

# CFD EXPERTS

Simulate the Future

[WWW.CFDEXPERTS.NET](http://WWW.CFDEXPERTS.NET)



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

# Ansys CFX-Pre User's Guide

---



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. FLEXIm 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

<b>1. CFX-Pre Basics</b> .....	33
1.1. Starting CFX-Pre .....	33
1.2. CFX-Pre Modes of Operation .....	35
1.3. Working with the CFX-Pre Interface .....	35
1.3.1. Viewer .....	36
1.3.2. CFX-Pre Workspace .....	37
1.3.2.1. Outline Tree View .....	37
1.3.2.1.1. General Considerations .....	37
1.3.2.1.2. Outline Tree View Structure .....	37
1.3.2.1.2.1. Outline Tree View Shortcut Menu Commands .....	40
1.3.2.2. Details View .....	43
1.3.3. Physics Message Window .....	45
1.3.3.1. Physics Errors from Old .def/.res Files .....	46
1.3.3.2. Physics Message Window Shortcut Menu Commands .....	46
1.3.4. Menu Bar .....	47
1.3.5. Toolbar .....	48
1.4. CFX-Pre File Types .....	48
<b>2. CFX-Pre 3D Viewer</b> .....	51
2.1. Object Visibility .....	51
2.2. 3D Viewer Modes and Commands .....	51
2.2.1. 3D Viewer Toolbar .....	52
2.2.2. Shortcut Menus .....	54
2.2.2.1. CFX-Pre 3D Viewer Shortcut Menu .....	54
2.2.2.1.1. Shortcuts for CFX-Pre (Viewer Background) .....	54
2.2.2.1.2. Shortcuts for CFX-Pre (Viewer Object) .....	55
2.2.3. Viewer Hotkeys .....	56
2.2.4. Mouse Button Mapping .....	57
2.2.5. Picking Mode .....	59
2.2.5.1. Selecting Objects .....	60
2.2.6. Labels and Markers .....	60
2.2.6.1. Labels .....	60
2.2.6.2. Boundary Markers .....	60
2.2.6.3. Boundary Vectors .....	61
2.2.6.4. Profile Vectors .....	61
2.2.6.5. Injection Region Vectors .....	61
2.2.6.6. Particle Injection Regions .....	61
2.3. Views and Figures .....	61
2.3.1. Switching to a View or Figure .....	62
2.3.2. Changing the Definition of a View or Figure .....	62
2.4. Stereo Viewer .....	62
<b>3. CFX-Pre File Menu</b> .....	65
3.1. New Case Command .....	65
3.2. Open Case Command .....	66
3.2.1. Recover Original Session .....	67
3.2.2. Opening Case (.cfx) Files .....	67
3.2.3. Opening CFX-Solver Input (.def, .mdef), Results (.res), Transient (.trn) or Backup (.bak) Files .....	67
3.2.4. Opening CCL (.ccl) Files .....	68
3.2.5. Opening Meshing (.cmdb or .dsdb) Files .....	68
3.2.6. Opening CFX-Mesh (.gtm) Files .....	68

3.3. Close Command .....	68
3.4. Save Case Command .....	68
3.5. Save Project Command .....	68
3.6. Refresh Command (Ansys Workbench only) .....	69
3.7. Save Case As Command .....	69
3.8. Reload Mesh Files Command .....	69
3.9. Import Mesh Command .....	69
3.10. Import CCL Command .....	70
3.10.1. Append or Replace .....	70
3.10.1.1. Append .....	70
3.10.1.2. Replace .....	70
3.10.1.2.1. Auto-load materials .....	71
3.11. Export CCL Command .....	71
3.11.1. Save All Objects .....	71
3.11.1.1. Sample of Saving CEL Expressions .....	71
3.12. Export CSV Command .....	72
3.13. Save Picture Command .....	72
3.14. Recent Case Files Submenu .....	75
3.15. Recent CCL Files Submenu .....	75
3.16. Recent Session Files Submenu .....	76
3.17. Quit Command .....	76
<b>4. CFX-Pre Edit Menu .....</b>	<b>77</b>
4.1. Undo and Redo .....	77
4.2. Options .....	78
4.2.1. CFX-Pre Options .....	78
4.2.1.1. General .....	79
4.2.1.1.1. Beta Options .....	80
4.2.1.1.1.1. Physics Beta Features .....	80
4.2.1.1.1.2. Topology Beta Features .....	80
4.2.1.1.2. Auto Generation .....	80
4.2.1.1.3. Physics .....	82
4.2.1.2. Graphics Style .....	83
4.2.1.2.1. Object Highlighting .....	83
4.2.1.2.2. Background .....	84
4.2.1.2.2.1. Color .....	84
4.2.1.2.2.2. Image .....	84
4.2.1.2.3. Colors .....	84
4.2.1.2.3.1. Labels .....	84
4.2.1.2.3.2. Legend Text and Turbo Axis .....	84
4.2.1.2.4. Visibility .....	84
4.2.1.2.4.1. Axis and Ruler Visibility .....	84
4.2.1.3. Render .....	85
4.2.1.4. Mesh .....	85
4.2.1.4.1. Mesh Import Options .....	85
4.2.1.5. Turbo .....	85
4.2.1.6. Labels and Markers .....	85
4.2.1.6.1. Labels .....	85
4.2.1.6.2. Boundary Markers .....	85
4.2.1.6.3. Boundary Vectors .....	86
4.2.1.6.4. Profile Vectors .....	86
4.2.1.6.5. Injection Region Vectors .....	86

