational movement of the mesh surrounding the subdomain. The mesh that is located on the inner cylindrical domain interface will be assigned the same physical properties as that of the rigid body.

1. Edit `Simulation` > `Flow Analysis 1` > `buoy` > `Default Fluid Fluid Interface Side 1`.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Location**	F74.27[a]
Boundary Details	Mesh Motion > **Option**	Rigid Body Solution
	Mesh Motion > **Rigid Body**	Rigid Body 1
	Mesh Motion > **Motion Constraints**	(Selected)
	Mesh Motion > Motion Constraints > **Motion Constraints**	Ignore Rotations[b]

a. This is the outer cylindrical domain interface.

b. Ignore Rotations constrains the outer domain interface to only translational motion.

3. Click **OK**.

4. Edit `Simulation` > `Flow Analysis 1` > `buoy` > `Default Fluid Fluid Interface Side 2`.

5. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Location**	F89.86[a]
Boundary Details	Mesh Motion > **Option**	Rigid Body Solution

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Mesh Motion > **Rigid Body**	Rigid Body 1
	Mesh Motion > **Motion Constraints**	(Selected)
	Mesh Motion > Motion Constraints > **Motion Constraints**	None
a. This is the inner cylindrical domain interface.		

6. Click **OK**.

30.5.1.3. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .

2. Configure the following setting(s):

Setting	Value
File name	Buoy_decoupled.def

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, **Solver Input File** is set.

4. Quit CFX-Pre, saving the simulation (`.cfx`) file.

30.5.2. Obtaining the Solution Using CFX-Solver Manager

When CFX-Pre has shut down and the CFX-Solver Manager has started, obtain a solution to the CFD problem by following the instructions below.

1. Ensure that the **Define Run** dialog box is displayed.

   Solver Input File should be set to `Buoy_decoupled.def`.

2. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. At the end of the run, a dialog box is displayed stating that the simulation has ended.

3. Select **Post-Process Results**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. If using stand-alone mode, select **Shut down CFX-Solver Manager**.

5. Click **OK**.

30.5.3. Viewing the Results Using CFD-Post

In this section, you will create a contour plot of `Water at 25 C.Volume Fraction` to show the water content of the fluid region, and to illustrate the interface between the air and water. An animation will then be created to show the movement of the rigid body in the fluid domain. Furthermore, the minimum face angle of the mesh will be calculated for comparison purposes between the simulations.

30.5.3.1. Loading a Contour Plot from the State File

In the first part of the tutorial, you created a plane and a contour plot, then saved a state file named `Buoy.cst`. You will load the resulting state file so that you do not have to create a new plane and contour plot:

1. Select **File** > **Load State** and choose the name `Buoy.cst`.

2. Click **Open**.

30.5.3.2. Creating a Keyframe Animation

A short animation of the rigid body and surrounding fluid region, starting from rest and given an initial velocity, will be created to show the complex motion of the rigid body and deformation of the mesh because of the waves created in the fluid region. You will record a short animation that can be played in a MPEG player.

1. Ensure that `Contour 1` is visible in the **3D Viewer** (make sure there is a check mark beside `Contour 1` in the **Outline** tree view).

2. Turn off the visibility of `Plane 1` and `Default Legend View 1` to better see the movement of the buoy.

3. Edit `Wireframe` and configure the following setting(s):

Tab	Setting	Value
Definition	**Show surface mesh**	(Selected)

4. Click the *Timestep Selector* . Select the 1st time step and click **Apply**.

5. Click *Animation* in the *Timestep Selector* dialog box.

   The **Animation** dialog box appears.

6. Set **Type** to **Keyframe Animation**.

7. Click *New* to create `KeyframeNo1`.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. Select `KeyframeNo1`, then set **# of Frames** to 93, then press **Enter** while the cursor is in the **# of Frames** box.

   Tip:

   Be sure to press **Enter** and confirm that the new number appears in the list before continuing.

9. Use the **Timestep Selector** to load the 280th time step.

10. In the **Animation** dialog box, click *New* to create `KeyframeNo2`.

11. Ensure that *More Animation Options* is pushed down to show more animation settings.

12. Select **Loop**.

13. Ensure that *Repeat forever* (next to **Repeat**) is not selected (not pushed down).

14. Select **Save Movie**.

15. Set **Format** to `MPEG1`.

16. Set **File name** to `Buoy_decoupled.mpg`.

17. If you want to save the animation to a location other than your working directory, click *Browse* (next to **Save Movie**) to set the path to a different directory and click **Save**.

    The movie filename (including the path) has been set, but the animation has not yet been produced.

18. Click *To Beginning* .

    This ensures that the animation will begin at the first keyframe.

19. After the first keyframe has been loaded, click *Play the animation* .

    - The MPEG will be created as the animation proceeds.

20. Close the **Animation** dialog box when the animation is complete.

30.5.3.3. Calculating the Minimum Mesh Face Angle

In this section you will calculate the Minimum Face Angle of the mesh, which is an indicator of the overall mesh quality during the deformation of the mesh. A Minimum Face Angle of greater than 15° is one indicator of a good quality mesh. However an angle of between 10° and 15° is also acceptable but may produce inaccuracies in that region of the mesh during the simulation.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click the *Timestep Selector* and load the 162nd time step.
2. Select **Tools** > **Mesh Calculator** or click the **Calculators** tab and select **Mesh Calculator**.
3. Configure the following setting(s):

Tab	Setting	Value
Mesh Calculator	**Function**	Minimum Face Angle

4. Click **Calculate**.

   You can also check other time steps to calculate the mesh quality throughout the simulation.
5. When you have finished, close the **Timestep Selector** dialog box.

30.6. Comparing the Two Cases Using CFD-Post

In this section you will compare two cases. First, compare the animations:

1. With `Buoy_decoupled_001.res` already loaded in CFD-Post, select **File** > **Load Results**.
2. In the **Load Results File** dialog box, select **Keep current cases loaded**, then select the file `Buoy_001.res`. Click **Open**.
3. Click the viewport icon and select .
4. Click the synchronize active view icon .
5. Right-click within the 3D view and select **Predefined Camera** > **View from +Z** to orient the view.

   Click *Fit View* to scale the buoy appropriately within the **3D Viewer**.
6. In the Outline tree, double-click **Case Comparison**.
7. In the **Case Comparison** editor, select **Case Comparison Active**, then ensure that both cases are set to `Current Step: 0`. Click **Apply**.
8. Ensure that in each of the views:
   - `Contour 1` is visible
   - The plane and default legend are hidden

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- `Wireframe` has **Show surface mesh** selected

Note:

To show/hide plots, toggle the check box next to the plot name in the **Outline** tree view. This toggles the visibility of the plot in the currently selected view. Because the synchronization of active views has been enabled, this also modifies the visibility of all other views to match the currently selected view.

9. Click the *Timestep Selector* .

10. Click *Animation* in the *Timestep Selector* dialog box.

    The **Animation** dialog box appears.

11. Set **Type** to **Keyframe Animation**.

12. Delete the two existing Keyframes using the delete icon , because they display the results from only one results file. Then set up the Keyframe Animation in the same way as for the animations you created previously in this tutorial.

13. Beside **Save Movie**, set the movie filename to `Buoy_comparison.mpg`.

14. Click *Stop* , then click *To Beginning* .

    This ensures that the animation will begin at the first keyframe.

15. After the first keyframe has been loaded, click *Play the animation* .

    - The MPEG will be created as the animation proceeds.

16. Close the **Animation** dialog box when the animation is complete.

Now compare the mesh deformations:

1. Click the *Timestep Selector* and load the 162nd time step.

   The top two views show the differences in the mesh deformation.

2. Select **Tools** > **Mesh Calculator** or click the **Calculators** tab and select **Mesh Calculator**.

3. Configure the following setting(s):

Tab	Setting	Value
Mesh Calculator	**Function**	Minimum Face Angle

4. Click **Calculate**. Results from both cases appear.

   You can also check other time steps to calculate the mesh quality throughout the simulation.

5. When you have finished, close the **Timestep Selector** dialog box and exit from CFD-Post.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 31: Time Transformation Method for an Inlet Disturbance Case

This tutorial includes:

31.1. Tutorial Features

31.2. Overview of the Problem to Solve

31.3. Preparing the Working Directory

31.4. Defining a Steady-state Case in CFX-Pre

31.5. Obtaining a Solution to the Steady-state Case

31.6. Defining a Transient Blade Row Case in CFX-Pre

31.7. Obtaining a Solution to the Transient Blade Row Case

31.8. Viewing the Time Transformation Results in CFD-Post

31.1. Tutorial Features

In this tutorial you will learn about:

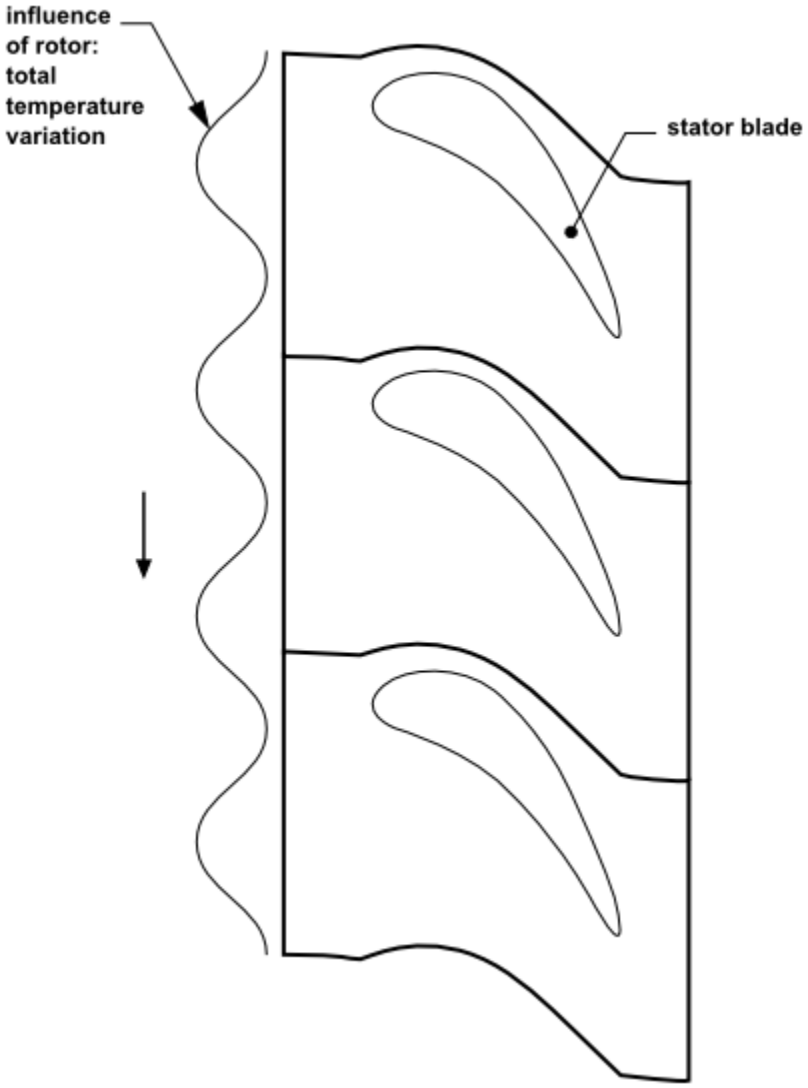
Component	Feature	Details
CFX-Pre	User Mode	Turbo Wizard
		General Mode
	Analysis Type	Transient Blade Row
	Fluid Type	Air Ideal Gas
	Domain Type	Single Domain
		Stationary Frame
	Turbulence Model	k-Epsilon
	Heat Transfer	Total Energy
	Boundary Conditions	Inlet (Subsonic)
		Outlet (Subsonic)
CFD-Post	Plots	Contour
		Animation

31.2. Overview of the Problem to Solve

The goal of this tutorial is to set up a transient blade row calculation to model an inlet disturbance (frozen gust) using the Time Transformation model. The tutorial uses an axial turbine to illustrate the basic concepts of setting up, running, and monitoring a transient blade row problem in Ansys CFX.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In this tutorial, the full geometry of the axial rotor-stator stage contains 21 stator blades and 28 rotor blades. The schematic below shows three stator blades along with the profile boundary showing a disturbance in the total temperature of the flow:



Rotational periodicity boundaries are used to enable only a small section of the full geometry to be modeled.

In your model, you should always try to obtain a pitch ratio as close to unity as possible to minimize approximations, but this must be weighted against computational resources. For this disturbance/passage geometry, 1/7 of the full wheel (4 disturbance pulses and 3 passages) would produce a pitch ratio of 1.0, but this would require a model about 3 times larger than in this tutorial example.

Using the Time Transformation method, you can work with pitch ratios near unity in order to minimize computational requirements, with little loss of accuracy. The acceptable range of pitch ratios varies, depending on the case. In this tutorial, the geometry that will be modeled consists of just a single blade passage from the stator, which is a 17.14° section (360°/21 blades). With only one stator blade, the rotor-stator pitch ratio is 4:3, which happens to fall within the acceptable range (as can be confirmed in the "Time Transformation stability limits" section of the CFX-Solver Output file for the second part of this tutorial).

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

The rotor is upstream of the stator, and creates a disturbance in the total temperature of the flow. The rotor will be modeled by applying a moving profile boundary condition at the inlet of the stator blade passage. In this case, the profile is of total temperature in a Gaussian distribution with a maximum that is 20% higher than the baseline value, and a pattern that repeats in the theta direction every 12.86° (360°/28 blades). To create a moving disturbance, the profile boundary is applied on a moving coordinate frame that rotates about the machine axis at 6300 rev/min. In this case, the machine axis is the Z axis. The rotation direction is positive using the right-hand rule as applied to the machine axis. The total temperature profile is implemented via CEL expressions that are provided in a `.ccl` file.

The outlet boundary condition is a static pressure profile, provided in a `.csv` file. It was obtained from a previous simulation of a downstream stage.

The flow is modeled as being turbulent and compressible.

The overall approach to solving this problem is:

1. Define the simulation using the Turbomachinery wizard in CFX-Pre.
2. Import the stator mesh, which was created in Ansys TurboGrid.
3. Enter the basic model definition.
4. Set the profile boundary conditions using CFX-Pre in General mode.
5. Run the steady-state simulation.
6. Modify the simulation to use the Time Transformation model.
7. Run the transient blade row simulation using the steady-state results as an initial guess.
8. Create contours of temperature and animate them in CFD-Post.

If this is the first tutorial you are running, it is important to review the following topics before beginning:

- Running Ansys CFX Tutorials Using Ansys Workbench (p. 34)
- Changing the Display Colors (p. 37)

31.3. Preparing the Working Directory

1. Create a working directory.

   Ansys CFX uses a working directory as the default location for loading and saving files for a particular session or project.

2. Download the `time_inlet_disturbance.zip` file here.
3. Unzip `time_inlet_disturbance.zip` to your working directory.

   Ensure that the following tutorial input files are in your working directory:

   - `TBRInletDistCEL.ccl`
   - `TBRInletDistOutlet.csv`

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- `TBRInletDistStator.gtm`

4. Set the working directory and start CFX-Pre.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

31.4. Defining a Steady-state Case in CFX-Pre

This tutorial uses the Turbomachinery wizard in CFX-Pre. This preprocessing mode is designed to simplify the setup of turbomachinery simulations.

1. In CFX-Pre, select **File** > **New Case**.
2. Select **TurboMachinery** and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `TimeInletDistIni`.
5. Click **Save**.
6. If you are notified that the file already exists, click **Overwrite**.

31.4.1. Basic Settings

1. In the **Basic Settings** panel, configure the following:

Setting	Value
Machine Type	Axial Turbine
Axes > **Rotation Axis**	 Z
Analysis Type > **Type**	 Steady State

   Leave the other settings at their default values.

2. Click **Next**.

31.4.2. Components Definition

As stated in the overview, this tutorial requires a single blade passage for the stator. You will define a single component and import its mesh.

1. Right-click in the blank area and select **Add Component** from the shortcut menu.
2. Create a new component of type `Stationary`, named `S1` and click **OK**.
3. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
Mesh > **File**	 TBRInletDistStator.gtm[a]

a. You may have to select the `CFX Mesh (*gtm *cfx)` option under **Files of type**.

4. Click **Next**.

31.4.3. Physics Definition

In this section, you will set properties of the fluid domain and some solver parameters.

1. In the **Physics Definition** panel, configure the following setting(s):

Setting	Value
Fluid	Air Ideal Gas
Model Data > **Reference Pressure**	 0 [atm] [a]
Model Data > **Heat Transfer**	 Total Energy
Model Data > **Turbulence**	 k-Epsilon
Inflow/Outflow Boundary Templates > **P-Total Inlet P-Static Outlet**	 (Selected)
Inflow/Outflow Boundary Templates > Inflow > **P-Total**	 200000 [Pa]
Inflow/Outflow Boundary Templates > Inflow > **T-Total**	 500 [K] [b]
Inflow/Outflow Boundary Templates > Inflow	 Cylindrical Components

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
> **Flow Direction**	
Inflow/Outflow Boundary Templates > **Inflow Direction (a,r,t)**	 1, 0, –0.4
Inflow/Outflow Boundary Templates > Outflow > **P-Static**	 175000 [Pa] [b]
a. To define the simulation using absolute pressure, set this value to 0 atm. b. These values are temporary. They will be replaced with profile data later in the tutorial.	

2. Continue to click **Next** until you reach `Final Operations`.
3. Set **Operation** to `Enter General Mode` because you will continue to define the simulation through settings not available in the Turbomachinery wizard.
4. Click **Finish**.

31.4.4. Modifying the Fluid Model Settings

You will include additional settings to improve the accuracy of the simulation.

1. Edit S1.
2. Configure the following setting(s):

Tab	Setting	Value
Fluid Models	Heat Transfer > **Incl. Viscous Work Term**	 (Selected)
	Turbulence > **High Speed (compressible) Wall Heat Transfer Model**	 (Selected)

3. Click **OK**.

31.4.5. Initializing Profile Boundary Conditions

The inlet and outlet boundary conditions are defined using profiles. Boundary profile data must be initialized before they can be used for boundary conditions.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select **File** > **Import** > **CCL**.
2. Select **Import Method** > **Append**.
3. From your working directory, select `TBRInletDistCEL.ccl`.
4. Click **Open**.
5. Select **Tools** > **Initialize Profile Data**.

   The **Initialize Profile Data** dialog box appears.
6. Beside **Profile Data File**, click *Browse* .

   The **Select Profile Data File** dialog box appears.
7. From your working directory, select `TBRInletDistOutlet.csv`.
8. Click **Open**.
9. Click **OK**.

   The profile data is read into memory.

Note:

After profile data has been initialized from a file, the profile data file should not be deleted or otherwise removed from its directory. By default, the full file path to the profile data file is stored in CFX-Pre, and the profile data file is read directly by CFX-Solver each time the solver is started or restarted.

31.4.6. Modifying Inlet and Outlet Boundary Conditions

Here, you will apply profiles to the inlet and outlet boundary conditions.

1. Edit `S1 Inlet`.
2. Configure the following setting(s):

Tab	Setting	Value
Boundary Details	Heat Transfer > **Option**	Total Temperature
	Heat Transfer > **Total Temperature**	TINLET [a]

a. Click the *Enter Expression* icon to specify the CEL expression.

3. Click **OK**.
4. Edit `S1 Outlet`.
5. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Profile Boundary Conditions > **Use Profile Data**	(Selected)
	Profile Boundary Setup > **Profile Name**	outlet

6. Click **Generate Values**.
7. Click **OK**.

31.4.7. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .
2. Configure the following setting(s):

Setting	Value
File name	TimeInletDistIni.def

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, the **Solver Input File** is set.

4. Save the simulation.

31.5. Obtaining a Solution to the Steady-state Case

At this point, CFX-Solver Manager is running.

1. Ensure that the **Define Run** dialog box is displayed.
2. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. This may take a long time, depending on your system. Eventually a dialog box is displayed.

3. Clear the check box next to **Post-Process Results** when the completion message appears at the end of the run.
4. Click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. If using stand-alone mode, quit CFX-Solver Manager.

31.6. Defining a Transient Blade Row Case in CFX-Pre

In this second part of the tutorial, you will modify the simulation from the first part of the tutorial in order to model the transient blade row.

31.6.1. Opening the Existing Case

This step involves opening the original simulation and saving it to a different location.

1. Ensure that the following tutorial input files are in your working directory:
   - `TBRInletDistOutlet.csv`
   - `TimeInletDistIni.cfx`
   - `TimeInletDistIni_001.res`
2. Set the working directory and start CFX-Pre if is it not already running.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).
3. If the original simulation is not already opened, then open `TimeInletDistIni.cfx`.
4. Save the case as `TimeInletDist.cfx` in your working directory.

31.6.2. Modifying the Analysis Type

Modify the analysis type as follows:

1. Edit `Analysis Type`.
2. Configure the following setting(s):

Setting	Value
Analysis Type > **Option**	 Transient Blade Row

3. Click **OK**.

31.6.3. Creating the Local Rotating Coordinate Frame

Create a local rotating coordinate frame that will be applied to the inlet boundary in order to cause the inlet boundary condition to rotate:

1. Select **Insert** > **Coordinate Frame**.
2. Accept the default name and click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Configure the following setting(s):

Setting	Value
Option	Axis Points
Coordinate Frame Type	Cartesian
Ref. Coord. Frame	Coord 0
Origin	0, 0, 0
Z Axis Point	0, 0, 1
X-Z Plane Pt	1, 0, 0
Frame Motion	(Selected)
Frame Motion > **Option**	Rotating
Frame Motion > **Angular Velocity**	VSignal [a]
Frame Motion > Axis Definition > **Option**	Coordinate Axis
Frame Motion > Axis Definition > **Rotation Axis**	Global Z
a. Click the *Enter Expression* icon to specify the CEL expression.	

4. Click **OK**.

31.6.4. Setting up a Transient Blade Row Model

You will set the simulation to be solved using the Time Transformation method.

1. Edit `Transient Blade Row Models`.
2. Set **Transient Blade Row Model** > **Option** to `Time Transformation`.
3. Under `Time Transformation`, click *Add new item*, accept the default name, and click **OK**.
4. Configure the following setting(s):

Setting	Value
Time Transformation	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
> Time Transformation 1 > **Option**	Rotational Flow Boundary Disturbance
Time Transformation > Time Transformation 1 > **Domain Name**	S1
Time Transformation > Time Transformation 1 > Signal Motion > **Option**	Rotating
Time Transformation > Time Transformation 1 > Signal Motion > **Coordinate Frame**	Coord 1
Time Transformation > Time Transformation 1 > External Passage Definition > **Passages in 360**	28
Time Transformation > Time Transformation 1 > External Passage Definition > **Pass. in Component**	1
Transient Method > Time Period > **Option**	Passing Period[a]
Transient Method > Time Steps > **Option**	Number of Timesteps per Period
Transient Method	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
> Time Steps > **Timesteps/Period**[b]	 60[c]
Transient Method > Time Duration > **Option**	 Number of Periods per Run
Transient Method > Time Duration > **Periods per Run**	 9

a. The passing period is automatically calculated as: 2 * pi / (Passages in 360 * Signal Angular Velocity). The **Passing Period** setting cannot be edited.

b. The number of time steps per period should always be larger than 2 * Number of Fourier Coefficients + 1 to be used for postprocessing.

c. The time step size is also automatically calculated as: Passing Period / Number of Timesteps per Period. The **Timestep** setting cannot be edited.

5. Click **OK**.

31.6.5. Applying the Local Rotating Frame to the Inlet Boundary

You can create a moving disturbance by applying a moving coordinate frame to a boundary.

Add rotational motion to the boundary condition values on the inlet by applying the local rotating coordinate frame that you made earlier:

1. Edit `S1 Inlet`.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Coordinate Frame**	(Selected)
	Coordinate Frame > **Coordinate Frame**	 Coord 1

3. Click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.6.6. Setting the Output Control and Creating Monitor Points

For transient blade row calculations, a minimal set of variables are selected to be computed using Fourier coefficients. It is convenient to postprocess total (stagnation) variables as well. Here, you will add `Total Pressure` and `Total Temperature` variables to the default list.

In addition, monitor points can be used to effectively compare the Time Transformation results against a reference case. They provide useful information on the quality of the reference phase and frequency produced in the simulation. They should also be used to monitor convergence and, as the simulation converges, the user points should display a periodic pattern.

Note:

- When comparing to a reference case, make sure monitor points are placed in the same relative locations with respect to the initial configuration in both cases.
- It is important to check that the solver equations are being solved correctly. Monitoring pressure provides feedback on the momentum equations while monitoring temperature provides feedback on the energy equations.

Set up the output control and create monitor points as follows:

1. Click *Output Control* .
2. Click the **Trn Results** tab.
3. Configure the following setting(s):

Setting	Value
Transient Blade Row Results > **Extra Output Variables List**	 (Selected)
Transient Blade Row Results > Extra Output Variables List > **Extra Output Var. List**	 Total Pressure, Total Temperature[a]
a. Click *Multi-select from extended list* ... and hold down the **Ctrl** key while selecting each of the listed variables.	

4. Click **Apply**.
5. Click the **Monitor** tab.
6. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
Monitor Objects > **Monitor Points and Expressions**	Create a monitor point named `Monitor Point 1`[a]
Monitor Objects > Monitor Points and Expressions > Monitor Point 1 > **Output Variables List**	Pressure, Temperature, Total Pressure, Total Temperature[b]
Monitor Objects > Monitor Points and Expressions > Monitor Point 1 > **Cartesian Coordinates**	(0.31878, 0.02789, 0.1)

a. To create a new item, you must first click the *Add new item* icon, then enter the name as required and click **OK**.

b. Click *Multi-select from extended list* and hold down the **Ctrl** key while selecting each of the listed variables.

7. Create additional monitor points with the same output variables. The names and Cartesian coordinates are listed below:

Name	Cartesian Coordinates
Monitor Point 2	(0.319220, 0.022322, 0.16)
Monitor Point 3	(0.312644, 0.064226, 0.162409)
Monitor Point 4	(0.316970, -0.0359315, 0.06)

8. Click **OK**.

31.6.7. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .

2. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
File name	TimeInletDist.def

3. Click **Save**.
4. Ignore the error message (the initial values will be specified in CFX-Solver Manager) and click **Yes** to continue.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, **Solver Input File** is set.
5. If using stand-alone mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

31.7. Obtaining a Solution to the Transient Blade Row Case

When CFX-Pre has shut down and the CFX-Solver Manager has started, obtain a solution to the CFD problem by following the instructions below. To reduce the simulation time, the simulation will be initialized using a steady-state case.

1. Ensure that the **Define Run** dialog box is displayed.
2. Ensure that **Solver Input File** is set to `TimeInletDist.def`.
3. Under the **Initial Values** tab, select **Initial Values Specification**.
4. Under **Initial Values Specification** > **Initial Values**, select `Initial Values 1`.
5. Under **Initial Values Specification** > **Initial Values** > **Initial Values 1 Settings** > **File Name**, click *Browse* .
6. Select `TimeInletDistIni_001.res` from your working directory.
7. Click **Open**.
8. Under **Initial Values Specification** > **Use Mesh From**, select `Solver Input File`.
9. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. This can take a long time depending on your system. Eventually a dialog box is displayed.

Note:

- Before the simulation begins, the "Transient Blade Row Post-processing Information" summary in the CFX-Solver Output file will display the time step range over which the solver will accumulate the Fourier coefficients.
- Similarly, the "Time Transformation Stability" summary in the CFX-Solver Output file displays whether the Passage/Signal pitch ratio is within the acceptable range.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- After the CFX-Solver Manager has run for a short time, you can track the monitor points you created in CFX-Pre by clicking the **Time Corrected User Points** tab that appears at the top of the graphical interface of CFX-Solver Manager.
- After the simulation has proceeded for some time, observe the periodic nature of the monitor point values.

---

10. When CFX-Solver is finished, select the check box next to **Post-Process Results**.
11. Click **OK**.

31.8. Viewing the Time Transformation Results in CFD-Post

In this section, you will work with the Fourier coefficients compressed data in transient blade row analysis. The solution variables are automatically set to the transient position corresponding to the end of the simulation.

31.8.1. Creating a Turbo Surface

1. You will see a dialog box named **Transient Blade Row Post-processing**. Click **OK**.
2. Click the **Turbo** tab.
3. A dialog box will ask if you want to auto-initialize all turbo components. Click **Yes**.
4. Select **Insert** > **Location** > **Turbo Surface**.
5. Change the name to `Span 50`.
6. Click **OK**.
7. Click **Apply**.
8. Turn off the visibility of `Span 50`.

31.8.2. Creating a Contour Plot

1. Click **Insert** > **Contour** and accept the default name.
2. Configure the following setting(s):

Tab	Setting	Value
Geometry	**Locations**	Span 50
	Variable	Temperature
	Range	User Specified
	Min	465 [K]
	Max	605 [K]

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	# of Contours	21

3. Click **Apply**.

31.8.3. Animating Temperature

Create an animation of the contour plot:

1. Click *Animation* .

   The **Animation** dialog box appears.

2. Set **Type** to **Timestep Animation**.

3. Click *To First Timestep* in order to load the first time step.

4. Click *Play the animation* .

5. When you have finished, quit CFD-Post.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 32: Fourier Transformation Method for an Inlet Disturbance Case

This tutorial includes:

32.1. Tutorial Features

32.2. Overview of the Problem to Solve

32.3. Preparing the Working Directory

32.4. Defining a Transient Blade Row Case in CFX-Pre

32.5. Obtaining a Solution to the Transient Blade Row Case

32.6. Viewing the Fourier Transformation Results in CFD-Post

Important:

This tutorial requires file `TimeInletDistIni_001.res`, which is produced by following tutorial Time Transformation Method for an Inlet Disturbance Case (p. 761).

32.1. Tutorial Features

In this tutorial you will learn about:

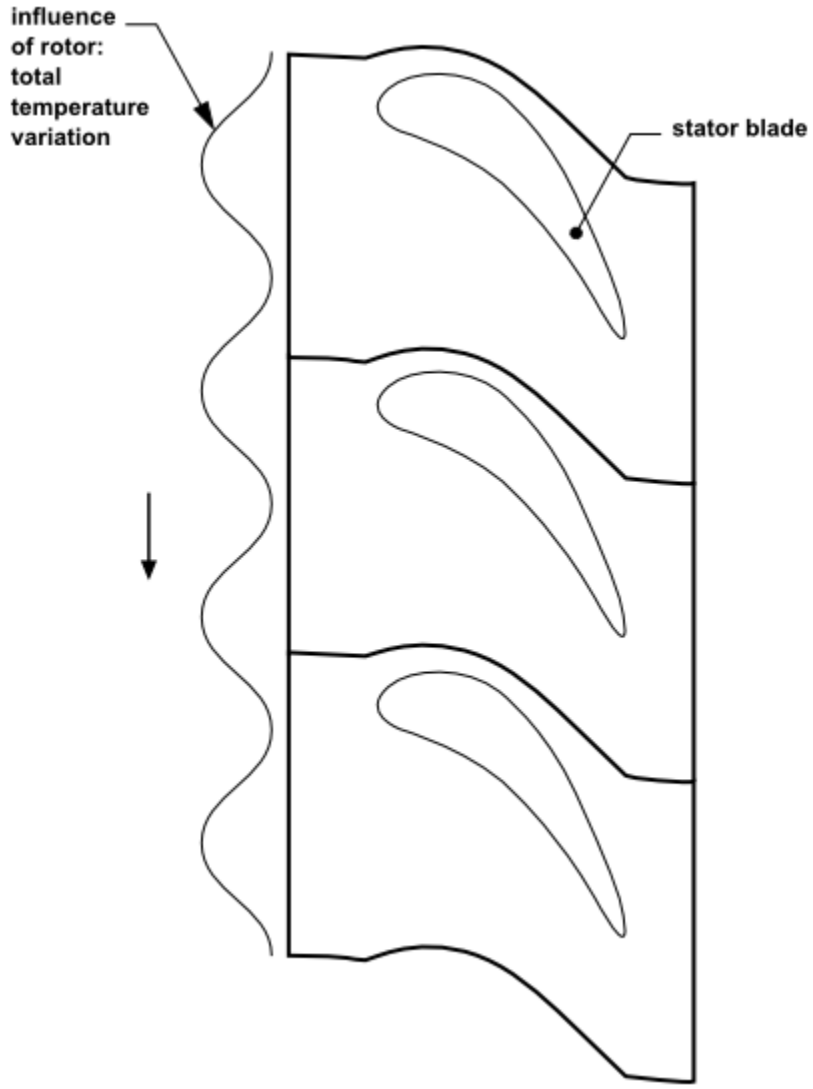
Component	Feature	Details
CFX-Pre	User Mode	Turbo Wizard
		General Mode
	Analysis Type	Transient Blade Row
	Fluid Type	Air Ideal Gas
	Domain Type	Single Domain
		Stationary Frame
	Turbulence Model	k-Epsilon
	Heat Transfer	Total Energy
	Boundary Conditions	Inlet (Subsonic)
		Outlet (Subsonic)
CFD-Post	Plots	Contour
		Animation

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.2. Overview of the Problem to Solve

The goal of this tutorial is to set up a transient blade row calculation to model an inlet disturbance (frozen gust) using the Fourier Transformation model. The tutorial uses an axial turbine to illustrate the basic concepts of setting up, running, and monitoring a transient blade row problem in Ansys CFX. The full geometry of the axial rotor-stator stage contains 21 stator blades and 28 rotor blades.

In this tutorial, rotational phase-shifted periodic boundaries are used to enable only a small section of the full geometry to be modeled. The schematic below shows three stator blades along with the profile boundary showing a disturbance in the total temperature of the flow:



The geometry to be modeled consists of the stator blade row. When using the Fourier Transformation model, two passages of the bladed geometry must be used. This is required to enable a clean signal to accumulate at the sampling interface between the two passages where the Fourier coefficients will also be accumulated. In the stator blade component, a 34.28° section is being modeled (2*360°/21 blades). The rotor is upstream of the stator and creates a disturbance in the total temperature of the flow, which is then imposed at the inlet.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

The flow is modeled as being turbulent and compressible. The inlet boundary condition serves to model the disturbance coming from the upstream rotor. It consists of a total temperature Gaussian profile with a pitch of 12.86° (360°/28 blades) and rotating about the Z axis at 6300 [rev min^-1]. The outlet boundary condition is a static pressure profile. The inlet and outlet boundary profiles are provided in `.csv` files. The outlet boundary profile was obtained from a previous simulation of a downstream stage.

When starting a new run, it is good practice to initialize Transient Blade Row simulations using results from steady-state cases. In this case, you will incorporate the steady-state results obtained from a previous tutorial. You will also use the Turbomachinery wizard feature, which facilitates the setup of a Fourier Transformation simulation. In order to do this, you have to:

1. Define the Transient Blade Row simulation using the Turbomachinery wizard in CFX-Pre.
2. Import the stator mesh, which was created in Ansys TurboGrid.
3. Enter the basic model definition.
4. Set the profile boundary conditions using CFX-Pre in General mode.
5. Run the transient blade row simulation using the steady-state results from Time Transformation Method for an Inlet Disturbance Case (p. 761) as an initial guess.

32.3. Preparing the Working Directory

1. Create a working directory.

   Ansys CFX uses a working directory as the default location for loading and saving files for a particular session or project.

2. Download the `fourier_inlet_disturbance.zip` file here.

3. Unzip `fourier_inlet_disturbance.zip` to your working directory.

   Ensure that the following tutorial input files are in your working directory:

   - `TBRInletDistInlet.csv`
   - `TBRInletDistOutlet.csv`
   - `TBRInletDistStator.gtm`
   - `TimeInletDistIni_001.res` (produced by following tutorial Time Transformation Method for an Inlet Disturbance Case (p. 761))

4. Set the working directory and start CFX-Pre.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.4. Defining a Transient Blade Row Case in CFX-Pre

The following sections describe the transient blade row simulation setup in CFX-Pre.

This tutorial uses the Turbomachinery wizard in CFX-Pre. This preprocessing mode is designed to simplify the setup of turbomachinery simulations.

1. In CFX-Pre, select **File** > **New Case**.
2. Select **TurboMachinery** and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `FourierInletDist`.
5. Click **Save**.

32.4.1. Basic Settings

1. In the **Basic Settings** panel, configure the following:

Setting	Value
Machine Type	Axial Turbine
Axes > **Rotation Axis**	Z
Analysis Type > **Type**	Transient Blade Row
Analysis Type > **Method**	Fourier Transformation

2. Click **Next**.

32.4.2. Components Definition

The Fourier Transformation method requires two stator blade passages. You will define a new component and import the stator mesh.

1. Right-click in the blank area and select **Add Component** from the shortcut menu.
2. Create a new component of type `Stationary`, named `S1` and click **OK**.
3. Configure the following setting(s):

Setting	Value
Mesh	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
> **File**	TBRInletDistStator.gtm[a]
a. You may have to select the `CFX Mesh (*gtm *cfx)` option under **Files of type**.	

4. Expand the **Passages and Alignment** section.
5. Click **Edit**.
6. Configure the following setting(s):

Setting	Value
Passages and Alignment > **Passages to Model**	 2

7. Click **Done**

   You will see that the stator blade passage is correctly replicated and the resulting mesh now contains two stator blade passages. This will also create the Sampling Interface (**S1 Internal Interface 1**) required for the Fourier Transformation model.

8. Click **Next**.

32.4.3. Physics Definition

In this section, you will set properties of the fluid domain and some solver parameters.

1. In the **Physics Definition** panel, configure the following setting(s):

Setting	Value
Fluid	Air Ideal Gas
Model Data > **Reference Pressure**	 0 [atm][a]
Model Data > **Heat Transfer**	 Total Energy
Model Data > **Turbulence**	 k-Epsilon
Inflow/Outflow Boundary Templates > **P-Total Inlet P-Static Outlet**	 (Selected)

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
Inflow/Outflow Boundary Templates > Inflow > **P-Total**	200000 [Pa]
Inflow/Outflow Boundary Templates > Inflow > **T-Total**	500 [K] [b]
Inflow/Outflow Boundary Templates > Inflow > **Flow Direction**	Cylindrical Components
Inflow/Outflow Boundary Templates > **Inflow Direction (a,r,t)**	1, 0, -0.4
Inflow/Outflow Boundary Templates > Outflow > **P-Static**	175000 [Pa] [b]
a. To define the simulation using absolute pressure, set this value to 0 atm. b. These values are temporary. They will be replaced with profile data later in the tutorial.	

2. Click **Next**.

   Under the **Interface Definition** section you can observe that both the Fourier coefficient sampling interface **S1 Internal Interface 1** as well as the phase shifted interface **S1 to S1 Periodic 1** are automatically created.

3. Click **Next**.

32.4.4. Disturbance Definition

In this section, you will specify the periodicity of the disturbance being imposed. In this case the inlet profile has a pitch of 12.857 [Degrees] or 1/28 of the wheel, so you need to specify 28 for the value of **Passages in 360**.

1. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
Disturbances > External Boundary > **Passages in 360**	 28

2. Continue clicking **Next** until the **Final Operations** panel is reached.
3. Ensure that **Operation** is set to `Enter General Mode` because you will continue to define the simulation through settings not available in the TurboMachinery wizard.
4. Click **Finish**.

Note:

You may ignore the physics validation errors for the moment. You will correct these errors in the steps that follow.

32.4.5. Modifying the Fluid Model Settings

You will include additional settings to improve the accuracy of the simulation.

1. Edit S1.
2. Configure the following setting(s):

Tab	Setting	Value
Fluid Models	Heat Transfer > **Incl. Viscous Work Term**	 (Selected)
	Turbulence > **High Speed (compressible) Wall Heat Transfer Model**	 (Selected)

3. Click **OK**.

32.4.6. Initializing Profile Boundary Conditions

The inlet and outlet boundary conditions are defined using profiles in your working directory. Boundary profile data must be initialized before they can be used for boundary conditions.

1. Select **Tools** > **Initialize Profile Data**.

   The **Initialize Profile Data** dialog box appears.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Beside **Profile Data File**, click *Browse* .

   The **Select Profile Data File** dialog box appears.

3. From your working directory, select `TBRInletDistOutlet.csv`.

4. Click **Open**.

5. Click **OK**.

   The outlet profile data is read into memory.

   Next, you will prepare the inlet profile. Since the supplied profile file, `TBRInletDistInlet.csv`, only covers a single passage, you need to expand the profile so that it covers at least both passages. In this case you will expand the profile so that it covers the full wheel.

6. Select **Tools** > **Edit Profile Data**.

   The **Edit Profile Data** dialog box appears.

7. Under **Source Profile**, click *Browse* .

   The **Select Profile Data File** dialog box appears.

8. From your working directory, select `TBRInletDistInlet.csv` and click **Open**.

9. Set **Write to Profile** to `TBRInletDistInlet_FullWheel.csv`.

10. Ensure that **Initialize New Profile After Writing** is selected so that the inlet profile data will be automatically initialized using the expanded profile.

11. In the **Transformations** frame, click *Add new item* , set **Name** to `Transformation 1`, and click **OK**.

12. Configure the following setting(s):

Setting	Value
Transformation 1 > **Option**	Expansion
Transformation 1 > Expansion Definition > **Rotation Option**	Principal Axis
Transformation 1 > Expansion Definition > **Axis**	Z
Transformation 1	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
> Expansion Definition > **Passages in Profile**	1
Transformation 1 > Expansion Definition > **Passages in 360**	28
Transformation 1 > Expansion Definition > **Expansion Option**	Expand to Full Circle
Transformation 1 > Expansion Definition > **Theta Offset**	0 [degree]

13. Click **OK**.

Note:

After profile data has been initialized from a file, the profile data file should not be deleted or otherwise removed from its directory. By default, the full file path to the profile data file is stored in CFX-Pre, and the profile data file is read directly by CFX-Solver each time the solver is started or restarted.

32.4.7. Creating the Local Rotating Coordinate Frame

Create a local rotating coordinate frame that will be applied to the inlet boundary in order to cause it to rotate:

1. Select **Insert** > **Coordinate Frame**.
2. Accept the default name and click **OK**.
3. Configure the following setting(s):

Setting	Value
Option	Axis Points
Coord Frame Type	Cartesian
Ref. Coordinate Frame	Coord 0
Origin	0, 0, 0
Z Axis Point	0, 0, 1
X-Z Plane Pt	1, 0, 0

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
Frame Motion	(Selected)
Frame Motion > **Option**	Rotating
Frame Motion > **Angular Velocity**	6300 [rev min^-1]
Frame Motion > Axis Definition > **Option**	Coordinate Axis
Frame Motion > Axis Definition > **Rotation Axis**	Global Z

4. Click **OK**.

32.4.8. Modifying Inlet and Outlet Boundary Conditions

Here, you will apply profiles to the inlet and outlet boundary conditions. In addition to this, you will also be applying the local rotating frame to the inlet boundary.

1. Edit `S1 Inlet`.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Profile Boundary Conditions > **Use Profile Data**	(Selected)
	Profile Boundary Setup > **Profile Name**	inletTo

3. Click **Generate Values**.

You can create a moving disturbance by applying a moving coordinate frame to a boundary. Add rotational motion to the boundary condition values on the inlet by applying the local rotating coordinate frame that you made earlier:

1. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Coordinate Frame**	(Selected)

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Coordinate Frame > **Coordinate Frame**	 Coord 1

2. Click **OK**.
3. Edit `S1 Outlet`.
4. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Profile Boundary Conditions > **Use Profile Data**	 (Selected)
	Profile Boundary Setup > **Profile Name**	 outlet

5. Click **Generate Values**.
6. Click **OK**.

32.4.9. Setting up a Transient Blade Row Model

In this section, you will make some modifications to the `Transient Blade Row Models` object:

1. Edit `Transient Blade Row Models`.
2. Configure the following setting(s):

Setting	Value
Fourier Transformation > Fourier Transformation 1 > Signal Motion > **Option**	 Rotating
Fourier Transformation > Fourier Transformation 1 > Signal Motion > **Coordinate Frame**	 Coord 1
Transient Method > Time Period	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
> **Option**[a]	Automatic
Transient Method > Time Steps > **Timestep Multiplier**[b][c]	20
Transient Method > Time Duration > **Option**	Number of Periods per Run
Transient Method > Time Duration > **Periods per Run**	10

a. The passing period is automatically calculated as: 2 * pi / (Passages in 360 * Signal Angular Velocity). The **Passing Period** setting cannot be edited.

b. The number of time steps per period should always be larger than 2 * Number of Fourier Coefficients + 1 to be used for postprocessing.

c. The time step size is also automatically calculated as: Passing Period / Number of Timesteps per Period. The **Timestep** setting cannot be edited.