4.2.1.6.6. Particle Injection Regions .....	86
4.2.1.7. User Interface .....	86
4.2.1.8. Extensions .....	87
4.2.1.9. Customization .....	87
4.2.1.10. Solve .....	87
4.2.1.11. Viewer .....	87
4.2.2. Common Options .....	87
4.2.2.1. Appearance .....	88
4.2.2.2. Viewer Setup .....	88
4.2.2.2.1. Mouse Mapping .....	89
4.2.2.3. Units .....	89
4.2.2.3.1. Additional Help on Units .....	90
<b>5. CFX-Pre Session Menu .....</b>	<b>91</b>
5.1. New Session Command .....	91
5.2. Start Recording and Stop Recording Commands .....	92
5.3. Play Session Command .....	92
<b>6. CFX-Pre Insert Menu .....</b>	<b>95</b>
<b>7. CFX-Pre Tools Menu .....</b>	<b>99</b>
7.1. Command Editor .....	99
7.2. Edit Profile Data .....	99
7.2.1. Profile Data Expansion (Option = Expansion) .....	100
7.2.1.1. Notes and Limitations of Profile Data Expansion .....	101
7.2.2. Profile Data Mapping (Option = Map to Mesh Regions) .....	102
7.2.2.1. Notes and Limitations of Profile Data Mapping .....	102
7.2.3. Profile Data Expansion with Mapping (Option = Mapped Expansion) .....	102
7.2.4. Mapping Profile Data to a File .....	103
7.2.5. Modifying Spatial Fields in Profile Data .....	104
7.2.6. Other Profile Transformation Options .....	105
7.3. Initialize Profile Data .....	106
7.4. Initialize FMU Instance .....	108
7.5. Initialize GT-SUITE Coupling .....	109
7.5.1. Boundary Association Tab .....	111
7.5.2. Material Association Tab .....	112
7.5.3. Turboshaft Setup Tab .....	112
7.5.4. Monitoring Tab .....	113
7.5.5. Function Names Tab .....	114
7.6. Macro Calculator .....	114
7.7. Solve .....	115
7.7.1. Write Solver Input File Command .....	115
7.8. Applications .....	116
7.9. Quick Setup Mode .....	116
7.10. Turbo Mode .....	116
<b>8. CFX-Pre Extensions Menu .....</b>	<b>117</b>
<b>9. CFX-Pre Help Menu .....</b>	<b>119</b>
<b>10. Importing and Transforming Meshes .....</b>	<b>121</b>
10.1. Importing Meshes .....	121
10.1.1. Importing Multiple Meshes .....	122
10.1.2. Common Import Options .....	122
10.1.2.1. Mesh Units .....	122
10.1.2.2. Assembly Prefix .....	122
10.1.2.3. Primitive Strategy .....	122

10.1.2.4. Ignore Invalid Degenerate Elements .....	123
10.1.2.5. Duplicate Node Checking .....	123
10.1.3. Supported Mesh File Types .....	123
10.1.3.1. Ansys Meshing Files .....	124
10.1.3.1.1. Named Selections .....	124
10.1.3.1.2. Contact Detection Settings .....	125
10.1.3.2. CFX-Mesh Files .....	126
10.1.3.3. CFX-Solver Input files .....	126
10.1.3.4. ICEM CFD Files .....	126
10.1.3.5. Ansys Files .....	126
10.1.3.6. Fluent Files .....	127
10.1.3.6.1. Override Default 2D Mesh Settings .....	127
10.1.3.6.1.1. Interpret 2D Mesh as .....	127
10.1.3.6.1.1.1. Axisymmetric .....	127
10.1.3.6.1.1.2. Planar .....	128
10.1.3.7. CGNS Files .....	128
10.1.3.7.1. Importing CGNS files into CFX .....	128
10.1.3.7.1.1. Method .....	128
10.1.3.7.1.2. Base (Base_t) .....	128
10.1.3.7.1.3. Zone (Zone_t) .....	128
10.1.3.7.1.4. Elements (ElementSection_t) .....	129
10.1.3.7.1.5. Element Types Supported .....	129
10.1.3.7.1.6. Boundary Conditions (BC_t) .....	129
10.1.3.7.1.7. Families (Family_t, FamilyBC_t, FamilyName_t) .....	129
10.1.3.7.1.8. Grid Connectivity (GridConnectivity_t and GridConnectivity1to1_t) .....	129
10.1.3.7.1.9. CGNS Data Ignored .....	130
10.1.3.7.2. Prefix regions with zone name .....	130
10.1.3.7.3. Create Regions From: Element Sections .....	130
10.1.3.7.4. Create Regions From: Boundary Conditions .....	130
10.1.3.7.5. Create Regions From: Families .....	131
10.1.3.7.6. Create Regions From: Connectivity Mappings .....	131
10.1.3.7.7. Example of Create Regions From .....	131
10.1.3.7.8. Read Only One CGNS Base .....	131
10.1.3.8. CFX-TASCflow Files .....	132
10.1.3.8.1. Convert 3D Region Labels to Regions .....	132
10.1.3.8.2. Grid Connections Processed (in the .grd file) .....	132
10.1.3.8.3. Grid Embedding .....	133
10.1.3.8.4. Retain Block-off .....	133
10.1.3.8.5. Regions in the .grd file .....	133
10.1.3.8.6. Boundary Conditions in .bcf File .....	134
10.1.3.8.7. Regions in the .gci File .....	134
10.1.3.8.8. Importing CFX-TASCflow TurboPre MFR Grids .....	134
10.1.3.8.9. Parameter File .....	134
10.1.3.9. CFX-4 Grid Files .....	134
10.1.3.9.1. Split Symmetry Planes .....	135
10.1.3.9.2. Import from Cylindrical Coordinates .....	135
10.1.3.9.3. Create 2D Regions on .....	135
10.1.3.9.4. Create 3D Regions on .....	135
10.1.3.9.5. Blocked Off Regions (SOLIDS) .....	136
10.1.3.9.6. Conducting Solid Regions (SOLCONS) .....	136
10.1.3.9.7. Import 2D Axisymmetric Mesh .....	136