3. Click **OK**.

32.4.10. Setting the Output Control and Creating Monitor Points

For transient blade row calculations, a minimal set of variables are selected to be computed using Fourier coefficients. It is convenient to postprocess total (stagnation) variables as well. Here, you will add `Total Pressure` and `Total Temperature` variables to the default list.

Monitor points can be used to effectively compare the Fourier Transformation results against a reference case. They provide useful information on the quality of the reference phase and frequency produced in the simulation. They should also be used to monitor convergence and, as the simulation converges, the user points should display a periodic pattern.

Note:

When comparing to a reference case, make sure monitor points are placed in the same relative locations with respect to the initial configuration in both cases.

It is important to check that the solver equations are being solved correctly. Monitoring pressure provides feedback on the momentum equations while monitoring temperature provides feedback on the energy equations.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Set up the output control and create monitor points as follows:

1. Click *Output Control* .
2. Click the **Trn Results** tab.
3. Configure the following setting(s):

Setting	Value
Transient Blade Row Results > **Extra Output Variables List**	 (Selected)
Transient Blade Row Results > Extra Output Variables List > **Extra Output Var. List**	 Total Pressure, Total Temperature[a]
a. Click *Multi-select from extended list* and hold down the **Ctrl** key while selecting each of the listed variables.	

4. Click **Apply**.
5. Click the **Monitor** tab.
6. Configure the following setting(s):

Setting	Value
Monitor Objects > **Monitor Points and Expressions**	 Create a monitor point named `Monitor Point 1`[a]
Monitor Objects > Monitor Points and Expressions > Monitor Point 1 > **Option**	 Cylindrical Coordinates
Monitor Objects > Monitor Points and Expressions > Monitor Point 1 > **Output Variables List**	 Pressure, Temperature, Total Pressure, Total Temperature[b]

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
Monitor Objects > Monitor Points and Expressions > Monitor Point 1 > **Position Axial Comp.**	0.1 [m]
Monitor Objects > Monitor Points and Expressions > Monitor Point 1 > **Position Radial Comp.**	0.32 [m]
Monitor Objects > Monitor Points and Expressions > Monitor Point 1 > **Position Theta Comp.**	5 [degree]
a. To create a new item, you must first click the *Add new item* icon, then enter the name as required and click **OK**. b. Click *Multi-select from extended list* and hold down the **Ctrl** key while selecting each of the listed variables.	

7. Create additional monitor points with the same output variables. The names and Cylindrical co-ordinates are listed below:

Name	Cylindrical Coordinates
Monitor Point 2	0.16 [m], 0.32 [m], 4 [degree]
Monitor Point 3	0.16 [m], 0.32 [m], 11.6 [degree]
Monitor Point 4	0.06 [m], 0.32 [m], -6.5 [degree]

8. Click **OK**.

32.4.11. Setting the Execution Control

Here you will prepare the case for execution and initialize the solution with steady-state results. Instead of obtaining steady-state results by setting up and running the steady-state solution for this case, which has a double passage configuration, you will import a results file from another steady-state

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

simulation, which happens to have a single passage (see Defining a Steady-state Case in CFX-Pre (p. 764)). Because that other simulation involves only a single passage, you will use replication control settings to apply those results to both passages in this simulation.

1. In the **Outline** tree view, right-click `Simulation Control` and select **Insert** > **Execution Control**.

2. Configure the following setting(s):

Tab	Setting	Value
Run Definition	Run Settings > **Double Precision**	(Selected)
Initial Values	**Initial Values Specification**	(Selected)
	Initial Values Specification > **Initial Values**	Initial Values 1
	Initial Values Specification > Initial Values > Initial Values 1 > **File Name**	TimeInletDistIni_001.res [a]
	Initial Values Specification > Initial Values > Initial Values 1 > **Interpolation Mapping**	Create an interpolation mapping object named `Interpolation Mapping 1` [b]
	Initial Values Specification > Initial Values > Initial Values 1 > Interpolation Mapping > Interpolation Mapping 1 > **Replication Control**	(Selected)
	Initial Values Specification	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Initial Values > Initial Values 1 > Interpolation Mapping > Interpolation Mapping 1 > Replication Control > **Passages in 360**	21
	Initial Values Specification > Initial Values > Initial Values 1 > Interpolation Mapping > Interpolation Mapping 1 > Replication Control > **Total Num. Instances**	2
	Initial Values Specification > **Initial Values Control**	(Selected)
	Initial Values Specification > Initial Values Control > **Continue History From**	(Selected)
a. You can select a file to load by clicking *Browse* and selecting the appropriate file from your working directory, then clicking **Open**. b. Add an item by clicking the *Add New Item* icon and clicking **OK** to accept the default name.		

3. Click **OK**.

32.4.12. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .

   The CFX-Solver input file `FourierInletDist.def` is created.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, **Solver Input File** is set.

2. Ignore the message and click **Yes** to continue.

3. If using stand-alone mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

32.5. Obtaining a Solution to the Transient Blade Row Case

Ensure that the **Define Run** dialog box is displayed in CFX-Solver Manager.

1. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. This can take a long time depending on your system. Eventually a dialog box is displayed.

   Note:

   - Before the simulation begins, the "Transient Blade Row Post-processing Information" summary in the CFX-Solver Output file displays the time step range over which the solver will accumulate the Fourier coefficients.
   - The CFX-Solver Output file contains a "Fourier Transformation Information" summary as well as the time step at which the full Fourier Transformation Model is activated.
   - Monitor points of similar values can be grouped together by right-clicking to the right of the **User Points** tab, selecting **New Monitor**, and clicking **OK**. In the **New Monitor** dialog box, you can set the name for the new monitor point and select the variables you want to monitor in the **Monitor Properties** dialog box.
   - After the simulation has proceeded for some time, observe the periodic nature of the monitor point values.
   - If the monitor points do not establish a periodic nature in a Fourier Transformation run, you can try applying frequency filtering.

     Frequency filtering is a powerful tool to deal with instabilities. It filters out all frequencies that are not harmonics of the blade passing frequency (or blade vibration frequency for flutter cases) and that could trigger instabilities. These typically occur in elongated domains where the amplitude of the periodic signal becomes very weak at the furthest point away from the source of the disturbance.

     To apply frequency filtering in CFX-Pre, edit the `Transient Blade Row Models` object; in the details view, on the **Advanced Options** tab, select **Fourier Transformation Control** > **Frequency Filtering**.

   ---

2. When CFX-Solver is finished, select the check box next to **Post-Process Results**.

3. Click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.6. Viewing the Fourier Transformation Results in CFD-Post

In this section, you will work with the Fourier coefficients compressed data in transient blade row analysis. The solution variables are automatically set to the transient position corresponding to the end of the simulation.

32.6.1. Creating a Turbo Surface

1. You will see a dialog box named **Transient Blade Row Post-processing**. Click **OK**.
2. Click the **Turbo** tab.
3. Click **Initialize All Components**.
4. A dialog box will ask if you want to auto-initialize all turbo components. Click **Yes**.
5. Select **Insert** > **Location** > **Turbo Surface**.
6. Change the name to `Span 50`.
7. Click **OK**.
8. Click **Apply**.
9. Turn off the visibility of `Span 50`.

32.6.2. Creating a Contour Plot

1. Click **Insert** > **Contour** and accept the default name.
2. Configure the following setting(s):

Tab	Setting	Value
Geometry	**Locations**	Span 50
	Variable	Temperature
	Range	User Specified
	Min	465 [K]
	Max	605 [K]
	# of Contours	21

3. Click **Apply**.

32.6.3. Animating Temperature

Create an animation of the contour plot:

1. Click *Animation* .

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Animation** dialog box appears.

2. Set **Type** to **Timestep Animation**.
3. Click *To First Timestep* in order to load the first time step.
4. Click *Play the animation* .
5. When you have finished, quit CFD-Post.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 33: Time Transformation Method for a Transient Rotor-stator Case

This tutorial includes:

33.1. Tutorial Features

33.2. Overview of the Problem to Solve

33.3. Preparing the Working Directory

33.4. Defining a Steady-state Case in CFX-Pre

33.5. Obtaining a Solution to the Steady-state Case

33.6. Defining a Transient Blade Row Case in CFX-Pre

33.7. Obtaining a Solution to the Transient Blade Row Case

33.8. Viewing the Time Transformation Results in CFD-Post

33.1. Tutorial Features

In this tutorial you will learn about:

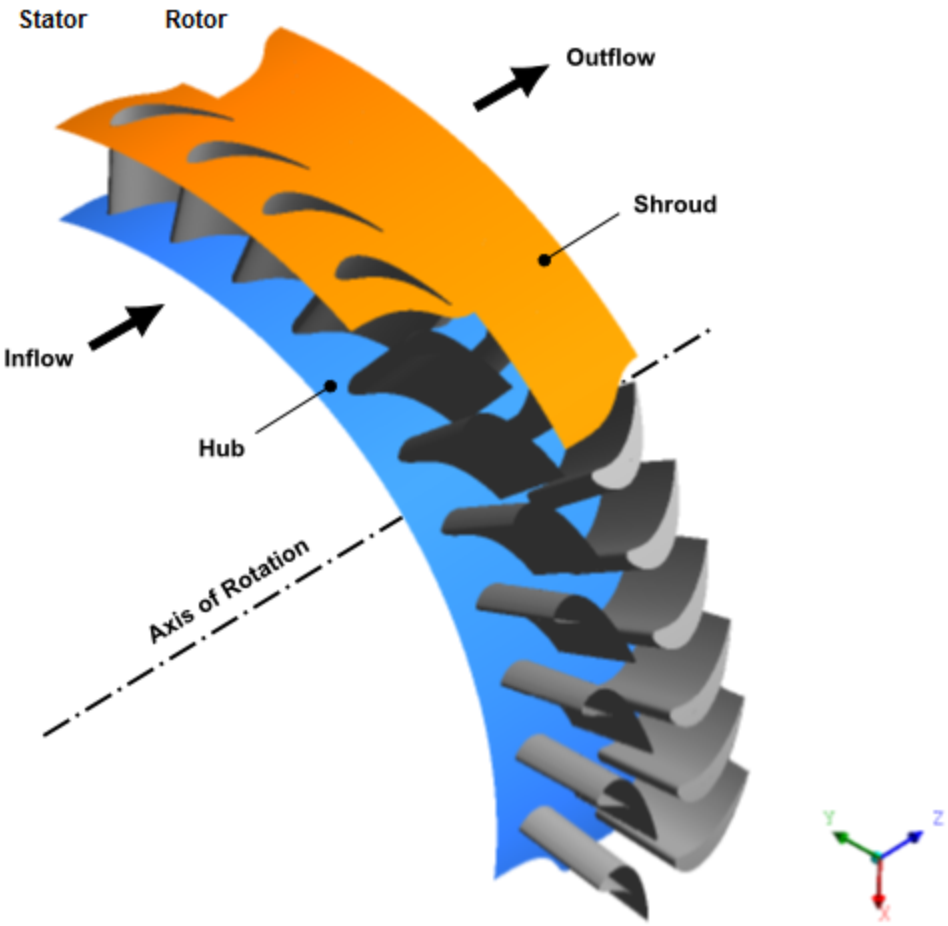
Component	Feature	Details
CFX-Pre	User Mode	Turbo Wizard
		General mode
	Analysis Type	Transient Blade Row
	Fluid Type	Air Ideal Gas
	Domain Type	Multiple Domains
		Rotating Frame of Reference
	Turbulence Model	Shear Stress Transport
	Heat Transfer	Total Energy
	Boundary Conditions	Inlet (Subsonic)
		Outlet (Subsonic)
		Wall (Counter Rotating)
CFD-Post	Plots	Vector
		Contour
		Data Instancing
		Time Chart
		Animation

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

33.2. Overview of the Problem to Solve

This tutorial sets up a transient blade row calculation using the Time Transformation model. It uses an axial turbine to illustrate the basic concepts of setting up, running, and monitoring a transient blade row problem in Ansys CFX. It also describes the postprocessing of transient blade row results using the tools provided in CFD-Post for this type of calculation.

The full geometry of the axial rotor-stator stage selected for modeling contains 36 stator blades and 42 rotor blades.



The geometry to be modeled consists of a single rotor blade passage and a single stator blade passage. Each rotor blade passage is an 8.571° section (360°/42 blades), while each stator blade passage is a 10° section (360°/36 blades). The pitch ratio at the interface between the rotor passage and the stator passage is 0.8571 (that is, 6/7).

For the Time Transformation method, you should always maintain an ensemble pitch ratio within a range of 0.75 to 1.4. Note that the range of permissible pitch ratios narrows significantly with slower rotation speed. A full machine analysis can be performed (modeling all rotor and stator blades), which always eliminates any pitch change, but will require significant computational time. For this rotor-stator geometry, a 1/6 machine section (7 rotor blades, 6 stator blades) would produce a pitch ratio of 1.0, but this would require a model about 7 times larger than in this tutorial example.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In this example, the rotor rotates about the Z axis at 3500 rev/min (positive rotation following the right hand rule) while the stator is stationary. Rotational periodicity boundaries are used to enable only a small section of the full geometry to be modeled.

The flow is modeled as being turbulent and compressible. Profile boundary conditions are used at the inlet and outlet. In this tutorial, the profiles are a function of the radial coordinate only. These profiles were obtained from previous simulations of the upstream and downstream stages.

The overall approach of this tutorial is to run a transient blade row simulation initialized with the results of a steady-state simulation. First, you will define a steady-state simulation using the Turbomachinery Wizard followed by General mode. While the steady-state simulation is running, you will modify a copy of it to define a transient blade row simulation that uses the Time Transformation model. After running the transient blade row simulation, you will create contour plots and an animation showing blade rotation.

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- Running Ansys CFX Tutorials Using Ansys Workbench (p. 34)
- Changing the Display Colors (p. 37)

33.3. Preparing the Working Directory

1. Create a working directory.

   Ansys CFX uses a working directory as the default location for loading and saving files for a particular session or project.

2. Download the `time_blade_row.zip` file here.

3. Unzip `time_blade_row.zip` to your working directory.

   Ensure that the following tutorial input files are in your working directory:

   - `TBRTurbineRotor.gtm`
   - `TBRTurbineStator.gtm`
   - `TBRInletProfile.csv`
   - `TBROutletProfile.csv`

4. Set the working directory and start CFX-Pre.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

33.4. Defining a Steady-state Case in CFX-Pre

This tutorial uses the Turbomachinery wizard in CFX-Pre. This preprocessing mode is designed to simplify the setup of turbomachinery simulations.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In CFX-Pre, select **File** > **New Case**.
2. Select **Turbomachinery** and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `TimeBladeRowIni`.
5. Click **Save**.
6. If you are notified that the file already exists, click **Overwrite**.

33.4.1. Basic Settings

1. In the **Basic Settings** panel, configure the following settings:

Setting	Value
Machine Type	Axial Turbine
Axes > **Rotation Axis**	Z
Analysis Type > **Type**	Steady State

2. Leave the other settings at their default values.
3. Click **Next**.

33.4.2. Components Definition

You will define two new components and import their respective meshes.

1. Right-click in the blank area and select **Add Component** from the shortcut menu.
2. Create a new component of type `Stationary` named `S1` and click **OK**.
3. Configure the following setting(s):

Setting	Value
Mesh > **File**	TBRTurbineStator.gtm [a]
a. You may have to select the `CFX Mesh (*gtm *cfx)` option under **Files of type**.	

4. Create a new component of type `Rotating`, named `R1` and click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Configure the following setting(s):

Setting	Value
Component Type > **Value**	3500 [rev min^-1] [a]
Mesh > **File**	TBRTurbineRotor.gtm
a. From the problem description.	

6. Click **Next**.

33.4.3. Physics Definition

In this section you will set properties of the fluid domain and some solver parameters.

1. In the **Physics Definition** panel, configure the following:

Setting	Value
Fluid	Air Ideal Gas
Model Data > **Reference Pressure**	0 [atm] [a]
Model Data > **Heat Transfer**	Total Energy
Model Data > **Turbulence**	Shear Stress Transport
Inflow/Outflow Boundary Templates > **P-Total Inlet P-Static Outlet**	(Selected)
Inflow/Outflow Boundary Templates > Inflow > **P-Total**	169000 [Pa][b]
Inflow/Outflow Boundary Templates > Inflow > **T-Total**	306 [K][b]

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
Inflow/Outflow Boundary Templates > Inflow > **Flow Direction**	 Normal to Boundary
Inflow/Outflow Boundary Templates > Outflow > **P-Static**	 110000 [Pa][b]
Interface > **Default Type**	 Stage (Mixing-Plane)

a. To define the simulation using absolute pressure, set this value to 0 atm.

b. These values are temporary. They will be replaced with profile data later in the tutorial.

2. Continue to click **Next** until you reach `Final Operations`.
3. Set **Operation** to `Enter General Mode` because you will continue to define the simulation through settings not available in the Turbomachinery wizard.
4. Click **Finish**.

33.4.4. Additional Fluid Model Settings

Verify the following settings, which affect the accuracy of the simulation:

1. Edit R1.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Domain Models > Domain Motion > **Alternate Rotation Model**	 (Selected)
Fluid Models	Heat Transfer > **Incl. Viscous Work Term**	 (Selected)

3. Click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

33.4.5. Initializing Profile Boundary Conditions

The inlet and outlet boundary conditions are defined using profiles in your working directory. Boundary profile data must be initialized before they can be used for boundary conditions.

1. Select **Tools** > **Initialize Profile Data**.

   The **Initialize Profile Data** dialog box appears.

2. Beside **Profile Data File**, click *Browse* .

   The **Select Profile Data File** dialog box appears.

3. From your working directory, select `TBRInletProfile.csv`.

4. Click **Open**.

5. Click **Apply**.

   The profile data is read into memory.

6. Beside **Profile Data File**, click *Browse* .

   The **Select Profile Data File** dialog box appears.

7. From your working directory, select `TBROutletProfile.csv`.

8. Click **Open**.

9. Click **OK**.

Note:

After profile data has been initialized from a file, the profile data file should not be deleted or otherwise removed from its directory. By default, the full filepath to the profile data file is stored in CFX-Pre, and the profile data file is read directly by CFX-Solver each time the solver is started or restarted.

33.4.6. Modifying Inlet and Outlet Boundary Conditions

1. Edit `S1 Inlet`.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Profile Boundary Conditions > **Use Profile Data**	 (Selected)
	Profile Boundary Setup	

Tab	Setting	Value
	> **Profile Name**	inlet

3. Click **Generate Values**.

   This causes the profile values of `Total Pressure` and `Total Temperature` to be applied at the nodes on the inlet boundary. It also causes entries to be made in the **Boundary Details** tab. In order to later reset the velocity values at the main inlet to match those that were originally read from the profile data file, revisit the **Basic Settings** tab for this boundary and click **Generate Values**.

4. Configure the following setting(s):

Tab	Setting	Value
Boundary Details	Mass and Momentum > **Option**	Total Pressure (stable)
	Mass and Momentum > **Relative Pressure**	inlet.Total Pressure(r)
	Flow Direction > **Option**	Cylindrical Components
	Flow Direction > **Axial Component**	1
	Flow Direction > **Radial Component**	0
	Flow Direction > **Theta Component**	0
	Heat Transfer > **Total Temperature**	inlet.Total Temperature(r)

5. Click **OK**.

6. Edit `R1 Outlet`.

7. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Frame Type**	Rotating

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Profile Boundary Conditions > **Use Profile Data**	 (Selected)
	Profile Boundary Setup > **Profile Name**	 outlet

8. Click **Generate Values**.

9. Configure the following setting(s):

Tab	Setting	Value
Boundary Details	Mass And Momentum > **Option**	 Static Pressure
	Mass and Momentum > **Relative Pressure**	 outlet.Pressure(r)

10. Click **OK**.

33.4.7. Visualizing the Profile Boundary Value

You can plot scalar profile values and vectors on inlet and outlet boundaries. In this section, you will edit a boundary so that you can visualize the pressure profile values at the inlet.

1. Edit `S1 Inlet`

2. Configure the following setting(s):

Tab	Setting	Value
Plot Options	**Boundary Contour**	(Selected)
	Profile Variable	Relative Pressure

3. Click **Apply**

CFX-Pre plots the total pressure radial profile at the inlet with the pressure values displayed in a legend.

33.4.8. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .

2. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
File name	TimeBladeRowIni.def

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, **Solver Input File** is set.

4. Save the simulation.

33.5. Obtaining a Solution to the Steady-state Case

Ensure that the **Define Run** dialog box is displayed in CFX-Solver Manager.

1. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. At the end of the run, a dialog box is displayed stating that the simulation has ended.

2. Ensure that the **Post-Process Results** check box is cleared.

3. Click **OK**.

33.6. Defining a Transient Blade Row Case in CFX-Pre

In the second part of the tutorial, you will modify the simulation from the first part of the tutorial to model the transient blade row.

33.6.1. Opening the Existing Case

This step involves opening the original simulation and saving it to a different location.

1. Ensure that the following tutorial input files are in your working directory:

   - `TBRInletProfile.csv`
   - `TBROutletProfile.csv`
   - `TimeBladeRowIni.cfx`
   - `TimeBladeRowIni_001.res`

2. Set the working directory and start CFX-Pre if it is not already running.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

3. If the original simulation is not already opened, then open `TimeBladeRowIni.cfx`.

4. Save the case as `TimeBladeRow.cfx` in your working directory.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

33.6.2. Modifying the Analysis Type

In this section, you will make use of the transient blade row feature.

Modify the analysis type as follows:

1. Edit `Analysis Type`.
2. Configure the following setting(s):

Setting	Value
Analysis Type > **Option**	Transient Blade Row
Analysis Type > Initial Time > **Option**	Automatic with Value
Analysis Type > Initial Time > **Time**	0 [s]

3. Click **OK**.

33.6.3. Modifying the Rotor–stator Interface

1. Edit `R1 to S1`.
2. Configure the following setting(s):

Setting	Value
Interface Models > Frame Change/Mixing Model > **Option**	Transient Rotor Stator

3. Click **OK**.

33.6.4. Setting up a Transient Blade Row Model

You will set the simulation to be solved using the Time Transformation method.

1. Edit `Transient Blade Row Models`.
2. Set **Transient Blade Row Model** > **Option** to `Time Transformation`.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Under `Time Transformation`, click *Add new item* , accept the default name, and click **OK**.

4. Configure the following setting(s):

Setting	Value
Transient Method > Time Period > **Option**	Passing Period
Transient Method > Time Period > **Domain**	S1
Transient Method > Time Steps > **Option**	Number of Timesteps per Period
Transient Method > Time Steps > **Timesteps/Period**	35
Transient Method > Time Duration > **Option**	Number of Periods per Run
Transient Method > Time Duration > **Periods per Run**	10

Note:

- The passing period is automatically calculated as: 2 * pi / (Passages in 360 * Signal Angular Velocity). The **Passing Period** setting cannot be edited.
- The number of time steps per period should always be larger than 2 * Number of Fourier Coefficients + 1 to be used for postprocessing.
- The time step size is also automatically calculated as: Passing Period / Number of Timesteps per Period. The **Timestep** setting cannot be edited.

5. Click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

33.6.5. Setting Output Control and Creating Monitor Points

For transient blade row calculations, a minimal set of variables are selected to be computed using Fourier coefficients. It is convenient to postprocess variables in the stationary frame when multiple frames of reference are present. Here, you will add the `Velocity in Stn Frame` and `Mach Number in Stn Frame` variables to the default list.

In addition, monitor points can be used to effectively compare the Time Transformation results against a reference case. They provide useful information on the quality of the reference phase and frequency produced in the simulation. They should be used to monitor convergence and, as the simulation converges, the user points should display a periodic pattern.

Note:

- When comparing to a reference case, make sure monitor points are placed in the same relative locations with respect to the initial configuration in both cases.
- It is important to check that the solver equations are being solved correctly. Monitoring pressure provides feedback on the momentum equations while monitoring temperature provides feedback on the energy equations.

Set up the output control and create monitor points as follows:

1. Click *Output Control* .
2. Click the **Trn Results** tab.
3. Configure the following setting(s):

Setting	Value
Transient Blade Row Results > **Extra Output Variables List**	(Selected)
Transient Blade Row Results > Extra Output Variables List > **Extra Output Var. List**	Velocity in Stn Frame, Mach Number in Stn Frame[a]

a. Click *Multi-select from extended list* and hold down the **Ctrl** key while selecting each of the listed variables.

4. Click the **Monitor** tab.
5. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
Monitor Objects	(Selected)
Monitor Objects > **Efficiency Output**	 (Cleared)

6. Create a monitor point named `rotor_P1`.
7. Under **Monitor Objects** > **Monitor Points and Expressions** > **rotor_P1**, configure the following settings:

Setting	Value
Option	Cylindrical Coordinates
Output Variables List	Pressure, Temperature, Total Pressure, Total Temperature, Velocity
Position Axial Comp.	0.211 [m]
Position Radial Comp.	0.2755 [m]
Position Theta Comp.	182 [degree]

8. Create an additional monitor point named `stator_P1`.
9. Under **Monitor Objects** > **Monitor Points and Expressions** > **stator_P1**, configure the following settings:

Setting	Value
Option	Cylindrical Coordinates
Output Variables List	Pressure, Temperature, Total Pressure, Total Temperature, Velocity
Position Axial Comp.	0.202 [m]
Position Radial Comp.	0.2755 [m]
Position Theta Comp.	178 [degree]

Note:

Transient blade row cases use monitor points to monitor the periodic fluctuating variable values. For diagnostic purposes, you should have several monitor points. Here, two monitor points will be used for demonstration purposes.

10. Click **OK**.

33.6.6. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Configure the following setting(s):

Setting	Value
File name	TimeBladeRow.def

3. Click **Save**.

4. Ignore the error message (the initial values will be specified in CFX-Solver Manager) and click **Yes** to continue.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, **Solver Input File** is set.

5. If using stand-alone mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

33.7. Obtaining a Solution to the Transient Blade Row Case

At this point, CFX-Pre has been shut down, and the **Define Run** dialog box is displayed in CFX-Solver Manager. You will now obtain a solution to the CFD problem. To reduce the simulation time, the simulation will be initialized using a steady-state case.

1. Ensure that the **Define Run** dialog box is displayed. If an error message appears, ignore it and click **Yes** to continue.

   Solver Input File should be set to `TimeBladeRow.def`.

2. Under the **Initial Values** tab, select **Initial Values Specification**.

3. Under **Initial Values Specification** > **Initial Values**, select `Initial Values 1`.

4. Under **Initial Values Specification** > **Initial Values** > **Initial Values 1 Settings** > **File Name**, click *Browse* .

5. Select `TimeBladeRowIni_001.res` from your working directory.

6. Click **Open**.

7. Under **Initial Values Specification** > **Use Mesh From**, select `Solver Input File`.

8. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. At the end of the run, a dialog box is displayed stating that the simulation has ended.

Note:

- Before the simulation begins, the "Transient Blade Row Post-processing Information" summary in the CFX-Solver Output file will display the time step range over which the solver will accumulate the Fourier coefficients.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Similarly, the "Time Transformation Stability" summary in the CFX-Solver Output file displays whether the rotor–stator pitch ratio is within the acceptable range.
- After the CFX-Solver Manager has run for a short time, you can track the monitor points you created in CFX-Pre by clicking the **Time Corrected User Points** tab that appears at the top of the graphical interface of CFX-Solver Manager.
- After the simulation has proceeded for some time, observe the periodic nature of the monitor point values.

---

9. When CFX-Solver is finished, select the check box next to **Post-Process Results**.
10. Click **OK**.

33.8. Viewing the Time Transformation Results in CFD-Post

In a transient blade row run, flow field variables are compressed using the Fourier coefficient method. These variables are accumulated at the end of the simulation. This enables you to navigate through any time instance, within the common period, without having to load multiple transient results files. By default CFD-Post displays results corresponding to the end the simulation.

To get started, follow these steps:

1. Start CFD-Post and load `TimeBladeRow_001.res`.
2. When CFD-Post opens, if you see the **Domain Selector** dialog box, ensure that all the domains are selected, then click **OK** to load the results from these domains.
3. If you see a message regarding transient blade row postprocessing, click **OK**.

33.8.1. Creating a Turbo Surface

Create a turbo surface to be used for making plots:

1. Click the **Turbo** tab.
2. If you see the **Turbo Initialization** dialog box, click **Yes**, otherwise click the **Initialize All Components** button, which is visible initially by default, or after double-clicking the **Initialization** object in the **Turbo** tree view.
3. Select **Insert** > **Location** > **Turbo Surface**.
4. Change the name to `Span 50`.
5. Configure the following setting(s):

Tab	Setting	Value
Geometry	Definition	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Method**	Constant Span
	Definition > **Value**	 0.5

6. Click **Apply**.
7. Turn off the visibility of `Span 50` by clearing its check box in the **Outline** tree view.

33.8.2. Creating a Vector Plot

1. Click **Insert** > **Vector** and accept the default name.
2. Configure the following setting(s):

Tab	Setting	Value
Geometry	Definition > **Locations**	 Span 50
	Definition > **Variable**	 Velocity in Stn Frame

3. Click **Apply**.

   The vector plot shows `Velocity in Stn Frame` values corresponding to the end of a common period.

   The rotor domain is in the angular position corresponding to its location after 10 passing periods. Now you will align the rotor with the stator, as it was in the solver input file.

4. Click *Timestep Selector* .
5. Select the 1st time step.
6. Click **Apply** to load the time step, and then click **Close** to exit the dialog box.

   The rotor blades move to their starting positions.

33.8.3. Creating a Contour Plot

1. Turn off the visibility of `Vector 1`.
2. Click **Insert** > **Contour** and accept the default name.
3. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Geometry	**Locations**	Span 50
	Variable	Pressure
	Range	Local
	# of Contours	21

4. Click **Apply**.

The contour plot shows `Pressure` values corresponding to the specified time step.

33.8.4. Creating a Variable Time Chart

In this section, you will compute and plot the magnitude of the forces that the flow applies on the rotor blade. For a transient blade row case, CFD-Post automatically reconstructs variables for the flow solution time based on the last time step. Intermediate time steps for time instances in the common period are located in the **Timestep Selector**. In Setting up a Transient Blade Row Model (p. 809), you set 35 time steps per stator blade passing period and there are six rotor blade passing periods in a common period. Therefore, the total number of intermediate time steps in the common period is 210. For this case, the solver has reconstructed results over two common periods (420 time steps). You will reduce the total number of time steps to 140 to speed up the generation of the time chart.

Reduce the number of time steps in the period:

1. Click *Timestep Selector* .

2. Configure the following setting(s):

Tab	Setting	Value
Timestep Selector	**Timestep Sampling**	Uniform
	Number of Timesteps	70

3. Click **Apply**.

   The **Timestep Selector** now shows a total of 140 steps over two common periods (shown under the *Phase* column).

Compute the forces on the blade:

1. Select **Insert** > **Expression**.

2. In the **Insert Expression** dialog box, type `forces on rotor blade`.

3. Click **OK**.

4. Set **Definition** to `sqrt(force_x()@ R1 Blade ^2 + force_y()@ R1 Blade ^2 + force_z()@ R1 Blade ^2)`

5. Click **Apply** to create the expression.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create a transient chart showing force:

1. Select **Insert** > **Chart** and accept the default name.
2. Configure the following setting(s):

Tab	Setting	Value
General	**XY - Transient or Sequence**	(Selected)
Data Series	Series 1 > Data Source > **Expression**	(Selected)
Data Series	Series 1 > Data Source > **Expression**	forces on rotor blade

3. Click **Apply**.

   A chart showing force on a single rotor blade against time is created, added to the chart object, and displayed in the **Chart Viewer**.

33.8.5. Setting up Data Instancing Transformations

In this section, you will create additional copies of the original passages that replicate mesh nodes at different locations with correct space and time interpolation values. After the data instancing process, CFD-Post will create additional mesh nodes proportional to the number of extra passages created, and populate them with solution variables correctly updated to their corresponding position in time and space.

1. From the **Outline** tree view, edit R1.
2. In the **Data Instancing** tab, set **Number of Data Instances** to 7.
3. Click **Apply**.
4. From the **Outline** tree view, edit S1.
5. In the **Data Instancing** tab, set **Number of Data Instances** to 6.
6. Click **Apply**.
7. Turn off the visibility of `Wireframe`.

On the **3D Viewer** tab, CFD-Post displays the group of blades corresponding to 1/6 of a full wheel (the minimum number of blades that makes a unity pitch ratio between stator and rotor passages).

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The data-dependent transient `forces on rotor blade` on `Chart 1` is still showing the result computed on a single blade passage. After you expand the number of rotor blades in the rotor passage to 7, the `R1 Blade` groups all 7 rotor blades together and the total force should be updated. To update the chart, click the **Refresh** button at the top of the **Chart Viewer**.

The `forces on rotor blade` expression is now being computed on all 7 blades in the extended number of passages in the rotor.

33.8.6. Setting up Graphical Instancing Transformations

In this section, you will create additional copies of the original domain passage. For the current example, the original passage was replicated by a number to assemble 1/6 of the wheel using the **Data Instancing** tool. Therefore, by applying the graphical instancing on R1 and S1 you will be making copies of the extended version of these passages.

1. From the **Outline** tree view, edit R1.
2. In the **Instancing** tab configure the following settings:

Tab	Setting	Value
Instancing	**Number of Graphical Instances**	6
	Instance Definition	Custom
	Passages per Component	7

3. Click **Apply**.
4. From the **Outline** tree view, edit S1.
5. In the **Instancing** tab configure the following settings:

Tab	Setting	Value
Instancing	**Number of Graphical Instances**	6
	Instance Definition	Custom
	Passages per Component	6

6. Click **Apply**.

   The Graphics Instancing feature makes graphical copies of objects and places them at an angular position computed using the **Number of Passages** and **Number of Passages per Component** on the **Instancing** panel. To complete the full wheel, you replicated the 1/6 wheel sector, which was obtained using data instancing, six times. On the **3D Viewer** tab, CFD-Post displays the pressure plot on `Span 50` over the full wheel.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

33.8.7. Animating the Movement of the Rotor Relative to the Stator

With the **Timestep Selector** set to time step `0`, you will make an animation showing the relative motion starting from this time step and lasting for one stator blade passing period.

1. Click the **3D Viewer** tab.
2. Position the geometry for the animation by right-clicking on a blank area in the viewer and selecting **Predefined Camera** > **View From -X**.
3. Click *Animation* .

   The **Animation** dialog box appears.
4. Set **Type** to **Keyframe Animation**.
5. Click *New* to create `KeyframeNo1`.
6. Select `KeyframeNo1`, then set **# of Frames** to `70`, then press **Enter** while in the **# of Frames** box.

   Tip:

   Be sure to press **Enter** and confirm that the new number appears in the list before continuing.

   This will place 70 intermediate frames between the keyframes, for a total of 72 frames.
7. Use the **Timestep Selector** to load time step `70` and then close the dialog box.
8. In the **Animation** dialog box, click *New* to create `KeyframeNo2`.
9. Click *More Animation Options* to expand the **Animation** dialog box.
10. Select **Save Movie**.
11. Specify a filename for the movie.
12. Set **Format** to `MPEG1`.
13. Click *To Beginning* to rewind the active keyframe to `KeyframeNo1`.

    Note:

    The active keyframe is indicated by the value appearing in the **F:** field in the middle of the **Animation** dialog box. In this case it will be `1`.

    Wait for CFD-Post to finish loading the objects for this frame before proceeding.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14. Click *Save animation state* and save the animation to a file. This will enable you to quickly restore the animation in case you want to make changes. Animations are not restored by loading ordinary state files (those with the `.cst` extension).

15. Click *Play the animation* .

    Note:

    It takes a while for the animation to be completed. To view the movie file, you will need to use a media player that supports the MPEG format.

    From the animation and plots, you can see that the flow is continuous across the interface. This is because CFD-Post is capable of interpolating the flow field variables to the correct time and position using the computed Fourier coefficients.

16. When you have finished, close the **Animation** dialog box and then close CFD-Post, saving the animation state at your discretion.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 34: Time Transformation Method for a Transient Rotor-stator Case with Conjugate Heat Transfer

This tutorial includes:

34.1. Tutorial Features

34.2. Overview of the Problem to Solve

34.3. Preparing the Working Directory

34.4. Defining a Transient Blade Row Case with Conjugate Heat Transfer in CFX-Pre

34.5. Obtaining a Solution to the Transient Blade Row Case

34.6. Viewing the Time Transformation Results in CFD-Post

Important:

This tutorial requires file `TimeBladeRowIni_001.res`, which is produced by following tutorial Time Transformation Method for a Transient Rotor-stator Case (p. 799).

34.1. Tutorial Features

In this tutorial you will learn about:

Component	Feature	Details
CFX-Pre	User Mode	Turbo Wizard
		General mode
	Analysis Type	Transient Blade Row
	Fluid Type	Air Ideal Gas
	CCL File	Import
	Domain Type	Multiple Domains
		Rotating Frame of Reference
	Turbulence Model	Shear Stress Transport
	Heat Transfer	Total Energy
	Boundary Conditions	Inlet (Subsonic)
		Outlet (Subsonic)
		Wall (Counter Rotating)

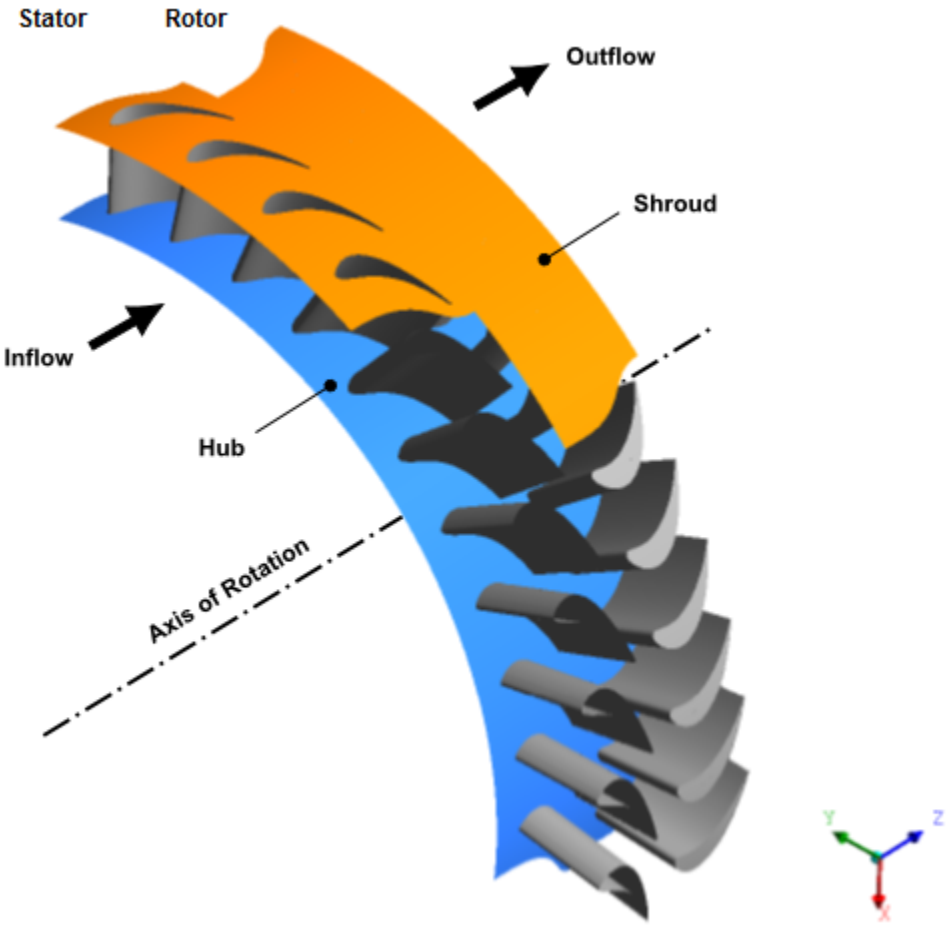
Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Component	Feature	Details
	Domain Interface	Fluid Solid
	Domain Solver Control	Timescale Factor
CFX-Solver Manager	Derived Variable	Average solid temperature
CFD-Post	Plots	Contour
		Data Instancing
		Time Chart

34.2. Overview of the Problem to Solve

This tutorial sets up a transient blade row calculation with conjugate heat transfer using the Time Transformation model. It uses an axial turbine to illustrate the basic concepts of setting up, running, and monitoring a transient blade row problem with conjugate heat transfer in Ansys CFX. It also describes the postprocessing of transient blade row and conjugate heat transfer results using the tools provided in CFD-Post for this type of calculation.

The full geometry of the axial rotor-stator stage selected for modeling contains 36 stator blades and 42 rotor blades.



Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The geometry to be modeled consists of a single rotor blade passage, a single stator blade passage, and a single solid rotor blade. Each rotor blade passage is an 8.571° section (360°/42 blades), while each stator blade passage is a 10° section (360°/36 blades). The pitch ratio at the interface between the rotor passage and the stator passage is 0.8571 (that is, 6/7).

For the Time Transformation method, you should always maintain an ensemble pitch ratio within a range of 0.75 to 1.4. Note that the range of permissible pitch ratios narrows significantly with slower rotation speed. A full machine analysis can be performed (modeling all rotor and stator blades), which always eliminates any pitch change, but will require significant computational time. For this rotor-stator geometry, a 1/6 machine section (7 rotor blades, 6 stator blades) would produce a pitch ratio of 1.0, but this would require a model about 7 times larger than in this tutorial example.

In this example, the rotor rotates about the Z axis at 3500 rev/min (positive rotation following the right hand rule) while the stator is stationary. Rotational periodicity boundaries are used to enable only a small section of the full geometry to be modeled.

The flow is modeled as being turbulent and compressible. Profile boundary conditions are used at the inlet and outlet. These profiles were obtained from previous simulations of the upstream and downstream stages. In this tutorial, a separate total temperature profile is applied to the stator inlet via CEL expressions that are provided in a .ccl file. This profile is of total temperature in a Gaussian distribution with a maximum that is 50% higher than the baseline value.

In this tutorial, conjugate heat transfer is modeled between the rotor blade passage and the solid rotor blade. The solid rotor blade domain is defined and the conjugate heat transfer interface is created. The timescale factor setting is adjusted in the domain solver control settings for the solid rotor blade to accelerate the solid thermal diffusion convergence.

The following steps outline the overall approach:

1. Define the transient blade row simulation using the Turbomachinery wizard in CFX-Pre.
2. Import the stator and rotor meshes, which were created in Ansys TurboGrid.
3. Enter the basic model definition.
4. Import the solid blade mesh, which was created in ICEM CFD.
5. Set the profile boundary conditions using CFX-Pre in General mode.
6. Create a fluid-solid interface to model conjugate heat transfer between the rotor blade passage and the solid rotor blade
7. Set the timescale factor in domain solver control for the solid rotor blade.
8. Run the transient blade row simulation using the steady-state results from Time Transformation Method for a Transient Rotor-stator Case (p. 799) as an initial guess.

34.3. Preparing the Working Directory

1. Create a working directory.

   Ansys CFX uses a working directory as the default location for loading and saving files for a particular session or project.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Download the `time_blade_row_cht.zip` file here.

3. Unzip `time_blade_row_cht.zip` to your working directory.

   Ensure that the following tutorial input files are in your working directory:

   - `TBRCHTCEL.ccl`
   - `TBRInletProfile.csv`
   - `TBROutletProfile.csv`
   - `TBRTurbineRotor.gtm`
   - `TBRTurbineSolid.cfx5`
   - `TBRTurbineStator.gtm`
   - `TimeBladeRowIni_001.res` (produced by following tutorial Time Transformation Method for a Transient Rotor-stator Case (p. 799))

4. Set the working directory and start CFX-Pre.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

34.4. Defining a Transient Blade Row Case with Conjugate Heat Transfer in CFX-Pre

This tutorial uses the Turbomachinery wizard in CFX-Pre. This preprocessing mode is designed to simplify the setup of turbomachinery simulations.

1. In CFX-Pre, select **File** > **New Case**.
2. Select **Turbomachinery** and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `TimeBladeRowCHT`.
5. Click **Save**.
6. If you are notified that the file already exists, click **Overwrite**.

34.4.1. Basic Settings

1. In the **Basic Settings** panel, configure the following settings:

Setting	Value
Machine Type	Axial Turbine
Axes	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
> **Coordinate Frame**	Coord 0
Axes > **Rotation Axis**	 Z
Analysis Type > **Type**	 Transient Blade Row
Analysis Type > **Method**	 Time Transformation

2. Click **Next**.

34.4.2. Components Definition

You will define two new components and import their respective meshes.

1. Right-click in the blank area and select **Add Component** from the shortcut menu.
2. Create a new component of type `Stationary` named `S1` and click **OK**.
3. Configure the following setting(s):

Setting	Value
Mesh > **File**	 TBRTurbineStator.gtm [a]
a. You may have to select the `CFX Mesh (*gtm *cfx)` option under **Files of type**.	

4. Create a new component of type `Rotating`, named `R1` and click **OK**.
5. Configure the following setting(s):

Setting	Value
Component Type > **Value**	 3500 [rev min^-1] [a]
Mesh > **File**	 TBRTurbineRotor.gtm
a. From the problem description.	

6. Click **Next**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

34.4.3. Physics Definition

In this section you will set properties of the fluid domain and some solver parameters.

1. In the **Physics Definition** panel, configure the following:

Setting	Value
Fluid	Air Ideal Gas
Model Data > **Reference Pressure**	0 [atm] [a]
Model Data > **Heat Transfer**	Total Energy
Model Data > **Turbulence**	Shear Stress Transport
Inflow/Outflow Boundary Templates > **P-Total Inlet P-Static Outlet**	(Selected)
Inflow/Outflow Boundary Templates > Inflow > **P-Total**	169000 [Pa][b]
Inflow/Outflow Boundary Templates > Inflow > **T-Total**	306 [K][b]
Inflow/Outflow Boundary Templates > Inflow > **Flow Direction**	Normal to Boundary
Inflow/Outflow Boundary Templates > Outflow > **P-Static**	110000 [Pa][b]
Interface > **Default Type**	Transient Rotor Stator

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
a. To define the simulation using absolute pressure, set this value to 0 atm. b. These values are temporary. They will be replaced with profile data later in the tutorial.	

2. Continue to click **Next** until you reach `Final Operations`.
3. Set **Operation** to `Enter General Mode` because you will continue to define the simulation through settings not available in the Turbomachinery wizard.
4. Click **Finish**.

34.4.4. Additional Fluid Model Settings

Verify the following settings, which affect the accuracy of the simulation:

1. Edit R1.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Domain Models > Domain Motion > **Alternate Rotation Model**	 (Selected)
Fluid Models	Heat Transfer > **Incl. Viscous Work Term**	 (Selected)

3. Click **OK**.

34.4.5. Importing the Solid Blade Mesh File

1. In the **Outline** tree view, right-click `Mesh` and select **Import Mesh** > **ICEM CFD**.

   The **Import Mesh** dialog box appears.

2. Configure the following setting(s):

Setting	Value
File name	TBRTurbineSolid.cfx5
Options > **Mesh Units**	 m[a]

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
a. This mesh was created using units of meters; however the units are not stored with this type of mesh. Set **Mesh Units** to `m` when importing the mesh into CFX-Pre so that the mesh remains the intended size.	

3. Click **Open**.

34.4.6. Creating the Solid Blade Domain

In this section you will define the solid domain for the blade and initialize the domain temperature to a reasonable value. You will also adjust the **Timescale Factor** setting in the solver control options. This setting is used in transient cases involving conjugate heat transfer to increase the thermal response of the solid. This is done to compensate for the large disparity in time scales between fluid convection and solid diffusion, which makes transient modeling of such problems otherwise impractical. The initialization and solver control options that are set when creating a domain apply only to that domain.

1. Rename `Default Domain` to `Rotor Solid Blade`.

2. Edit `Rotor Solid Blade`.

3. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Location and Type > **Domain Type**	Solid Domain
	Solid Definitions > Solid 1 > **Material**	Aluminum [a]
	Domain Models > Domain Models > **Option**	Rotating
	Domain Models > Domain Motion > **Angular Velocity**	3500 [rev min^-1]
	Domain Models > Passage Definition > **Pass. in Component**	1
	Domain Models	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Passage Definition > **Passages in 360**	 42
Initialization	**Domain Initialization**	(Selected)
	Domain Initialization > Initial Conditions > Temperature > **Option**	 Automatic with Value
	Domain Initialization > Initial Conditions > Temperature > **Temperature**	 300 [K]
Solver Control	**Domain Solver Control**	(Selected)
	Domain Solver Control > Timescale Control > **Timescale Factor**	 1.0e6[b]

a. Click the *Ellipsis* icon ... to open the **Material** dialog box.

b. The timescale factor is defined as: solid time scale / Δt, where Δt is the simulation time-step size necessary to resolve flow features of interest. Increasing the timescale factor to 1.0e6 from the default 1 increases the physical time scale of this transient run inside the solid domain by a factor of 1.0e6.

4. Click **OK**.

34.4.7. Modifying the Boundary Conditions

34.4.7.1. Initializing Profile Boundary Conditions

The inlet and outlet boundary conditions are defined using profiles in your working directory and several CEL expressions. Boundary profile data must be initialized before they can be used for boundary conditions.

1. Select **Tools** > **Initialize Profile Data**.

   The **Initialize Profile Data** dialog box appears.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Beside **Profile Data File**, click *Browse* .

   The **Select Profile Data File** dialog box appears.

3. From your working directory, select `TBRInletProfile.csv`.
4. Click **Open**.
5. Click **Apply**.

   The profile data is read into memory.

6. Under **Data File**, click *Browse* .
7. From your working directory, select `TBROutletProfile.csv`.
8. Click **Open**.
9. Click **OK**.
10. Select **File** > **Import** > **CCL**.
11. Ensure that **Import Method** is set to **Append**.
12. Select `TBRCHTCEL.ccl`.
13. Click **Open**.

Note:

After profile data has been initialized from a file, the profile data file should not be deleted or otherwise removed from its directory. By default, the full filepath to the profile data file is stored in CFX-Pre, and the profile data file is read directly by CFX-Solver each time the solver is started or restarted.

34.4.7.2. Modifying Inlet and Outlet Boundary Conditions

1. Edit `S1 Inlet`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Profile Boundary Conditions > **Use Profile Data**	(Selected)
	Profile Boundary Setup > **Profile Name**	inlet

3. Click **Generate Values**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This causes the profile values of `Total Pressure` and `Total Temperature` to be applied at the nodes on the inlet boundary. It also causes entries to be made in the **Boundary Details** tab. In order to later reset the velocity values at the main inlet to match those that were originally read from the profile data file, revisit the **Basic Settings** tab for this boundary and click **Generate Values**.

4. Configure the following setting(s):

Tab	Setting	Value
Boundary Details	Mass and Momentum > **Option**	Total Pressure (stable)
	Mass and Momentum > **Relative Pressure**	inlet.Total Pressure(r)
	Flow Direction > **Option**	Cylindrical Components
	Flow Direction > **Axial Component**	1
	Flow Direction > **Radial Component**	0
	Flow Direction > **Theta Component**	0
	Heat Transfer > **Total Temperature**	TINLET

5. Click **OK**.

6. Edit `R1 Outlet`.

7. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Frame Type**	Rotating
	Profile Boundary Conditions > **Use Profile Data**	(Selected)
	Profile Boundary Setup > **Profile Name**	outlet

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. Click **Generate Values**.

9. Configure the following setting(s):

Tab	Setting	Value
Boundary Details	Mass And Momentum > **Option**	 Static Pressure
	Mass and Momentum > **Relative Pressure**	 outlet.Pressure(r)

10. Click **OK**.

34.4.7.3. Visualizing the Profile Boundary Value

You can plot scalar profile values and vectors on inlet and outlet boundaries. In this section, you will edit a boundary so that you can visualize the pressure profile values at the inlet.

1. Edit `S1 Inlet`

2. Configure the following setting(s):

Tab	Setting	Value
Plot Options	**Boundary Contour**	(Selected)
	Profile Variable	Total Temperature

3. Click **Apply**

CFX-Pre plots the Total Pressure radial profile at the inlet with the pressure values displayed in a legend.

34.4.7.4. Creating the Blade Base Boundary

Here you will edit the default boundary in `Rotor Solid Blade` to create the hub boundary for the blade.

1. Rename `Rotor Solid Blade Default` to `Blade Base`.

2. Edit `Blade Base`.

3. On the **Basic Settings** tab, set **Location** to `Hub 2`.

4. On the **Boundary Details** tab, ensure that **Heat Transfer > Option** is set to `Adiabatic`.

5. Click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

34.4.8. Creating the Conjugate Heat Transfer Interface

1. Click **Insert** > **Domain Interface** and, in the dialog box that appears, set **Name** to `CHT Interface` and click **OK**.

2. Configure the following setting(s) of `CHT Interface`:

Tab	Setting	Value
Basic Settings	Interface Type	Fluid Solid
	Interface Side 1 > **Domain (Filter)**	Rotor Solid Blade
	Interface Side 1 > **Region List**	R1_BLADE, TE, TIP [a]
	Interface Side 2 > **Domain (Filter)**	R1
	Interface Side 2 > **Region List**	BLADE 2
	Interface Models > **Option**	General Connection[b]

a. The only region not selected is HUB 2, which is the blade surface attached to the hub.

b. As long as the connection is general, the interface will enable heat transfer.

3. Click **OK**.

   An error message related to the domain R1 will appear in the message window. This error occurs because the 2D Region BLADE 2 in R1 is being used in both a boundary and a domain interface. This error is resolved in the next step.

4. Delete the boundary `R1 Blade` in the domain R1.

34.4.9. Setting up a Transient Blade Row Model

You will set the simulation to be solved using the Time Transformation method.

1. Edit `Transient Blade Row Models`.

2. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
Transient Method > Time Period > **Option**	Passing Period
Transient Method > Time Period > **Domain**	S1
Transient Method > Time Steps > **Option**	Number of Timesteps per Period
Transient Method > Time Steps > **Timesteps/Period**	35
Transient Method > Time Duration > **Option**	Number of Periods per Run
Transient Method > Time Duration > **Periods per Run**	10

Note:

- The passing period is automatically calculated as: 2 * pi / (Passages in 360 * Signal Angular Velocity). The **Passing Period** setting cannot be edited.
- The number of time steps per period should always be larger than 2 * Number of Fourier Coefficients + 1 to be used for postprocessing.
- The time step size is also automatically calculated as: Passing Period / Number of Timesteps per Period. The **Timestep** setting cannot be edited.

3. Click **OK**.

34.4.10. Setting Output Control and Creating Monitor Points

For transient blade row calculations, a minimal set of variables are selected to be computed using Fourier coefficients. It is convenient to postprocess variables in the stationary frame when multiple

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

frames of reference are present. Here, you will add the `Velocity in Stn Frame` and `Mach Number in Stn Frame` variables to the default list. You will also add the `Wall Heat Flux` variable to the default list to observe the results of the conjugate heat transfer.

In addition, monitor points can be used to effectively compare the Time Transformation results against a reference case. They provide useful information on the quality of the reference phase and frequency produced in the simulation. They should be used to monitor convergence and, as the simulation converges, the user points should display a periodic pattern.

Note:

- When comparing to a reference case, make sure monitor points are placed in the same relative locations with respect to the initial configuration in both cases.
- It is important to check that the solver equations are being solved correctly. Monitoring pressure provides feedback on the momentum equations while monitoring temperature provides feedback on the energy equations.

Set up the output control and create monitor points as follows:

1. Click *Output Control* .
2. Click the **Trn Results** tab.
3. Configure the following setting(s):

Setting	Value
Transient Blade Row Results > **Extra Output Variables List**	(Selected)
Transient Blade Row Results > Extra Output Variables List > **Extra Output Var. List**	Velocity in Stn Frame, Mach Number in Stn Frame, Wall Heat Flux[a]
a. Click *Multi-select from extended list* and hold down the **Ctrl** key while selecting each of the listed variables.	

4. Click the **Monitor** tab.
5. Configure the following setting(s):

Setting	Value
Monitor Objects	(Selected)

Setting	Value
Monitor Objects > **Efficiency Output**	 (Cleared)

6. Create a monitor point named `rotor_P1`.
7. Under **Monitor Objects** > **Monitor Points and Expressions** > **rotor_P1**, configure the following settings:

Setting	Value
Option	Cylindrical Coordinates
Output Variables List	Pressure, Temperature, Total Pressure, Total Temperature, Velocity
Position Axial Comp.	0.211 [m]
Position Radial Comp.	0.2755 [m]
Position Theta Comp.	182 [degree]

8. Create an additional monitor point named `stator_P1`.
9. Under **Monitor Objects** > **Monitor Points and Expressions** > **stator_P1**, configure the following settings:

Setting	Value
Option	Cylindrical Coordinates
Output Variables List	Pressure, Temperature, Total Pressure, Total Temperature, Velocity
Position Axial Comp.	0.202 [m]
Position Radial Comp.	0.2755 [m]
Position Theta Comp.	178 [degree]

10. Create an additional monitor point named `solid_P1`.
11. Under **Monitor Objects** > **Monitor Points and Expressions** > **solid_P1**, configure the following settings:

Setting	Value
Option	Cylindrical Coordinates
Output Variables List	Temperature
Position Axial Comp.	0.224 [m]
Position Radial Comp.	0.2755 [m]

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
Position Theta Comp.	181 [degree]

Note:

Transient blade row cases use monitor points to monitor the periodic fluctuating variable values. For diagnostic purposes, you should have several monitor points. Here, three monitor points will be used for demonstration purposes.

12. Click **OK**.

34.4.11. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .
2. Configure the following setting(s):

Setting	Value
File name	TimeBladeRowCHT.def

3. Click **Save**.
4. Ignore the error message (the initial values for R1 and S1 will be specified in CFX-Solver Manager) and click **Yes** to continue.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, **Solver Input File** is set.
5. If using stand-alone mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

34.5. Obtaining a Solution to the Transient Blade Row Case

At this point, CFX-Pre has been shut down, and the **Define Run** dialog box is displayed in CFX-Solver Manager. You will now obtain a solution to the CFD problem. To reduce the simulation time, the simulation will be initialized using a steady-state case.

1. Ensure that the **Define Run** dialog box is displayed. If an error message appears, ignore it and click **Yes** to continue.

   Solver Input File should be set to `TimeBladeRowCHT.def`.
2. Under the **Initial Values** tab, select **Initial Values Specification**.
3. Under **Initial Values Specification** > **Initial Values**, select `Initial Values 1`.
4. Under **Initial Values Specification** > **Initial Values** > **Initial Values 1 Settings** > **File Name**, click *Browse* .

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Select `TimeBladeRowIni_001.res` from your working directory.
6. Click **Open**.
7. Under **Initial Values Specification** > **Use Mesh From**, select `Solver Input File`.
8. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. At the end of the run, a dialog box is displayed stating that the simulation has ended.

Note:

- Before the simulation begins, the "Transient Blade Row Post-processing Information" summary in the CFX-Solver Output file will display the time step range over which the solver will accumulate the Fourier coefficients.
- Similarly, the "Time Transformation Stability" summary in the CFX-Solver Output file displays whether the rotor–stator pitch ratio is within the acceptable range.
- After the CFX-Solver Manager has run for a short time, you can track the monitor points you created in CFX-Pre by clicking the **Time Corrected User Points** tab that appears at the top of the graphical interface of CFX-Solver Manager.
- After the simulation has proceeded for some time, observe the periodic nature of the monitor point values.

---

9. While the solver is running, create and then monitor a derived variable for the average solid temperature over the last period:

   a. Open the **New Derived Variable** dialog box in any of these ways:

   - Select **Workspace** > **New Derived Variable**.
   - Select **Workspace** > **Workspace Properties**.

     The **Workspace Properties** dialog box appears.

     Select the **Derived Variables** tab.

     Click *New* .
   - Right-click in the chart area and select **Monitor Properties** from the context menu.

     The **Monitor Properties** dialog box appears.

     Select the **Derived Variables** tab.

     Click *New* .

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. Set **Name** to `Moving Average` and click **OK**.

   The **Derived Variable Properties** dialog box appears.

c. Set **Statistics** > **Statistics Type** to `Arithmetic Average`.

d. Ensure that the settings specify that the average is over a moving interval of one time period, then click **OK**.

   The derived variable is created.

e. Click **OK** to dismiss the **Workspace Properties** dialog box, if applicable.

f. In the **Monitor Properties** dialog box (right-click in the chart area and select **Monitor Properties** from the context menu), select the **Plot Lines** tab.

g. Expand the tree to `USER POINT` > `Temperature` > `solid_P1`, then right-click `solid_P1` and select **Add Derived Plot Line For Variable** > **Moving Average**.

h. Click **OK** to dismiss the **Monitor Properties** dialog box.

i. After at least one interval has passed, observe that the **User Points** tab shows a plot for `solid_P1 (Temperature), Moving Average`.

10. When CFX-Solver is finished, select the check box next to **Post-Process Results**.

11. Click **OK**.

34.6. Viewing the Time Transformation Results in CFD-Post

In a transient blade row run, flow field variables are compressed using the Fourier coefficient method. These variables are accumulated at the end of the simulation. This enables you to navigate through any time instance, within the common period, without having to load multiple transient results files. By default CFD-Post displays results corresponding to the end the simulation.

To get started, follow these steps:

1. Start CFD-Post and load `TimeBladeRowCHT_001.res`.

2. When CFD-Post opens, if you see the **Domain Selector** dialog box, ensure that all the domains are selected, then click **OK** to load the results from these domains.

3. If you see a message regarding transient blade row postprocessing, click **OK**.

4. Click *Timestep Selector* .

5. Select the 1st time step.

6. Click **Apply** to load the time step, and then click **Close** to exit the dialog box.

   The rotor blades move to their starting positions.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

34.6.1. Creating a Slice Plane

Create a plane to be used for making plots:

1. Select **Insert** > **Location** > **Plane**. Accept the default name and click **OK**.
2. Configure the following setting(s):

Tab	Setting	Value
Geometry	**Domains**	Rotor Solid Blade
	Definition > **Method**	ZX Plane
	Definition > **Y**	-0.27 [m]

3. Click **Apply**.
4. Turn off the visibility of `Plane 1` by clearing its check box in the **Outline** tree view.

34.6.2. Creating a Contour Plot

1. Click **Insert** > **Contour** and accept the default name.
2. Configure the following setting(s):

Tab	Setting	Value
Geometry	**Locations**	Plane 1
	Variable	Temperature.Trnavg[a]
	Range	Local
	# of Contours	31

a. Click the *Ellipsis* icon to open the selection dialog box, then select `Temperature.Trnavg` under the `Transient Statistics > Arithmetic Average` branch. Click **OK**.

3. Click **Apply**.

   The contour plot shows `Temperature` values corresponding to arithmetic averages calculated using the results from all time steps.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

34.6.3. Creating a Variable Time Chart

In this section, you will compute and plot the magnitude of the heat flow on the rotor blade. For a transient blade row case, CFD-Post automatically reconstructs variables for the flow solution time based on the last time step. Intermediate time steps for time instances in the common period are located in the **Timestep Selector**. In Setting up a Transient Blade Row Model (p. 833), you set 35 time steps per stator blade passing period and there are six stator blade passing periods in a common period. Therefore, the total number of intermediate time steps in the common period is 210. For this case, the solver has reconstructed results over two common periods (420 time steps). You will reduce the total number of time steps to 140 to speed up the generation of the time chart.

Reduce the number of time steps in the period:

1. Click *Timestep Selector* .
2. Configure the following setting(s):

Tab	Setting	Value
Timestep Selector	**Timestep Sampling**	Uniform
	Number of Timesteps	70

3. Click **Apply**.

   The **Timestep Selector** now shows a total of 140 steps over two common periods (shown under the *Phase* column).

Compute the heat flow on the blade:

1. Select **Insert** > **Expression**.
2. In the **Insert Expression** dialog box, type `heat flow on solid blade`.
3. Click **OK**.
4. Set **Definition**, to `areaInt(Wall Heat Flux)@CHT Interface Side 1`
5. Click **Apply** to create the expression.

Create a transient chart showing heat flow:

1. Select **Insert** > **Chart** and accept the default name.
2. Configure the following setting(s):

Tab	Setting	Value
General	**XY - Transient or Sequence**	(Selected)
Data Series	Series 1 > Data Source	

Tab	Setting	Value
	> **Expression**	(Selected)
	Series 1 > Data Source > **Expression**	 heat flow on solid blade

3. Click **Apply**.

   A chart showing heat flow on a single rotor blade against time is created, added to the chart object, and displayed in the **Chart Viewer**.

34.6.4. Setting up Data Instancing Transformations

In this section, you will create additional copies of the original passages that replicate mesh nodes at different locations with correct space and time interpolation values. After the data instancing process, CFD-Post will create additional mesh nodes proportional to the number of extra passages created, and populate them with solution variables correctly updated to their corresponding position in time and space.

1. From the **Outline** tree view, edit `Contour 1`.
2. In the **Geometry** tab, set **Location** to `CHT Interface Side 1`.
3. Click **Apply**.
4. From the **Outline** tree view, edit `Rotor Solid Blade`.
5. In the **Data Instancing** tab, set **Number of Data Instances** to 7.
6. Click **Apply**.
7. Turn off the visibility of `Wireframe`.

On the **3D Viewer** tab, CFD-Post displays the group of rotor blades corresponding to 1/6 of a full wheel (the minimum number of blades that makes a unity pitch ratio between stator and rotor passages).

The data-dependent transient `heat flow on solid blade` on `Chart 1` is still showing the result computed on a single solid blade. After you expand the number of solid rotor blades to 7, the `CHT Interface Side 1` groups all 7 rotor blades together and the total heat flow should be updated. To update the chart, click the **Refresh** button at the top of the **Chart Viewer**.

The `heat flow on solid blade` expression is now being computed on all 7 solid rotor blades.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 35: Fourier Transformation Method for a Transient Rotor-stator Case

A single-stage turbomachinery configuration with pitch change can be modeled using the Fourier Transformation method. This method is particularly useful for modeling a large-pitch-change configuration and when it is not possible to use the Time Transformation method.

This tutorial includes:

35.1. Tutorial Features

35.2. Overview of the Problem to Solve

35.3. Preparing the Working Directory

35.4. Defining and Obtaining a Solution for the Time Integration Solution Method Case

35.5. Defining and Obtaining a Solution for the Harmonic Balance Solution Method Case

35.6. Postprocessing the Transient Rotor-stator Solution

Important:

This tutorial requires file `TimeBladeRowIni_001.res`, which is produced by following tutorial Time Transformation Method for a Transient Rotor-stator Case (p. 799).

35.1. Tutorial Features

In this tutorial you will learn about:

Component	Feature	Details
CFX-Pre	User Mode	Turbo Wizard
		General mode
	Analysis Type	Transient Blade Row
		Fourier Transformation Pitch Change Model
		Time Integration Solution Method
		Harmonic Balance Solution Method
	Fluid Type	Air Ideal Gas
	Domain Type	Multiple Domains
		Rotating Frame of Reference

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

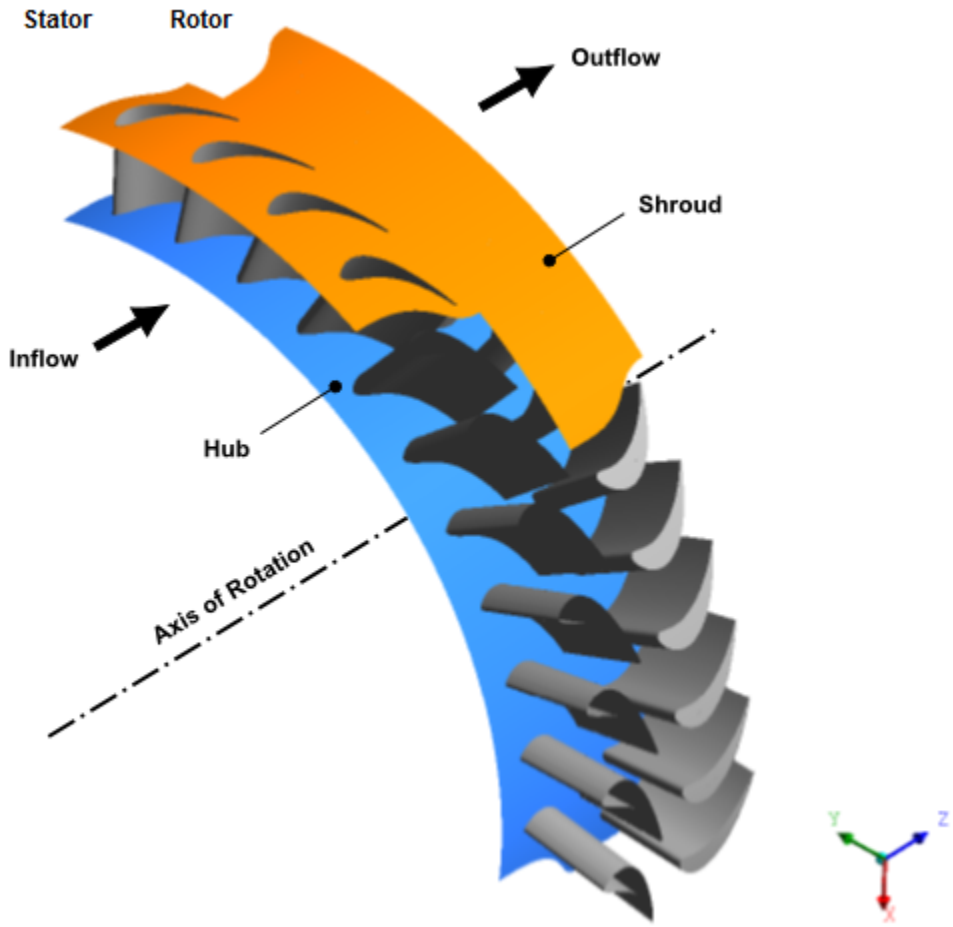
Component	Feature	Details
	Turbulence Model	Shear Stress Transport
	Heat Transfer	Total Energy
	Boundary Conditions	Inlet (Subsonic)
		Outlet (Subsonic)
		Wall (Counter Rotating)
CFD-Post	Plots	Vector
		Contour
		Data Instancing
		Time Chart
		Animation

35.2. Overview of the Problem to Solve

The goal of this tutorial is to set up a transient blade row calculation using the Fourier Transformation model. It uses an axial turbine to illustrate the basic concepts of setting up, running, and monitoring a transient blade row problem in Ansys CFX. It also describes the postprocessing of transient blade row results using the tools provided in CFD-Post for this type of calculation.

The full geometry of the axial rotor-stator stage selected for modeling contains 36 stator blades and 42 rotor blades.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The geometry to be modeled consists of a pair of rotor blade passages and a pair of stator blade passages. Pairs of passages are needed because the Fourier Transformation method uses a double-passage strategy. Each rotor blade passage is an 8.571° section (360°/42 blades), while each stator blade passage is a 10° section (360°/36 blades). The pitch ratio at the interface between the pair of rotor passages and the pair of stator passages is 0.8571 (that is, 6/7).

You should always try to obtain a pitch ratio as close to 1 as possible in your model to minimize approximations, but this must be weighed against computational resources. A full machine analysis (modeling all rotor and stator blades) eliminates pitch change, but requires significant computational time. For this rotor-stator geometry, a 1/6 machine section (7 rotor blades, 6 stator blades) would produce a pitch ratio of 1.0, but this would require a model about 3 times larger than in this tutorial example.

In this example, the rotor rotates about the Z axis at 3500 rev/min (positive rotation following the right hand rule) while the stator is stationary. Rotational periodicity boundaries with phase lag are used to enable only a small section of the full geometry to be modeled.

The flow is modeled as being turbulent and compressible. Profile boundary conditions are used at the inlet and outlet. In this tutorial, the profiles are a function of radial coordinate only. These profiles were obtained from previous simulations of the upstream and downstream stages.

The following steps outline the overall approach:

1. Define the transient blade row simulation using the Turbomachinery wizard in CFX-Pre.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Import the stator and rotor meshes, which were created in Ansys TurboGrid.
3. Enter the basic model definition.
4. Set the profile boundary conditions using CFX-Pre in General mode.
5. Run the transient blade row simulation using the steady-state results from Time Transformation Method for a Transient Rotor-stator Case (p. 799) as an initial guess.

35.3. Preparing the Working Directory

1. Create a working directory.

   Ansys CFX uses a working directory as the default location for loading and saving files for a particular session or project.

2. Download the `fourier_blade_row.zip` file here.
3. Unzip `fourier_blade_row.zip` to your working directory.

   Ensure that the following tutorial input files are in your working directory:

   - `TBRInletProfile.csv`
   - `TBROutletProfile.csv`
   - `TBRTurbineRotor.gtm`
   - `TBRTurbineStator.gtm`
   - `TimeBladeRowIni_001.res` (produced by following tutorial Time Transformation Method for a Transient Rotor-stator Case (p. 799))

4. Set the working directory and start CFX-Pre.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

35.4. Defining and Obtaining a Solution for the Time Integration Solution Method Case

This tutorial uses the Turbomachinery wizard in CFX-Pre. This preprocessing mode is designed to simplify the setup of turbomachinery simulations.

1. In CFX-Pre, select **File** > **New Case**.
2. Select **Turbomachinery** and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `FourierBladeRowTime`.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Click **Save**.

6. If you are notified that the file already exists, click **Overwrite**.

35.4.1. Basic Settings

1. In the **Basic Settings** panel, configure the following settings:

Setting	Value
Machine Type	Axial Turbine
Axes > **Coordinate Frame**	 Coord 0
Axes > **Rotation Axis**	 Z
Analysis Type > **Type**	 Transient Blade Row
Analysis Type > **Method**	 Fourier Transformation

2. Click **Next**.

35.4.2. Components Definition

The Fourier Transformation method requires two rotor blade passages and two stator blade passages. You will define two new components and import their respective meshes.

1. Right-click in the blank area and select **Add Component** from the shortcut menu.

2. Create a new component of type `Stationary` named `S1` and click **OK**.

3. Configure the following setting(s):

Setting	Value
Mesh > **File**	 TBRTurbineStator.gtm [a]
a. You may have to select the `CFX Mesh (*gtm *cfx)` option under **Files of type**.	

4. Expand the **Passage and Alignment** section and click **Edit**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Configure the following setting(s):

Setting	Value
Passage and Alignment > **Passages to Model**	 2

After clicking **Done**, you will see that the stator blade passage is correctly replicated and the resulting mesh contains two stator blade passages as required by the Fourier Transformation model. This also creates the Sampling Interface (**S1 Internal Interface 1**).

6. Right-click in the blank area and select **Add Component** from the shortcut menu.

7. Create a new component of type `Rotating`, named `R1` and click **OK**.

8. Configure the following setting(s):

Setting	Value
Component Type > **Value**	 3500 [rev min^-1] [a]
Mesh > **File**	 TBRTurbineRotor.gtm
a. From the problem description.	

9. Expand the **Passage and Alignment** section and click **Edit**.

10. Configure the following setting(s):

Setting	Value
Passage and Alignment > **Passages to Model**	 2

After clicking **Done**, you will see that the rotor blade passage is correctly replicated and the resulting mesh contains two rotor blade passages as required by the Fourier Transformation model. This also creates the Sampling Interface (**R1 Internal Interface 1**).

11. Click **Next**.

35.4.3. Physics Definition

In this section, you will set properties of the fluid domain and some solver parameters.

1. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
Fluid	Air Ideal Gas
Model Data > **Reference Pressure**	0 [atm] [a]
Model Data > **Heat Transfer**	Total Energy
Model Data > **Turbulence**	Shear Stress Transport
Inflow/Outflow Boundary Templates > **P-Total Inlet P-Static Outlet**	(Selected)
Inflow/Outflow Boundary Templates > Inflow > **P-Total**	169000 [Pa] [b]
Inflow/Outflow Boundary Templates > Inflow > **T-Total**	306 [K] [b]
Inflow/Outflow Boundary Templates > Inflow > **Flow Direction**	Normal to Boundary
Inflow/Outflow Boundary Templates > Outflow > **P-Static**	110000 [Pa] [b]
Interface > **Default Type**	Transient Rotor Stator

a. A reference pressure of 0 atm defines the simulation in absolute pressure.

b. These values are temporary. They will be replaced with profile data later in the tutorial.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click **Next**.

   Under the Interface Definition section, you can observe that both the Fourier coefficient sampling interfaces **S1 Internal Interface 1** and **R1 Internal Interface 1**, as well as, the Phase-shifted interfaces **S1 to S1 Periodic 1** and **R1 to R1 Periodic 1** are automatically created.

3. Click **Next**.

35.4.4. Disturbance Definition

In this section, you will specify the type of disturbance being imposed.

1. Configure the following setting(s):

Setting	Value
Disturbance > Fourier Transformation > **Type**	Rotor Stator
Disturbance > Fourier Transformation > **Domain Interface**	R1 to S1

2. Continue to click **Next** until you reach `Final Operations`.

3. Set **Operation** to `Enter General Mode` because you will continue to define the simulation through settings not available in the Turbomachinery wizard.

4. Click **Finish**.

35.4.5. Additional Fluid Model Settings

Verify the following settings, which affect the accuracy of the simulation:

1. Edit R1.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Domain Models > Domain Motion > **Alternate Rotation Model**	(Selected)
Fluid Models	Heat Transfer > **Incl. Viscous Work Term**	(Selected)

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click **OK**.

35.4.6. Initializing Profile Boundary Conditions

The inlet and outlet boundary conditions are defined using profiles in your working directory. Boundary profile data must be initialized before they can be used for boundary conditions.

1. Select **Tools** > **Initialize Profile Data**.

   The **Initialize Profile Data** dialog box appears.

2. Beside **Profile Data File**, click *Browse* .

   The **Select Profile Data File** dialog box appears.

3. From your working directory, select `TBRInletProfile.csv`.
4. Click **Open**.
5. Click **Apply**.

   The profile data is read into memory.

6. Under **Data File**, click *Browse* .
7. From your working directory, select `TBROutletProfile.csv`.
8. Click **Open**.
9. Click **OK**.

Note:

After profile data has been initialized from a file, the profile data file should not be deleted or otherwise removed from its directory. By default, the full file path to the profile data file is stored in CFX-Pre, and the profile data file is read directly by CFX-Solver each time the solver is started or restarted.

35.4.7. Modifying Inlet and Outlet Boundary Conditions

1. Edit `S1 Inlet`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Profile Boundary Conditions > **Use Profile Data**	 (Selected)
	Profile Boundary Setup	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Profile Name**	inlet

3. Click **Generate Values**.

   This causes the profile values of `Total Pressure` and `Total Temperature` to be applied at the nodes on the inlet boundary. It also causes entries to be made in the **Boundary Details** tab. In order to later reset the velocity values at the main inlet to match those that were originally read from the profile data file, revisit the **Basic Settings** tab for this boundary and click **Generate Values**.

4. Configure the following setting(s):

Tab	Setting	Value
Boundary Details	Mass and Momentum > **Option**	Total Pressure (stable)
	Mass and Momentum > **Relative Pressure**	inlet.Total Pressure(r)
	Flow Direction > **Option**	Cylindrical Components
	Flow Direction > **Axial Component**	1
	Flow Direction > **Radial Component**	0
	Flow Direction > **Theta Component**	0
	Heat Transfer > **Option**	Total Temperature
	Heat Transfer > **Total Temperature**	inlet.Total Temperature(r)

5. Click **OK**.

6. Edit `R1 Outlet`.

7. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Basic Settings	**Frame Type**	Rotating
	Profile Boundary Conditions > **Use Profile Data**	(Selected)
	Profile Boundary Setup > **Profile Name**	outlet

8. Click **Generate Values**.
9. Configure the following setting(s):

Tab	Setting	Value
Boundary Details	Mass and Momentum > **Option**	Static Pressure
	Mass and Momentum > **Relative Pressure**	outlet.Pressure(r)

10. Click **OK**.

35.4.8. Setting up a Transient Blade Row Model

You will set the simulation to be solved using the Fourier Transformation method.

1. Edit `Transient Blade Row Models`.
2. Configure the following setting(s):

Setting	Value
Transient Method > Time Period > **Option**	Automatic
Transient Method > Time Steps > **Option**	Timestep Multiplier
Transient Method > Time Steps > **Timestep Multiplier**	5
Transient Method	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
> Time Duration > **Option**	 Number of Periods per Run
Transient Method > Time Duration > **Periods per Run**	 20

Note:

- The passing period is automatically calculated as: 2 * pi / (Number of Blades * Angular Velocity). The **Passing Period** setting cannot be edited.
- The number of time steps per period should always be larger than 2 * Number of Fourier Coefficients + 1 to be used for postprocessing. When the time step size is set using the `Timestep Multiplier` option, the number of time steps per period is the product of the values of **Min Timesteps / Per.** and **Timestep Multiplier**.
- The time step size is also automatically calculated as: Passing Period / Number of Time Steps per Period. The **Timestep** setting cannot be edited.

3. Click **OK**.

35.4.9. Setting Output Control and Creating Monitor Points

For transient blade row calculations, a minimal set of variables are selected to be computed using Fourier coefficients. It is convenient to postprocess variables in the stationary frame when multiple frames of reference are present. Here, you will add the `Velocity in Stn Frame` and `Mach Number in Stn Frame` variables to the default list.

In addition, monitor points can be used to effectively compare the Fourier Transformation results against a reference case. They provide useful information on the quality of the reference phase and frequency produced in the simulation. As the simulation converges, the user points should display a periodic pattern.

Note:

- When comparing to a reference case, make sure monitor points are placed in the same relative locations with respect to the initial configuration in both cases.
- It is important to check that the solver equations are being solved correctly. Monitoring pressure provides feedback on the momentum equations while monitoring temperature provides feedback on the energy equations.
- For diagnostic purposes, you should have several monitor points. Here, two monitor points will be used for demonstration purposes.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Set up the output control and create monitor points as follows:

1. Click *Output Control* .
2. Click the **Trn Results** tab.
3. Configure the following setting(s):

Setting	Value
Transient Blade Row Results > **Extra Output Variables List**	(Selected)
Transient Blade Row Results > Extra Output Variables List > **Extra Output Var. List**	Velocity in Stn Frame, Mach Number in Stn Frame[a]
a. Click *Multi-select from extended list* and hold down the **Ctrl** key while selecting each of the listed variables.	

4. Click the **Monitor** tab.
5. Configure the following setting(s):

Setting	Value
Monitor Objects	(Selected)
Monitor Objects > **Efficiency Output**	(Cleared)

6. Create a monitor point named `rotor_P1`.
7. Under **Monitor Objects** > **Monitor Points and Expressions** > **rotor_P1**, configure the following settings:

Setting	Value
Option	Cylindrical Coordinates
Output Variables List	Pressure, Temperature, Total Pressure, Total Temperature, Velocity
Position Axial Comp.	0.211 [m]
Position Radial Comp.	0.2755 [m]
Position Theta Comp.	182 [degree]

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. Create an additional monitor point named `stator_P1`.

9. Under **Monitor Objects** > **Monitor Points and Expressions** > **stator_P1**, configure the following settings:

Setting	Value
Option	Cylindrical Coordinates
Output Variables List	Pressure, Temperature, Total Pressure, Total Temperature, Velocity
Position Axial Comp.	0.202 [m]
Position Radial Comp.	0.2755 [m]
Position Theta Comp.	178 [degree]

10. Click **OK**.

11. Save the simulation.

35.4.10. Setting the Execution Control

Here you will prepare the case for execution and initialize the solution with steady-state results. Instead of obtaining steady-state results by setting up and running the steady-state solution for this case, which has a double passage configuration, you will import a results file from another steady-state simulation, which happens to have a single passage (see Defining a Steady-state Case in CFX-Pre (p. 764)). Because that other simulation involves only a single passage, you will use replication control settings to apply those results to both passages in this simulation.

1. In the **Outline** tree view, right-click `Simulation Control` and select **Insert** > **Execution Control**.

2. Configure the following setting(s):

Tab	Setting	Value
Run Definition	Run Settings > **Double Precision**	 (Selected)
Initial Values	**Initial Values Specification**	(Selected)
	Initial Values Specification > **Initial Values**	 Initial Values 1
	Initial Values Specification > Initial Values > Initial Values 1 > **File Name**	 TimeBladeRowIni_001.res [a]

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Initial Values Specification > **Initial Values Control**	(Selected)
	Initial Values Specification > Initial Values Control > **Continue History From**	(Selected)

a. You can select a file to load by clicking *Browse* and selecting the appropriate file from your working directory, then clicking **Open**.

3. On the **Initial Values** tab, create an interpolation mapping item named `R1 Interpolation`.

4. Under **Initial Values Specification** > **Initial Values** > **Initial Values 1** > **Interpolation Mapping** > **R1 Interpolation**, configure the following settings:

Setting	Value
Source Location	R1
Target Location	R1
Replication Control	(Selected)
Replication Control > **Passages in 360**	42
Replication Control > **Total Num. Instances**	2

5. Create a second interpolation mapping item named `S1 Interpolation`.

6. Under **Initial Values Specification** > **Initial Values** > **Initial Values 1** > **Interpolation Mapping** > **S1 Interpolation**, configure the following settings:

Setting	Value
Source Location	S1
Target Location	S1
Replication Control	(Selected)
Replication Control > **Passages in 360**	36
Replication Control > **Total Num. Instances**	2

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Click **OK**.

35.4.11. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .

   The CFX-Solver input file `FourierBladeRowTime.def` is created by default.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, the **Solver Input File** is set.

2. If using stand-alone mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

35.4.12. Obtaining a Solution for the Time Integration Solution Method Case

At this point, CFX-Pre has been shut down, and the **Define Run** dialog box is displayed in CFX-Solver Manager. The initial values file has already been specified (see Setting the Execution Control (p. 856)). You will now obtain a solution to the CFD problem.

1. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. This can take a long time depending on your system. Eventually a dialog box is displayed.

   Note:

   - Before the simulation begins, the "Transient Blade Row Post-processing Information" summary in the CFX-Solver Output file will display the time step range over which the solver will accumulate the Fourier coefficients.
   - During the run, the "Fourier Transformation Information" summary in the CFX-Solver Output file displays the time step at which the full Fourier Transformation Model is activated.
   - Monitor points of similar values can be grouped together by right-clicking to the right of the **User Points** tab, selecting **New Monitor**, and clicking **OK**. In the **New Monitor** dialog box, you can name the new monitor point and select the variables to monitor in the **Monitor Properties** dialog box.
   - After the simulation has proceeded for some time, observe the periodic nature of the monitor point values.

   ---

2. When CFX-Solver is finished, clear the check box next to **Post-Process Results**.

3. Click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.4.12.1. Confirming Convergence Using Derived Variables

If you did not observe periodic monitor behavior within 20 passing periods of the stator, you should create statistical derived variables to verify that the solution *approaches* periodicity by following the procedure below:

1. Select **Workspace** > **New Monitor**.
2. Change the name to `Stator Pressure`.
3. Click **OK**.

   The **Monitor Properties** dialog box appears.
4. On the **Plot Lines** tab, select `USER POINTS` > `Pressure` > `stator_P1`.
5. Click **Apply**.

   A plot line of pressure (in Pa) at `stator_P1` versus accumulated time step appears.
6. Click the **Derived Variables** tab.
7. Create a derived variable named `Avg Over Passing Period`.

   The **Derived Variable Properties** dialog box appears.
8. Configure the following setting(s):

Setting	Value
Statistics > **Statistics Type**	Arithmetic Average
Statistics > **Interval Option**	Moving Interval
Statistics > Interval Definition > **Option**	Timesteps
Statistics > Interval Definition > **Number of Timesteps**	30

9. Click **OK**.
10. Return to the **Monitor Properties** dialog box.
11. In the **Workspace Derived Variables** list box, select `Avg Over Passing Period`.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12. Click **Apply**.

    A plot line of the average pressure at `stator_P1` over a moving passing period appears.

13. Create a derived variable named `Max Over Passing Period`.

    The **Derived Variable Properties** dialog box appears.