10.1.3.9.8. Importing MFR Grids .....	137
10.1.3.10. CFX-BladeGenPlus Files .....	137
10.1.3.11. PATRAN Neutral Files .....	137
10.1.3.12. IDEAS Universal Files .....	137
10.1.3.13. GridPro/az3000 Grid Files .....	138
10.1.3.14. NASTRAN Files .....	138
10.1.3.15. Pointwise Gridgen Files .....	138
10.1.3.16. User Import .....	138
10.2. Mesh Tree View .....	139
10.2.1. Shortcut Menu Commands for Meshes and Regions .....	140
10.3. Deleting Meshes and Mesh Components from the Tree View .....	140
10.4. Transform Mesh Command .....	140
10.4.1. Target Location .....	141
10.4.2. Reference Coordinate Frame .....	141
10.4.3. Transformation: Rotation .....	141
10.4.3.1. Rotation Option: Principal Axis .....	142
10.4.3.2. Rotation Option: Rotation Axis .....	142
10.4.3.3. Rotation Angle Option .....	142
10.4.3.3.1. Specified .....	142
10.4.3.3.2. Full Circle .....	142
10.4.3.3.3. Two Points .....	142
10.4.4. Transformation: Translation .....	143
10.4.4.1. Method: Deltas .....	143
10.4.4.2. Method: Vector .....	143
10.4.5. Transformation: Scale .....	143
10.4.5.1. Method: Uniform .....	143
10.4.5.2. Method: Non Uniform .....	143
10.4.5.3. Scale Origin .....	143
10.4.5.4. Apply Scale To .....	144
10.4.6. Transformation: Reflection .....	144
10.4.6.1. Method .....	144
10.4.6.2. Apply Reflection To .....	144
10.4.7. Transformation: Turbo Rotation .....	145
10.4.7.1. Rotation Option: Principal Axis .....	145
10.4.7.2. Rotation Option: Rotation Axis .....	145
10.4.7.3. Rotation Axis Options .....	145
10.4.8. Multiple Copies .....	145
10.4.8.1. # of Copies .....	146
10.4.8.2. Delete Original .....	146
10.4.9. Advanced Options .....	146
10.4.9.1. Glue Adjacent Meshes .....	146
10.4.10. Automatic Transformation Preview .....	147
10.5. Gluing Meshes Together .....	147
10.6. Mesh Editor .....	148
10.7. Render Options .....	148
10.7.1. Render Options Dialog Box .....	148
10.7.1.1. Symbols .....	148
10.7.1.1.1. Symbol .....	149
10.7.1.1.2. Symbol Color .....	149
10.7.1.1.3. Symbol Size .....	149
10.7.1.2. Draw Faces .....	149



10.7.1.3. Face Color .....	149
10.7.1.4. Transparency .....	149
10.7.1.5. Draw Mode/Surface Drawing .....	149
10.7.1.5.1. Flat Shading .....	149
10.7.1.5.2. Smooth Shading .....	149
10.7.1.6. Face Culling .....	150
10.7.1.6.1. Front Faces .....	150
10.7.1.6.2. Back Faces .....	150
10.7.1.6.3. No Culling .....	150
10.7.1.7. Lighting .....	150
10.7.1.8. Specular .....	150
10.7.1.9. Draw Lines .....	150
10.7.1.10. Edge Angle/Render Edge Angle .....	150
10.7.1.11. Line Width .....	150
10.7.1.12. Line Color .....	151
10.7.1.13. Visibility .....	151
10.7.2. Render Options - Multiple 2D Regions .....	151
10.7.3. Render Options - Profile Geometry .....	151
10.7.3.1. Color .....	151
10.7.3.2. Lines .....	151
10.7.3.3. Transparency .....	152
10.7.3.4. Properties .....	152
10.8. Mesh Topology in CFX-Pre .....	152
10.8.1. Assemblies, Primitive Regions, and Composite Regions .....	152
10.8.1.1. Composite Regions .....	153
10.8.1.1.1. Applications of the Composite Regions .....	153
10.8.2. Domain and Subdomain Locations .....	153
10.8.3. Boundary Condition and Domain Interface Locations .....	154
10.8.4. Importing Multi-domain Cases .....	154
10.9. Advanced Topic: cfx5gtmconv Application .....	154
<b>11. Regions</b> .....	<b>155</b>
11.1. Primitive Regions .....	155
11.2. Composite Regions .....	155
11.3. Using Regions in CFX-Pre .....	156
11.4. Editing Regions in CFX-Pre .....	156
11.4.1. Defining and Editing Primitive Regions .....	156
11.4.1.1. Advanced Options .....	158
11.4.2. Defining and Editing Composite Regions .....	158
11.4.2.1. Union .....	159
11.4.2.2. Alias .....	159
11.5. Applications of Composite Regions .....	159
<b>12. Analysis Type</b> .....	<b>161</b>
12.1. Basic Settings Tab .....	161
12.1.1. Analysis Type Settings .....	161
12.1.1.1. Steady State .....	161
12.1.1.2. Transient .....	161
12.1.1.2.1. Time Duration .....	161
12.1.1.2.2. Time Steps .....	162
12.1.1.2.3. Initial Time .....	162
12.1.1.3. Transient Blade Row .....	163
<b>13. Domains</b> .....	<b>165</b>

13.1. Creating New Domains .....	165
13.2. The Details View for Domain Objects .....	166
13.3. Using Multiple Domains .....	166
13.3.1. Multiple Fluid Domains .....	167
13.3.2. Multiple Solid Domains .....	167
13.4. User Interface .....	167
13.4.1. Basic Settings Tab .....	168
13.4.1.1. Location and Type .....	168
13.4.1.1.1. Location .....	168
13.4.1.1.2. Domain Type .....	168
13.4.1.1.3. Coordinate Frame .....	169
13.4.1.2. Fluid and Particle Definitions and Solid Definitions .....	169
13.4.1.2.1. Morphology .....	169
13.4.1.2.1.1. Mean Diameter .....	170
13.4.1.2.1.2. Minimum Volume Fraction .....	170
13.4.1.2.1.3. Maximum Packing .....	170
13.4.1.2.1.4. Restitution Coefficient .....	170
13.4.1.2.1.5. Particle Diameter Distribution .....	170
13.4.1.2.1.6. Particle Shape Factors .....	170
13.4.1.2.1.7. Particle Diameter Change .....	170
13.4.1.2.1.7.1. Swelling Model .....	170
13.4.1.3. Particle Tracking .....	171
13.4.1.4. Domain Models .....	171
13.4.1.4.1. GT-SUITE Domain Models .....	171
13.4.1.4.2. Pressure: Reference Pressure .....	172
13.4.1.4.3. Buoyancy: Option .....	172
13.4.1.4.4. Domain Motion .....	172
13.4.1.4.5. Mesh Deformation .....	174
13.4.1.4.6. Passage Definition .....	175
13.4.2. Fluid Models Tab .....	175
13.4.2.1. Multiphase Options .....	176
13.4.2.1.1. Homogeneous Model .....	176
13.4.2.1.2. Free Surface Model .....	176
13.4.2.1.3. Multiphase Reactions .....	176
13.4.2.2. Heat Transfer .....	176
13.4.2.2.1. Homogeneous Model .....	176
13.4.2.2.2. Heat Transfer: Option .....	176
13.4.2.3. Turbulence .....	177
13.4.2.3.1. Homogeneous Model .....	177
13.4.2.3.2. Turbulence: Option .....	178
13.4.2.3.3. High Speed (compressible) Wall Heat Transfer Model .....	179
13.4.2.3.4. Buoyancy Turbulence .....	179
13.4.2.3.5. Wall Function .....	179
13.4.2.3.6. Turbulent Flux Closure for Heat Transfer .....	179
13.4.2.3.7. Transitional Turbulence .....	180
13.4.2.3.8. Advanced Turbulence Control .....	180
13.4.2.4. Reaction or Combustion Model .....	182
13.4.2.4.1. Soot Model .....	183
13.4.2.5. Thermal Radiation Model .....	183
13.4.2.6. Electromagnetic Model .....	184
13.4.2.7. Component Details .....	185