14. Configure the following setting(s):

Setting	Value
Statistics > **Statistics Type**	Maximum
Statistics > **Interval Option**	Moving Interval
Statistics > Interval Definition > **Option**	Timesteps
Statistics > Interval Definition > **Number of Timesteps**	30

15. Click **OK**.

16. Return to the **Monitor Properties** dialog box.

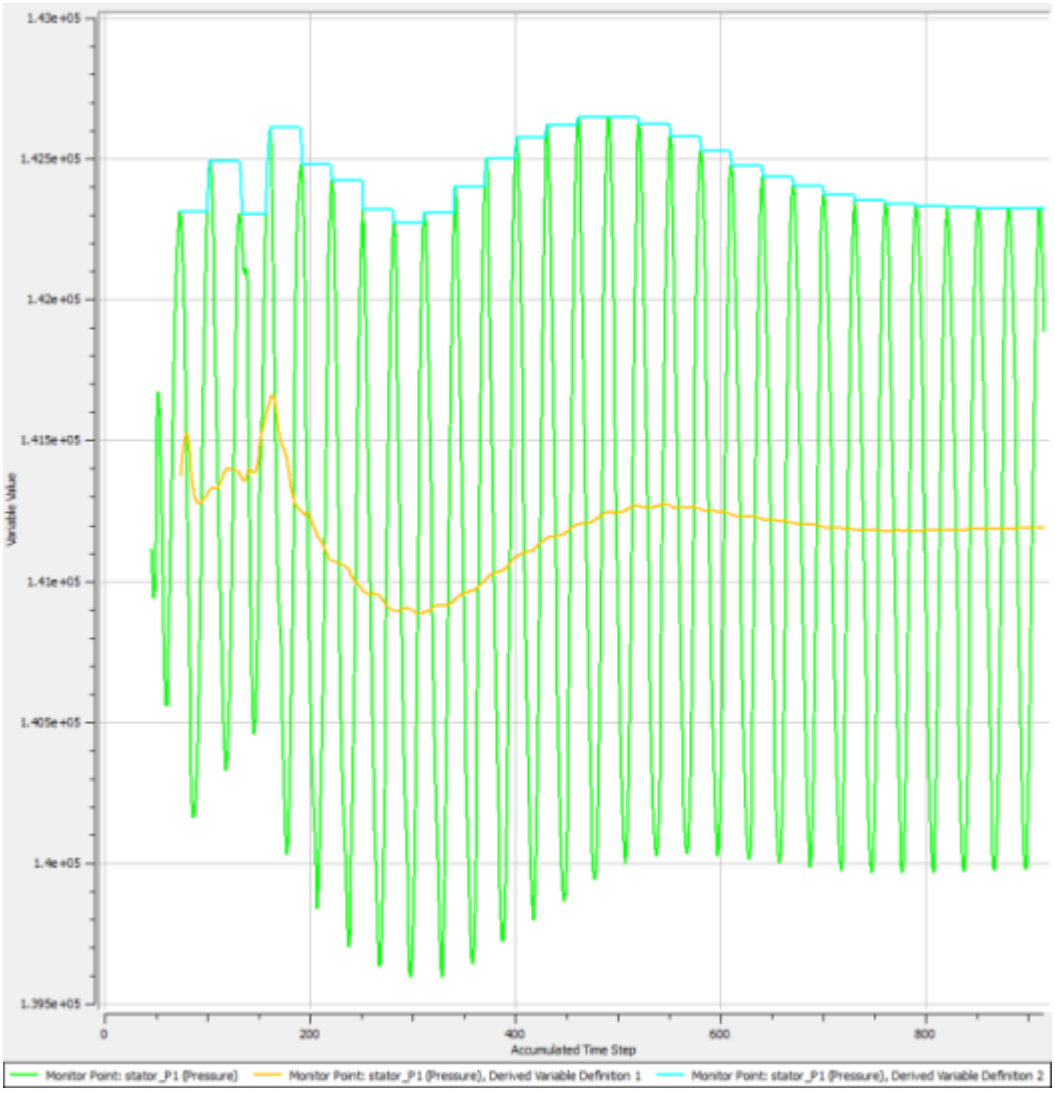
17. In the **Workspace Derived Variables** list box, select `Max Over Passing Period`.

18. Click **OK**.

    A plot line of the maximum pressure at `stator_P1` over a moving passing period appears.

Both derived variable plot lines should appear to approach stable values. If you had set the simulation to run for 25 passing periods, the plot monitor would resemble the image below:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



35.5. Defining and Obtaining a Solution for the Harmonic Balance Solution Method Case

In this part of the tutorial, you will modify the Time Integration solution method case that was set up in the previous part of the tutorial in order to use the Harmonic Balance solution method. As in the previous part, the result from the steady-state simulation is used as an initial guess to speed convergence.

35.5.1. Opening the Existing Case

This step involves opening the Time Integration simulation and saving it to a different location.

1. Ensure that the following tutorial input files are in your working directory:
   - `FourierBladeRowTime.cfx`
   - `TimeBladeRowIni_001.res`

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Set the working directory and start CFX-Pre if it is not already running.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

3. If the Time Integration simulation is not already opened, then open `FourierBladeRow Time.cfx`.

4. Save the case as `FourierBladeRowHarmonic.cfx` in your working directory.

There are many common steps between setting up Time Integration and Harmonic Balance Transient Rotor-stator cases, including using the same Fourier Transformation pitch change model. Here, only the differences are highlighted.

35.5.2. Modifying the Transient Blade Row Model

In this section, you will change the transient method to `Harmonic Balance`.

1. In the **Outline** tree view, edit `Flow Analysis 1` > `Transient Blade Row Models`.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Transient Method** > **Option**	Harmonic Balance
	Transient Method > **Number of Modes**	3

   Note that, in the **Transient Blade Row Models** details view, on the **Basic Settings** tab, the settings for **Time Period**, **Time Steps** and **Time Duration** have disappeared.

3. Click **OK**.

There is no need to specify the period involved when selecting `Harmonic Balance` in combination with the Fourier Transformation model. The period in each domain is obtained by the blade passing frequency, which is proportional to the blade count in the adjacent row.

35.5.3. Modifying the Solver Control

1. Click *Solver Control* .

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Transient Scheme > **Option**	Harmonic Balance
	Convergence Control	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Min. Iterations**	1
	Convergence Control > **Max. Iterations**	 200
	Convergence Control > Fluid Timescale Control > **Timescale Control**	 Physical Timescale
	Convergence Control > Fluid Timescale Control > **Physical Timescale**	 ((1 [rev] / 42) / 3500 [rev min^-1]) / 20

3. Click **OK**.

The physical timescale is usually set as a fraction of the smallest time period involved. In this case 1/20th of the rotor blade passing period is used as the physical timescale. A larger physical timescale speeds convergence but can lower stability. In general, if the solution becomes unstable or does not converge, you should use a smaller fraction when computing the physical timescale.

35.5.4. Modifying the Output Control

In preparation for a Harmonic Forced Response analysis during postprocessing, you will set CFX-Pre to export a harmonic of the pressure on the rotor's surface.

In order to do this, you will indicate the frequency of excitation of the rotor by specifying an engine order value, which represents the number of disturbances (or pulses) per revolution of the engine.

In this case, any given rotor blade is excited by 36 disturbances per engine revolution: one disturbance from each of the 36 stator blades.

1. Click *Output Control* .
2. Click the **Export** tab.
3. Under **Export Results**, click *Add new item* .

   The **Insert Export Results** dialog box appears.
4. Accept the default name by clicking **OK**.
5. Under **Export Results** > **Export Results 1** > **Export Format**, ensure that **Option** is set to `CFX CSV BC Profile` and select **Include Topology**.

   When selected, the **Include Topology** setting causes connectivity data to be included so that the resulting surface data can be used as a locator in CFD-Post, for example for a contour plot.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Under **Export Results** > **Export Results 1** > **Export Surface**, click *Add new item* .

   The **Insert Export Surface** dialog box appears.

7. Accept the default name by clicking **OK**.

8. Configure the following settings under **Export Results** > **Export Results 1** > **Export Surface** > **Export Surface 1**:

Setting	Value
Option	Harmonic Forced Response
Location Type > **Option**	`Boundary`
Location Type > **Boundary**	R1 Blade
Excitation Frequency > **Option**	Engine Order
Excitation Frequency > **Engine Order**	36

9. Click **OK**.

Later in this tutorial, when the CFX-Solver has finished its calculations, the solver will write file `ExportResults1_<a number>.csv` where `ExportResults1` is the given name of the surface (under "`[Name]`", as shown below) and *<a number>* is the applicable outer loop iteration number (padded with leading zeros to make 6 digits).

After the exported file is written, its contents will look similar to the following:

```
[Name]
Export Surface 1

[Parameters]
Ncompt = 1
Nnodes = 3680
Rotation Axis From = 0.0000000 [m], 0.0000000 [m], 0.0000000 [m]
Rotation Axis To   = 0.0000000 [m], 0.0000000 [m], 1.0000000 [m]
Rotating Speed     = 366.51914 [s^-1 rad]
Frequency          = 2100.0000 [Hz]
Engine Order       = 36

[Spatial Fields]
x, y, z

[Data]
x [ m ], y [ m ], z [ m ], Real Pressure [ kg m^-1 s^-2 ], Imaginary Pressure [ kg m^-1 s^-2 ], Node Number
-2.44994841E-001,  1.58908837E-003,  2.15076035E-001, -5.99750474E+001, -1.75136536E+002,         0
-2.45487356E-001,  1.59036453E-003,  2.15079039E-001, -7.26132011E+001, -2.26077478E+002,         1
-2.46203489E-001,  1.59221122E-003,  2.15083406E-001, -1.52475786E+002, -1.82970865E+002,         2
-2.47237201E-001,  1.59486280E-003,  2.15089716E-001, -2.65408934E+002, -3.06071903E+001,         3
-2.48715507E-001,  1.59862583E-003,  2.15098751E-001, -2.95003528E+002,  5.97159931E+001,         4
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
-2.50804203E-001,  1.60388403E-003,  2.15111541E-001, -2.81566347E+002,  1.12160789E+002,         5
.
.
.
-2.97890454E-001, -3.19597847E-002,  2.58500217E-001, -4.62603071E+001, -1.13633158E+002,      3675
-2.97474064E-001, -3.56276826E-002,  2.57683679E-001, -2.49212458E+001, -1.51912000E+001,      3676
-2.97497251E-001, -3.54335493E-002,  2.57406431E-001, -2.59207308E+001, -2.29944420E+001,      3677
-2.97524623E-001, -3.52029722E-002,  2.57077487E-001, -2.78774687E+001, -3.31765348E+001,      3678
-2.97558085E-001, -3.49189960E-002,  2.56672866E-001, -2.92674110E+001, -4.02254086E+001,      3679

[Faces]
       148,       147,      1260,      1261
      1261,      1260,        84,        85
       149,       148,      1261,      1262
      1262,      1261,        85,        86
       150,       149,      1262,      1263
      1263,      1262,        86,        87
       151,       150,      1263,      1264
      1264,      1263,        87,        88
.
.
.
```

Note that the connectivity data consists of the `Node Number` data column and the `[Faces]` section.

35.5.5. Modifying Execution Control

1. Click *Execution Control* .
2. Configure the following setting(s):

Tab	Setting	Value
Run Definition	Input File Settings > **Solver Input File**	FourierBladeRowHarmonic.def [a]

a. You do not need to set the full path unless you are saving the solver file somewhere other than the working directory.

3. Confirm that the rest of the execution control settings are set appropriately.
4. Click **OK**.

35.5.6. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .

   The CFX-Solver input file `FourierBladeRowHarmonic.def` is created by default.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, the **Solver Input File** is set.

2. If using stand-alone mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

35.5.7. Obtaining a Solution for the Harmonic Balance Solution Method Case

At this point, CFX-Pre has been shut down, and the **Define Run** dialog box is displayed in CFX-Solver Manager. The initial values file has already been specified (see Setting the Execution Control (p. 856)). You will now obtain a solution to the CFD problem.

1. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. This can take a long time depending on your system. Eventually a dialog box is displayed.

2. When CFX-Solver is finished, clear the check box next to **Post-Process Results**.

3. Click **OK**.

35.6. Postprocessing the Transient Rotor-stator Solution

The postprocessing steps outlined here can be equally used on either of the Transient Rotor-stator solutions obtained in Defining and Obtaining a Solution for the Time Integration Solution Method Case (p. 846) and Defining and Obtaining a Solution for the Harmonic Balance Solution Method Case (p. 861). The results should be similar and the differences between the two will be minimized by comparing a time-resolved transient case to a modal-resolved harmonic balance case (that is, a case where increasing the number of modes or time planes does not result in solution change).

In a transient blade row run, flow field variables are compressed using the Fourier coefficient method. These variables are accumulated at the end of the simulation. This enables you to navigate through any time instance, within the common period, without having to load multiple transient results files. By default CFD-Post displays results corresponding to the end the simulation.

To get started, follow these steps:

1. Start CFD-Post and load results from either `FourierBladeRowTime_001.res` or `FourierBladeRowHarmonic_001.res`.

2. When CFD-Post opens, if you see the **Domain Selector** dialog box, ensure that all the domains are selected, then click **OK** to load the results from these domains.

3. If you see a message regarding transient blade row postprocessing, click **OK**.

35.6.1. Creating a Turbo Surface

Create a turbo surface to be used for making plots:

1. Click the **Turbo** tab.

2. If you see the **Turbo Initialization** dialog box, click **Yes**, otherwise click the **Initialize All Components** button, which is visible initially by default, or after double-clicking the **Initialization** object in the **Turbo** tree view.

3. Select **Insert** > **Location** > **Turbo Surface**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Change the name to `Span 50`.
5. Configure the following setting(s):

Tab	Setting	Value
Geometry	Definition > **Method**	Constant Span
	Definition > **Value**	0.5

6. Click **Apply**.
7. Turn off the visibility of `Span 50` by clearing its check box in the **Outline** tree view.

35.6.2. Creating a Vector Plot

1. Click **Insert** > **Vector** and accept the default name.
2. Configure the following setting(s):

Tab	Setting	Value
Geometry	Definition > **Locations**	Span 50
	Definition > **Variable**	Velocity in Stn Frame

3. Click **Apply**.

   The vector plot shows `Velocity in Stn Frame` values corresponding to the end of a common period.

   Now you will align the rotor with the stator, as it was in the solver input file.

4. Click *Timestep Selector* .
5. Select the 1st time step.
6. Click **Apply** to load the time step, and then click **Close** to exit the dialog box.

   The rotor blades move to their starting positions.

35.6.3. Creating a Contour Plot

1. Turn off the visibility of `Vector 1`.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click **Insert** > **Contour** and accept the default name.

3. Configure the following setting(s):

Tab	Setting	Value
Geometry	**Locations**	Span 50
	Variable	Pressure
	Range	Local
	# of Contours	21

4. Click **Apply**.

The contour plot shows `Pressure` values corresponding to the specified time step.

35.6.4. Creating a Variable Time Chart

In this section, you will compute and plot the magnitude of the forces that the flow applies on the rotor blade. For a transient blade row case, CFD-Post automatically reconstructs variables for the flow solution time based on the last time step. Intermediate time steps for time instances in the common period are located in the **Timestep Selector**.

For the time integration solution method case, you set 35 time steps per stator blade passing period in Setting up a Transient Blade Row Model (p. 853). Because there are six stator blade passing periods in a common period, the total number of intermediate time steps in the common period is 210. Because the solver has reconstructed results over four common periods, there are 840 time steps. If you are currently postprocessing the time integration solution method case, reduce the total number of time steps over four periods from 840 to 280, as follows, to speed generation of the time chart:

1. Click *Timestep Selector* .

2. Configure the following setting(s):

Tab	Setting	Value
Timestep Selector	**Timestep Sampling**	Uniform
	Number of Timesteps	70

3. Click **Apply**.

   The **Timestep Selector** now shows a total of 280 steps over four common periods (shown under the *Phase* column).

Compute the net force on the pair of rotor blades in the rotor domain (that is, the net force on region `R1 Blade`):

1. Select **Insert** > **Expression**.

2. In the **Insert Expression** dialog box, type `forces on rotor blades`.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click **OK**.

4. Set **Definition** to `sqrt(force_x()@ R1 Blade ^2 + force_y()@ R1 Blade ^2 + force_z()@ R1 Blade ^2)`

5. Click **Apply** to create the expression.

Create a transient chart showing force:

1. Select **Insert** > **Chart** and accept the default name.

2. Configure the following setting(s):

Tab	Setting	Value
General	**XY - Transient or Sequence**	(Selected)
Data Series	Series 1 > Data Source > **Expression**	(Selected)
	Series 1 > Data Source **> Expression**	forces on rotor blades

3. If you are postprocessing the time integration solution method case, configure the following settings to plot only the last common period:

Tab	Setting	Value
X Axis	Axis Range > **Determine ranges automatically**	(Cleared)
	Axis Range > **Min**	0.00857143
	Axis Range > **Max**	0.0114286

4. Click **Apply**.

   The chart displayed in the **Chart Viewer** shows the net force on the modeled pair of rotor blades as a function of time.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.6.5. Viewing the Harmonic Forced Response Results

1. Select **File** > **Import** > **Import Surface, Line or Point Data**.

   The **Import Surface, Line or Point Data** dialog box appears.

2. Click *Browse* .

3. Select the Export Results (`.csv`) file from the run directory for the Harmonic Analysis: `FourierBladeRowHarmonic_001`.

4. Click **Open**.

5. In the **Import Surface, Line or Point Data** dialog box, ensure that **Import As** is set to `Surface or Line`.

6. Click **OK**.

   The surface appears in the **3D Viewer**.

You can use the surface as a locator for a contour plot of `Real Pressure` or `Imaginary Pressure`.

35.6.6. Setting up Data Instancing Transformations

Data instancing uses Fourier interpolation to reproduce other passages that are not included in a simulation. In this section, you will use data instancing to replicate enough passages in the rotor and stator domains to recover unity pitch ratio across the rotor-stator interface.

1. From the **Outline** tree view, edit `R1`.

2. In the **Data Instancing** tab, set **Number of Data Instances** to 7.

3. Click **Apply**.

   On the **3D Viewer** tab, CFD-Post displays 14 rotor blades.

4. From the **Outline** tree view, edit `S1`.

5. In the **Data Instancing** tab, set **Number of Data Instances** to 6.

6. Click **Apply**.

   On the **3D Viewer** tab, CFD-Post displays 12 stator blades.

CFD-Post now computes `forces on rotor blades` on all 14 blade instances in `R1 Blade`. However, `Chart 1` will not reflect this change automatically. To update `Chart 1`:

- At the top of the **Chart Viewer**, click the **Refresh** button.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.6.7. Setting up Graphical Instancing Transformations

In this section, you will use graphical instancing to visually complete the full wheel. You will make copies of the wheel segment that was completed using data instancing.

1. From the **Outline** tree view, turn off the visibility of `Wireframe`.
2. Edit R1.
3. Configure the following setting(s):

Tab	Setting	Value
Instancing	**Number of Graphical Instances**	3
	Instance Definition	Custom
	Passages per Component	14

4. Click **Apply**.
5. Edit S1.
6. Configure the following setting(s):

Tab	Setting	Value
Instancing	**Number of Graphical Instances**	3
	Instance Definition	Custom
	Passages per Component	12

7. Click **Apply**.

   The Graphics Instancing feature makes graphical copies of objects and places them at an angular position computed using the **Number of Passages** and **Number of Passages per Component** on the **Instancing** panel. To complete the full wheel, you replicated the 1/3 wheel sector, which was obtained using data instancing, three times. On the **3D Viewer** tab, CFD-Post displays the pressure plot on `Span 50` over the full wheel.

35.6.8. Animating the Movement of the Rotor Relative to the Stator

With the **Timestep Selector** set to time step `0`, you will make an animation showing the relative motion starting from this time step and lasting for one stator blade passing period.

1. Click the **3D Viewer** tab.
2. Position the geometry for the animation by right-clicking on a blank area in the viewer and selecting **Predefined Camera** > **View From -X**.
3. Click *Animation* .

   The **Animation** dialog box appears.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Set **Type** to **Keyframe Animation**.

5. Click *New* to create `KeyframeNo1`.

6. Select `KeyframeNo1`, then set **# of Frames** to `70`, then press **Enter** while in the **# of Frames** box.

   Tip:

   Be sure to press **Enter** and confirm that the new number appears in the list before continuing.

   This will place 70 intermediate frames between the keyframes, for a total of 72 frames.

7. Use the **Timestep Selector** to load time step `70` and then close the dialog box.

8. In the **Animation** dialog box, click *New* to create `KeyframeNo2`.

9. Click *More Animation Options* to expand the **Animation** dialog box.

10. Select **Save Movie**.

11. Specify a filename for the movie.

12. Set **Format** to `MPEG1`.

13. Click *To Beginning* to rewind the active keyframe to `KeyframeNo1`.

    Note:

    The active keyframe is indicated by the value appearing in the **F:** field in the middle of the **Animation** dialog box. In this case it will be `1`.

    Wait for CFD-Post to finish loading the objects for this frame before proceeding.

14. Click *Save animation state* and save the animation to a file. This will enable you to quickly restore the animation in case you want to make changes. Animations are not restored by loading ordinary state files (those with the `.cst` extension).

15. Click *Play the animation* .

    Note:

    It takes a while for the animation to be completed. To view the movie file, you will need to use a media player that supports the MPEG format.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

From the animation and plots, you will see that the flow is continuous across the interface. This is because CFD-Post is capable of interpolating the flow field variables to the correct time and position using the computed Fourier coefficients.

16. When you have finished, close the **Animation** dialog box and then close CFD-Post, saving the animation state at your discretion.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 36: Fourier Transformation Method for a Blade Flutter Case

This tutorial includes:

36.1. Tutorial Features

36.2. Overview of the Problem to Solve

36.3. Preparing the Working Directory

36.4. Defining and Obtaining a Solution for the Steady-state Case

36.5. Defining and Obtaining a Solution for the Time Integration Solution Method Case

36.6. Defining and Obtaining a Solution for the Harmonic Balance Solution Method Case

36.7. Postprocessing the Blade Flutter Solution

36.1. Tutorial Features

In this tutorial you will learn about:

Component	Feature	Details
CFX-Pre	User Mode	General mode
	Analysis Type	Transient Blade Row
		Fourier Transformation Pitch Change Model
		Time Integration Solution Method
		Harmonic Balance Solution Method
	Fluid Type	Air Ideal Gas
	Domain Type	Multiple Domains
		Rotating Frame of Reference
	Turbulence Model	k-Epsilon
	Heat Transfer	Total Energy
	Boundary Conditions	Inlet (Subsonic)
		Outlet (Subsonic)
		Wall (Counter Rotating)
	Mesh Motion	Periodic Motion

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Component	Feature	Details
		Sliding Mesh
CFD-Post	Plots	Contour
		Isosurface
		Vectors
		Transient Blade Row Expansion

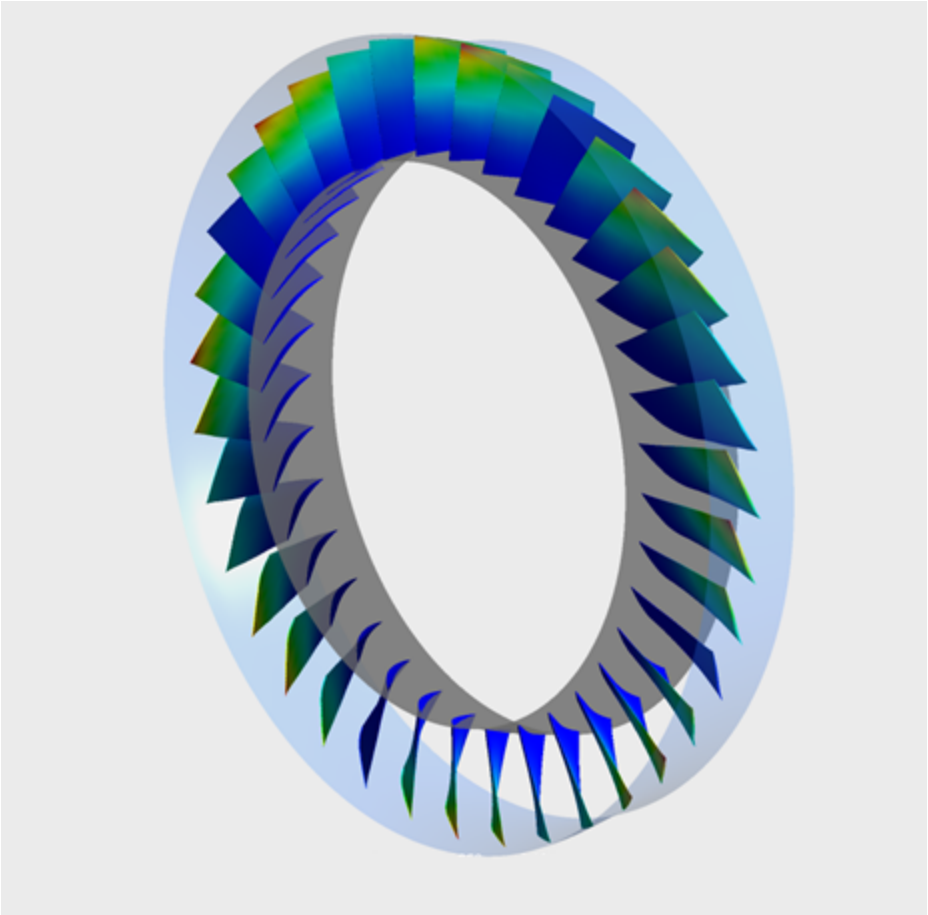
36.2. Overview of the Problem to Solve

The goal of this tutorial is to set up transient blade row blade flutter simulations using both time integration and harmonic balance transient methods in combination with the Fourier Transformation pitch change model. An integral step of blade flutter modeling is the calculation of the aerodynamic damping factor as a function of the possible nodal diameters (radial lines of symmetry around the circumference) for the component being modeled. When the number of passages in the component is an integer multiplier of the nodal diameter (ND), the number of blade passages required to model a given nodal diameter can be substantially reduced by using the rotational periodic boundary conditions. This eliminates the need to model the full component. By using the Fourier Transformation model, the number of passages required can be kept to a minimum of two for all nodal diameters. The Harmonic balance transient method and the Fourier Transformation pitch change model can each cut solution time independently; when combined, the solution time is further reduced.

This tutorial uses an axial compressor to illustrate the basic concepts of setting up, running, and monitoring a transient blade row calculation with blade motion in CFX. The full geometry consists of one rotor containing 36 blades as seen in Figure 36.1: Single-row Reference Case Containing 36 Blades (p. 877) below.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 36.1: Single-row Reference Case Containing 36 Blades



For non-zero nodal diameters, there is a finite inter-blade phase angle (IBPA) between neighboring blades. This phase difference between the blades is defined as:

$$IBPA=\frac{2\pi ND}{N_{BL}}$$

where N_{BL} is the number of blades and $0 \le ND \le N_{BL}/2$ for an even number of blades and $0 \le ND \le (N_{BL}-1)/2$ for an odd number of blades.

The following table compares the number of passages per component required to model a given nodal diameter when using periodic boundary conditions or the Fourier Transformation approach:

Nodal Diameter	IBPA [deg]	Number of Passages per Component to Model	
		Reference Case (Rotational Periodicity)	Fourier Transformations
0	0	1	2
1	10	36	2
2	20	18	2

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Nodal Diameter	IBPA [deg]	Number of Passages per Component to Model	
		Reference Case (Rotational Periodicity)	Fourier Transformations
3	30	12	2
4	40	9	2
5	50	36	2
6	60	6	2
7	70	36	2
8	80	9	2
9	90	4	2

For this tutorial, you will model a nodal diameter of four using the Fourier transformation approach with only two passages. The equivalent model using the periodic boundary conditions (reference case) requires nine passages, that is, a quarter of the original rotor.

The machine is rotating at 1800 [rad s^-1]. The inlet boundary condition is modeled as Total Pressure and Total Temperature in the stationary frame, with a specified flow direction in cylindrical components. The outlet boundary condition is set to an average static pressure of 138 [kPa], varying in the radial direction only. The inlet boundary profile is provided in a `.csv` file.

The blade vibration is modeled as forced periodic motion at a fixed frequency with a specified inter-blade phase angle. The frequency and displacement profile (mode shape) are obtained from cyclic symmetry calculations in Ansys Mechanical using a single blade model, and exported to a `.csv` file. For this case the vibration frequency is 1152.13 [Hz], and the maximum displacement for the mode shape is 0.00129 [m]. In order to use this single blade mode shape for multiple blade flow simulations, the profile must be replicated around the machine axis. This replicated profile contains a sector number identifying every copied section from the original profile. This sector number increases following the right hand rule around the machine axis. The sector number information can be used to determine the direction of the phase shift; that is, it can be used to determine whether the blade displacement is initiated on the blade with the higher or lower theta position.

The surface of revolution mesh motion boundary condition is used at the shroud to model the sliding of the mesh along the surface.

The vibration of the blades with IBPA results in a traveling wave pattern. For a model that has a rotating component:

- A Forward Traveling Wave (FTW) is a wave traveling in the direction of machine rotation, and is associated with a positive IBPA value.
- A Backward Traveling Wave (BTW) is a wave traveling in the direction opposite to machine rotation, and is associated with a negative IBPA value.

This tutorial models IBPA=+40°, which represents Nodal Diameter=4 for a forward traveling wave.

The hub surface nodes are set to be stationary. The shroud surface nodes are allowed to follow the blade displacement.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- Running Ansys CFX Tutorials Using Ansys Workbench (p. 34)
- Changing the Display Colors (p. 37)

36.3. Preparing the Working Directory

1. Create a working directory.

   Ansys CFX uses a working directory as the default location for loading and saving files for a particular session or project.

2. Download the `fourier_blade_flutter.zip` file here.

3. Unzip `fourier_blade_flutter.zip` to your working directory.

   Ensure that the following tutorial input files are in your working directory:

   - `R37ATM_60k.gtm`
   - `R37_inlet.csv`
   - `R37_mode1_1p.csv`

4. Set the working directory and start CFX-Pre.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

36.4. Defining and Obtaining a Solution for the Steady-state Case

This section describes the steady-state simulation setup for blade flutter in CFX-Pre and subsequent use of CFX-Solver Manager to obtain a solution. Although, the effect of mesh motion on a steady-state run is minimal, it will provide you with the initial conditions for the Fourier Transformation Blade Flutter case.

1. In CFX-Pre, select **File** > **New Case**.
2. Select **General** and click **OK**.
3. Select **File** > **Save Case As**.
4. Set **File name** to `FourierBladeFlutterIni.cfx`.
5. Click **Save**.

36.4.1. Importing the Mesh

1. In the **Outline** tree view, right-click `Mesh` and select **Import Mesh** > **CFX Mesh**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Import Mesh** dialog box appears.

2. Configure the following setting(s):

Setting	Value
Filename	R37ATM_60k.gtm

3. Click **Open**.

   This file contains a single passage mesh. The Fourier Transformation method requires two passages for any IBPA number.

4. In the **Outline** tree view, right-click **Mesh** > `R37ATM_60k.gtm` and select **Transform Mesh**.

   The **Mesh Transformation Editor** dialog box appears.

5. Set **Transformation** to **Turbo Rotation**.

6. Configure the following setting(s):

Setting	Value
Rotation Option	Principal Axis
Axis	Z
Passages per Mesh	1
Passages to Model	2
Passages in 360	36

7. Click **Apply** and close the **Mesh Transformation Editor** dialog box.

36.4.2. Expanding Profile Data

The profile describing the frequency and blade mode shape for one blade is provided with this tutorial. In preparation for a two-passage Fourier Transformation setup, you will expand this profile and initialize it to be used for boundary condition specifications.

1. Select **Tools** > **Edit Profile Data**.

   The **Edit Profile Data** dialog box appears.

2. Under **Source Profile**, click *Browse* .

   The **Select Profile Data File** dialog box appears.

3. From your working directory, select `R37_mode1_1p.csv` and click **Open**.

4. Set **Write to Profile** to `R37_mode1_36p.csv`.

5. Ensure that **Initialize New Profile After Writing** is selected so that the mode1 profile data will be automatically initialized using the expanded profile.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. In the **Transformations** frame, click *Add new item* , set **Name** to `Transformation 1`, and click **OK**.

7. Configure the following setting(s):

Setting	Value
Transformation 1 > **Option**	Expansion
Transformation 1 > Expansion Definition > **Rotation Option**	Principal Axis
Transformation 1 > Expansion Definition > **Axis**	Z
Transformation 1 > Expansion Definition > **Passages in Profile**	1
Transformation 1 > Expansion Definition > **Passages in 360**	36
Transformation 1 > Expansion Definition > **Expansion Option**	Expand to Full Circle
Transformation 1 > Expansion Definition > **Theta Offset**	0 [degree]

8. Click **OK**.

You have expanded the profile data coverage from one passage to 36 passages. The expanded profile file is divided into sectors, as indicated by an extra data column named `Sector Tag`. Sector 1 contains the original data, which covers one passage. Sector 2 contains data for a second passage. The coordinates and displacement vector components in sector 2 are rotated appropriately compared to the corresponding data in sector 1.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.4.3. Initializing Profile Data

The `inflow` and `mode1` functions are defined using profiles found in the `.csv` files in your working directory.

You have already initialized the profile data for the `mode1` functions (`meshdisptot x(Initial X, Initial Y, Initial Z)`, `meshdisptot y(Initial X, Initial Y, Initial Z)`, `meshdisptot z(Initial X, Initial Y, Initial Z)`) during the profile expansion process, due to use of the **Initialize New Profile After Writing** option in the **Edit Profile Data** dialog box.

You will now initialize the profile data for the `inflow` functions (`Total Pressure(r)`, `Total Temperature(r)`, `Velocity Axial(r)`, `Velocity Circumferential(r)`, and `Velocity Radial(r)`):

1. Select **Tools** > **Initialize Profile Data**.

   The **Initialize Profile Data** dialog box appears.

2. Beside **Profile Data File**, click *Browse* .

   The **Select Profile Data File** dialog box appears.

3. From your working directory, select `R37_inlet.csv`.

4. Click **Open**.

5. Click **OK**.

   The `inflow` profile data is read into memory.

Note:

After profile data has been initialized from a file, the profile data file should not be deleted or otherwise removed from its directory. By default, the full file path to the profile data file is stored in CFX-Pre, and the profile data file is read directly by CFX-Solver each time the solver is started or restarted.

36.4.4. Creating the Domain

The fluid domain used for this simulation contains Air as an Ideal Gas. In addition to this, you will also set mesh motion for the blades.

1. Select **Insert** > **Domain** from the main menu.

   The **Insert Domain** dialog box appears.

2. Set **Name** to R1 and click **OK**.

3. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Basic Settings	Location and Type > **Location**	 Entire Rotor Passage
	Fluid and Particle Definitions... > Fluid 1 > **Material**	 Air Ideal Gas
	Domain Models > Pressure > **Reference Pressure**	 0 [atm]
	Domain Models > Domain Motion > **Option**	 Rotating
	Domain Models > Domain Motion > **Angular Velocity**	 -1800 [radian s^-1][a]
	Domain Models > Domain Motion > **Alternate Rotation Model**	 (Selected)
	Domain Models > Mesh Deformation > **Option**	 Regions of Motion Specified
	Domain Models > Mesh Deformation > **Displacement Rel. To**	 Initial Mesh
	Domain Models > Mesh Deformation > Mesh Motion Model	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Option**	Displacement Diffusion
Fluid Models	Heat Transfer > **Option**	 Total Energy
	Turbulence > **Option**	 Shear Stress Transport
	Turbulence > **Wall Function**	 Automatic
	Turbulence > Advanced Turbulence Control > **Reattachment Modification**	 (Selected)
	Turbulence > Advanced Turbulence Control > Reattachment Modification > **Option**	 Reattachment Production
a. Notice that a negative angular velocity is used because the machine rotates clockwise with respect to the axis of rotation		

4. Click **OK**.

36.4.5. Creating the Boundaries

36.4.5.1. Inlet Boundary

1. Create a new boundary named `R1 Inlet`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Inlet
	Location	Entire Rotor INFLOW
	Frame Type	Stationary

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Profile Boundary Conditions > **Use Profile Data**	 (Selected)
	Profile Boundary Setup > **Profile Name**	 Inflow
	Profile Boundary Setup > **Generate Values**	 (Click) This causes profile values to be applied at the nodes on the inlet boundary. It also causes entries to be made in the **Boundary Details** tab. In order to later reset the velocity values at the inlet to match those that were originally read from the BC Profile file, revisit the **Basic Settings** tab for this boundary and click **Generate Values**.
Boundary Details	Mesh Motion > **Option**	 Stationary
	Mass and Momentum > **Option**	 Stat. Frame Tot. Press.
	Mass and Momentum > **Relative Pressure**	 Inflow.Total Pressure(r)[a]
	Flow Direction	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Option**	Cylindrical Components
	Flow Direction > **Axial Component**	Inflow.Velocity Axial(r)[a]
	Flow Direction > **Radial Component**	Inflow.Velocity Radial(r)[a]
	Flow Direction > **Theta Component**	Inflow.Velocity Circumferential(r)[a]
	Turbulence > **Option**	Medium (Intensity = 5%)
	Heat Transfer > **Option**	Stat. Frame Total Temp.
	Heat Transfer > **Stat. Frame Tot. Temp.**	Inflow.Total Temperature(r)[a]
a. Click the *Enter Expression* icon to specify the CEL expression.		

3. Click **OK**.

36.4.5.2. Outlet Boundary

1. Create a new boundary named `R1 Outlet`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Outlet
	Location	Entire Rotor OUTFLOW

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Frame Type	Stationary
Boundary Details	Mesh Motion > **Option**	 Stationary
	Mass and Momentum > **Option**	 Average Static Pressure
	Mass and Momentum > **Relative Pressure**	 138 [kPa]
	Mass and Momentum > **Pres. Profile Blend**	 1
	Pressure Averaging > **Option**	 Radial Equilibrium
	Pressure Averaging > Radial Reference Position > **Option**	 Specified Radius
	Pressure Averaging > Radial Reference Position > **Specified Radius**	 0.215699 [m]

3. Click **OK**.

36.4.5.3. Wall Boundaries

The hub, shroud and blade of the fluid region all require wall boundaries.

1. Create a new boundary named R1 Hub.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Wall
	Location	Entire Rotor HUB
	Frame Type	Rotating
Boundary Details	Mesh Motion	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Option**	Stationary

3. Click **OK**.
4. Create a new boundary named `R1 Shroud`.
5. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Wall
	Location	Entire Rotor SHROUD
	Frame Type	Rotating
Boundary Details	Mesh Motion > **Option**	Surface of Revolution
	Mesh Motion > Axis Definition > **Option**	Coordinate Axis
	Mesh Motion > Axis Definition > **Rotation Axis**	Global Z
	Mass and Momentum > **Wall Velocity**	(Selected)
	Mass and Momentum > Wall Velocity > **Option**	Counter Rotating Wall

6. Click **OK**.
7. Create a new boundary named `R1 Blade`.
8. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Wall

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Location	Entire Rotor BLADE
	Frame Type	Rotating
	Profile Boundary Conditions > **Use Profile Data**	 (Selected)
	Profile Boundary Setup > **Profile Name**	 mode1
Boundary Details	Mesh Motion > **Option**	 Stationary

9. Click **OK**.

36.4.6. Creating Domain Interfaces

You will now create a pair of fluid-fluid domain interfaces along the tip gap for each blade.

1. Click **Insert** > **Domain Interface** and, in the dialog box that appears, set **Name** to `R1 Blade Tip Gap` and click **OK**.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Interface Type**	Fluid Fluid
	Interface Side 1 > **Domain (Filter)**	 R1
	Interface Side 1 > **Region List**	 Rotor SHROUD TIP GGI SIDE 1
	Interface Side 2 > **Domain (Filter)**	 R1
	Interface Side 2 > **Region List**	 Rotor SHROUD TIP GGI SIDE 2
	Interface Models > **Option**	 General Connection
Mesh Connection	Mesh Connection Method	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Mesh Connection > **Option**	 GGI

3. Click **OK**.
4. Click **Insert** > **Domain Interface** and, in the dialog box that appears, set **Name** to `R1 Blade Tip Gap 2` and click **OK**.
5. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Interface Type**	Fluid Fluid
	Interface Side 1 > **Domain (Filter)**	 R1
	Interface Side 1 > **Region List**	 Rotor SHROUD TIP GGI SIDE 1 2
	Interface Side 2 > **Domain (Filter)**	 R1
	Interface Side 2 > **Region List**	 Rotor SHROUD TIP GGI SIDE 2 2
	Interface Models > **Option**	 General Connection
Mesh Connection	Mesh Connection Method > Mesh Connection > **Option**	 GGI

6. Click **OK**.
7. Click **Insert** > **Domain Interface** and, in the dialog box that appears, set **Name** to `R1 to R1 Periodic` and click **OK**.
8. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Basic Settings	**Interface Type**	Fluid Fluid
	Interface Side 1 > **Domain (Filter)**	R1
	Interface Side 1 > **Region List**	Rotor PER1
	Interface Side 2 > **Domain (Filter)**	R1
	Interface Side 2 > **Region List**	Rotor PER2 2
	Interface Models > **Option**	Rotational Periodicity
	Interface Models > Axis Definition > **Option**	Coordinate Axis
	Interface Models > Axis Definition > **Rotation Axis**	Global Z
Mesh Connection	Mesh Connection Method > Mesh Connection > **Option**	GGI

9. Click **OK**.

In addition to the two fluid-fluid interfaces, the Fourier Transformation method requires a domain interface between the two passages. This interface method will be used by the Fourier Transformation method to collect information about the flow. The data will then be transferred back to the rotational periodic boundaries with the proper time lag.

Note:

The periodic and sampling interfaces must use the GGI mesh connection.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10. Click **Insert** > **Domain Interface** and, in the dialog box that appears, set **Name** to `R1 Sampling Interface` and click **OK**.

11. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Interface Type**	Fluid Fluid
	Interface Side 1 > **Domain (Filter)**	R1
	Interface Side 1 > **Region List**	Rotor PER2
	Interface Side 2 > **Domain (Filter)**	R1
	Interface Side 2 > **Region List**	Rotor PER1 2
	Interface Models > **Option**	General Connection
Mesh Connection	Mesh Connection Method > Mesh Connection > **Option**	GGI

12. Click **OK**.

Fourier Transformation periodic boundary condition mappings are affected by the mesh motion applied to the periodic interfaces. You can prevent this by changing the mesh motion options for the `Periodic` and `Sampling` interfaces to stationary.

1. In the **Outline** tree view, edit `Flow Analysis 1` > `R1` > `R1 to R1 Periodic Side 1`.

2. Configure the following setting(s):

Tab	Setting	Value
Boundary Details	Mesh Motion > **Option**	Stationary

3. Click **OK**.

4. Repeat step 2 for `R1 to R1 Periodic Side 2`, `R1 Sampling Interface Side 1`, and `R1 Sampling Interface Side 2`.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.4.7. Writing the CFX-Solver Input (.def) File

1. Click *Write Solver Input File* .
2. Configure the following setting(s):

Setting	Value
File name	FourierBladeFlutterIni.def

3. Click **Save**.
4. Save the simulation.

36.4.8. Obtaining a Solution for the Steady-state Case

From the Ansys CFX launcher, start the CFX-Solver Manager. In CFX-Solver Manager:

1. Select **File** > **Define Run**

   The **Define Run** dialog box is displayed.

2. Under **Solver Input File**, click *Browse* and, in the dialog box that appears, select `FourierBladeFlutterIni.def` and click **Open**.
3. Select **Double Precision**.
4. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. At the end of the run, a dialog box is displayed stating that the simulation has ended.

5. Ensure that **Post-Process Results** is cleared.
6. Click **OK**.

36.5. Defining and Obtaining a Solution for the Time Integration Solution Method Case

In this part of the tutorial, you will modify the steady-state simulation from the first part of the tutorial in order to add the transient blade flutter/moving mesh details and set up the case to use the Fourier Transformation method. The result from the steady-state simulation is used as an initial guess to speed up the convergence for the transient simulation.

36.5.1. Opening the Existing Case

This step involves opening the original simulation and saving it to a different location.

1. Ensure that the following tutorial input files are in your working directory:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- `FourierBladeFlutterIni.cfx`
- `FourierBladeFlutterIni_001.res`
- `R37_inlet.csv`
- `R37_mode1_36p.csv`

2. Set the working directory and start CFX-Pre if it is not already running.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

3. If the original simulation is not already opened, then open `FourierBladeFlutterIni.cfx`.
4. Save the case as `FourierBladeFlutterTime.cfx` in your working directory.