13.4.2.8. Additional Variable Details .....	185
13.4.3. Polydispersed Fluid Tab .....	186
13.4.4. Fluid Specific Models Tab .....	186
13.4.4.1. Fluid List Box .....	186
13.4.4.2. Kinetic Theory .....	186
13.4.4.3. Heat Transfer Model .....	187
13.4.4.4. Turbulence Model .....	187
13.4.4.5. Turbulent Wall Functions .....	188
13.4.4.6. Combustion Model .....	188
13.4.4.7. Erosion Model .....	188
13.4.4.8. Fluid Buoyancy Model .....	188
13.4.4.9. Solid Pressure Model .....	188
13.4.4.10. Component Details .....	188
13.4.4.11. Additional Variable Models .....	188
13.4.5. Fluid Pair Models Tab .....	189
13.4.5.1. Fluid Pair List box .....	189
13.4.5.2. Particle Coupling .....	189
13.4.5.3. Surface Tension Coefficient .....	189
13.4.5.4. Surface Tension Force Model .....	190
13.4.5.5. Interphase Transfer Model .....	190
13.4.5.5.1. Particle Model .....	190
13.4.5.5.2. Mixture Model .....	190
13.4.5.5.3. Free Surface Model .....	190
13.4.5.5.4. None .....	190
13.4.5.6. Momentum Transfer .....	190
13.4.5.6.1. Drag Force .....	190
13.4.5.6.2. Particle User Source .....	191
13.4.5.6.3. Lift Force .....	191
13.4.5.6.4. Virtual Mass Force .....	191
13.4.5.6.5. Wall Lubrication Force .....	191
13.4.5.6.6. Turbulent Dispersion Force .....	191
13.4.5.6.7. Pressure Gradient Force .....	192
13.4.5.7. Turbulence Transfer .....	192
13.4.5.8. Heat Transfer .....	192
13.4.5.9. Mass Transfer .....	192
13.4.5.10. Additional Variable Pairs .....	193
13.4.5.11. Component Pairs .....	193
13.4.5.11.1. Eulerian   Eulerian Pairs .....	193
13.4.5.11.2. Continuous   Particle Pairs .....	194
13.4.5.12. Particle Breakup .....	194
13.4.5.13. Particle Collision .....	194
13.4.6. Solid Models Tab .....	195
13.4.6.1. Heat Transfer .....	195
13.4.6.2. Thermal Radiation Model .....	196
13.4.6.3. Electromagnetic Model .....	196
13.4.6.4. Additional Variables Models .....	196
13.4.6.5. Solid Motion .....	196
13.4.7. Porosity Settings Tab .....	197
13.4.7.1. Area Porosity .....	197
13.4.7.2. Volume Porosity .....	197
13.4.7.3. Loss Models .....	197

13.4.7.4. Contact Area Model .....	198
13.4.7.5. Fluid Solid Area Density .....	198
13.4.7.6. Fluid Solid Heat Transfer .....	199
13.4.7.7. Additional Variable Pair Details .....	199
13.4.8. Particle Injection Regions Tab .....	199
13.4.8.1. Particle Injection Regions List Box .....	199
13.4.8.2. Coordinate Frame .....	199
13.4.8.3. Fluid: List Box .....	200
13.4.8.4. [fluid name] Check Box .....	200
13.4.8.4.1. Injection Method .....	200
13.4.8.4.1.1. Settings for Cone Definition .....	201
13.4.8.4.1.2. Settings for Injection Velocity .....	201
13.4.8.4.1.3. Settings for Particle Primary Breakup .....	202
13.4.9. Boundary Models Tab .....	203
13.4.10. Initialization Tab .....	203
13.4.11. Solver Control Tab .....	204
13.4.11.1. Timescale Control for Solid Domains (Transient Cases; Transient Blade Row Cases with Harmonic Balance) .....	204
<b>14. Domain Interfaces</b> .....	<b>205</b>
14.1. Creating and Editing a Domain Interface .....	205
14.1.1. Domain Interface: Basic Settings Tab .....	206
14.1.1.1. Interface Type .....	206
14.1.1.2. Interface Side 1/2 .....	206
14.1.1.2.1. Domain (Filter) .....	206
14.1.1.2.2. Region List .....	207
14.1.1.3. Interface Models .....	207
14.1.1.3.1. Interface Model Option: Translational Periodicity .....	207
14.1.1.3.2. Interface Model Option: Rotational Periodicity .....	207
14.1.1.3.3. Interface Model Option: General Connection .....	207
14.1.1.3.3.1. Frame Change/Mixing Model .....	207
14.1.1.3.3.1.1. Option .....	207
14.1.1.3.3.1.2. Frozen Rotor: Rotational Offset Check Box .....	208
14.1.1.3.3.1.3. Stage: Pressure Profile Decay Check Box .....	208
14.1.1.3.3.1.4. Stage: Downstream Velocity Constraint .....	208
14.1.1.3.3.2. Pitch Change Options .....	208
14.1.1.3.3.2.1. Pitch Change: Value: Pitch Ratio .....	208
14.1.1.3.3.2.2. Pitch Change: Specified Pitch Angles: Pitch Angle Side 1/2 .....	208
14.1.2. Domain Interface: Additional Interface Models Tab .....	209
14.1.2.1. Mass And Momentum .....	209
14.1.2.1.1. Conservative Interface Flux .....	209
14.1.2.1.2. No Slip Wall .....	210
14.1.2.1.2.1. No Slip Wall: Wall Velocity .....	210
14.1.2.1.3. Free Slip Wall .....	210
14.1.2.1.4. Side Dependent .....	210
14.1.2.2. Heat Transfer .....	210
14.1.2.2.1. Conservative Interface Flux .....	210
14.1.2.2.2. Side Dependent .....	211
14.1.2.3. Electric Field .....	211
14.1.2.3.1. Conservative Interface Flux .....	211
14.1.2.3.2. Side Dependent .....	211
14.1.2.4. Additional Variable .....	211

14.1.2.4.1. Conservative Interface Flux .....	211
14.1.2.4.2. Side Dependent .....	212
14.1.2.5. Conditional Connection Control .....	212
14.1.3. Domain Interface: Solid Interface Models Tab .....	212
14.1.4. Domain Interface: Mesh Connection Tab .....	213
14.1.4.1. Mesh Connection Method .....	213
14.1.4.1.1. Mesh Connection: Option .....	213
14.1.4.1.2. Intersection Control .....	213
14.1.4.1.2.1. Intersection Control: Option .....	214
<b>15. Boundary Conditions .....</b>	<b>217</b>
15.1. Default Boundary Condition .....	217
15.2. Creating and Editing a Boundary Condition .....	218
15.2.1. Boundary Basic Settings Tab .....	219
15.2.1.1. Boundary Type .....	219
15.2.1.2. Location .....	219
15.2.1.3. Coordinate Frame .....	219
15.2.1.4. Frame Type .....	220
15.2.1.5. Profile Boundary Conditions .....	220
15.2.1.5.1. Use Profile Data .....	220
15.2.1.6. Use GT-SUITE Coupling .....	220
15.2.2. Boundary Details Tab .....	221
15.2.2.1. Boundary Details: Inlet .....	222
15.2.2.1.1. Flow Regime: Inlet .....	222
15.2.2.1.2. Mesh Motion: Inlet .....	222
15.2.2.2. Boundary Details: Outlet .....	222
15.2.2.2.1. Flow Regime: Outlet .....	222
15.2.2.2.2. Mass and Momentum: Outlet .....	222
15.2.2.2.3. Pressure Averaging: Outlet .....	222
15.2.2.2.4. Thermal Radiation: Outlet .....	222
15.2.2.2.5. Mesh Motion: Outlet .....	222
15.2.2.3. Boundary Details: Opening .....	223
15.2.2.3.1. Mass and Momentum: Opening .....	223
15.2.2.3.2. Flow Direction: Opening .....	223
15.2.2.3.3. Loss Coefficient: Opening .....	223
15.2.2.3.4. Turbulence: Opening .....	223
15.2.2.3.5. Heat Transfer: Opening .....	223
15.2.2.3.6. Thermal Radiation: Opening .....	223
15.2.2.3.7. Component Details: Opening .....	223
15.2.2.3.8. Mesh Motion: Opening .....	223
15.2.2.4. Boundary Details: Wall .....	223
15.2.2.4.1. Wall Boiling Model: Wall .....	223
15.2.2.4.2. Mass And Momentum: Wall .....	224
15.2.2.4.2.1. Slip Model Settings .....	224
15.2.2.4.2.2. Shear Stress Settings .....	224
15.2.2.4.2.3. Wall Velocity Settings .....	224
15.2.2.4.2.3.1. Axis Definition .....	224
15.2.2.4.3. Wall Roughness .....	224
15.2.2.4.4. Solid Motion: Wall .....	225
15.2.2.4.5. Heat Transfer: Wall .....	225
15.2.2.4.6. Wall Contact Model .....	225
15.2.2.4.7. Thermal Radiation: Wall .....	225