36.5.2. Modifying the Analysis Type

Modify the analysis type as follows:

1. Edit `Analysis Type`.
2. Configure the following setting(s):

Setting	Value
Analysis Type > **Option**	 Transient Blade Row

3. Click **OK**.

36.5.3. Modifying the Domain

Modify the domain as follows:

1. In the **Outline** tree view, edit `Flow Analysis 1` > `R1`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Domain Models > Passage Definition > **Pass. in Component**	 2
	Domain Models > Passage Definition > **Passages in 360**	 36

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click **OK**.

36.5.4. Creating Expressions for Frequency and Scaling Factor

Next, you will create expressions defining the frequency, maximum periodic displacement and scaling factor that will be used in the blade boundary definition.

1. From the main menu, select **Insert** > **Expressions, Functions and Variables** > **Expression**.
2. In the **Insert Expression** dialog box, type `VibrationFrequency`.
3. Click **OK**.
4. Set **Definition** to `1152.13 [Hz]`.
5. Click **Apply** to create the expression.

You will create an expression defining the maximum periodic displacement.

1. Create an expression called `MaxPeriodicDisplacement`.
2. Set **Definition** to `0.0015 [m]`.
3. Click **Apply**.

You will use the maximum periodic displacement from above to calculate the scaling factor. The scaling factor is chosen as the maximum amplitude the blade will deform, normalized by the maximum amplitude of the mode shape provided. The maximum amplitude for the blade is set to approximately 2% of the maximum span of the blade.

1. Create an expression called `ScalingFactor`.
2. Set **Definition** to `MaxPeriodicDisplacement/0.00129[m]`.
3. Click **Apply**.

36.5.5. Modifying the R1 Blade Boundary

1. In the **Outline** tree view, edit `Flow Analysis 1` > `R1` > `R1 Blade`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Boundary Type**	Wall
	Location	Entire Rotor BLADE
	Frame Type	Rotating
	Profile Boundary Conditions > **Use Profile Data**	 (Selected)

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Profile Boundary Setup > **Profile Name**	mode1
	Profile Boundary Setup > **Generate Values**	(Click) This sets the options under the **Boundary Details** tab.
Boundary Details	Mesh Motion > **Option**	Periodic Displacement
	Mesh Motion > Periodic Displacement > **Option**	Cartesian Components
	Mesh Motion > Periodic Displacement > **X Component**	mode1.meshdisptotx(Initial X,Initial Y,Initial Z)
	Mesh Motion > Periodic Displacement > **Y Component**	mode1.meshdisptoty(Initial X,Initial Y,Initial Z)
	Mesh Motion > Periodic Displacement > **Z Component**	mode1.meshdisptotz(Initial X,Initial Y,Initial Z)
	Mesh Motion > Periodic Displacement > **Frequency**	mode1.Frequency()
	Mesh Motion	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> Periodic Displacement > **Scaling**	ScalingFactor[a]
	Mesh Motion > Periodic Displacement > Phase Angle > **Option**	Nodal Diameter (Phase Angle Multiplier)
	Mesh Motion > Periodic Displacement > Phase Angle > **Nodal Diameter Mag.**	4
	Mesh Motion > Periodic Displacement > Phase Angle > **Traveling Wave Dir.**	Forward
	Mesh Motion > Periodic Displacement > Phase Angle > **Passage Number**	mode1.Sector Tag(Initial X,Initial Y,Initial Z)
a. Click the *Enter Expression* icon to specify the CEL expression.		

3. Click **OK**.

36.5.6. Setting up a Transient Blade Row Model

In this section, you will set the simulation to be solved using the Fourier Transformation method.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the **Outline** tree view, edit `Flow Analysis 1` > `Transient Blade Row Models`.
2. Configure the following setting(s):

Setting	Value
Transient Blade Row Model > **Option**	Fourier Transformation

3. Under **Fourier Transformation**, click *Add new item* , accept the default name, and click **OK**.
4. Configure the following setting(s):

Setting	Value
Fourier Transformation > Fourier Transformation 1 > **Option**	Blade Flutter
Fourier Transformation > Fourier Transformation 1 > **Phase Corrected Intf.**	R1 to R1 Periodic
Fourier Transformation > Fourier Transformation 1 > **Sampling Dom. Intf.**	R1 Sampling Interface
Fourier Transformation > Fourier Transformation 1 > **Blade Boundary**	R1 Blade
Transient Method > **Option**	Time Integration
Transient Method > Time Period > **Option**	Value
Transient Method > Time Period > **Period**	1/VibrationFrequency[a]
Transient Method	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
> Time Steps > **Option**	Number of Timesteps per Period
Transient Method > Time Steps > **Timesteps/Period**	72[b]
Transient Method > Time Duration > **Option**	Number of Periods per Run
Transient Method > Time Duration > **Periods per Run**	10
a. Click the *Enter Expression* icon to specify the CEL expression. b. The number of time steps per period is an integer that is a multiple of the number of blades divided by the nodal diameter. This guarantees that both blades will go through the same deformations within the period.	

5. Click **OK**.

36.5.7. Setting Output Control and Creating Monitor Points

In this section you will create monitor points to monitor flow properties, integrated flow quantities, and mesh displacement. Monitor points provide useful information on the quality of the reference phase and frequency produced by the simulation. These monitor points should also be used to monitor convergence and patterns during the simulation.

Note:

- When comparing your Fourier Transformation plots to those from the reference case, make sure the monitor points are placed in the same relative locations with respect to the initial configuration in both cases.
- Monitoring pressure and velocity provides feedback on the momentum equations, while monitoring temperature provides feedback on the energy equations. Monitor points help check that the solver equations are being solved correctly.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Set up the solver to output transient results files. The transient blade row analysis type offers the Fourier compression method of storing transient periodic data.

1. Click *Output Control* .
2. Click the **Trn Results** tab.
3. Configure the following setting(s):

Setting	Value
Transient Blade Row Results > **Extra Output Variables List**	(Selected)
Transient Blade Row Results > Extra Output Variables List > **Extra Output Var. List**	Total Pressure, Total Temperature, Total Mesh Displacement, Wall Work Density, Wall Power Density[a]

a. Click *Multi-select from extended list* and hold down the **Ctrl** key while selecting each of the listed variables.

4. Click **Apply**.
5. Click the **Monitor** tab.
6. Select **Monitor Objects**.
7. Add an Efficiency Output monitor:

Setting	Value
Monitor Objects > **Efficiency Output**	(Selected)
Monitor Objects > Efficiency Output > **Option**	Output to Solver Monitor
Monitor Objects > Efficiency Output > **Inflow Boundary**	R1 Inlet

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
Monitor Objects > Efficiency Output > **Outflow Boundary**	R1 Outlet
Monitor Objects > Efficiency Output > **Efficiency Type**	Compression
Monitor Objects > Efficiency Output > **Value**	Total to Total

8. You will set up three types of monitors for this simulation. Firstly, you will create a set of monitor points to monitor variables at specific cylindrical coordinates within the domain. Cylindrical coordinates are useful in turbomachinery applications because they allow you to place monitor points with the same relative position inside different passages by shifting the theta component by the equivalent passage pitch. Next, you will create a second set of monitors to monitor the values of expressions. Finally, you will create a third set of monitors to monitor aerodynamic damping.

   Create monitor points by configuring the following settings:

Setting	Value
Monitor Objects > **Monitor Points and Expressions**	Create a monitor point named `LE1pass1` [a]
Monitor Objects > Monitor Points and Expressions > LE1pass1 > **Option**	Cylindrical Coordinates
Monitor Objects > Monitor Points and Expressions > LE1pass1 > **Output Variables List**	Pressure, Temperature, Total Pressure, Total Temperature, Velocity, Velocity in Stn Frame[b]
Monitor Objects	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
> Monitor Points and Expressions > LE1pass1 > **Position Axial Comp.**	0 [m]
Monitor Objects > Monitor Points and Expressions > LE1pass1 > **Position Radial Comp.**	0.23 [m]
Monitor Objects > Monitor Points and Expressions > LE1pass1 > **Position Theta Comp.**	-7.5 [degree]
a. To create a new item, you must first click the *Add new item* icon, then enter the name as required and click **OK**. b. Click *Multi-select from extended list* and hold down the **Ctrl** key while selecting each of the listed variables.	

9. Click **Apply**.

10. Create additional monitor points with the same output variables. The names and cylindrical coordinates are listed below:

Name	Coordinates
LE1pass2	(0 [m], 0.23 [m], 2.5 [degree])
LE2pass1	(0 [m], 0.23 [m], -2.5 [degree])
LE2pass2	(0 [m], 0.23 [m], 7.5 [degree])
TE1pass1	(0.05 [m], 0.23 [m], 0 [degree])
TE1pass2	(0.05 [m], 0.23 [m], 10 [degree])
TE2pass1	(0.05 [m], 0.23 [m], 5 [degree])
TE2pass2	(0.05 [m], 0.23 [m], 15 [degree])

11. Create additional monitor points with the following expressions:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Expression
Force on Blade	force()@REGION:Rotor BLADE
Force on Blade 2	force()@REGION:Rotor BLADE 2
Max Displ Blade	maxVal(Total Mesh Displacement)@REGION:Rotor BLADE
Max Displ Blade 2	maxVal(Total Mesh Displacement)@REGION:Rotor BLADE 2
Power on Blade	areaInt(Wall Power Density)@REGION:Rotor BLADE
Power on Blade 2	areaInt(Wall Power Density)@REGION:Rotor BLADE 2
Work on Blade	areaInt(Wall Work Density)@REGION:Rotor BLADE
Work on Blade 2	areaInt(Wall Work Density)@REGION:Rotor BLADE 2

12. Create aerodynamic damping monitors by configuring the following settings:

Setting	Value
Monitor Objects > **Aerodynamic Damping**	Create an aerodynamic damping object named `Aerodynamic Damping 1`.
Monitor Objects > Aerodynamic Damping > Aerodynamic Damping 1 > **Option**	Full Period Integration
Monitor Objects > Aerodynamic Damping > Aerodynamic Damping 1 > Location Type > **Option**	Mesh Regions
Monitor Objects > Aerodynamic Damping > Aerodynamic Damping 1	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
> Location Type > **Location**	 Rotor BLADE
Monitor Objects > **Aerodynamic Damping**	 Create an aerodynamic damping object named `Aerodynamic Damping 2`.
Monitor Objects > Aerodynamic Damping > Aerodynamic Damping 2 > **Option**	 Full Period Integration
Monitor Objects > Aerodynamic Damping > Aerodynamic Damping 2 > Location Type > **Option**	 Mesh Regions
Monitor Objects > Aerodynamic Damping > Aerodynamic Damping 2 > Location Type > **Location**	 Rotor BLADE 2
Monitor Objects > **Aerodynamic Damping**	 Create an aerodynamic damping object named `Aerodynamic Damping 3`.
Monitor Objects > Aerodynamic Damping > Aerodynamic Damping 3 > **Option**	 Moving Integration Interval
Monitor Objects > Aerodynamic Damping	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
> Aerodynamic Damping 3 > Location Type > **Option**	 Mesh Regions
Monitor Objects > Aerodynamic Damping > Aerodynamic Damping 3 > Location Type > **Location**	 Rotor BLADE

13. Click **OK**.

36.5.8. Writing the CFX-Solver Input (.def) File

1. Click *Write Solver Input File* .

2. Configure the following setting(s):

Setting	Value
File name	FourierBladeFlutterTime.def

3. Click **Save**.

   If a **Physics Validation Summary** message appears, indicating one global warning, click **Yes** to continue. The warning is about initial values, which will be supplied at run-time.

4. Save the case.

36.5.9. Obtaining a Solution for the Time Integration Solution Method Case

To reduce the simulation time for the blade flutter case, the simulation will be initialized using the steady-state case.

In CFX-Solver Manager:

1. Click **File** > **Define Run**.

2. Under **Solver Input File**, click *Browse* and select `FourierBladeFlutterTime.def`.

3. Select **Double Precision**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. On the **Initial Values** tab, select **Initial Values Specification**.
5. Under **Initial Values Specification** > **Initial Values**, select `Initial Values 1`.
6. Under **Initial Values Specification** > **Initial Values** > **Initial Values 1 Settings** > **File Name**, click *Browse* .
7. Select `FourierBladeFlutterIni_001.res` from your working directory.
8. Click **Open**.
9. Set **Initial Values Specification** > **Use Mesh From**, to `Solver Input File`.
10. Click **Start Run**.

    CFX-Solver runs and attempts to obtain a solution. This can take a long time depending on your system. Eventually a dialog box is displayed.

    Before the simulation begins, the "Transient Blade Row Post-processing Information" summary in the CFX-Solver Output file will display the time step range over which the solver will accumulate the Fourier coefficients. A "Fourier Transformation Stability" summary appears in the CFX-Solver Output file, as well as the time step at which the full Fourier Transformation model is activated.
11. When CFX-Solver is finished, you can optionally skip to Postprocessing the Blade Flutter Solution (p. 911).

36.6. Defining and Obtaining a Solution for the Harmonic Balance Solution Method Case

In this part of the tutorial, you will modify the Time Integration solution method case that was set up in the previous part of the tutorial in order to use the Harmonic Balance solution method. As in the previous part, the result from the steady-state simulation is used as an initial guess to speed convergence.

36.6.1. Opening the Existing Case

This step involves opening the Time Integration simulation and saving it to a different location.

1. Ensure that the following tutorial input files are in your working directory:
   - `FourierBladeFlutterTime.cfx`
   - `FourierBladeFlutterIni_001.res`
   - `R37_inlet.csv`
   - `R37_mode1_36p.csv`
2. Set the working directory and start CFX-Pre if it is not already running.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. If the Time Integration simulation is not already opened, then open `FourierBladeFlutter Time.cfx`.

4. Save the case as `FourierBladeFlutterHarmonic.cfx` in your working directory.

There are many common steps between setting up Time Integration and Harmonic Balance Flutter cases, including using the same Fourier Transformation pitch change model. Here, only the differences are highlighted.

36.6.2. Modifying the Transient Blade Row Model

In this section, you will change the transient method to `Harmonic Balance`.

1. In the **Outline** tree view, edit `Flow Analysis 1` > `Transient Blade Row Models`.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Transient Method** > **Option**	Harmonic Balance
	Transient Method > **Number of Modes**	3

   Note that, in the **Transient Blade Row Models** details view, on the **Basic Settings** tab, the settings for **Time Period**, **Time Steps** and **Time Duration** have disappeared.

3. Click **OK**.

Most blade flutter cases require only one mode. However, if the flow contains discontinuities like shocks that happen to oscillate with the vibration, then retaining 3 modes will be necessary for an accurate solution.

There is no need to specify the period involved when selecting `Harmonic Balance` in combination with the Fourier Transformation model. It will be obtained from the usual location for a blade flutter run (that is, in the **Mesh Motion** settings for the vibrating blade).

36.6.3. Modifying the Solver Control

1. Click *Solver Control* .

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Transient Scheme > **Option**	Harmonic Balance
	Convergence Control	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Min. Iterations**	1
	Convergence Control > **Max. Iterations**	200
	Convergence Control > Fluid Timescale Control > **Timescale Control**	Physical Timescale
	Convergence Control > Fluid Timescale Control > **Physical Timescale**	1/(15*VibrationFrequency)

3. Click **OK**.

The time periods involved in this case are:

- The blade vibrating period
- The period defined by the rotating domain

The physical timescale is usually set as a fraction of the smallest time period involved. In this case, the blade vibrating period is the smallest time period involved, and 1/15th of that period is used as the physical timescale. A larger physical timescale speeds convergence but can lower stability. In general, if the solution becomes unstable or does not converge, you should use a smaller fraction when computing the physical timescale.

36.6.4. Modifying the Output Control

In this section you will create monitor points to monitor flow properties, integrated flow quantities, and mesh displacement. Monitor points provide useful information on the quality of the reference phase and frequency produced by the simulation. These monitor points should also be used to monitor convergence and patterns during the simulation.

Note:

- When comparing your Fourier Transformation plots to those from the reference case, make sure the monitor points are placed in the same relative locations with respect to the initial configuration in both cases.
- Monitoring pressure and velocity provides feedback on the momentum equations, while monitoring temperature provides feedback on the energy equations. Monitor points help check that the solver equations are being solved correctly.

Set up the solver to output transient results files. The transient blade row analysis type offers the Fourier compression method of storing transient periodic data.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click *Output Control* .
2. Click the **Trn Results** tab.
3. Configure the following setting(s):

Setting	Value
Transient Blade Row Results > **Extra Output Variables List**	 (Selected)
Transient Blade Row Results > Extra Output Variables List > **Extra Output Var. List**	 Total Pressure, Total Temperature, Total Mesh Displacement, Wall Work Density, Wall Power Density[a]

a. Click *Multi-select from extended list* and hold down the **Ctrl** key while selecting each of the listed variables.

4. Click **Apply**.
5. Click the **Monitor** tab.

   You have already set up monitors for variables, values of expressions, and aerodynamic damping. The aerodynamic damping monitors need to be modified.

6. Modify the aerodynamic damping monitors by configuring the following settings:

Setting	Value
Monitor Objects > **Aerodynamic Damping**	 Select the aerodynamic damping object named `Aerodynamic Damping 1`.
Monitor Objects > Aerodynamic Damping > Aerodynamic Damping 1 > **Option**	 Fourier Integration
Monitor Objects > Aerodynamic Damping > Aerodynamic Damping 1	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
> Location Type > **Option**	 Mesh Regions
Monitor Objects > Aerodynamic Damping > Aerodynamic Damping 1 > Location Type > **Location**	 Rotor BLADE
Monitor Objects > **Aerodynamic Damping**	 Select the aerodynamic damping object named `Aerodynamic Damping 2`.
Monitor Objects > Aerodynamic Damping > Aerodynamic Damping 2 > **Option**	 Fourier Integration
Monitor Objects > Aerodynamic Damping > Aerodynamic Damping 2 > Location Type > **Option**	 Mesh Regions
Monitor Objects > Aerodynamic Damping > Aerodynamic Damping 2 > Location Type > **Location**	 Rotor BLADE 2
Monitor Objects > **Aerodynamic Damping**	 Delete the aerodynamic damping object named `Aerodynamic Damping 3`. (Note that this object is a copy of `Aerodynamic Damping 1`.)

7. Click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.6.5. Writing the CFX-Solver Input (.def) File

1. Click *Write Solver Input File* .
2. Configure the following setting(s):

Setting	Value
File name	FourierBladeFlutterHarmonic.def

3. Click **Save**.
4. Save the case.

36.6.6. Obtaining a Solution for the Harmonic Balance Solution Method Case

To reduce the simulation time for the blade flutter case, the simulation will be initialized using the steady-state case.

In CFX-Solver Manager:

1. Click **File** > **Define Run**.
2. Under **Solver Input File**, click *Browse* and select `FourierBladeFlutterHarmonic.def`.
3. Select **Double Precision**.
4. On the **Initial Values** tab, select **Initial Values Specification**.
5. Under **Initial Values Specification** > **Initial Values**, select `Initial Values 1`.
6. Under **Initial Values Specification** > **Initial Values** > **Initial Values 1 Settings** > **File Name**, click *Browse* .
7. Select `FourierBladeFlutterIni_001.res` from your working directory.
8. Click **Open**.
9. Set **Initial Values Specification** > **Use Mesh From**, to `Solver Input File`.
10. Click **Start Run**.

    CFX-Solver runs and attempts to obtain a solution. This can take a long time depending on your system. Eventually a dialog box is displayed.

36.7. Postprocessing the Blade Flutter Solution

The postprocessing steps outlined here can be equally used on either of the Blade Flutter solutions obtained in Defining and Obtaining a Solution for the Time Integration Solution Method Case (p. 893) and Defining and Obtaining a Solution for the Harmonic Balance Solution Method Case (p. 906). The

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

results should be similar and the differences between the two will be minimized by comparing a time-resolved transient case to a modal-resolved harmonic balance case (that is, a case where increasing the number of modes or time planes does not result in solution change).

36.7.1. Viewing Results in CFX-Solver Manager

Some of the monitor points of the Time Integration case show instantaneous values that oscillate about certain values. The corresponding monitor points of the Harmonic Balance case show values that are time-averaged over the vibration period.

1. Select **Workspace** > **New Monitor** and accept the default name.
2. Under the **Plot Lines** tab, expand the **USER POINT** branch and select `Work on Blade`.
3. Click **Apply**.
4. On the **Range Settings** tab, set **Plot Data By** to `Simulation Time`.

   This displays a simulation time history of the work on blade 1.
5. Click the **Aerodynamic Damping** tab.

   Observe the aerodynamic damping monitors. The monitor values represent mechanical work done by the blade on the fluid over the last period of mesh motion. If the monitor values remain positive (after the case has converged), then the vibration is damped (for the frequency being studied).
6. Click the **Efficiency** tab.

   Observe the monitor points related to efficiency.
7. For the harmonic results only, additional performance monitors are available:

   a. Click the **Turbo Output Ratios** tab.

      This displays the time-averaged inflow-to-outflow ratios of total pressure and total temperature, using the boundaries that you defined ealier in the efficiency definition.

   b. Click the **Turbo Output Mass Flows** tab.

      This displays the time-averaged mass flow at the outflow boundary that you defined earlier in the efficiency definition.
8. When done with the CFX-Solver Manager, you can continue examining the solutions with CFD-Post.

36.7.2. Viewing Results in CFD-Post

A transient blade row analysis calculation creates a number of solution variables in addition to those added in Setting Output Control and Creating Monitor Points (p. 899). These variables are compressed using a discrete Fourier Transformation and the corresponding coefficients are stored in the results file. CFD-Post can expand this transformation for the variable of interest at any given time value. The

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

time step selector shows time values that are representative of the values used by the solver. In addition to the existing time values, additional time values can be added or removed as deemed necessary.

In this section, you will create a few plots to illustrate the use of the time step selector for a transient blade row analysis. You will also create a user defined variable for the total wall work, and use that variable to create a contour and an animation of the blade.

Note:

When CFD-Post starts, you may see a message regarding transient blade row postprocessing. If you do, click **OK**.

36.7.2.1. Displaying Total Wall Work on the Blade

In CFD-Post, with results loaded:

1. Select **Insert** > **Variable** and set the name to `Total Wall Work`.
2. Configure the following setting(s):

Name	Setting	Value
Total Wall Work	**Method**	Expression
	Scalar	(Selected)
	Expression	Wall Work Density * Area
	Calculate Global Range	(Selected)

3. Click **Apply** to create the new variable.

   You can review the new `Total Wall Work` variable on the **Variables** tab, under the `User Defined` branch.

36.7.2.2. Creating a Contour Plot for Total Wall Work on the Blade

1. Click **Insert** > **Contour** and accept the default name.
2. Configure the following setting(s):

Tab	Setting	Value
Geometry	**Locations**	R1 Blade
	Variable	Total Wall Work
	Range	Local
	# of Contours	21
Render	**Show Contour Lines**	(Selected)
	Constant Coloring	(Selected)

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Color Mode	Default

3. Click **Apply**.

The contour plot shows instantaneous values for `Total Wall Work`.

36.7.2.3. Creating an Animation for Total Wall Work on the Blade

Using the contour plot created above, you will now create an animation of the `Total Wall Work` on the blade for the first phase.

1. Click *Timestep Selector* .

   The **Timestep Selector** dialog box appears.

2. Set **Timestep Sampling** to `Uniform`.

3. Select the time value of `0 [s]`.

4. Click **Apply**.

5. Select **Tools** > **Animation** or click *Animation* .

   The **Animation** dialog box appears.

6. Set **Type** to **Timestep Animation**.

7. Ensure that **Control By** is set to `Timestep`.

8. Select **Specify Range for Animation**.

9. Set **Start Timestep** to `0` and **End Timestep** to `10`.

10. Select **Save Movie**.

11. Set **Format** to `MPEG1`.

12. Click *Browse* next to **Save Movie** to set a path and filename for the movie file.

    If the file path is not given, the file will be saved in the directory from which CFD-Post was launched.

13. Click **Save**.

    The movie filename (including path) is set, but the movie is not yet created.

14. If **Current Timestep** is not `0` (shown in the **Animation** dialog box), click *To First Timestep* to load it.

    Wait for CFD-Post to finish loading the objects for this frame before proceeding.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15. Click *Play the animation* .

    The movie will be created as the animation proceeds. This will be slow, since a time step must be loaded and objects must be created for each frame. To view the movie file, you need to use a viewer that supports the MPEG format.

16. When you have finished, close CFD-Post.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 37: Time Transformation Method for a 1.5-stage Transient Rotor-stator Case

This tutorial includes:

37.1. Tutorial Features

37.2. Overview of the Problem to Solve

37.3. Preparing the Working Directory

37.4. Defining a Steady-state Case in CFX-Pre

37.5. Obtaining a Solution to the Steady-state Case

37.6. Defining a Transient Blade Row Case in CFX-Pre

37.7. Obtaining a Solution to the Transient Blade Row Case

37.8. Viewing the Time Transformation Results in CFD-Post

37.1. Tutorial Features

In this tutorial you will learn about:

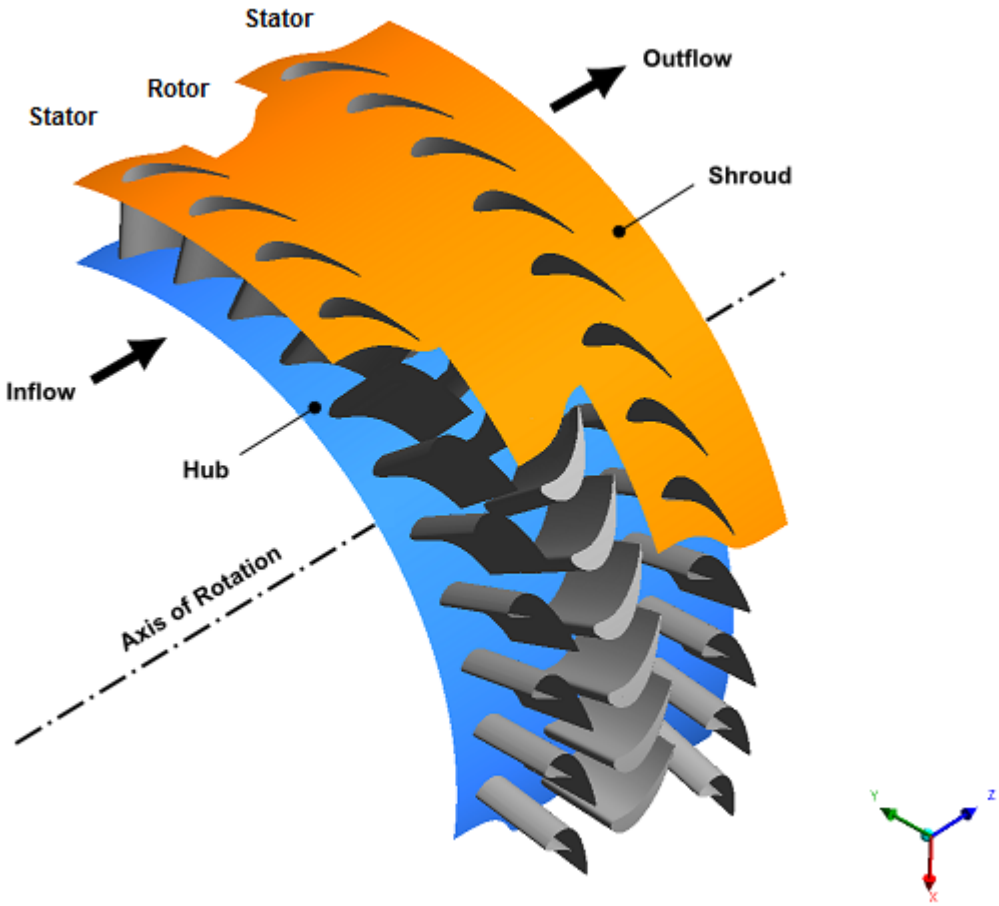
Component	Feature	Details
CFX-Pre	User Mode	Turbo Wizard
		General mode
	Analysis Type	Transient Blade Row
	Fluid Type	Air Ideal Gas
	Domain Type	Multiple Domains
		Rotating Frame of Reference
	Turbulence Model	Shear Stress Transport
	Heat Transfer	Total Energy
	Boundary Conditions	Inlet (Subsonic)
		Outlet (Subsonic)
		Wall (Counter Rotating)
CFD-Post	Plots	Vector
		Contour
		Data Instancing
		Time Chart
		Animation

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.2. Overview of the Problem to Solve

This tutorial sets up a 1.5-stage transient blade row calculation using the Time Transformation model. It uses a 1.5-stage machine to illustrate the basic concepts of setting up, running, and monitoring a transient blade row problem in Ansys CFX. It also describes the postprocessing of multistage transient blade row results using the tools provided in CFD-Post for this type of calculation.

The full geometry of the 1.5 stages selected for modeling contains 36 blades in the first stator row, 42 blades in the rotor row, and 37 blades in the second stator row.



The geometry to be modeled consists of a single rotor blade passage and two stator blade passages (one from each stator row). Each rotor blade passage is an 8.571° section (360°/42 blades), while the blade passages from the first stator row are 10° sections (360°/36 blades) and the blade passages from the second stator row are 9.730° sections (360°/37 blades). The pitch ratio at the interface between the rotor passage and the first stator passage is 0.8571 (that is, 6/7). The pitch ratio at the interface between the rotor passage and the second stator passage is 0.8810 (that is, 37/42).

For the Time Transformation method, you should always maintain an ensemble pitch ratio within a range of 0.75 to 1.4. Note that the range of permissible pitch ratios narrows significantly with slower rotation speed. A full machine analysis can be performed (modeling all rotor and stator blades), which always eliminates any pitch change, but will require significant computational time. For this geometry, only a full machine analysis can produce a pitch ratio of 1.0.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In this example, the rotor rotates about the Z axis at 3500 rev/min (positive rotation following the right hand rule) while the stator rows are stationary. Rotational periodicity boundaries are used to enable only a small section of the full geometry to be modeled.

The flow is modeled as being turbulent and compressible. Profile boundary conditions are used at the inlet and outlet. In this tutorial, the profiles are a function of the radial coordinate only. These profiles were obtained from previous simulations of the upstream and downstream stages.

The overall approach of this tutorial is to run a transient blade row simulation initialized with the results of a steady-state simulation. First, you will define a steady-state simulation using the Turbomachinery Wizard followed by General mode. While the steady-state simulation is running, you will modify a copy of it to define a transient blade row simulation that uses the Time Transformation model. After running the transient blade row simulation, you will create contour plots and an animation showing blade rotation.

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- Running Ansys CFX Tutorials Using Ansys Workbench (p. 34)
- Changing the Display Colors (p. 37)

37.3. Preparing the Working Directory

1. Create a working directory.

   Ansys CFX uses a working directory as the default location for loading and saving files for a particular session or project.

2. Download the `m_time_blade_row.zip` file here.

3. Unzip `m_time_blade_row.zip` to your working directory.

   Ensure that the following tutorial input files are in your working directory:

   - `TBRInletProfile.csv`
   - `TBROutlet2Profile.csv`
   - `TBRTurbineRotor.gtm`
   - `TBRTurbineStator.gtm`
   - `TBRTurbineStator2.gtm`

4. Set the working directory and start CFX-Pre.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.4. Defining a Steady-state Case in CFX-Pre

This tutorial uses the Turbomachinery wizard in CFX-Pre. This preprocessing mode is designed to simplify the setup of turbomachinery simulations.

1. In CFX-Pre, select **File** > **New Case**.
2. Select **Turbomachinery** and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `MTimeBladeRowIni`.
5. Click **Save**.
6. If you are notified that the file already exists, click **Overwrite**.

37.4.1. Basic Settings

1. In the **Basic Settings** panel, configure the following settings:

Setting	Value
Machine Type	Axial Turbine
Axes > **Rotation Axis**	Z
Analysis Type > **Type**	Steady State

2. Leave the other settings at their default values.
3. Click **Next**.

37.4.2. Components Definition

You will define three new components and import their respective meshes.

1. Right-click in the blank area and select **Add Component** from the shortcut menu.
2. Create a new component of type `Stationary` named `S1` and click **OK**.
3. Configure the following setting(s):

Setting	Value
Mesh > **File**	TBRTurbineStator.gtm [a]

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
a. You may have to select the `CFX Mesh (*gtm *cfx)` option under **Files of type**.	

4. Create a new component of type `Rotating`, named `R1` and click **OK**.

5. Configure the following setting(s):

Setting	Value
Component Type > **Value**	3500 [rev min^-1] [a]
Mesh > **File**	TBRTurbineRotor.gtm
a. From the problem description.	

6. Create a new component of type `Stationary` named `S2` and click **OK**.

7. Configure the following setting(s):

Setting	Value
Mesh > **File**	TBRTurbineStator2.gtm

8. Click **Next**.

37.4.3. Physics Definition

In this section you will set properties of the fluid domain and some solver parameters.

1. In the **Physics Definition** panel, configure the following:

Setting	Value
Fluid	Air Ideal Gas
Model Data > **Reference Pressure**	0 [atm] [a]
Model Data > **Heat Transfer**	Total Energy
Model Data	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
> **Turbulence**	Shear Stress Transport
Inflow/Outflow Boundary Templates > **P-Total Inlet P-Static Outlet**	(Selected)
Inflow/Outflow Boundary Templates > Inflow > **P-Total**	169000 [Pa][b]
Inflow/Outflow Boundary Templates > Inflow > **T-Total**	306 [K][b]
Inflow/Outflow Boundary Templates > Inflow > **Flow Direction**	Normal to Boundary
Inflow/Outflow Boundary Templates > Outflow > **P-Static**	110000 [Pa][b]
Interface > **Default Type**	Stage

a. To define the simulation using absolute pressure, set this value to 0 atm.

b. These values are temporary. They will be replaced with profile data later in the tutorial.

2. Continue to click **Next** until you reach `Final Operations`.

3. Set **Operation** to `Enter General Mode` because you will continue to define the simulation through settings not available in the Turbomachinery wizard.

4. Click **Finish**.

37.4.4. Additional Fluid Model Settings

Verify the following settings, which affect the accuracy of the simulation:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Edit R1.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Domain Models > Domain Motion > **Alternate Rotation Model**	(Selected)
Fluid Models	Heat Transfer > **Incl. Viscous Work Term**	(Selected)

3. Click **OK**.

37.4.5. Initializing Profile Boundary Conditions

The inlet and outlet boundary conditions are defined using profiles in your working directory. Boundary profile data must be initialized before they can be used for boundary conditions.

1. Select **Tools** > **Initialize Profile Data**.

   The **Initialize Profile Data** dialog box appears.

2. Beside **Profile Data File**, click *Browse* .

   The **Select Profile Data File** dialog box appears.

3. From your working directory, select `TBRInletProfile.csv`.
4. Click **Open**.
5. Click **Apply**.

   The profile data is read into memory.

6. Beside **Profile Data File**, click *Browse* .

   The **Select Profile Data File** dialog box appears.

7. From your working directory, select `TBROutlet2Profile.csv`.
8. Click **Open**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9. Click **OK**.

Note:

After profile data has been initialized from a file, the profile data file should not be deleted or otherwise removed from its directory. By default, the full filepath to the profile data file is stored in CFX-Pre, and the profile data file is read directly by CFX-Solver each time the solver is started or restarted.

37.4.6. Modifying Inlet and Outlet Boundary Conditions

1. Edit `S1 Inlet`.

2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Profile Boundary Conditions > **Use Profile Data**	(Selected)
	Profile Boundary Setup > **Profile Name**	inlet

3. Click **Generate Values**.

   This causes the profile values of `Total Pressure` and `Total Temperature` to be applied at the nodes on the inlet boundary. It also causes entries to be made in the **Boundary Details** tab. In order to later reset the velocity values at the main inlet to match those that were originally read from the profile data file, revisit the **Basic Settings** tab for this boundary and click **Generate Values**.

4. Configure the following setting(s):

Tab	Setting	Value
Boundary Details	Mass and Momentum > **Option**	Total Pressure (stable)
	Mass and Momentum > **Relative Pressure**	inlet.Total Pressure(r)
	Flow Direction > **Option**	Cylindrical Components
	Flow Direction > **Axial Component**	1

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	Flow Direction > **Radial Component**	 0
	Flow Direction > **Theta Component**	 0
	Heat Transfer > **Total Temperature**	 inlet.Total Temperature(r)

5. Click **OK**.
6. Edit `S2 Outlet`.
7. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Profile Boundary Conditions > **Use Profile Data**	 (Selected)
	Profile Boundary Setup > **Profile Name**	 outlet

8. Click **Generate Values**.
9. Configure the following setting(s):

Tab	Setting	Value
Boundary Details	Mass And Momentum > **Option**	 Static Pressure
	Mass and Momentum > **Relative Pressure**	 outlet.Pressure(r)

10. Click **OK**.

37.4.7. Visualizing the Profile Boundary Value

You can plot scalar profile values and vectors on inlet and outlet boundaries. In this section, you will edit a boundary so that you can visualize the pressure profile values at the inlet.

1. Edit `S1 Inlet`
2. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Plot Options	**Boundary Contour**	(Selected)
	Profile Variable	Relative Pressure

3. Click **Apply**

CFX-Pre plots the Total Pressure radial profile at the inlet with the pressure values displayed in a legend.

37.4.8. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .
2. Configure the following setting(s):

Setting	Value
File name	MTimeBladeRowIni.def

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, **Solver Input File** is set.

4. Save the simulation.

37.5. Obtaining a Solution to the Steady-state Case

Ensure that the **Define Run** dialog box is displayed in CFX-Solver Manager.

1. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. At the end of the run, a dialog box is displayed stating that the simulation has ended.

2. Ensure that the **Post-Process Results** check box is cleared.
3. Click **OK**.

37.6. Defining a Transient Blade Row Case in CFX-Pre

In the second part of the tutorial, you will modify the simulation from the first part of the tutorial to model the transient blade row.

37.6.1. Opening the Existing Case

This step involves opening the original simulation and saving it to a different location.

1. Ensure that the following tutorial input files are in your working directory:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- `MTimeBladeRowIni.cfx`
- `MTimeBladeRowIni_001.res`
- `TBRInletProfile.csv`
- `TBROutlet2Profile.csv`

2. Set the working directory and start CFX-Pre if it is not already running.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

3. If the original simulation is not already opened, then open `MTimeBladeRowIni.cfx`.

4. Save the case as `MTimeBladeRow.cfx` in your working directory.

37.6.2. Modifying the Analysis Type

In this section, you will make use of the transient blade row feature.

Modify the analysis type as follows:

1. Edit `Analysis Type`.

2. Configure the following setting(s):

Setting	Value
Analysis Type > **Option**	Transient Blade Row
Analysis Type > Initial Time > **Option**	Automatic with Value
Analysis Type > Initial Time > **Time**	0 [s]

3. Click **OK**.

37.6.3. Modifying the Rotor–stator Interfaces

1. Edit `R1 to S1`.

2. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
Interface Models > Frame Change/Mixing Model > **Option**	 Transient Rotor Stator

3. Click **OK**.
4. Edit `S2 to R1`.
5. Configure the following setting(s):

Setting	Value
Interface Models > Frame Change/Mixing Model > **Option**	 Transient Rotor Stator

6. Click **OK**.

37.6.4. Setting up a Transient Blade Row Model

You will set the simulation to be solved using the Time Transformation method.

1. Edit `Transient Blade Row Models`.
2. Set **Transient Blade Row Model** > **Option** to `Time Transformation`.
3. Under `Time Transformation`, click *Add new item* , accept the default name, and click **OK**.
4. Configure the following setting(s):

Setting	Value
Transient Method > Time Period > **Option**	 Passing Period
Transient Method > Time Period > **Domain**	 R1
Transient Method	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
> Time Steps > **Option**	Number of Timesteps per Period
Transient Method > Time Steps > **Timesteps/Period**	60
Transient Method > Time Duration > **Option**	Number of Periods per Run
Transient Method > Time Duration > **Periods per Run**	10

Note:

- The passing period is automatically calculated using 2 * pi / (Number of Blades * Angular Velocity). The **Passing Period** setting cannot be edited.
- The number of time steps per period should always be larger than 2 * Number of Fourier Coefficients + 1 for use during postprocessing.
- The time step size is also automatically calculated as: Passing Period / Number of Timesteps per Period. The **Timestep** setting cannot be edited.

5. Under `Time Transformation`, click *Add new item* , accept the default name, and click **OK**.
6. Configure the following setting(s):

Setting	Value
Time Transformation > **Time Transformation 2**	(Selected)
Time Transformation 2 > **Domain Interface**	S2 to R1

7. Click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.6.5. Setting Output Control and Creating Monitor Points

For transient blade row calculations, a minimal set of variables are selected to be computed using Fourier coefficients. It is convenient to postprocess variables in the stationary frame when multiple frames of reference are present. Here, you will add the `Velocity in Stn Frame` and `Mach Number in Stn Frame` variables to the default list.

In addition, monitor points can be used to effectively compare the Time Transformation results against a reference case. They provide useful information on the quality of the reference phase and frequency produced in the simulation. They should be used to monitor convergence and, as the simulation converges, the user points should display a periodic pattern.

Note:

- When comparing to a reference case, make sure monitor points are placed in the same relative locations with respect to the initial configuration in both cases.
- It is important to check that the solver equations are being solved correctly. Monitoring pressure provides feedback on the momentum equations while monitoring temperature provides feedback on the energy equations.

Set up the output control and create monitor points as follows:

1. Click *Output Control* .
2. Click the **Trn Results** tab.
3. Configure the following setting(s):

Setting	Value
Transient Blade Row Results > **Extra Output Variables List**	(Selected)
Transient Blade Row Results > Extra Output Variables List > **Extra Output Var. List**	Velocity in Stn Frame, Mach Number in Stn Frame[a]

a. Click *Multi-select from extended list* and hold down the **Ctrl** key while selecting each of the listed variables.

4. Click the **Monitor** tab.
5. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
Monitor Objects	(Selected)
Monitor Objects > **Efficiency Output**	 (Cleared)

6. Create a monitor point named `rotor_P1`.
7. Under **Monitor Objects** > **Monitor Points and Expressions** > **rotor_P1**, configure the following settings:

Setting	Value
Option	Cylindrical Coordinates
Output Variables List	Pressure, Temperature, Total Pressure, Total Temperature, Velocity
Position Axial Comp.	0.211 [m]
Position Radial Comp.	0.2755 [m]
Position Theta Comp.	182 [degree]

8. Create an additional monitor point named `stator_P1`.
9. Under **Monitor Objects** > **Monitor Points and Expressions** > **stator_P1**, configure the following settings:

Setting	Value
Option	Cylindrical Coordinates
Output Variables List	Pressure, Temperature, Total Pressure, Total Temperature, Velocity
Position Axial Comp.	0.202 [m]
Position Radial Comp.	0.2755 [m]
Position Theta Comp.	178 [degree]

10. Create an additional monitor point named `rotor_P2`.
11. Under **Monitor Objects** > **Monitor Points and Expressions** > **rotor_P2**, configure the following settings:

Setting	Value
Option	Cylindrical Coordinates
Output Variables List	Pressure, Temperature, Total Pressure, Total Temperature, Velocity
Position Axial Comp.	0.27 [m]
Position Radial Comp.	0.2755 [m]

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
Position Theta Comp.	176 [degree]

12. Create an additional monitor point named `stator_P2`.

13. Under **Monitor Objects** > **Monitor Points and Expressions** > **stator_P2**, configure the following settings:

Setting	Value
Option	Cylindrical Coordinates
Output Variables List	Pressure, Temperature, Total Pressure, Total Temperature, Velocity
Position Axial Comp.	0.28 [m]
Position Radial Comp.	0.2755 [m]
Position Theta Comp.	174 [degree]

Note:

Transient blade row cases use monitor points to monitor the periodic fluctuating variable values. For diagnostic purposes, you should have several monitor points. Here, four monitor points are used for demonstration purposes.

14. Click **OK**.

37.6.6. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .

2. Configure the following setting(s):

Setting	Value
File name	MTimeBladeRow.def

3. Click **Save**.

4. Ignore the error message (the initial values will be specified in CFX-Solver Manager) and click **Yes** to continue.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, **Solver Input File** is set.

5. If using stand-alone mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.7. Obtaining a Solution to the Transient Blade Row Case

At this point, CFX-Pre has been shut down, and the **Define Run** dialog box is displayed in CFX-Solver Manager. You will now obtain a solution to the CFD problem. To reduce the simulation time, the simulation will be initialized using a steady-state case.