15.2.2.4.8. Mesh Motion: Wall .....	225
15.2.2.5. Boundary Details: Symmetry .....	225
15.2.2.6. Boundary Details: Interfaces .....	226
15.2.2.7. Mesh Motion .....	226
15.2.2.8. Electric Field .....	226
15.2.3. Nonoverlap Conditions Tab .....	227
15.2.4. Boundary Fluid Values Tab .....	227
15.2.4.1. Fluid Values: Turbulence .....	227
15.2.4.1.1. Intensity and Length Scale .....	228
15.2.4.1.2. Intensity and Eddy Viscosity Ratio .....	228
15.2.4.1.3. k and Epsilon .....	228
15.2.4.1.4. Intensity and Auto Compute Length .....	228
15.2.4.2. Fluid Values: Volume Fraction .....	228
15.2.4.3. Fluid Values: Heat Transfer .....	229
15.2.4.4. Fluid Values for Inlets and Openings .....	229
15.2.4.4.1. Multiphase .....	229
15.2.4.4.2. MUSIG settings .....	230
15.2.4.4.3. Particle Tracking Settings for Inlets and Openings .....	230
15.2.4.4.3.1. Phase List .....	230
15.2.4.4.3.2. Particle Behavior .....	230
15.2.4.4.3.3. Mass and Momentum .....	230
15.2.4.4.3.4. Particle Position .....	230
15.2.4.4.3.5. Particle Locations .....	230
15.2.4.4.3.6. Number of Positions .....	230
15.2.4.4.3.7. Particle Mass Flow .....	230
15.2.4.4.3.8. Particle Diameter Distribution .....	230
15.2.4.4.3.9. Heat Transfer .....	230
15.2.4.4.3.10. Component Details .....	231
15.2.4.5. Fluid Values for Outlets .....	231
15.2.4.6. Fluid Values for Walls .....	231
15.2.4.6.1. Particle Tracking Settings for Walls .....	231
15.2.4.6.2. Settings for Particle-Wall Interaction .....	231
15.2.4.7. Fluid Values for Interfaces .....	232
15.2.4.7.1. Fluid-Solid Interface, Fluid Side .....	232
15.2.4.7.2. Fluid-Fluid and Periodic Interfaces .....	233
15.2.5. Boundary Solid Values Tab .....	233
15.2.5.1. Heat Transfer .....	233
15.2.5.1.1. Adiabatic .....	233
15.2.5.1.2. Fixed Temperature .....	233
15.2.5.1.3. Heat Flux .....	233
15.2.5.1.4. Heat Transfer Coefficient .....	233
15.2.5.1.5. Conservative Interface Flux .....	233
15.2.5.2. Additional Variables .....	233
15.2.6. Fluid Pair Values Tab .....	234
15.2.7. Boundary Sources Tab .....	234
15.2.8. Boundary Plot Options Tab .....	234
15.2.8.1. Boundary Contour .....	234
15.2.8.2. Boundary Vector .....	234
15.3. Interface Boundary Conditions .....	234
15.4. Symmetry Boundary Conditions .....	235
15.5. Working with Boundary Conditions .....	235

15.5.1. Boundary Condition Visualization .....	235
15.5.2. Profile Data and CEL Functions .....	236
15.5.2.1. Types of Discrete Profiles .....	236
15.5.2.2. Multiphase Boundary Condition Example .....	236
<b>16. Initialization .....</b>	<b>237</b>
16.1. Using the User Interface .....	237
16.1.1. Domain: Initialization Tab .....	237
16.1.1.1. Domain Initialization .....	238
16.1.2. Global Settings and Fluid Settings Tabs .....	238
16.1.2.1. Coordinate Frame Check Box .....	238
16.1.2.1.1. Coordinate Frame Check Box: Coordinate Frame .....	238
16.1.2.2. Frame Type Check Box .....	238
16.1.2.2.1. Frame Type .....	238
16.1.2.3. Initial Conditions: Velocity Type .....	239
16.1.2.4. Initial Conditions: Cartesian Velocity Components .....	239
16.1.2.4.1. Option .....	239
16.1.2.4.2. Velocity Scale Check Box .....	239
16.1.2.4.2.1. Velocity Scale Check Box: Value .....	239
16.1.2.4.3. U, V, W .....	239
16.1.2.5. Initial Conditions: Cylindrical Velocity Components .....	240
16.1.2.5.1. Option .....	240
16.1.2.5.2. Velocity Scale Check Box .....	240
16.1.2.5.3. Axial Component, Radial Component, Theta Component .....	240
16.1.2.6. Initial Conditions: Static Pressure .....	240
16.1.2.6.1. Option .....	240
16.1.2.6.2. Relative Pressure .....	240
16.1.2.7. Initial Conditions: Turbulence .....	240
16.1.2.7.1. Fractional Intensity .....	241
16.1.2.7.2. Eddy Viscosity Ratio .....	241
16.1.2.7.3. Eddy Length Scale .....	242
16.1.2.7.4. Turbulence Kinetic Energy .....	242
16.1.2.7.5. Turbulence Eddy Dissipation .....	242
16.1.2.7.6. Turbulence Eddy Frequency .....	242
16.1.2.7.7. Reynolds Stress Components .....	243
16.1.2.8. Initial Conditions: Temperature .....	243
16.1.2.9. Initial Conditions: Radiation Intensity .....	243
16.1.2.9.1. Option .....	243
16.1.2.10. Initial Conditions: Mixture Fraction .....	244
16.1.2.10.1. Option .....	244
16.1.2.10.2. Mixture Fraction .....	244
16.1.2.11. Initial Conditions: Mixture Fraction Variance .....	244
16.1.2.11.1. Option .....	244
16.1.2.11.2. Mix. Fracn. Variance .....	244
16.1.2.12. Initial Conditions: Component Details .....	244
16.1.2.12.1. List Box .....	244
16.1.2.12.2. [component name]: Option .....	245
16.1.2.12.3. [component name]: Mass Fraction .....	245
16.1.2.13. Initial Conditions: Additional Variable Details .....	245
16.1.2.14. Fluid Specific Initialization .....	245
16.1.2.15. Fluid Specific Initialization: List Box .....	245
16.1.2.16. Fluid Specific Initialization: [fluid name] Check Box .....	245