1. Ensure that the **Define Run** dialog box is displayed. If an error message appears, ignore it and click **Yes** to continue.

   Solver Input File should be set to `MTimeBladeRow.def`.

2. Under the **Initial Values** tab, select **Initial Values Specification**.

3. Under **Initial Values Specification** > **Initial Values**, select `Initial Values 1`.

4. Under **Initial Values Specification** > **Initial Values** > **Initial Values 1 Settings** > **File Name**, click *Browse* .

5. Select `MTimeBladeRowIni_001.res` from your working directory.

6. Click **Open**.

7. Under **Initial Values Specification** > **Use Mesh From**, select `Solver Input File`.

8. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution. At the end of the run, a dialog box is displayed stating that the simulation has ended.

Note:

- Before the simulation begins, the "Transient Blade Row Post-processing Information" summary in the CFX-Solver Output file will display the time step range over which the solver will accumulate the Fourier coefficients.
- Similarly, the "Time Transformation Stability" summary in the CFX-Solver Output file displays whether the rotor-stator pitch ratio is within the acceptable range.
- After the CFX-Solver Manager has run for a short time, you can track the monitor points you created in CFX-Pre by clicking the **Time Corrected User Points** tab that appears at the top of the graphical interface of CFX-Solver Manager.
- Monitor points of similar values can be grouped together by right-clicking on any monitor tab and selecting **New Monitor**. Change **Type** to `Plot Monitor` and click **OK**. Expand `USER POINT` and select the points of interest (for example, all the pressure points), then click **OK**.
- After the simulation has proceeded for some time, observe the periodic nature of the monitor point values.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9. When CFX-Solver is finished, select the check box next to **Post-Process Results**.

10. Click **OK**.

37.8. Viewing the Time Transformation Results in CFD-Post

In a transient blade row run, flow field variables are compressed using the Fourier coefficient method. These variables are accumulated at the end of the simulation. This enables you to navigate through any time instance, within the common period, without having to load multiple transient results files. By default CFD-Post displays results corresponding to the end the simulation.

To get started, follow these steps:

1. Start CFD-Post and load `MTimeBladeRow_001.res`.

2. When CFD-Post opens, if you see the **Domain Selector** dialog box, ensure that all the domains are selected, then click **OK** to load the results from these domains.

3. If you see a message regarding transient blade row postprocessing, click **OK**.

37.8.1. Creating a Turbo Surface

Create a turbo surface to be used for making plots:

1. Click the **Turbo** tab.

2. If you see the **Turbo Initialization** dialog box, click **Yes**, otherwise click the **Initialize All Components** button, which is visible initially by default, or after double-clicking the **Initialization** object in the **Turbo** tree view.

3. Select **Insert** > **Location** > **Turbo Surface**.

4. Change the name to `Span 50`.

5. Configure the following setting(s):

Tab	Setting	Value
Geometry	Definition > **Method**	Constant Span
	Definition > **Value**	0.5

6. Click **Apply**.

7. Turn off the visibility of `Span 50` by clearing its check box in the **Outline** tree view.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.8.2. Creating a Vector Plot

1. Click **Insert** > **Vector** and accept the default name.
2. Configure the following setting(s):

Tab	Setting	Value
Geometry	Definition > **Locations**	Span 50
	Definition > **Variable**	Velocity in Stn Frame

3. Click **Apply**.

   The vector plot shows `Velocity in Stn Frame` values corresponding to the end of a common period.

   The rotor domain is in the angular position corresponding to its location after 10 passing periods. Now you will align the rotor with the stator, as it was in the solver input file.

4. Click *Timestep Selector* .
5. Select the 1st time step.
6. Click **Apply** to load the time step, and then click **Close** to exit the dialog box.

   The rotor blades move to their starting positions.

37.8.3. Creating a Blade-to-Blade Plot

1. Turn off the visibility of `Vector 1`.
2. In the **Turbo** tab, edit `Plots` > `Blade-to-Blade`.
3. Configure the following setting(s):

Setting	Value
Span	0.5
Plot Type	Contour
Variable	Pressure
Range	Local
# of Contours	21

4. Click **Apply**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The contour plot shows `Pressure` values corresponding to the specified time step.

37.8.4. Creating a Chart of Force on a Rotor Blade versus Time

In this section, you will compute and plot the magnitude of the forces that the flow applies on the rotor blade. For a transient blade row case, CFD-Post automatically reconstructs variables for the flow solution time based on the last time step. Intermediate time steps for time instances in the common period are located in the **Timestep Selector**. In Setting up a Transient Blade Row Model (p. 928), you set 60 time steps per rotor blade passing period and there are 42 rotor blade passing periods in a common period. Therefore, the total number of intermediate time steps in the common period is 2520. For this case, the solver has reconstructed results over one common period (2520 time steps). You will reduce the total number of time steps to 420 to speed up the generation of the time chart.

Reduce the number of time steps in the period:

1. Click *Timestep Selector* .
2. Configure the following setting(s):

Setting	Value
Timestep Sampling	Uniform
Number of Timesteps	420

3. Click **Apply**.

   The **Timestep Selector** now shows a total of 420 steps over one common period (shown under the *Phase* column).

Compute the forces on the blade:

1. Select **Insert** > **Expression**.
2. In the **Insert Expression** dialog box, type `forces on rotor blade`.
3. Click **OK**.
4. Set **Definition** to `sqrt(force_x()@ R1 Blade ^2 + force_y()@ R1 Blade ^2 + force_z()@ R1 Blade ^2)`
5. Click **Apply** to create the expression.

Create a transient chart showing force:

1. Select **Insert** > **Chart** and accept the default name.
2. Configure the following setting(s):

Tab	Setting	Value
General	**XY - Transient or Sequence**	(Selected)

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Data Series	Series 1 > Data Source > **Expression**	 (Selected)
	Series 1 > Data Source > **Expression**	 forces on rotor blade

3. Click **Apply**.

   A chart showing the force on a single rotor blade versus time is created and displayed in the **Chart Viewer**.

37.8.5. Creating a Chart of a Solution Monitor and Fourier Coefficient Data for Pressure versus Time

In this section, you will plot a solution monitor.

First, create a chart of the pressure at monitor point `stator_P1` versus time:

1. From the main menu, select **Insert** > **Chart**.
2. Accept the default name by clicking **OK**.
3. Configure the following setting(s):

Tab	Setting	Value
General	**XY - Transient or Sequence**	(Selected)
Data Series	Series 1 > **Name**	 stator_P1 Pressure Solver Data
	stator_P1 Pressure Solver Data > Data Source > **Monitor Data**	 (Selected)
	stator_P1 Pressure Solver Data > Monitor Variable Selection	

Tab	Setting	Value
	> Y Axis > **Variable**	Monitor Point: stator_P1 (Pressure) [a]
a. Click *Browse Variables* ... to open the **Monitor Data Y Variable** dialog box. From the tree in the dialog box, select `User Point` > `Pressure` > `stator_P1`. Click **OK**.		

4. Click **Apply**.

   A chart showing the raw solver pressure data at monitor point `stator_P1` versus time is created and displayed in the **Chart Viewer**.

Next, create another data series that will show the pressure signal reconstructed from Fourier coefficients at the same point in the stator as monitor point `stator_P1`.

Start by adding a point:

1. From the main menu, select **Insert** > **Location** > **Point**.
2. Accept the default name by clicking **OK**.
3. Set **Method** to XYZ.
4. Set **Point** coordinates to `-0.2778`, `0.01044`, `0.2021`.
5. Click **Apply**.

   Note the location of `Point 1` in the viewer.

Next, create the chart series:

1. Edit `Chart 2`.
2. In the **Chart 2** details view, on the **Data Series** tab, click *New* to add a new series.
3. Configure the following setting(s):

Tab	Setting	Value
Data Series	Series 2 > **Name**	stator_P1 Pressure Fourier Coefficients
	stator_P1 Pressure Fourier Coefficients > Data Source	Point 1

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Location**	
X Axis	Axis Range > **Determine ranges automatically**	(Cleared)
	Axis Range > **Min**	0
	Axis Range > **Max**	0.00415405
Y Axis	Data Selection > **Variable**	Pressure

4. Click **Apply**.

   The chart now also shows the pressure at point `Point 1` versus time, as represented by the Fourier coefficient data. The reconstructed pressure signal is based on Fourier coefficients and the blade passing period and is therefore an approximation of the raw solver pressure data.

37.8.6. Setting up Data Instancing Transformations

In this section, you will create additional copies of the original passages that replicate mesh nodes at different locations with correct space and time interpolation values. After the data instancing process, CFD-Post will create additional mesh nodes proportional to the number of extra passages created, and populate them with solution variables correctly updated to their corresponding position in time and space.

1. From the **Outline** tree view, edit `R1`.
2. In the **Data Instancing** tab, set **Number of Data Instances** to 42.
3. Click **Apply**.
4. From the **Outline** tree view, edit `S1`.
5. In the **Data Instancing** tab, set **Number of Data Instances** to 36.
6. Click **Apply**.
7. From the **Outline** tree view, edit `S2`.
8. In the **Data Instancing** tab, set **Number of Data Instances** to 37.
9. Click **Apply**.
10. Turn off the visibility of `Wireframe`.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

On the **3D Viewer** tab, CFD-Post displays the group of blades corresponding to a full wheel (the minimum number of blades that makes a unity pitch ratio between stator and rotor passages).

If you click the **Turbo** tab, then the **3D Viewer** shows the contour plot that you made earlier. Notice that the contour plot is now expanded to include the full wheel. Notice also that the solution varies around the wheel.

The data-dependent transient `forces on rotor blade` on `Chart 1` is still showing the result computed on a single blade passage. After you expand the number of rotor blades in the rotor passage to 42, the `R1 Blade` groups all 42 rotor blades together and the total force should be updated. To update the chart, click the **Refresh** button at the top of the **Chart Viewer**.

The `forces on rotor blade` expression is now being computed on all 42 blades in the extended number of passages in the rotor.

37.8.7. Animating the Movement of the Rotor Relative to the Stator

With the **Timestep Selector** set to time step `0`, you will make an animation showing the relative motion starting from this time step and lasting for one stator blade passing period.

1. Click the **3D Viewer** tab.
2. Position the geometry for the animation by right-clicking on a blank area in the viewer and selecting **Predefined Camera** > **View From -X**.
3. Click *Animation* .

   The **Animation** dialog box appears.
4. Set **Type** to **Keyframe Animation**.
5. Click *New* to create `KeyframeNo1`.
6. Select `KeyframeNo1`, then set **# of Frames** to `70`, then press **Enter** while in the **# of Frames** box.

   Tip:

   Be sure to press **Enter** and confirm that the new number appears in the list before continuing.

   This will place 70 intermediate frames between the keyframes, for a total of 72 frames.
7. Use the **Timestep Selector** to load time step `70` and then close the dialog box.
8. In the **Animation** dialog box, click *New* to create `KeyframeNo2`.
9. Click *More Animation Options* to expand the **Animation** dialog box.
10. Select **Save Movie**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11. Specify a filename for the movie.
12. Set **Format** to `MPEG1`.
13. Click *To Beginning* to rewind the active keyframe to `KeyframeNo1`.

    Note:

    The active keyframe is indicated by the value appearing in the **F:** field in the middle of the **Animation** dialog box. In this case it will be `1`.

    Wait for CFD-Post to finish loading the objects for this frame before proceeding.

14. Click *Save animation state* and save the animation to a file. This will enable you to quickly restore the animation in case you want to make changes. Animations are not restored by loading ordinary state files (those with the `.cst` extension).
15. Click *Play the animation* .

    Note:

    It takes a while for the animation to be completed. To view the movie file, you will need to use a media player that supports the MPEG format.

    From the animation and plots, you can see that the flow is continuous across the interface. This is because CFD-Post is capable of interpolating the flow field variables to the correct time and position using the computed Fourier coefficients.

16. When you have finished, close the **Animation** dialog box and then close CFD-Post, saving the animation state at your discretion.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 38: Film Cooling in a Turbine

This tutorial includes:

38.1. Tutorial Features

38.2. Overview of the Problem to Solve

38.3. Preparing the Working Directory

38.4. Defining the Case Using CFX-Pre

38.5. Obtaining the Solution Using CFX-Solver Manager

38.6. Viewing the Results Using CFD-Post

Important:

This tutorial requires file `TimeBladeRowIni_001.res`, which is produced by following tutorial Time Transformation Method for a Transient Rotor-stator Case (p. 799).

38.1. Tutorial Features

In this tutorial you will learn about:

- Initializing profile data.
- Expanding profile data.
- Defining injection regions and showing the injection positions in CFX-Pre and CFD-Post.
- Monitoring expressions during a solver run.
- Using streamlines in CFD-Post to trace the flow field from multiple specified points.

Component	Feature	Details
CFX-Pre	Injection Regions	Circular Hole
	Profile Data	Initialization
		Editing
	Analysis Type	Transient Blade Row
	Fluid Type	Air Ideal Gas
	Domain Type	Multiple Domains
		Rotating Frame of Reference
	Turbulence Model	Shear Stress Transport
	Heat Transfer	Total Energy

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Component	Feature	Details
	Boundary Conditions	Inlet (Subsonic)
		Outlet (Subsonic)
		Wall (Counter Rotating)
CFD-Post	Plots	Contour
		Point Cloud
		Streamline

38.2. Overview of the Problem to Solve

This tutorial models film cooling in a turbine using the Injection Regions feature of CFX.

You will build upon the geometry and setup of tutorial Time Transformation Method for a Transient Rotor-stator Case (p. 799), adjusting the inlet temperature and adding cooling holes to the rotor and hub. Near the end of this tutorial, you will produce a contour plot and streamlines to show the effect of injecting relatively cool air through the cooling holes.

If this is the first tutorial you are working with, it is important to review the following topics before beginning:

- Running Ansys CFX Tutorials Using Ansys Workbench (p. 34)
- Changing the Display Colors (p. 37)

38.3. Preparing the Working Directory

1. Create a working directory.

   Ansys CFX uses a working directory as the default location for loading and saving files for a particular session or project.

2. Download the `film_cooling.zip` file here.

3. Unzip `film_cooling.zip` to your working directory.

   Ensure that the following tutorial input files are in your working directory:

   - `FCInjectionPositions.csv`
   - `TBRInletProfile.csv`
   - `TBROutletProfile.csv`
   - `TBRTurbineRotor.gtm`
   - `TBRTurbineStator.gtm`
   - `TimeBladeRowIni_001.res` (produced by following tutorial Time Transformation Method for a Transient Rotor-stator Case (p. 799))

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Set the working directory and start CFX-Pre.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

38.4. Defining the Case Using CFX-Pre

1. Open the results file named `TimeBladeRowIni_001.res`.
2. Save the case as `FilmCooling.cfx` in your working directory.

38.4.1. Changing the Inlet Temperature

Adjust the inlet temperature as follows:

1. Edit `S1 Inlet`.
2. On the **Boundary Details** tab, set **Heat Transfer** > **Total Temperature** to `1000 [K]`.
3. Click **OK**.

38.4.2. Initializing Profile Data

File `FCInjectionPositions.csv` is a profile data file that contains information about specific injection positions. You can optionally view this file's contents using a text editor. The file contains 3 function definitions, each consisting of a name, optional parameter data, a declaration of the spatial field variables, and table data. Note that the function names, parameter names, and table column labels must be CEL-compatible; these names may contain spaces but must not contain the underscore character or any symbols.

Load `FCInjectionPositions.csv`:

1. Select **Tools** > **Initialize Profile Data**.

   The **Initialize Profile Data** dialog box appears.

2. Beside **Profile Data File**, click *Browse* .

   The **Select Profile Data File** dialog box appears.

3. Select `FCInjectionPositions.csv` and click **Open**.
4. In the **Initialize Profile Data** dialog box, select all three **Visibility** check boxes so that, in the viewer, symbols will appear at the (planned) injection positions.

   Note that the symbol visibility can also be controlled in the **Outline** tree view, by selecting or clearing the check boxes under `Simulation` > `Expressions, Functions and Variables` > `User Functions`.

5. Click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The profile data is loaded and initialized. Note that an additional three profile functions appear in the **Outline** tree view, under `Simulation` > `Expressions, Functions and Variables` > `User Functions`:

- `Hub Cooling`
- `Rotor Pressure Surface Cooling`
- `Rotor Suction Surface Cooling`

As an exercise, adjust the user function symbols that show the injection positions:

1. In the **Outline** tree view, right-click `Simulation` > `Expressions, Functions and Variables` > `User Functions` > `Hub Cooling` and select **Render** > **Properties**.

   The **Render Options** dialog box appears. Note that there are settings for faces, lines, and symbols.

2. Set **Symbol** to `Ball`.

3. Set **Symbol Size** to `0.5`.

4. Click **OK**.

   In the viewer, the symbols that mark the injection positions (cooling hole locations) on the hub are now shown as balls.

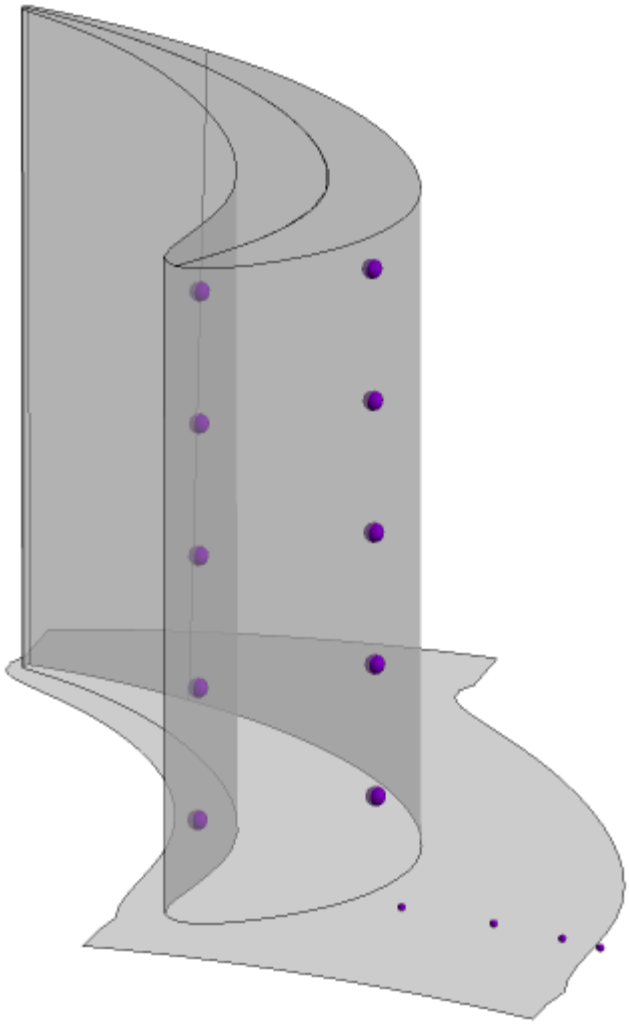
5. Using the same procedure, change the shape and size of the user function symbols that show the injection positions on the rotor pressure and suction surfaces.

6. If the curled-arrow symbols that mark interfaces are shown in the viewer, hide them by right-clicking the viewer background and deselecting **Markers** > **Interface**.

7. Note that, as shown in Figure 38.1: Injection Positions (p. 947), one of the injection positions on the hub is slightly outside the modeled passage; any such injection position is effectively relocated (by the CFX-Solver) to a corresponding position between the periodic interfaces. You can therefore create injection positions without having to know the exact locations of the periodic interfaces.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 38.1: Injection Positions



Note:

- If the injection positions (cooling hole locations) require rotation, added instances, or other transformations, you can use the **Edit Profile Data** dialog box to edit the profile data file. For details, see Edit Profile Data in the *CFX-Pre User's Guide*.
- If you are editing the profile data with an application other than CFX-Pre or a plain text editor, ensure that you preserve a reasonable number of significant digits in the data.

38.4.3. Preparing an Expanded Profile for Later Use

In the previous section, you noted that one of the injection positions is slightly outside the modeled passage.

Later in this tutorial, you will require a modified copy of the profile that, within the modeled passage in particular, has all of the modeled injection positions.

Create an expanded version of the profile:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select **Tools** > **Edit Profile Data**.

   The **Edit Profile Data** dialog box appears.

2. Under **Source Profile**, click *Browse* .

   The **Select Profile Data File** dialog box appears.

3. From your working directory, select `FCInjectionPositions.csv` and click **Open**.

4. Set **Write to Profile** to `FCInjectionPositions_2p.csv`.

5. **IMPORTANT:** Ensure that **Initialize New Profile After Writing** is cleared in order to prevent the profile data from being initialized with the expanded profile.

6. In the **Transformations** frame, click *Add new item* , set **Name** to `Transformation 1`, and click **OK**.

7. Configure the following setting(s):

Setting	Value
Transformation 1 > **Option**	Expansion
Transformation 1 > Expansion Definition > **Rotation Option**	Principal Axis
Transformation 1 > Expansion Definition > **Axis**	Z
Transformation 1 > Expansion Definition > **Passages in Profile**	1
Transformation 1 > Expansion Definition > **Passages in 360**	42
Transformation 1 > Expansion Definition > **Expansion Option**	Spec. Number of Passages
Transformation 1	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
> Expansion Definition > **Pass. in Expansion**	 2
Transformation 1 > Expansion Definition > **Theta Offset**	 -8.57143 [degree]

8. Click **OK**.

The new profile (which is not applied to the simulation, but which is available as file `FCInjection-Positions_2p.csv`) covers 2 passages. This profile will be needed in the postprocessing section of this tutorial. Although the expanded profile contains many unneeded points outside the modeled passage, it (importantly) contains *all* of the injection positions *within the modeled passage*.

38.4.4. Mapping Data to Injection Region Settings

As a best practice, you will create associations between the user function data and injection region settings. This will prepare CFX-Pre to automatically configure many of the injection region settings later in this tutorial. Note that these associations are not required; you can always choose to specify any or all of the injection region settings manually.

Edit the three relevant user functions as follows:

1. In the **Outline** tree view, right-click `Simulation` > `Expressions, Functions and Variables` > `User Functions` > `Hub Cooling` and select **Edit**.
2. Under **Parameters**, select **MassFlow** (in the list).
3. Select **MassFlow** > **Parameter List**.
4. Set **Parameter List** to `Injection Mass Flow Rate`.
5. Also, make the following associations for user function `Hub Cooling`:
   - With **Value Fields** set to `Cooling Temperature`, set **Parameter List** to `Stationary Frame Total Temperature`.
   - With **Value Fields** set to `Hole Diameter`, set **Parameter List** to `Diameter`.
6. Click **OK**.
7. Make the following associations for user function `Rotor Pressure Surface Cooling` (clicking **OK** when done):
   - With **Parameters** set to `Diameter`, ensure that **Parameter List** is set to `Diameter`.
   - With **Parameters** set to `InjectionTemperature`, set **Parameter List** to `Stationary Frame Total Temperature`.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- With **Value Fields** set to `Mass Flow`, set **Parameter List** to `Injection Mass Flow Rate`.

8. Make the same associations for user function `Rotor Suction Surface Cooling` as for user function `Rotor Pressure Surface Cooling` (clicking **OK** when done).

38.4.5. Adding Cooling Holes to the Rotor and Hub

Now that you have initialized the profile data and have created associations between the profile data and injection region settings, it is time to use the profile data to create cooling holes.

Add cooling holes to the rotor as follows:

1. Select **Insert** > **Injection Region** > **in R1**.

   The **Insert Injection Region** dialog box appears.

2. Set the name to `Rotor Pressure Surface Cooling Holes` and click **OK**.

3. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	**Region Type**	Inlet
	Injection Location Type > **Option**	Selected 2D Regions
	Injection Location Type > **Location**	BLADE 2
	Injection Positions > **Option**	From Profile
	Injection Positions > **Profile Name**	Rotor Pressure Surface Cooling
	Injection Positions > **Generate Values**	(Click) This fills in settings on the **Injection Details** tab using the associations that you defined earlier in the `Hub Cooling` user function.
	Coordinate Frame	Coord 0
	Frame Type	Rotating

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Injection Details	Injection Model > **Option**	 Circular Hole
	Injection Model > **Diameter**	 Rotor Pressure Surface Cooling.Diameter()
	Mass And Momentum > **Option**	 Injection Mass Flow Rate
	Mass and Momentum > **Mass Flow Rate**	 Rotor Pressure Surface Cooling.Mass Flow(x,y,z)
	Mass and Momentum > **Mass Flow Rate Area**	 Per Injection Position
	Flow Direction > **Option**	 Normal to Boundary Condition
	Heat Transfer > **Option**	 Stationary Frame Total Temperature
	Heat Transfer > **Stat. Frame Tot. Temp**	 Rotor Pressure Surface Cooling.InjectionTemperature()

4. Click **OK**.

Using the same procedure, add an injection region, `Rotor Suction Surface Cooling Holes`, for the cooling holes on the rotor suction surface, using the `Rotor Suction Surface Cooling` profile data for the injection positions, diameter, and mass flow.

Add cooling holes to the hub as follows:

1. Select **Insert** > **Injection Region** > **in R1**.

   The **Insert Injection Region** dialog box appears.

2. Set the name to **Hub Cooling Holes** and click **OK**.

3. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Basic Settings	**Region Type**	Inlet
	Injection Location Type > **Option**	Selected 2D Regions
	Injection Location Type > **Location**	Entire HUB 2
	Injection Positions > **Option**	From Profile
	Injection Positions > **Profile Name**	Hub Cooling
	Injection Positions > **Generate Values**	(Click)
	Coordinate Frame	Coord 0
	Frame Type	Rotating
Injection Details	Injection Model > **Option**	Circular Hole
	Injection Model > **Diameter**	Hub Cooling.Hole Diameter(x,y,z)
	Mass And Momentum > **Option**	Injection Mass Flow Rate
	Mass and Momentum > **Mass Flow Rate**	Hub Cooling.MassFlow()
	Mass and Momentum > **Mass Flow Rate Area**	As Specified (This indicates that the amount specified is the total over all 4 modeled holes on the hub.)
	Flow Direction	

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	> **Option**	Normal to Boundary Condition
	Heat Transfer > **Option**	 Stationary Frame Total Temperature
	Heat Transfer > **Stat. Frame Tot. Temp**	 Hub Cooling.Cooling Temperature(x,y,z)

4. Click **OK**.

38.4.6. Using an Injection Region as a Locator in a CEL Expression

An injection region can serve as a locator in a CEL expression in CFX-Pre and in CFX-Solver. Such an expression could be used, for example, to create a solver monitor.

As an exercise, create a solver monitor that monitors an expression for the area average of pressure on the hub cooling holes.

1. Select **Insert** > **Solver** > **Output Control**.

   The **Output Control** details view appears.

2. On the **Monitor** tab, under **Monitor Points and Expressions**, click *Add new item* .

   The **Insert Monitor Point** dialog box appears.

3. Set **Name** to `Average Pressure on Hub Cooling Holes` and click **OK**.

4. Under **Monitor Points and Expressions** > **Average Pressure on Hub Cooling Holes**:

   1. Set **Option** to `Expression`.

   2. Set **Expression Value** to `areaAve(Pressure)@Hub Cooling Holes`.

5. Click **Apply**.

As an exercise, create the following monitors that will be used later in this tutorial to confirm that there is conservation of mass in the rotor passage:

Monitor Name	Expression Value
Rotor Pressure Surface	massFlow()@Rotor Pressure Surface Cooling Holes

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Monitor Name	Expression Value
Cooling Hole Mass Flow	
Rotor Suction Surface Cooling Hole Mass Flow	massFlow()@Rotor Suction Surface Cooling Holes
Hub Cooling Hole Mass Flow	massFlow()@Hub Cooling Holes
Rotor Passage Inlet Mass Flow	massFlow()@REGION:Entire INFLOW 2
Rotor Passage Outlet Mass Flow	massFlow()@REGION:Entire OUTFLOW 2
Net Flow into Rotor Passage	massFlow()@Rotor Pressure Surface Cooling Holes + massFlow()@Rotor Suction Surface Cooling Holes + massFlow()@Hub Cooling Holes + massFlow()@REGION:Entire INFLOW 2 + massFlow()@REGION:Entire OUTFLOW 2

38.4.7. Modifying Execution Control

Because the current case was built upon the results of another case, a definition file (*.def) name is stored in the Execution Control settings. Update that name to reflect the current case as follows:

1. Click *Execution Control* .
2. Configure the following setting(s):

Tab	Setting	Value
Run Definition	Input File Settings > **Solver Input File**	FilmCooling.def [a]

a. You do not need to set the full path unless you are saving the solver file somewhere other than the working directory.

3. Confirm that the rest of the execution control settings are set appropriately.
4. Click **OK**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

38.4.8. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, **Solver Input File** is set.

2. If using stand-alone mode, quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

38.5. Obtaining the Solution Using CFX-Solver Manager

1. In CFX-Solver Manager, ensure that the **Define Run** dialog box is displayed.
2. Under the **Initial Values** tab, select **Initial Values Specification**.
3. Under **Initial Values Specification** > **Initial Values**, select `Initial Values 1`.
4. Under **Initial Values Specification** > **Initial Values** > **Initial Values 1 Settings** > **File Name**, click *Browse* .
5. Select `TimeBladeRowIni_001.res` from your working directory.
6. Click **Open**.
7. Click **Start Run**.

   CFX-Solver runs and attempts to obtain a solution.

8. As an exercise, show monitors for the mass flow through each set of cooling holes:

   1. In the CFX-Solver Manager, select **Workspace** > **New Monitor**.

      The **New Monitor** dialog box appears.

   2. Set **Name** to `Injection Regions`.
   3. Set **Type** to `Plot Monitor`.
   4. Click **OK**.

      The **Monitor Properties: Injection Regions** dialog box appears.

   5. Under `Plot Line Variable`, expand `FLOW` > `INJECTION REGION`, then select the following:

      - `Hub Cooling Holes` > `P-Mass Injection Region Flow on Hub Cooling Holes (R1)`
      - `Rotor Pressure Surface Cooling Holes` > `P-Mass Injection Region Flow on Rotor Pressure Surface Cooling Holes (R1)`

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

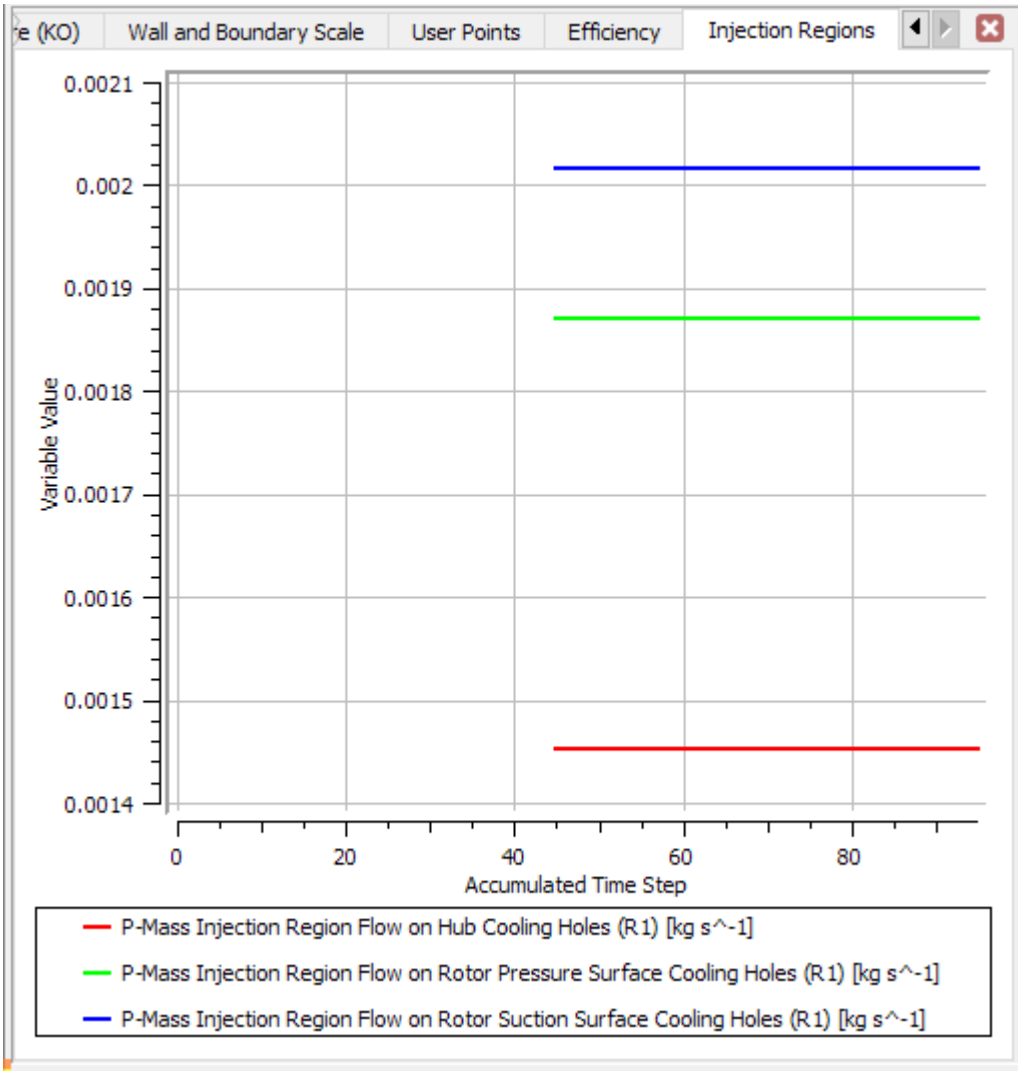
- `Rotor Suction Surface Cooling Holes > P-Mass Injection Region Flow on Rotor Suction Surface Cooling Holes (R1)`

6. Click **OK**.

   As shown in Figure 38.2: Mass Flow through each Injection Region (p. 956), a new monitor plot appears in the CFX-Solver Manager showing three horizontal lines; these lines correspond to the total mass flows through:

   - The 4 hub cooling holes
   - The 5 rotor pressure surface cooling holes
   - The 5 rotor suction surface cooling holes.

Figure 38.2: Mass Flow through each Injection Region



9. Show the monitors that you created earlier, in CFX-Pre:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click the **User Points** tab.

   Observe that the monitor **Average Pressure on Hub Cooling Holes** is plotted against `Accumulated Time Step`, as shown by the topmost plot line in Figure 38.3: User Points versus Accumulated Time Step (p. 958).

   The other monitors that you defined are also shown, but not currently at an appropriate scale for viewing.

2. Right-click inside the **User Points** tab and select **Monitor Properties**.

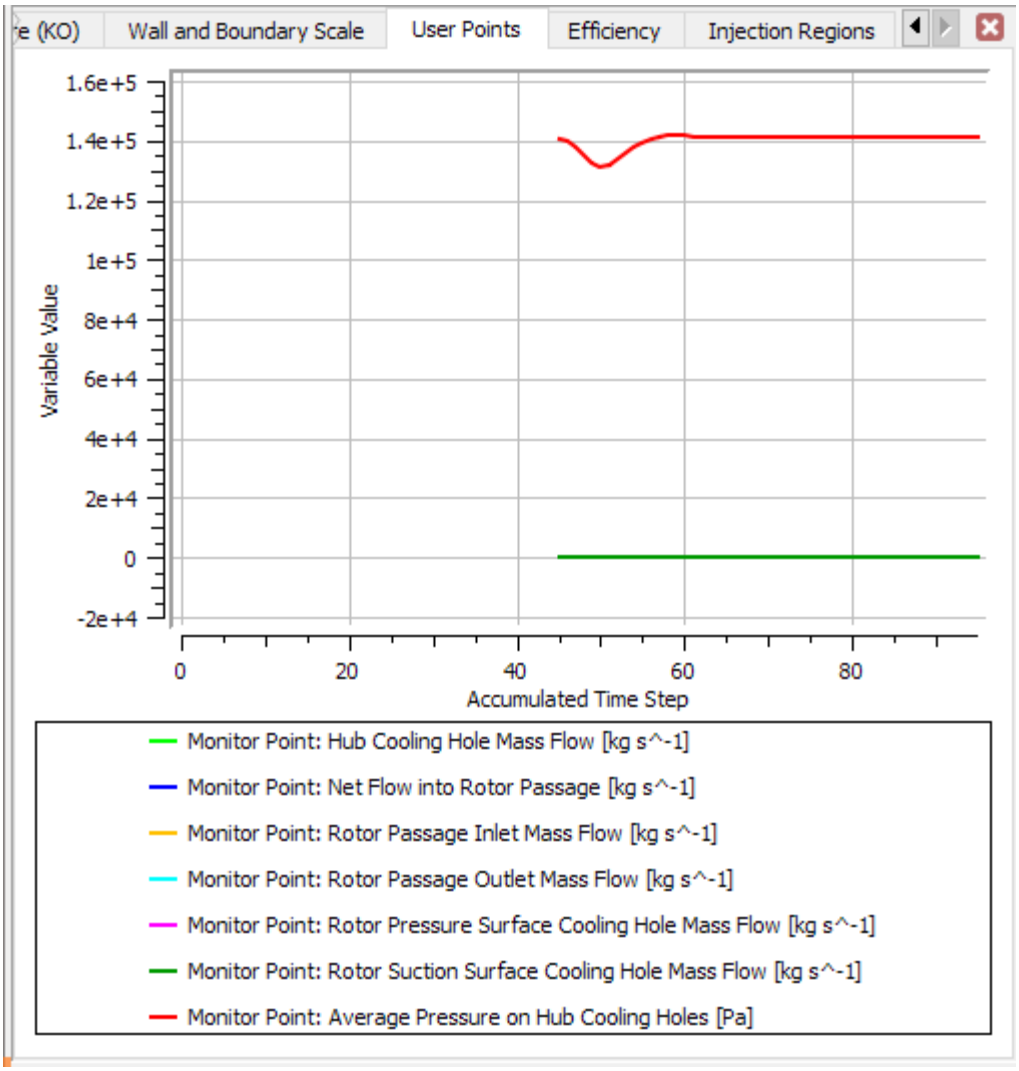
   The **Monitor Properties: User Points** dialog box appears.

3. In the dialog box, on the **Plot Lines** tab, clear `USER POINT` > `Average Pressure on Hub Cooling Holes`, then click **OK**.

   With the plot line **Average Pressure on Hub Cooling Holes** omitted, the chart is now re-scaled so that the remaining plot lines are easier to see, as shown in Figure 38.4: Conservation-related User Points versus Accumulated Time Step (p. 959). The plot line `Net Flow into Rotor Passage` approaches zero, indicating that mass flow is conserved in the rotor passage, as expected.

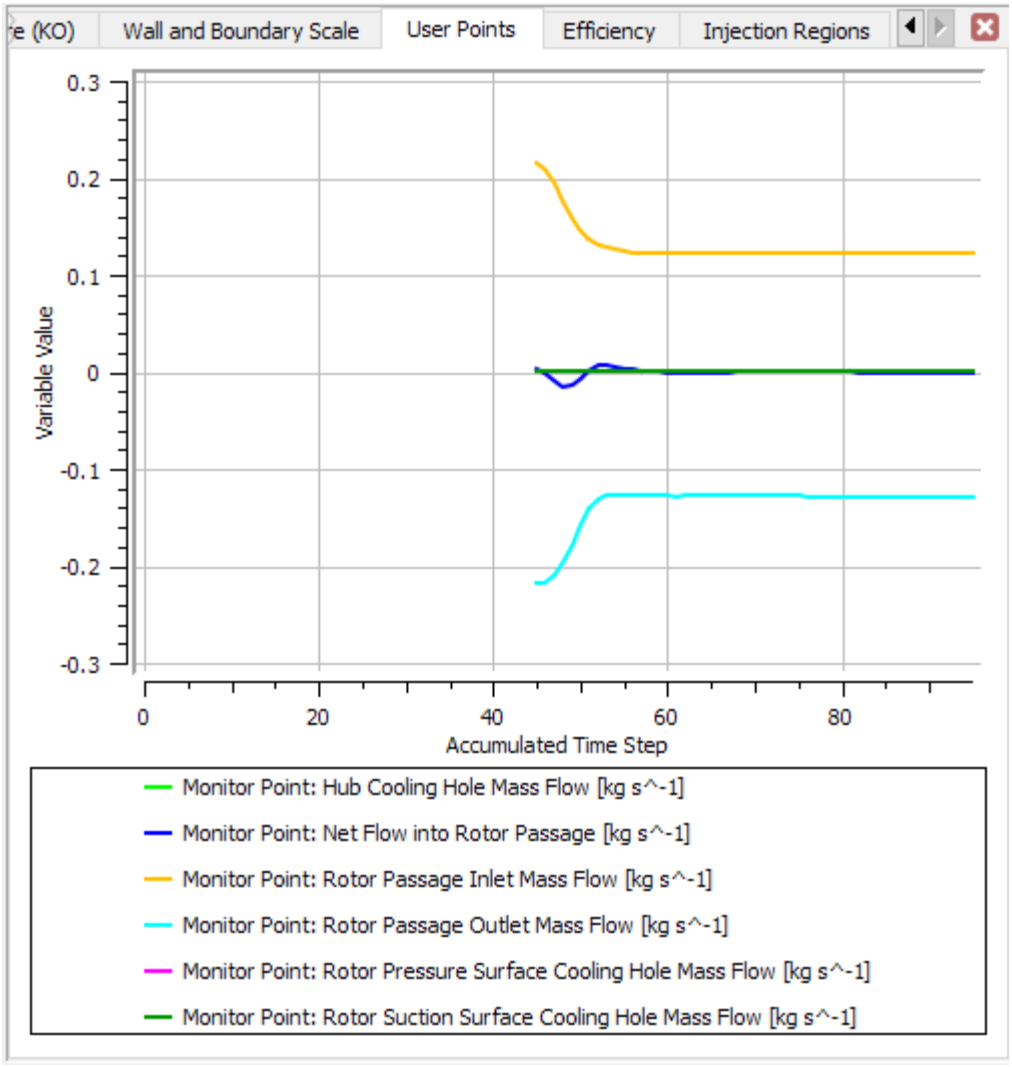
Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 38.3: User Points versus Accumulated Time Step



Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 38.4: Conservation-related User Points versus Accumulated Time Step



At the end of the run, a dialog box is displayed stating that the simulation has ended.

10. When CFX-Solver is finished, select the check box next to **Post-Process Results**.
11. If using stand-alone mode, select the check box next to **Shut down CFX-Solver Manager**.
12. Click **OK**.

38.6. Viewing the Results Using CFD-Post

In the following sections, you will:

- Show the mesh lines in the rotor domain.
- Create a contour plot that shows the temperature on the hub and rotor.
- Use the expanded profile data that you prepared earlier (`FCInjectionPositions_2p.csv`) to create point clouds that represent the cooling hole locations.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Plot streamlines that originate from all cooling hole locations in the modeled rotor passage.

38.6.1. Showing the Mesh Lines

1. In the **Outline** tree view, select the check boxes for `R1 Blade` and `R1 Hub`.
2. Edit `R1 Blade`.
3. In the details view, on the **Render** tab, clear **Show Faces**, select **Show Mesh Lines**, and click **Apply**.
4. Edit `R1 Hub` similarly.
5. Turn off the visibility of `User Locations and Plots` > `Wireframe`.

38.6.2. Creating a Contour Plot of Temperature

1. Click **Insert** > **Contour** and accept the default name.
2. Configure the following setting(s):

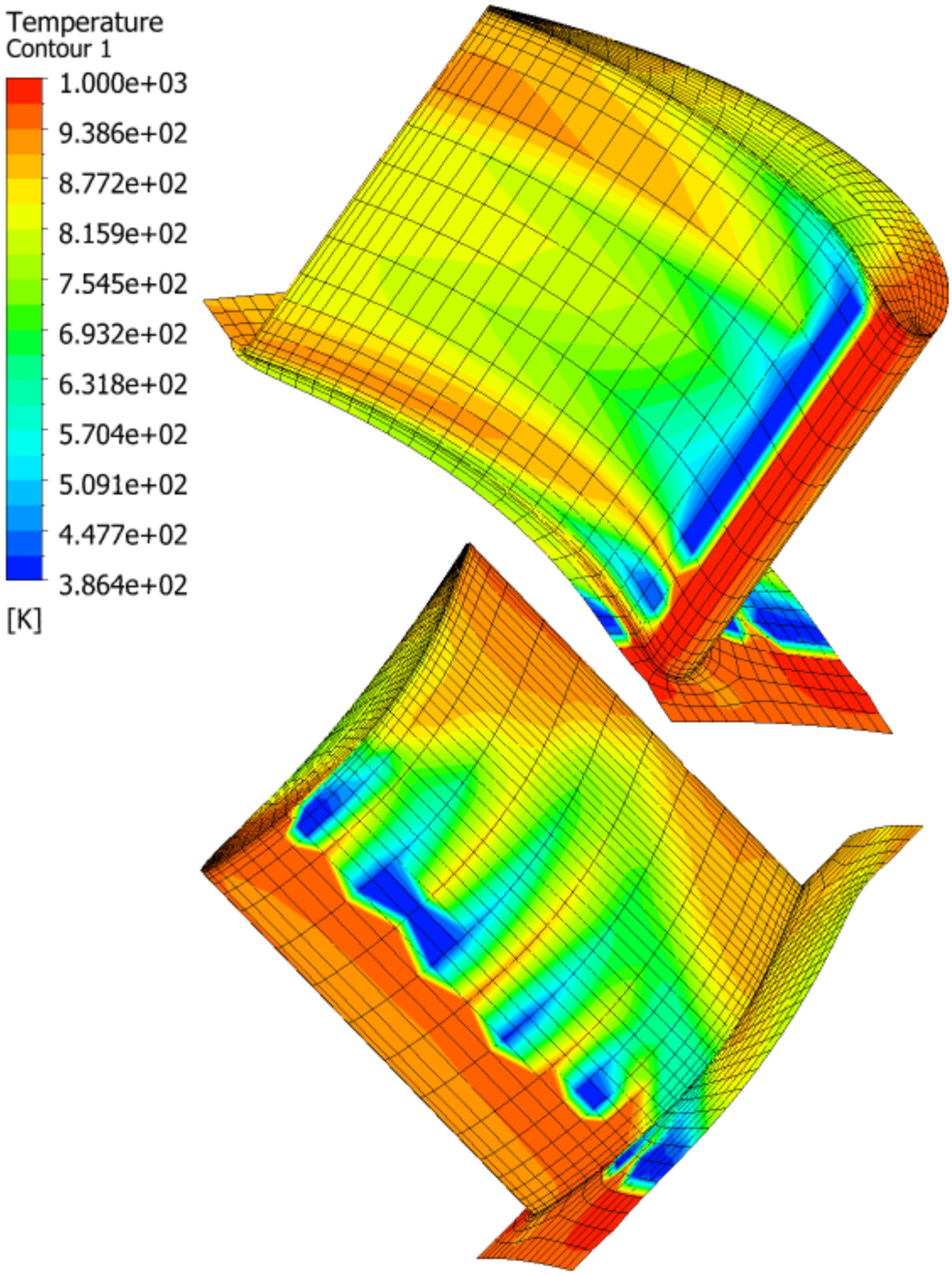
Tab	Setting	Value
Geometry	**Locations**	R1 Blade,R1 Hub
	Variable	Temperature
	Range	Global
	# of Contours	21
Render	Show Contour Bands > **Lighting**	 (Cleared)

3. Click **Apply**.

The contour plot shows `Temperature` values on the rotor and hub. As shown in Figure 38.5: Temperature on Blade and Hub (p. 961), the effect of film cooling is apparent.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 38.5: Temperature on Blade and Hub



Increasing the mesh resolution would better resolve the circular shape of the cooling holes. A case with film cooling is typically simulated using a finer mesh than used here. As a best practice, there

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

should be 3 to 5 elements across the hole diameter for any given cooling hole. As expected, flow quantities are conserved, regardless of the mesh resolution.

Note:

For a coarse mesh in which the mesh faces at an injection position are larger than the hole size, you must set, in the **Solver Control** details view, on the **Advanced Options** tab, **Injection Control** > **Surface Injection Control** > **Surface Overlap Type** to `Nearest Face`. Note that, as a side effect of using this option, each cooling hole is resolved by only one mesh face. Also note that, under this option, any given mesh face is permitted to resolve multiple cooling holes.

38.6.3. Showing the Cooling Hole Locations using Point Clouds

In this section, you will create point clouds to show the injection positions.

Later in this tutorial, you will use these point clouds as starting locations for streamlines. As mentioned earlier in this tutorial, one of the injection positions is outside the mesh. Because no streamlines can be produced from locations outside the mesh, make the point clouds using the expanded profile that you prepared earlier:

1. Select **File** > **Import** > **Import Surface, Line or Point Data**.

   The **Import Surface, Line or Point Data** dialog box appears.

2. Set **File** to `FCInjectionPositions_2p.csv`.

3. Set **Import As** to `Point Cloud`.

4. Click **OK**.

   In the **Outline** tree view, under `User Locations and Plots`, three point cloud objects appear. These objects correspond to the three profile data sets in `FCInjectionPositions_2p.csv`, which represent all cooling hole locations (in two passages) for the hub, rotor pressure surface, and rotor suction surface.

5. Edit `User Locations and Plots` > `Imported Hub Cooling`.

6. In the details view, on the **Symbol** tab, set **Symbol** to `Ball` and **Symbol Size** to `0.25`.

7. Click **Apply**.

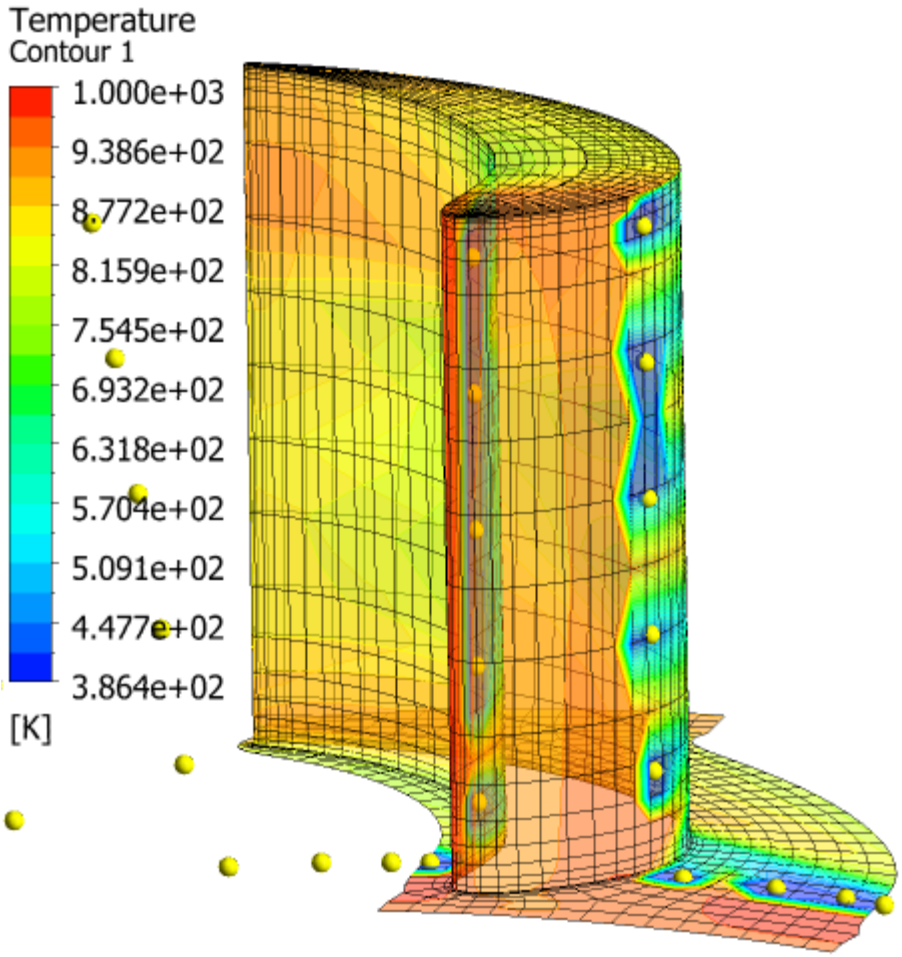
   The viewer shows ball symbols at the cooling hole locations on the hub.

8. Change the symbols similarly for the other two point cloud objects.

You should see all cooling hole locations inside the modeled passage (as well as many outside the passage), as shown in Figure 38.6: Hole Locations shown using Point Cloud (p. 963).

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 38.6: Hole Locations shown using Point Cloud



38.6.4. Creating Streamlines from the Point Clouds

In this section, you will produce streamlines from the point clouds, which contain all of the cooling hole locations within the modeled passage (and others outside the passage). Streamlines will start from, and only from, the points that are within the passage.

1. From the main menu, select **Insert** > **Streamline** .

   The **Insert Streamline** dialog box appears.

2. Accept the default name and click **OK**.

3. Set **Definition** > **Start From** to `Imported Hub Cooling,Imported Rotor Pressure Surface Cooling,Imported Rotor Suction Surface Cooling`.

4. To ensure that every injection hole in the passage has an associated streamline, set **Definition** > **Reduction** to `Reduction Factor` and **Definition** > **Factor** to `1.0`.

5. Ensure that **Cross Periodics** is selected.

6. Click the **Color** tab.

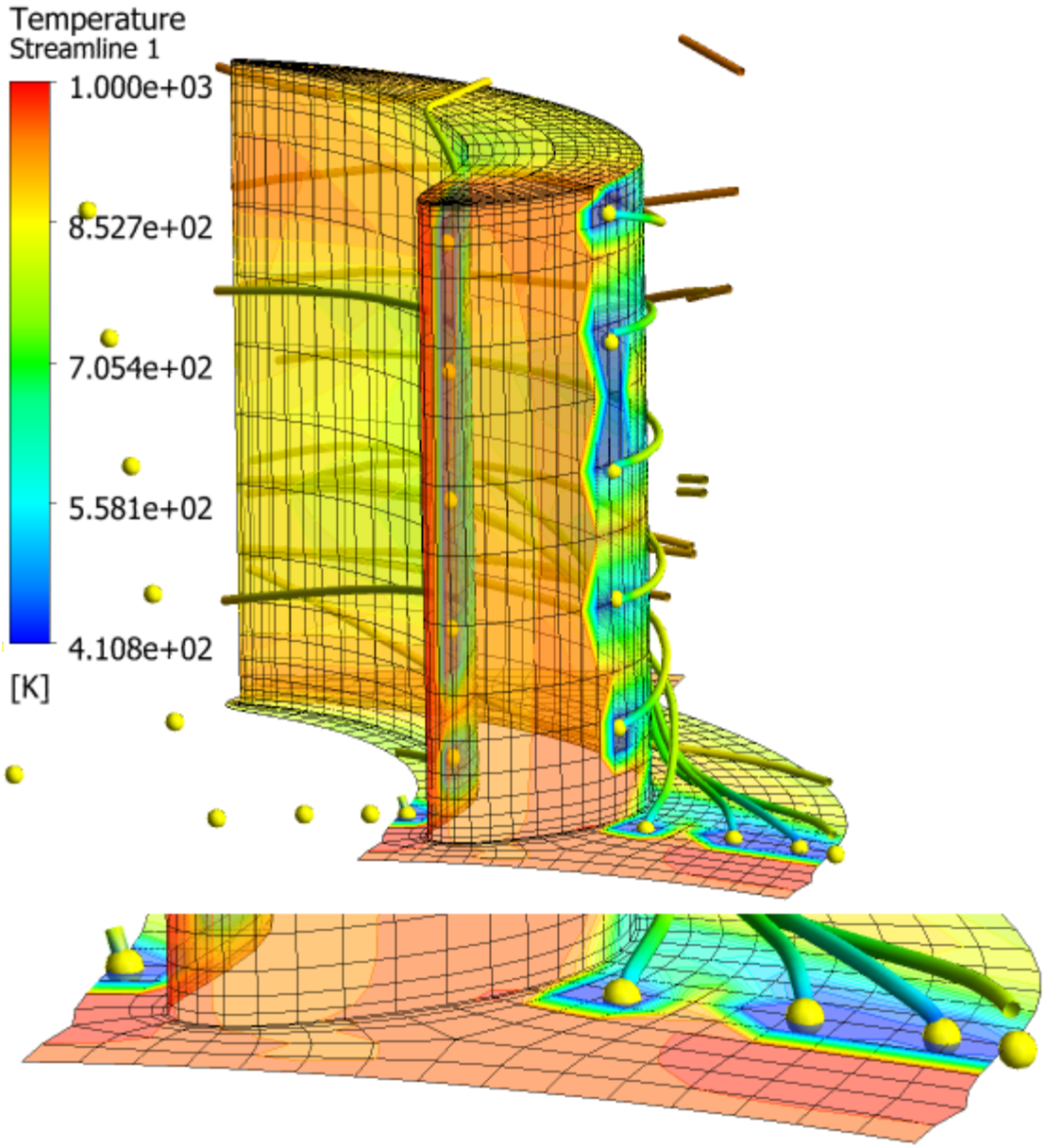
Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Set **Mode** to `Variable`.
8. Set **Variable** to `Temperature`.
9. Set **Range** to `Global`.
10. Click the **Symbol** tab.
11. Ensure that **Show Streams** is selected.
12. Set **Show Streams** > **Stream Type** to `Tube`.
13. Set **Show Streams** > **Tube Width** to `1.0`.
14. Click **Apply**.

    Streamlines start from each cooling hole in the passage, as shown in Figure 38.7: Streamlines starting from Cooling Holes (p. 965).

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 38.7: Streamlines starting from Cooling Holes



In Figure 38.7: Streamlines starting from Cooling Holes (p. 965), the bottom right point (shown as a ball) marks a specified cooling hole location. That cooling hole is outside the modeled passage, but took effect (in the simulation) at the corresponding location (bottom left) within the passage, as confirmed by the contour plot. A rotated copy of the point (made using an expanded version of the profile data) was placed at that corresponding location so that a streamline would be produced for that cooling hole. Because the streamline was configured to cross periodic boundaries, the streamline exits at the bottom left then resumes at the bottom right.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 39: Operating Map for a Centrifugal Compressor

This tutorial includes:

39.1. Tutorial Features

39.2. Overview of the Problem to Solve

39.3. Preparing the Working Directory

39.4. Defining the Case Using CFX-Pre

39.5. Obtaining the Solution Using CFX-Solver Manager

39.6. Viewing the Results Using CFD-Post

39.1. Tutorial Features

In this tutorial you will learn about:

- Controlling a simulation with operating points.
- Working with operating map charts in CFD-Post.

Component	Feature	Details
CFX-Pre	User Mode	General mode
	Analysis Type	Steady State
	Fluid Type	Air Ideal Gas
	Domain Type	Single Domain
	Turbulence Model	Shear Stress Transport
	Heat Transfer	Total Energy
	Boundary Conditions	Inlet (Subsonic)
		Outlet (Subsonic)
		Wall: No-Slip
		Wall: Adiabatic
	Timescale	Physical Timescale
CFX-Solver Manager	Define Run Dialog Box	Operating Points Tab
	Operating Point Run History Page	Operating Point Parameter Table
CFD-Post	Operating Points Viewer	Operating Point Parameter Table
		Operating Map Chart

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

39.2. Overview of the Problem to Solve

This tutorial uses the operating point functionality of Ansys CFX to simulate a centrifugal compressor running over a range of speeds and mass flow rates; each combination of speed and mass flow rate constitutes an operating point.

You will use CFX-Pre and CFD-Post to create operating map charts from the simulation results. Such operating map charts can, in principle, reveal performance characteristics of the centrifugal compressor.

In CFX, operating maps can be generated using an intermediate mathematical construct known as a *response surface*. As an exercise, you will check quality metrics of a response surface; these metrics are an indication of the accuracy of the resulting operating maps. You will also check the quality of a response surface visually.

Using a provided case as a starting point, you will:

- Define operating point input parameter values in a table function, with each row of table data to be used in prescribing a corresponding operating point.
- Define operating point output parameters as monitors in the `Output Control` object.
- Define the `Operating Points` object, including an optional pre-defined operating map chart definition, using the previously defined table function as the source of operating point input parameters.
- Modify the case to make use of the operating point input parameters.
- Run the operating point case, which involves the CFX-Solver running one simulation (operating point job) for each of the prescribed operating points.
- In CFD-Post, add contour lines to the operating map chart that was pre-defined in CFX-Pre.
- Generate a new operating map chart in CFD-Post to check the quality of a response surface that was used to construct the original operating map chart.

Note:

Operating point cases are not supported in Ansys Workbench.

39.3. Preparing the Working Directory

1. Create a working directory.

   Ansys CFX uses a working directory as the default location for loading and saving files for a particular session or project.

2. Download the `Compressor_OM.zip` file here.

3. Unzip `Compressor_OM.zip` to your working directory.

   Ensure that the following tutorial input files are in your working directory:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- `Compressor_OM.ccl`
- `Compressor_OM.gtm`

4. Set the working directory and start CFX-Pre.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

39.4. Defining the Case Using CFX-Pre

1. Select **File** > **New Case**.
2. Select **General** and click **OK**.
3. Select **File** > **Import** > **Mesh**.

   The **Import Mesh** dialog box appears.
4. Set **Files of type** to `CFX Mesh (*gtm *cfx)`.
5. Set **File name** to `Compressor_OM.gtm` and click **Open**.
6. Select **File** > **Import** > **CCL**.

   The **Import CCL** dialog box appears.
7. Set **File name** to `Compressor_OM.ccl`, ensure that **Import Method** is set to **Replace**, and click **Open**.
8. Select **File** > **Save Case**.

   The **Save Case** dialog box appears.
9. Set **File name** to `Compressor_OM.cfx` and click **Save**.

39.4.1. Creating a Table Function for Operating Point Input Parameters

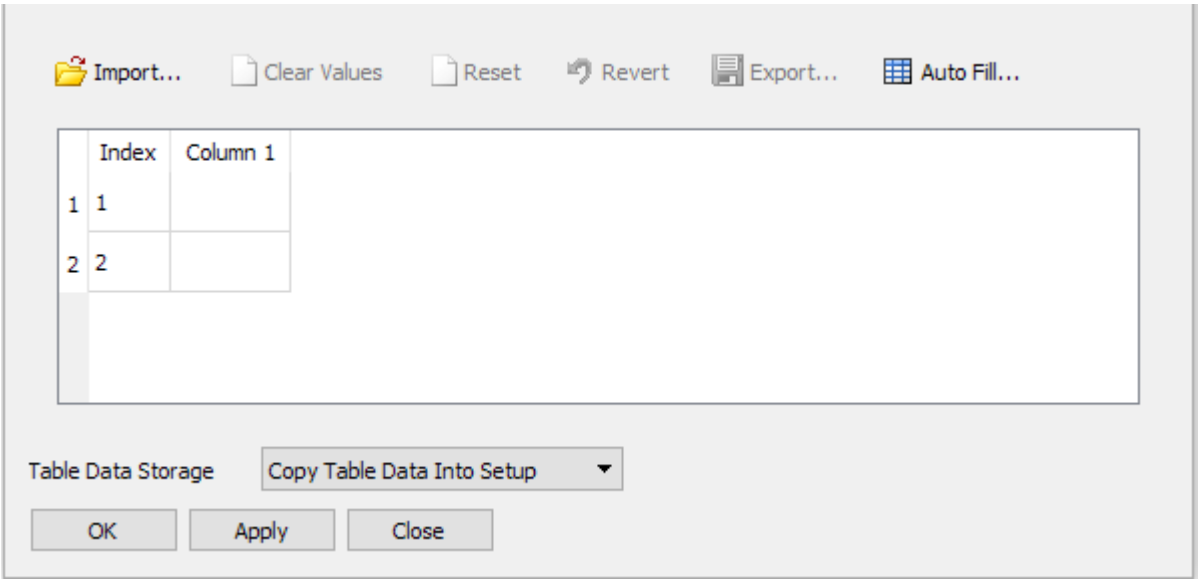
In this section, you will generate a table function that will provide input parameter values for operating points. You will use the **Auto Fill Values** dialog box to automatically generate data for the table function.

1. In the **Outline** tree view, right-click `Simulation` > `Expressions, Functions and Variables` > `User Functions` and select **Insert** > **Table Function**.

   The **Insert Function** dialog box appears.
2. Set **Name** to `Speeds and Flow Rates` and click **OK**.

   The table editor appears.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Edit the table header as follows:

   a. Right-click column header `Column 1` and select **Edit Column Header**.

      The **Edit Column Header** dialog box appears.

   b. Configure the following settings:

Setting	Value
Name	Speed
Units	rev min^-1

   c. Click **OK**.

      The column header is renamed. You might have to resize the column (drag the column header's right edge using the mouse) to see the entire label.

   d. Right-click column header `Speed [rev min^-1]` and select **Insert After**.

      A new column appears, named `Column 2`.

   e. Right-click column header `Column 2` and select **Edit Column Header**.

      The **Edit Column Header** dialog box appears.

   f. Configure the following settings:

Setting	Value
Name	CorrMassFlowRate
Units	kg s^-1

      Note that spaces are permitted in column names.

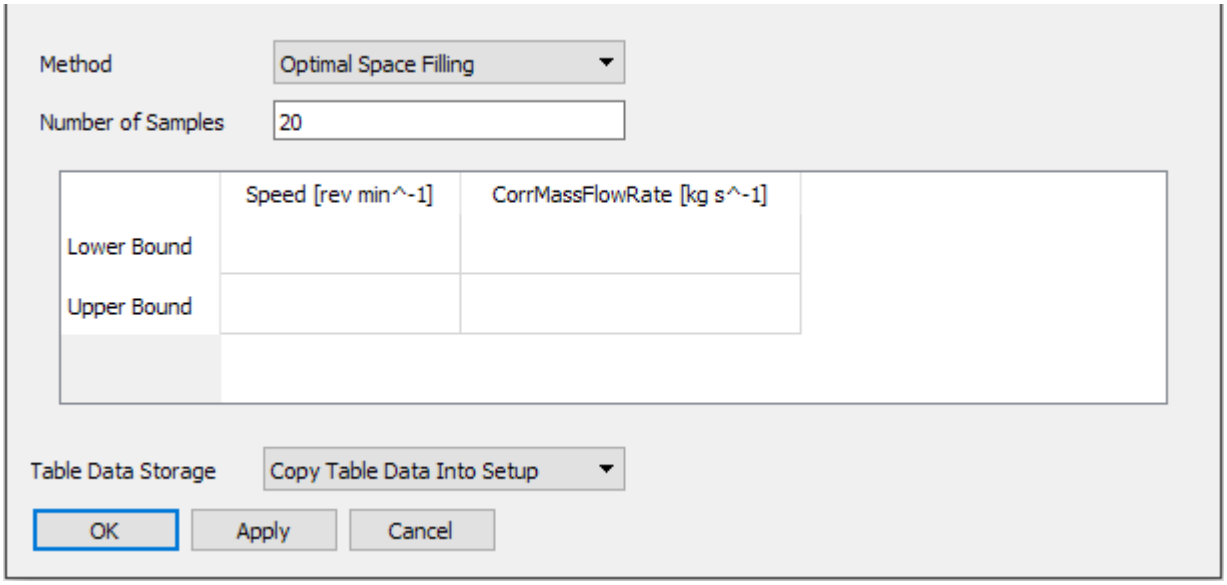
Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

g. Click **OK**.

The column header is renamed. Again, you might have to resize the column to see the entire label.

4. Click **Auto Fill**.

The **Auto Fill Values** dialog box appears (replacing the table editor).



5. Set **Number of Samples** to `30`.

6. Configure the following settings, clicking once in each cell before typing a value, or clicking twice (but not double-clicking) before pasting a value with **Ctrl** + **v**:

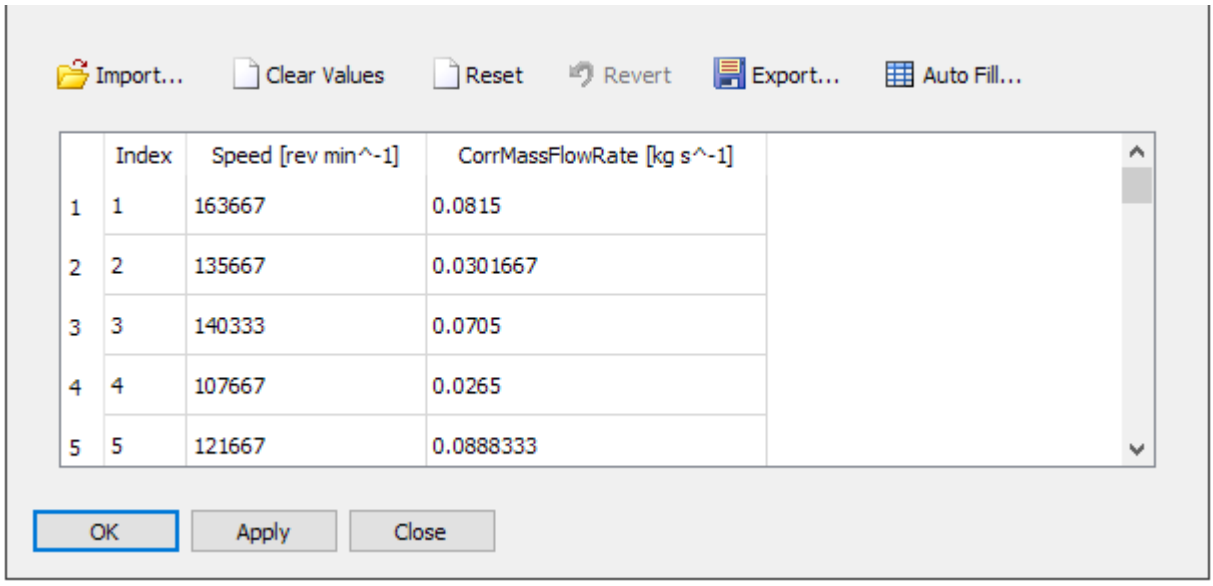
Row	Column	Value
Lower Bound	Speed	40000
Upper Bound	Speed	180000
Lower Bound	CorrMassFlowRate	0.01
Upper Bound	CorrMassFlowRate	0.12

7. Ensure that **Table Data Storage** is set to `Copy Table Data Into Setup`.

This will keep the table data with the case instead of writing it to an external (referenced) file. An external file enables other software to update the table data, but creates a need to keep the external file with the case.

8. Click **Apply** to close the **Auto Fill Values** dialog box and return to the table editor.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note that, as expected, the table is populated with values. The table data is also initialized, and a table function is now listed in the **Outline** tree view under `Simulation` > `Expressions, Functions and Variables` > `User Functions`.

Note that, if there are values in the table editor at the time you click **Auto Fill**, the ranges of those values are used to set the corresponding limits in the **Auto Fill Values** dialog box. If you click **OK** or **Apply** in the **Auto Fill Values** dialog box, automatically generated values replace any existing values in the table editor. You can click **Cancel** in the **Auto Fill Values** dialog box to return to the table editor without changing any existing values in the table editor.

9. Click **OK** to close the table editor.

   Note that, at any time, you could right-click the table function in the **Outline** tree view and select **Edit Table Data** to re-invoke the table editor with the table data displayed and ready to edit, export, and so on.

   Note also that, had you clicked **OK** instead of **Apply** in the **Auto Fill Values** dialog box, the result would be the same except that you would not be returned to the table editor.

You now have a table function ready to provide input parameters to the case.

39.4.2. Defining the Operating Point Output Parameters

In this section, you will define several solution monitors that will result in corresponding operating point output parameters in the simulation results.

1. Select **Insert** > **Solver** > **Output Control**.

   The **Output Control** details view appears.

2. On the **Monitor** tab, under **Monitor Points and Expressions**, click *Add new item* .

   The **Insert Monitor Point** dialog box appears.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Set **Name** to `Isentropic Efficiency` and click **OK**.
4. Under **Monitor Points and Expressions** > **Isentropic Efficiency**:
   a. Set **Option** to `Expression`.
   b. Set **Expression Value** to `Isentropic efficiency`.

      The latter is an expression that is provided with the case.
5. Click **Apply**.
6. Create the following other monitors based on expressions that are provided with the case:

Monitor Name	Expression Value
Mass flow rate	Outlet mass flow rate
Outlet temperature	Outlet Temperature
Total Pressure Ratio	Total pressure ratio

7. When finished, click **OK** in the **Output Control** details view.

39.4.3. Creating the Operating Points Object

An operating point case requires the `Operating Points` object to be defined in CFX-Pre. In this object, you can optionally pre-define one or more operating map charts.

Create the `Operating Points` object and pre-define an operating map chart as follows:

1. Click **Insert** > **Solver** > **Operating Points**.

   The **Operating Points** details view appears.
2. Configure the following setting(s):

Tab	Setting	Value
Operating Points	Option	From Table
	Table Name	Speeds and Flow Rates
	Operating Map Definitions	(Create a new operating map named `PR vs Flow`.)
	Operating Map Definitions PR vs Flow > **x Variable**	CorrMassFlowRate
	Operating Map Definitions	Total Pressure Ratio

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
	PR vs Flow > **y Variable**	

3. Click **OK**.

   The `Operating Points` object appears in the **Outline** tree view under `Simulation Control`.

39.4.4. Applying the Operating Point Input Parameters to the Case

In this section, you will modify the case to make use of the table function data.

1. In the **Outline** tree view, edit `Simulation` > `Flow Analysis 1` > `R1`.
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Domain Motion > **Angular Velocity**	-1*Speeds and Flow Rates.Speed(Index) [a]
a. In order to enter an expression, you must first click *Enter Expression* .		

3. Click **OK**.
4. Edit `R1 Outlet`.
5. Configure the following setting(s):

Tab	Setting	Value
Boundary Details	Mass And Momentum > **Option**	Exit Corrected Mass Flow Rate
	Mass And Momentum > **Mass Flow Rate**	Speeds and Flow Rates.CorrMassFlowRate(Index)
	Mass And Momentum > **Mass Flow Rate Area**	Total for All Sectors

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Click **OK**.

39.4.4.1. Setting Solver Control

1. Click *Solver Control* .
2. Configure the following setting(s):

Tab	Setting	Value
Basic Settings	Convergence Control > Fluid Timescale Control > **Timescale Control**	Physical Timescale
	Convergence Control > Fluid Timescale Control > **Physical Timescale**	1 / abs(Speeds and Flow Rates.Speed(Index))

3. Click **OK**.

39.4.5. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .

   The **Write Solver Operating Points Input File** dialog box appears (unless the **Simulation Control** > **Execution Control** object exists, in which case you can edit that object to plan the name of the solver input file).

2. Configure the following setting(s):

Setting	Value
File name	Compressor_OM.mdef

3. Click **Save**.

   CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, **Solver Input File** is set.

4. Quit CFX-Pre, saving the simulation (`.cfx`) file at your discretion.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

39.5. Obtaining the Solution Using CFX-Solver Manager

When CFX-Pre has shut down and the CFX-Solver Manager has started, obtain a solution to the operating point run by following the instructions below.

1. Ensure that the **Define Run** dialog box is displayed.

   Solver Input File should be set to `Compressor_OM.mdef`.

2. On the **Operating Points** tab, optionally set **Concurrency** > **Option** to **Maximum Number of Concurrent Jobs** and **Concurrency** > **Max. Concurrent Jobs** to 2.

   The allowed number of concurrent jobs depends on license availability.

   Note that the **Operating Points** tab contains other settings that can be used to manage restarts of interrupted runs.

3. Click **Start Run**.

   CFX-Solver runs and attempts to obtain one solution for each operating point.

   The operating point run history page can be seen in the `Operating Point Run Compressor OM 001` workspace, on the **Run Progress** tab. This page includes the current operating point parameter table, which shows the status of each operating point job as well as the input parameters and output parameters. The operating point parameter table also appears in CFD-Post, as will be seen later.

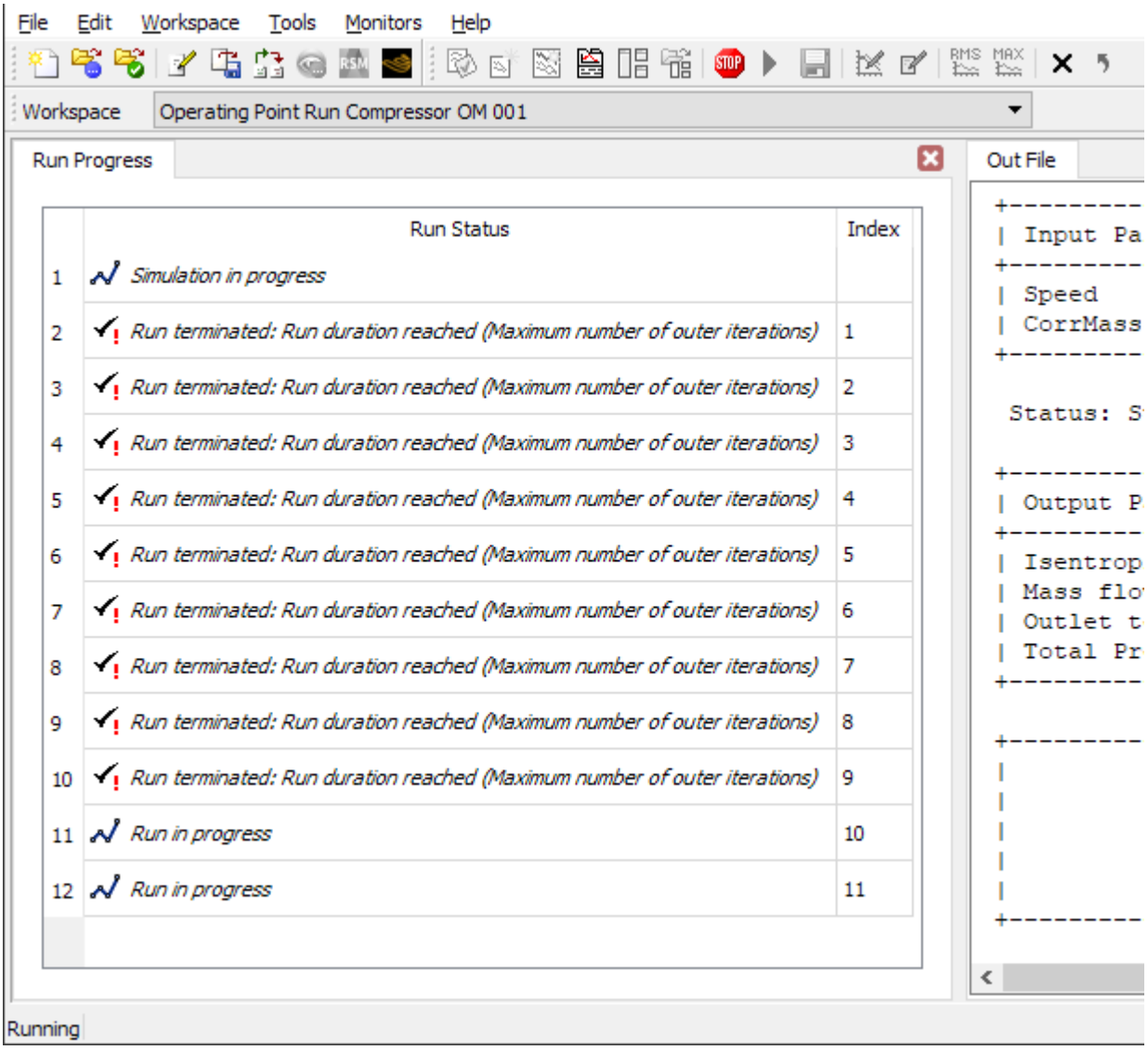
   The operating point parameter table is essentially the operating point input parameter table (defined in CFX-Pre) with added columns showing output parameter values (each output parameter corresponding to a monitor defined in CFX-Pre). You can export the operating point parameter table by right-clicking a cell and selecting the appropriate command.

4. Set **Workspace** to `Operating Point Index 1`.

   You can see residuals and, on the **User Points** tab, monitor points for the selected operating point job (`Operating Point Index 1`).

5. Set **Workspace** back to `Operating Point Run Compressor OM 001` to monitor the overall progress.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note that each run is terminated due to reaching the maximum number of outer iterations. The maximum number was set low by design in order to limit the time required to complete this tutorial.

At the end of the run, a dialog box is displayed stating that the simulation has ended.

6. Select **Post-Process Results**.
7. Select **Shut down CFX-Solver Manager**.
8. Click **OK**.

39.6. Viewing the Results Using CFD-Post

39.6.1. Viewing and Modifying the Pre-defined Operating Map Chart

While setting up the case in CFX-Pre, you pre-defined an operating map chart named `PR vs Flow` with x and y axes of `CorrMassFlowRate` and `Total Pressure Ratio`, respectively. In this

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

section, you will view that chart, temporarily show response points on it, then add contour lines to it.

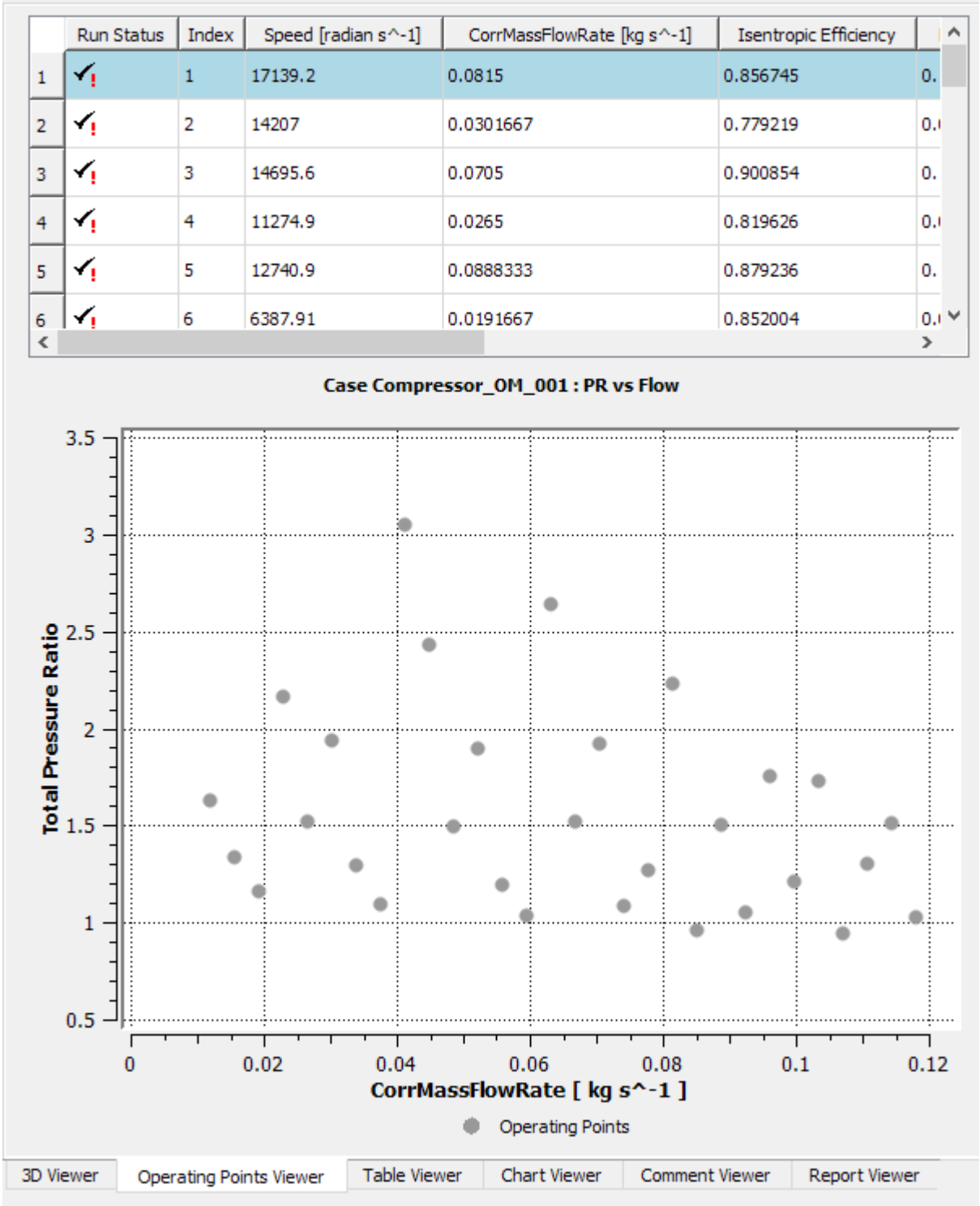
1. Edit `Cases` > `Compressor_OM_001` > `Operating Maps` > `PR vs Flow`.

   The **Operating Points Viewer** shows the operating point parameter table and the operating map chart that you pre-defined earlier.

   As mentioned earlier, the operating point parameter table is essentially the operating point input parameter table (defined in CFX-Pre) with added columns showing output parameter values (each output parameter corresponding to a monitor defined in CFX-Pre). Each row of the table represents an operating point. You can double-click any row in the operating point parameter table to postprocess the 3D results in the same way as for a non-operating-point case (provided that a mesh exists for the selected operating point). Filter settings are available to filter out certain rows of the operating point parameter table. You can export the operating point parameter table from CFD-Post by right-clicking a cell in the table and selecting **Export Table Data to CSV**.

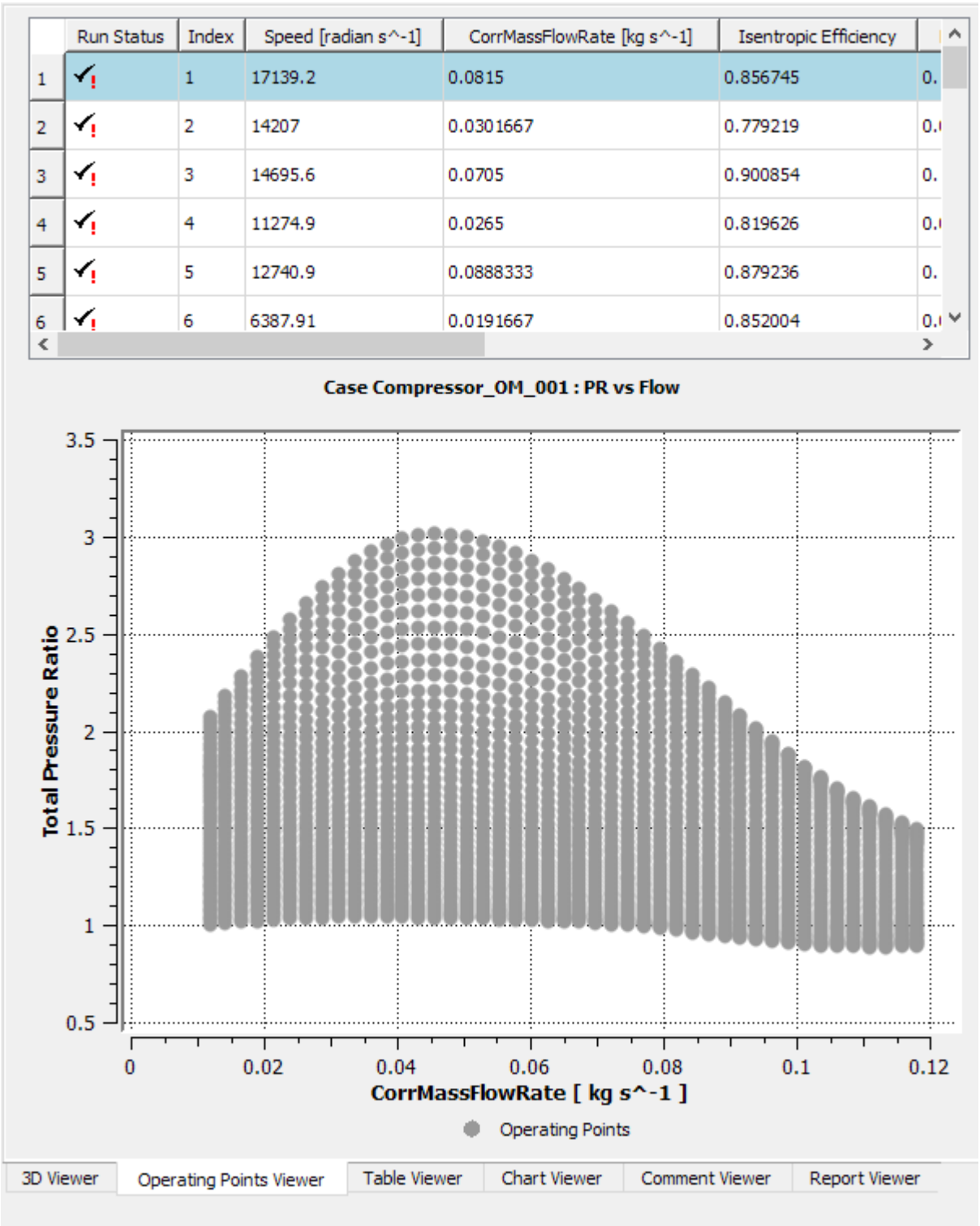
   The operating map chart initially shows the operating points.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. In the details view, on the **Chart Data** tab, set **Chart Data Source** > **Source Data** to `Response Points` and click **Apply**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



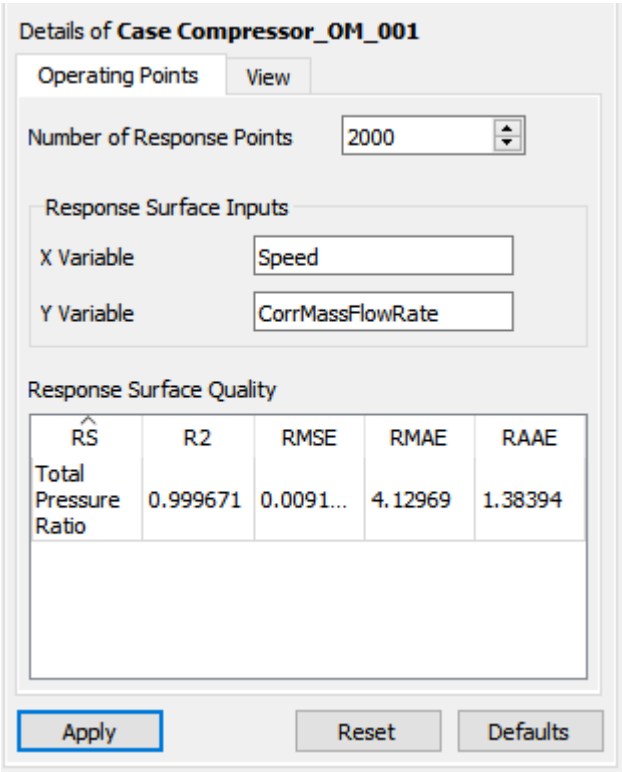
The operating map chart now shows response points, which are sampling points on a response surface (the response surface for `Total Pressure Ratio`). The response surface is constructed based on the set of operating points. Using a response surface can yield smoother looking plots. However, it is important to check the quality of the response surface to ensure that the plots that use it are not misleading.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. In the details view, on the **Chart Data** tab, set **Chart Data Source** > **Source Data** to `Operating Points` and click **Apply**.

4. In the **Outline** tree view, edit `Cases` > `Compressor_OM_001`.

   The details view for the case appears.



   Note that the number of response points sampled from the response surface is adjustable from the **Number of Reponse Points** setting.

   Recall that the input parameters that were defined in the operating point input parameter table in CFX-Pre were `Speed` and `CorrMassFlowRate`. The table shown in the **Operating Points Viewer** in CFD-Post (for pre-defined operating map chart `PR vs Flow`) has columns for those input parameters and additional columns for the output parameters, which came from the solution monitors that you defined earlier. When an operating map chart needs response points for one of those output parameters, a corresponding response surface is generated and quality metrics for that response surface appear as a row in the **Reponse Surface Quality** table. In this case, there is a row for `Total Pressure Ratio` because you made an operating map chart that involved `Total Pressure Ratio` sourced from response points. Later in this tutorial, you will examine the quality of the `Total Pressure Ratio` response surface visually: by plotting contours of constant value on the response surface and overlaying them with operating points.

   Note that the **Reponse Surface Quality** table does not list the input parameters, which in this case are `Speed` and `CorrMassFlowRate`. Response surfaces are not generated for the input parameters.

5. Once again, edit `Cases` > `Compressor_OM_001` > `Operating Maps` > `PR vs Flow`.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Define a chart series for operating map chart PR vs Flow to show contours of variable Speed as follows:

   a. Click *New* .

      Settings appear for a new chart series for operating map chart PR vs Flow.

   b. Set **Name** to Speed.

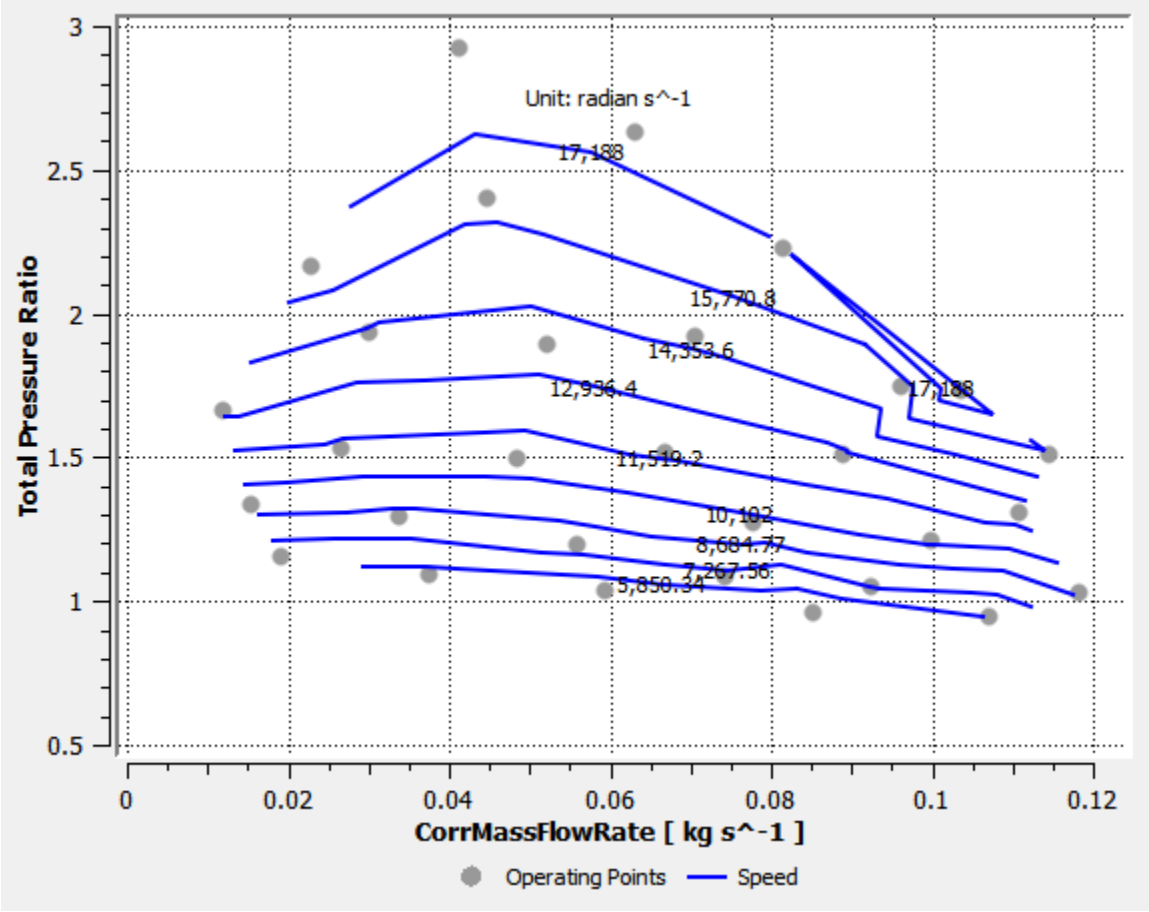
   c. Leave **Type** set to Contour Lines and **Parent Series** set to Operating Points.

      Contour lines are always plotted using the same axes as the parent series.

   d. Set **Variable** to Speed.

   e. Leave **Contour Data Source** > **Source Data** set to Operating Points.

   f. Click **Apply**.

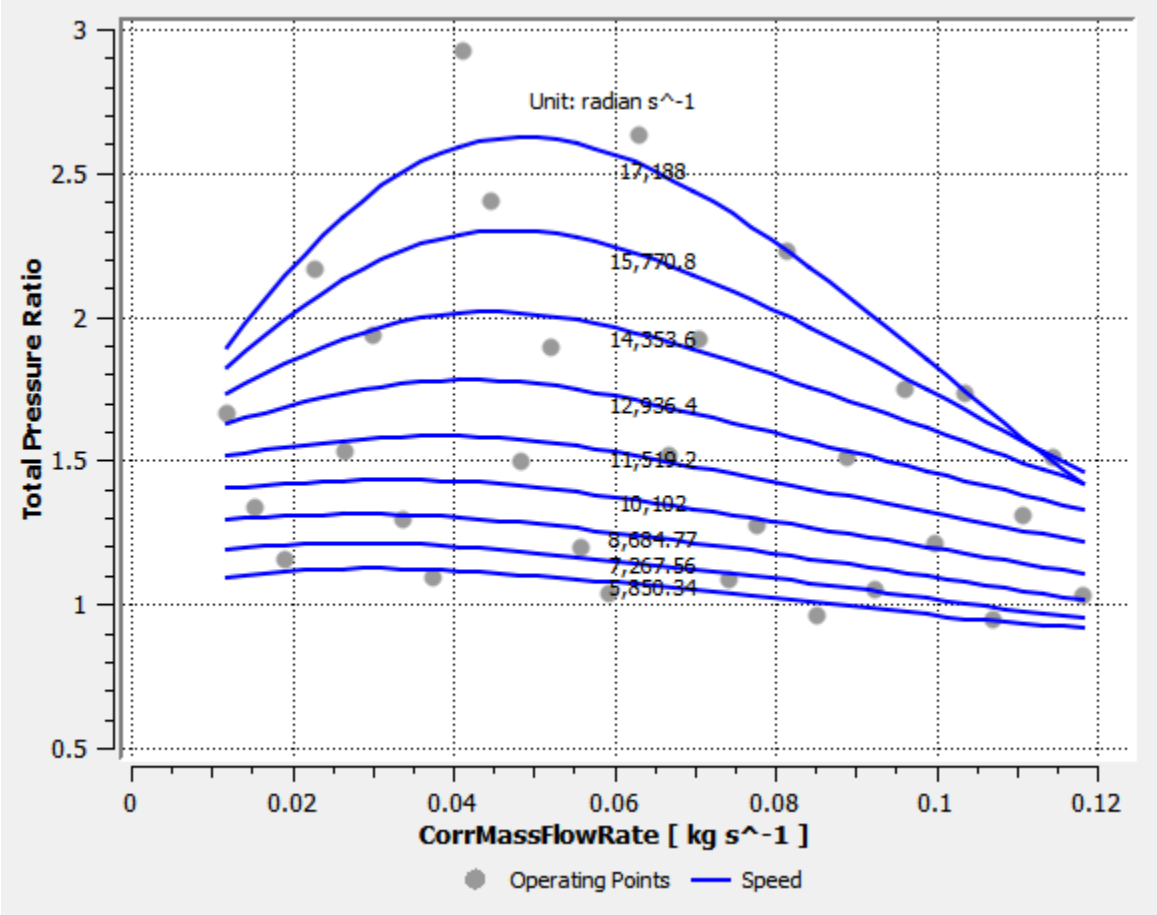


   Contours of variable Speed appear in the operating map chart.

   g. Change **Contour Data Source** > **Source Data** to Response Points.

   h. Click **Apply**.

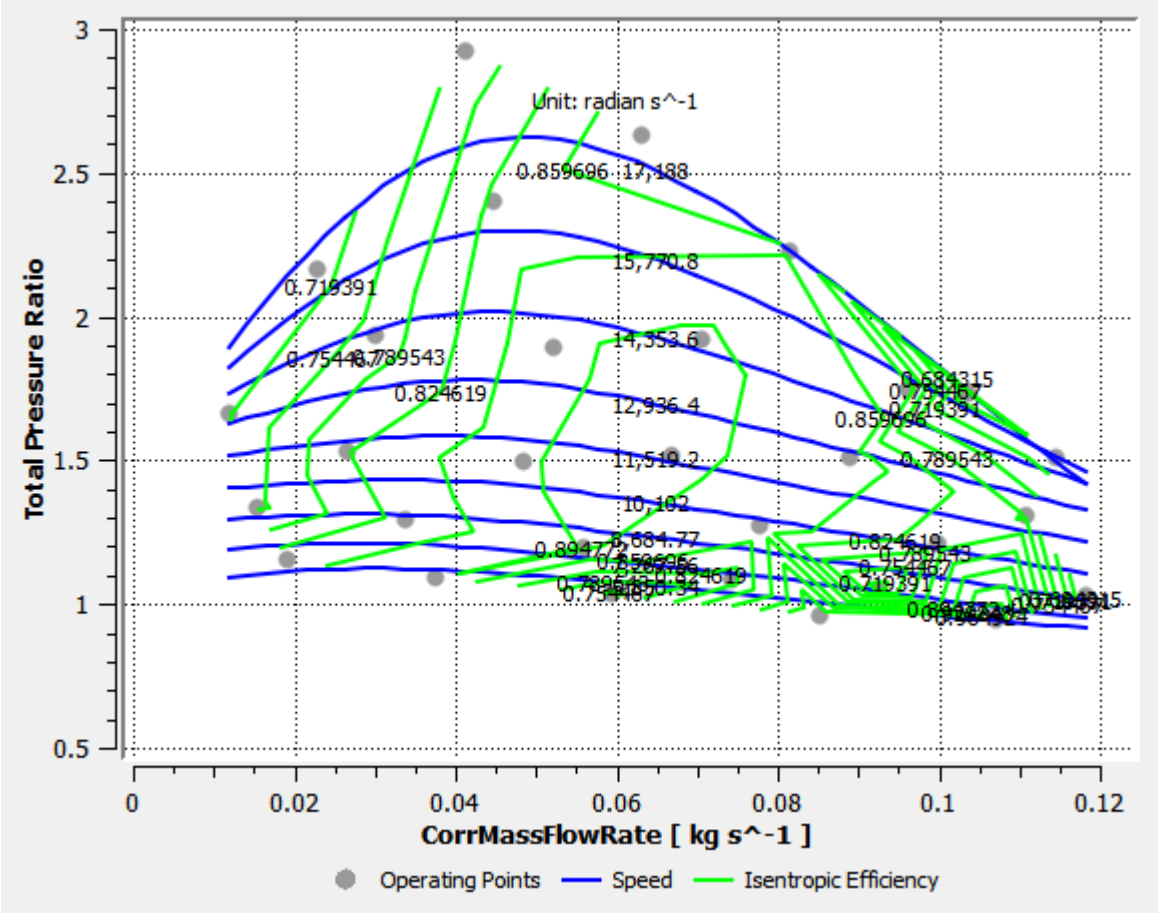
Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note how the contours become smoother. They are now created using the response surface for `Total Pressure Ratio`.

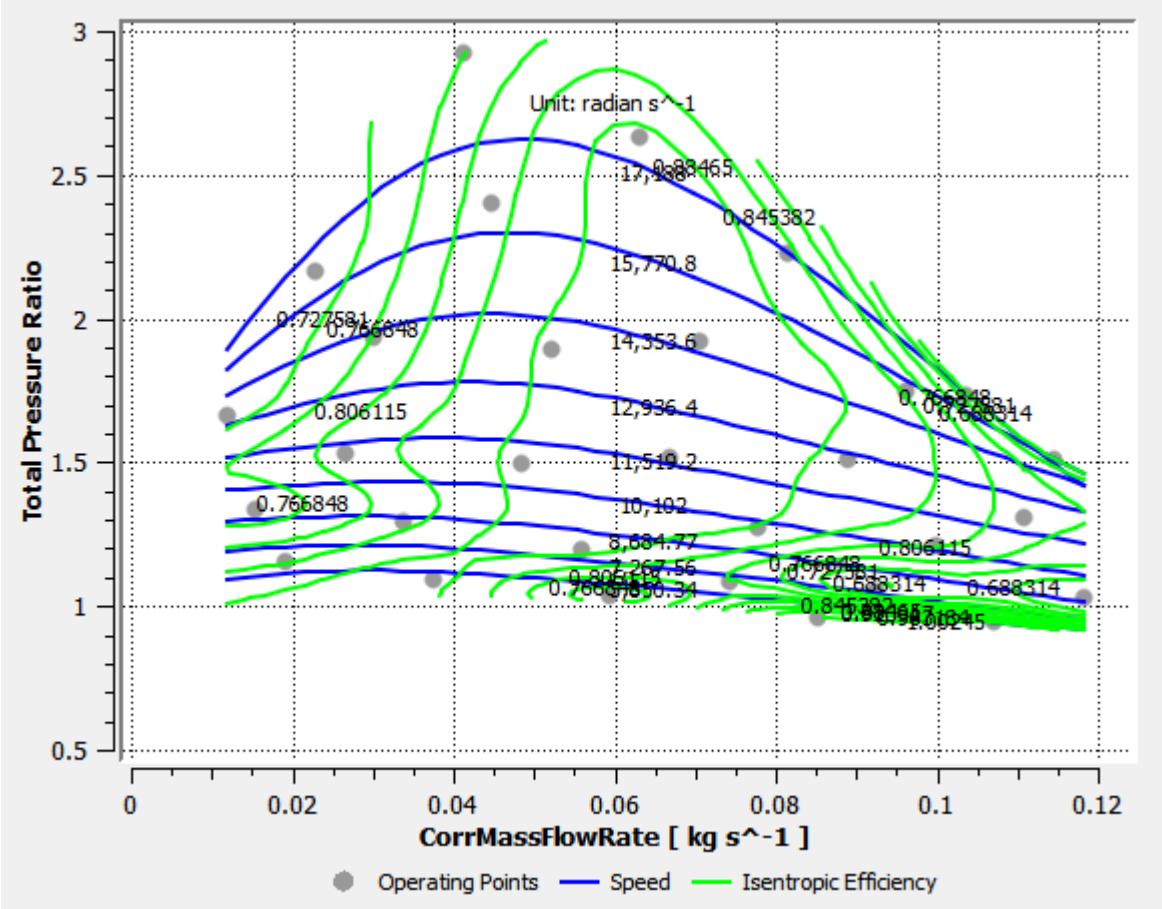
7. Using the technique of the previous step, define another chart series to show contours of variable `Isentropic Efficiency` based on, at first, operating points...

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



...then response points...

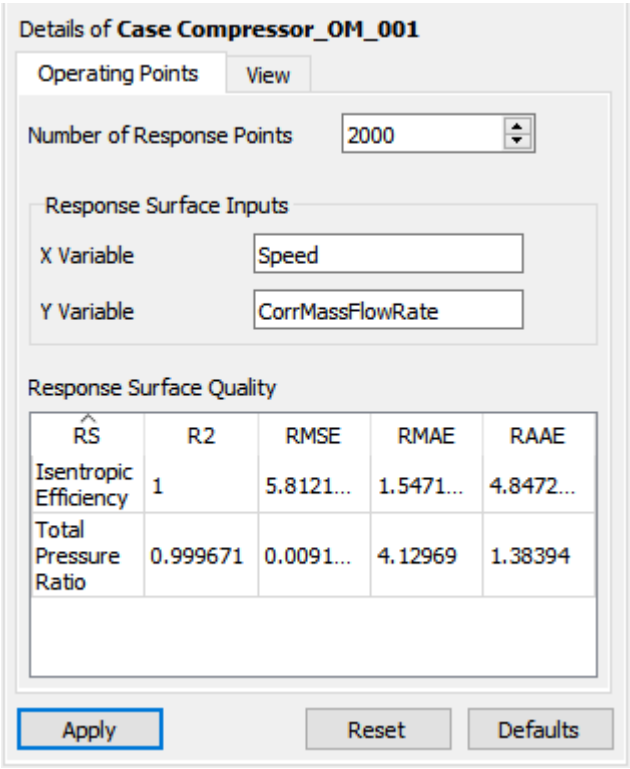
Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



...again, noting the visual difference.

Now that you have defined a plot that uses a response surface for variable `Isentropic Efficiency`, a corresponding row exists in the **Reponse Surface Quality** table.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



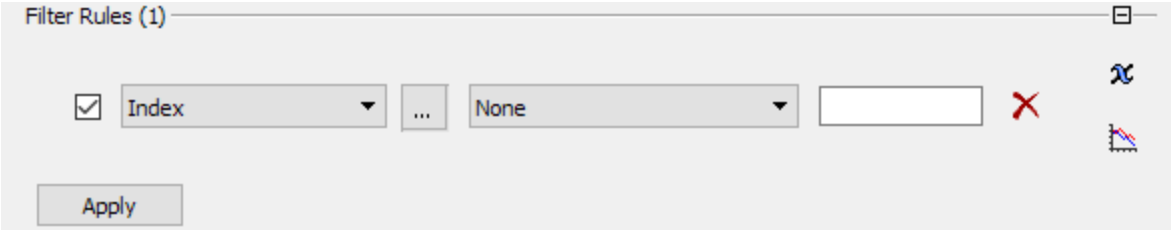
8. As mentioned earlier, filter settings are available to filter out certain rows of the operating point parameter table. These filter settings also cause operating points to be omitted from the operating map chart and, optionally, all response surfaces that underlie the chart.

   As an exercise, create a filter rule that requires operating points to have a total pressure ratio greater than one in order to be used for the operating map chart or response surfaces.

   a. From the toolbar, click *Show operating points filtering controls* to show the **Filter Rules** area of the **Operating Points Viewer**.

   b. Click *Add new variable rule* .

      A new line appears, which looks like the following:



      The check box on the left controls whether or not the rule takes effect and the extent of its effect:

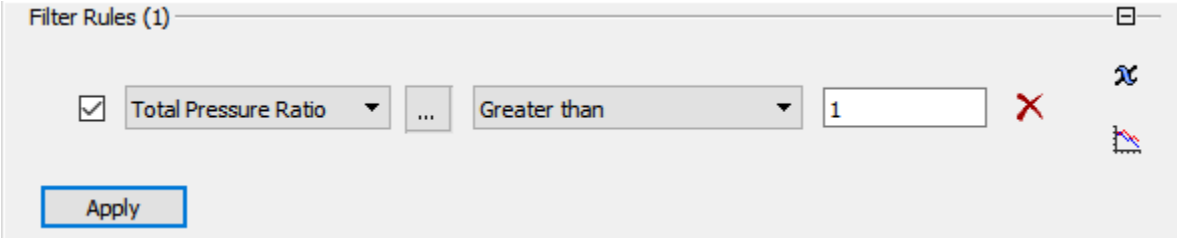
      - A cleared check box (☐) specifies that the filter rule should be ignored.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- A partially selected check box (▣) specifies that all operating points that do not satisfy the filter rule should be excluded from the operating point parameter table and the operating map chart.
- A fully selected check box (☑) specifies that all operating points that do not satisfy the filter rule should be excluded from the operating point parameter table, the operating map chart, and the generation of all of the response surfaces that underlie the operating map chart.

c. Leave the check box fully selected so that the filter rule applies to the operating map chart and its underlying response surfaces.

d. Set the filter rule to require that the total pressure ratio is greater than one, as shown below.

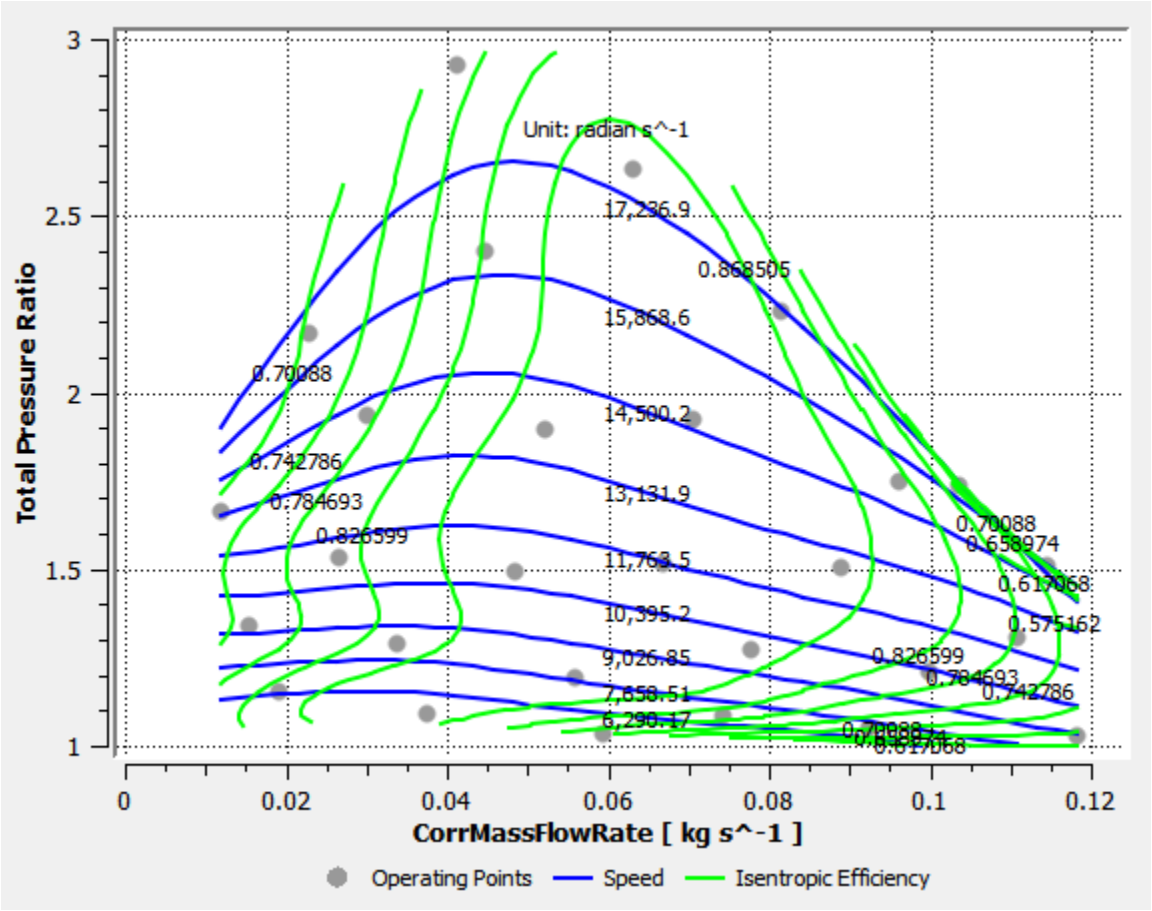


e. Click **Apply**.

Some operating points are filtered out; these points are now excluded from the table, the operating map chart, and the response surfaces that underlie the chart.

The operating map chart now appears as follows:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note that operating maps are included in the `Report` object.

39.6.2. Visually Checking the Quality of a Response Surface by Creating a New Operating Map Chart

As an exercise, create an operating map chart that can be used to visualize how well the response surface for `Total Pressure Ratio` fits the underlying operating point data:

1. In the **Outline** tree view, right-click `Cases` > `Compressor_OM_001` > `Operating Maps` and select **Insert** > **Operating Map Chart**.

   The **Insert Chart** dialog box appears.

2. Set **Name** to `Efficiency RS` and click **OK**.

   In the details view, one series is already in the list but requires a definition.

3. Set **Name** to `Operating Points`.

4. Leave **Type** set to `Scatter Points`.

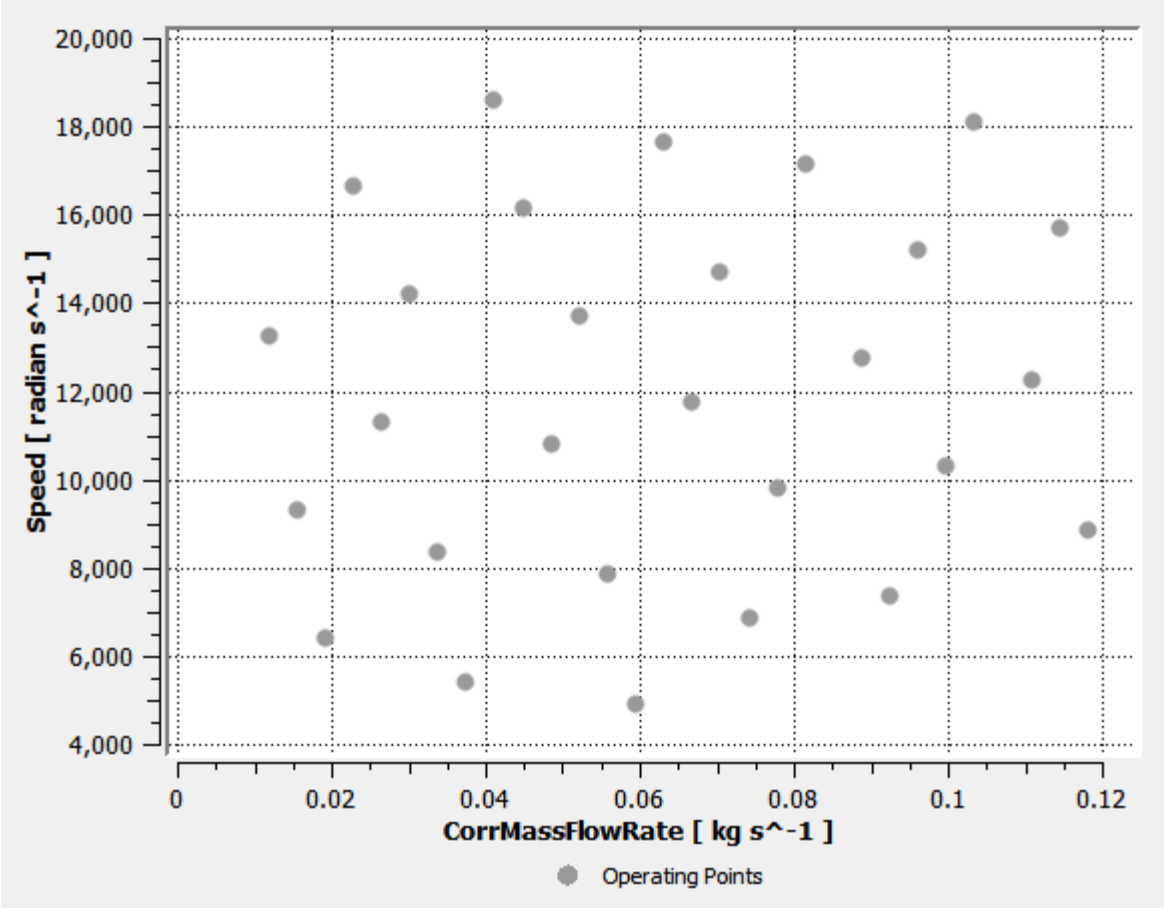
5. Under **Chart Definition**, set **x Variable** and **y Variable** to `CorrMassFlowRate` and `Speed`, respectively.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These two variables are the operating point input parameters.

6. Leave **Chart Data Source** > **Source Data** set to `Operating Points`.
7. Click **Apply**.

   The operating points appear in the operating map chart.

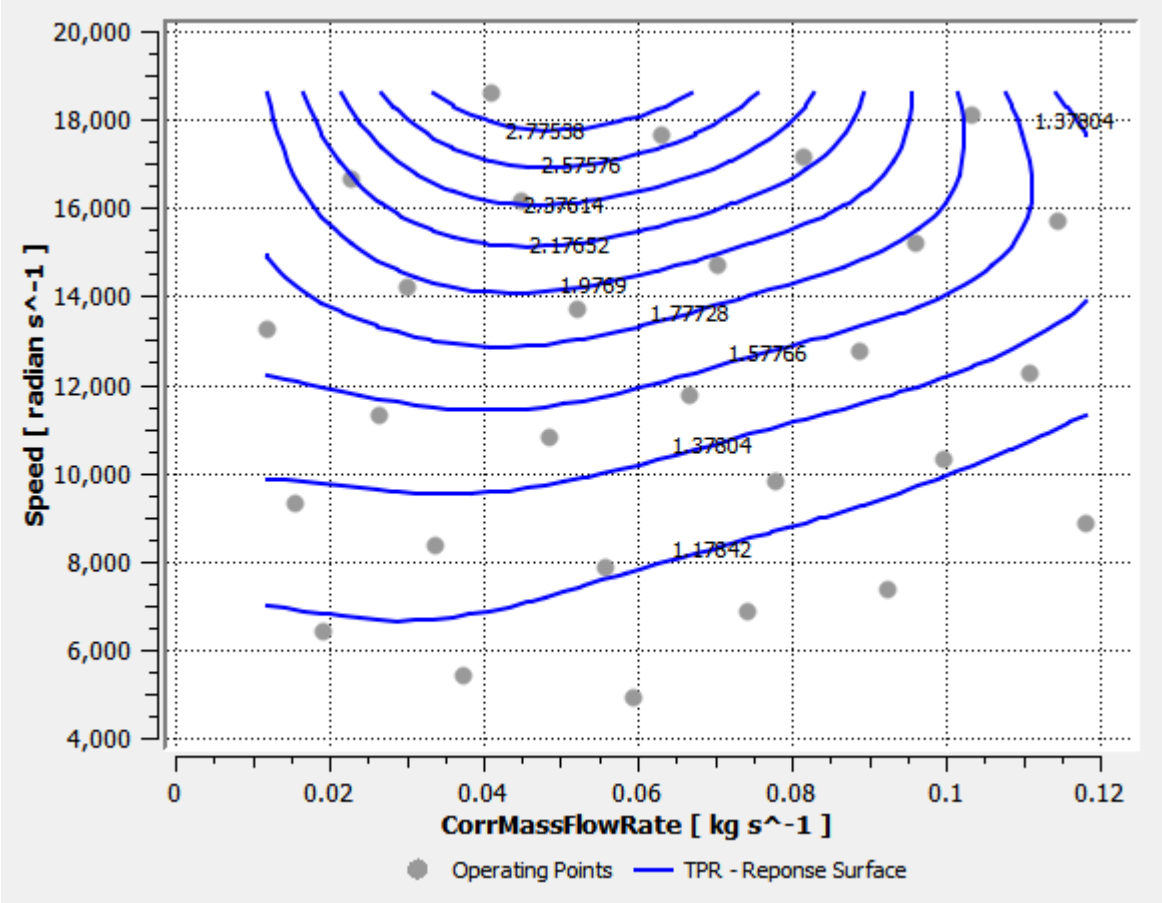


8. Click *New*

   The details view is ready to define the next series.

9. Set **Name** to `TPR - Reponse Surface`.
10. Leave **Type** set to `Contour Lines`.
11. Leave **Contour Definition** > **Parent Series** set to `Operating Points`.
12. Set **Variable** to `Total Pressure Ratio`.
13. Set **Contour Data Source** > **Source Data** to `Response Points`.
14. Click **Apply**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Contours that follow constant values on the response surface for `Total Pressure Ratio` appear in the operating map chart.

You can see the operating point values of `Total Pressure Ratio` by referring to the operating point parameter table. Ideally, these values should be closely approximated by the response surface.

In general, you can change the quality of a response surface by changing:

- The number of operating points,
- The input parameter values for existing operating points,
- Filter rules that affect the response surfaces (due to having a fully selected check box), and
- The number of response points.

The key to a good response surface is a good distribution of input parameters.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 40: Flow in an Intake Manifold using GT-SUITE Coupling

This tutorial includes:

40.1. Tutorial Features

40.2. Overview of the Problem to Solve

40.3. Preparing the Working Directory

40.4. Defining the Case Using CFX-Pre

40.5. Obtaining the Solution Using CFX-Solver Manager

Important:

You must have GT-SUITE installed in order to run this tutorial.

This tutorial was tested to work with GT-SUITE v2019.

40.1. Tutorial Features

In this tutorial you will learn about:

- Importing a CCL file to set up a case in CFX-Pre.
- Initializing GT-SUITE Coupling.

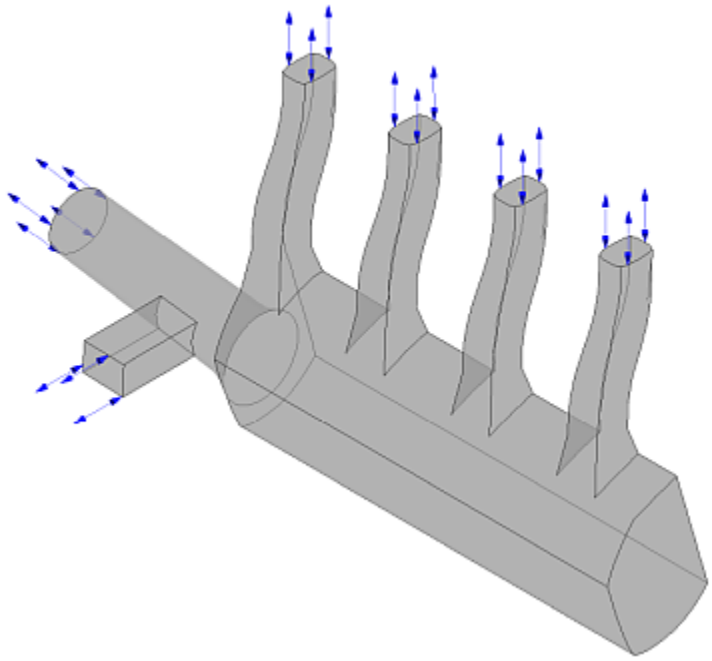
Component	Feature	Details
CFX-Pre	User Mode	General mode
	Analysis Type	Transient
	Fluid Type	General Fluid
	Domain Type	Single Domain
	Turbulence Model	k-Epsilon
	Boundary Conditions	Opening (GT-SUITE Coupling)

40.2. Overview of the Problem to Solve

This tutorial models transient flow through a manifold by coupling CFX with GT-SUITE.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The GT-SUITE model is a one-dimensional model of a 4-cylinder engine. The 3D mesh to be coupled is for an intake manifold. The flow consists of a mixture of indolene, nitrogen, oxygen, and combustion products CO_2 and water.



The approach to this tutorial is to load a transient simulation from CCL, then couple that simulation with GT-SUITE, exchanging information at the domain boundaries.

40.3. Preparing the Working Directory

1. Create a working directory.

   Ansys CFX uses a working directory as the default location for loading and saving files for a particular session or project.

2. Copy the following GT-SUITE `.gtm` file from your GT-SUITE installation into your working directory. Note that the path might not be exactly as shown.

   - `C:\Program Files (x86)\GTI\v2019\tutorials\Co-Simulation_And_User-Code\CFDcoupling\IntakeManifold\CFDCoupling_IntakeManifold.gtm`

3. Download the `intake_manifold_GT.zip` file here.

4. Unzip `intake_manifold_GT.zip` to your working directory.

5. Ensure that the following tutorial input files are in your working directory:

   - `IntakeManifold.msh`
   - `IntakeManifold.ccl`

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- `CFDCoupling_IntakeManifold.gtm`

6. Set the working directory and start CFX-Pre.

   For details, see Setting the Working Directory and Starting Ansys CFX in Stand-alone Mode (p. 33).

40.4. Defining the Case Using CFX-Pre

The tutorial follows this general workflow for setting up the case in CFX-Pre:

40.4.1. Importing a Mesh
40.4.2. Importing the Intake Manifold CCL File
40.4.3. Initializing GT-SUITE Coupling
40.4.4. Editing the Boundaries
40.4.5. Writing the CFX-Solver Input (.def) File

1. In CFX-Pre, select **File** > **New Case**.
2. Ensure that **General** is selected and click **OK**.
3. Select **File** > **Save Case As**.
4. Under **File name**, type `IntakeManifold`.
5. Click **Save**.

40.4.1. Importing a Mesh

1. Select **File** > **Import** > **Mesh**.

   The **Import Mesh** dialog box appears.

2. Under **Files of type**, select `FLUENT (*cas *msh)`.
3. From your working directory, select `IntakeManifold.msh`.
4. Click **Open**.

40.4.2. Importing the Intake Manifold CCL File

CFX Command Language (CCL) consists of commands used to carry out actions in CFX-Pre, CFX-Solver Manager, and CFD-Post. The flow analysis settings as well as the materials required for the simulation will be imported from the CCL file. You will review, then import, the contents of the `IntakeManifold.ccl` file.

Note:

The physics for a simulation can be saved to a CCL (CFX Command Language) file at any time by selecting **File** > **Export** > **CCL**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Open `IntakeManifold.ccl` with a text editor and take the time to look at the information it contains.

   The CCL sets up the following materials:

   - `GT Mixture`
   - `indolene combust`
   - `indolene vap`
   - `lump N2`
   - `n2 vap`
   - `o2 EGR`
   - `o2 backflow`
   - `o2 vap`
   - `prod co2`
   - `prod h2o`

2. In CFX-Pre, select **File** > **Import** > **CCL**.

   The **Import CCL** dialog box appears.

3. Under **Import Method**, select **Append**. This option appends the changes to the existing case.

   Note:

   The **Replace** option is useful if you have defined the physics and want to update or replace the existing physics using the newly imported CCL.

4. From your working directory, select `IntakeManifold.ccl`.

5. Click **Open**.

   The CCL is loaded as indicated by the status bar in the bottom right corner of the window. After a short pause, the CCL and the **Outline** tree view are updated.

6. In the **Outline** tree view, expand the `Materials` branch under `Simulation` to confirm that new objects have been added after importing the CCL file.

40.4.3. Initializing GT-SUITE Coupling

1. Select **Tools** > **Initialize GT-SUITE Coupling**.

   The **Initialize GT-SUITE Coupling** dialog box appears.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Ensure that **Create new GT-SUITE model** is selected.

3. Beside **Model File**, click *Browse* .

   The **Select GT-SUITE File** dialog box appears.

4. Select `CFDCoupling_IntakeManifold.gtm` and click **Open**.

5. Optionally, select **Specify GT-SUITE Installation** and specify the installation directory.

   If you do not select this option, the default installation directory will be used.

6. Click **Read Setup From File**.

   The GT-SUITE coupling information is loaded, and additional options appear for further configuration.

7. On the **Boundary Association** tab, ensure that **Option** is set to `Specify Boundary Names`.

   With this option, GT-SUITE interfaces are mapped to CFX boundaries according to the CFX boundary names in the **CFX Boundary** column of the two-column mapping table.

8. Ensure that **Allow new boundary names** and **Create missing boundaries** are selected.

   Together, these settings automatically create any non-existent mapped boundaries, giving them the specified names. In this case, six boundaries will be created.

9. On the **Material Association** tab, ensure that **Option** is set to `Specify Component Names`.

   This option allows you to manually specify component names to map from GT-SUITE materials to CFX materials. In this case, the mapping requires no adjustments.

10. On the **Function Names** tab, ensure that all six GT-SUITE patches are listed under **GT-SUITE Interface or Turboshaft Name**, with a corresponding GT-SUITE Patch function name listed under **CFX Function Name**.

    The listed GT-SUITE Patch functions will be created. These functions are used to exchange data between CFX boundaries and the corresponding GT-SUITE patches.

11. Click **OK**.

    GT-SUITE initialization is carried out, resulting in the creation of a GT-SUITE model, several GT-SUITE Patch functions, and boundaries, all of which appear in the **Outline** tree view.

40.4.4. Editing the Boundaries

The coupled boundaries have been created but require location information.

1. In the **Outline** tree view, right-click `Simulation` > `Flow Analysis 1` > `Default Domain` > `Cyl 1` and select **Edit**.

2. Configure the following setting(s):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tab	Setting	Value
Basic Settings	**Location**	outlet cyl 1

3. Click **OK**.
4. In a similar way, edit each of the other boundaries to specify a location as shown below.

Boundary	Location
`Cyl 2`	outlet cyl 2
`Cyl 3`	outlet cyl 3
`Cyl 4`	outlet cyl 4
EGR	inlet egr
`Intake`	inlet air

40.4.5. Writing the CFX-Solver Input (.def) File

1. Click *Define Run* .
2. Configure the following setting(s):

Setting	Value
File name	IntakeManifold.def

3. Click **Save**.

   The CFX-Solver input file (`IntakeManifold.def`) is created. CFX-Solver Manager automatically starts and, on the **Define Run** dialog box, **Solver Input File** is set.
4. When you are finished, select **File** > **Quit** in CFX-Pre.
5. Click **Save & Quit** if prompted, to save `IntakeManifold.cfx`.

40.5. Obtaining the Solution Using CFX-Solver Manager

You will now generate a solution for the coupled simulation that you just prepared.

40.5.1. Starting the Run

At this point, CFX-Solver Manager is running, and the **Define Run** dialog box is displayed, with the CFX-Solver input file set.

1. Click **Start Run**.
2. Select the check box next to **Post-Process Results** when the completion message appears at the end of the run.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. If using stand-alone mode, select the check box next to **Shut down CFX-Solver Manager**.
4. Click **OK**.

Postprocessing is left as a further exercise.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.