16.1.2.17. Fluid Specific Initialization: [fluid name] Check Box: Initial Conditions .....	245
16.1.2.17.1. Velocity Type .....	245
16.1.2.17.2. Volume Fraction: Option .....	246
16.1.2.17.3. Volume Fraction: Volume Fraction .....	246
16.1.2.18. Solid Specific Initialization .....	246
16.1.2.19. Solid Specific Initialization: List Box .....	246
16.1.2.20. Solid Specific Initialization: [solid name] Check Box .....	246
<b>17. Subdomains</b> .....	247
17.1. Creating and Editing a Subdomain .....	247
17.2. Subdomains Tab .....	248
17.3. Basic Settings Tab .....	248
17.3.1. Location .....	248
17.3.2. Coordinate Frame .....	248
17.4. Sources Tab .....	249
17.4.1. Single-Phase Fluid Sources .....	249
17.4.1.1. Momentum Source/Porous Loss .....	249
17.4.1.2. Equation Sources .....	250
17.4.1.2.1. Component Mass Fractions .....	250
17.4.1.2.2. Additional Variables .....	250
17.4.1.2.3. Continuity .....	250
17.4.1.2.4. Turbulence Quantities .....	251
17.4.1.2.5. Energy .....	251
17.4.2. Bulk Sources for Multiphase Simulations .....	251
17.4.3. Multiplying Sources by Porosity .....	251
17.5. Fluids Tab .....	252
17.5.1. Particle Absorption .....	252
17.6. Mesh Motion Tab .....	252
<b>18. Injection Regions</b> .....	253
18.1. Creating and Editing an Injection Region .....	253
18.2. User Interface .....	254
18.2.1. Basic Settings Tab .....	254
18.2.1.1. Region Type .....	254
18.2.1.2. Injection Location Type .....	254
18.2.1.3. Injection Positions .....	255
18.2.1.4. Coordinate Frame .....	255
18.2.2. Injection Details Tab .....	256
18.2.2.1. Injection Model .....	256
18.2.2.1.1. Hole Direction .....	256
18.2.2.2. Mass and Momentum .....	257
18.2.2.3. Flow Direction .....	258
18.2.2.4. Turbulence .....	258
18.2.2.5. Heat Transfer .....	258
18.2.2.6. Additional Variables .....	259
18.2.3. Injection Distribution Tab .....	259
18.2.3.1. Injection Distribution Basics .....	259
18.2.3.2. Injection Distribution Settings for the Jet in Crossflow Option .....	260
18.2.3.2.1. The Inflow Boundary Setting .....	260
18.2.3.2.2. Injection Trajectory .....	261
18.2.3.2.3. Injection Shape .....	263
18.2.3.2.4. Injection Variables .....	264
18.2.3.2.4.1. Uniform .....	265



18.2.3.2.4.2. Quadratic Radial Polynomial .....	265
18.2.3.2.4.3. Value .....	265
18.2.3.2.4.4. Standard Jet Velocity .....	265
18.2.3.2.4.5. Cartesian Vector Components .....	266
18.2.3.2.4.6. Cylindrical Vector Components .....	266
18.2.4. Fluid Values Tab .....	266
18.2.5. Plot Options Tab .....	267
18.2.5.1. Color by Parameter Value .....	267
18.2.5.2. Injection Region Vector .....	267
18.3. User Functions for Injection Regions .....	268
18.3.1. Using CFD-Post to Design an Injection Region Data File .....	268
18.4. Supported Capabilities and Limitations .....	269
18.5. Running Cases that involve Injection Regions .....	270
18.6. Postprocessing Injection Regions .....	270
<b>19. Source Points .....</b>	<b>271</b>
19.1. Basic Settings Tab .....	271
19.2. Sources Tab .....	272
19.2.1. Single-Phase Fluid Sources .....	272
19.2.1.1. Component Mass Fractions .....	272
19.2.1.2. Additional Variables .....	272
19.2.1.3. Continuity .....	272
19.2.1.3.1. Continuity Option .....	273
19.2.1.3.2. Additional Variables .....	273
19.2.1.3.3. Component Mass Fractions .....	273
19.2.1.3.4. Temperature .....	273
19.2.1.3.5. Velocity .....	273
19.2.1.4. Energy .....	273
19.2.1.5. Turbulence Eddy Dissipation or Turbulence Kinetic Energy .....	273
19.2.2. Multiphase Bulk Sources .....	273
19.2.3. Multiplying Sources by Porosity .....	273
19.3. Fluid Sources Tab .....	274
19.4. Sources in Solid Domains .....	274
19.5. Source Points and Mesh Deformation .....	274
<b>20. Rigid Bodies .....</b>	<b>275</b>
20.1. Rigid Body User Interface .....	275
20.1.1. Insert Rigid Body Dialog Box .....	275
20.1.2. Basic Settings Tab .....	276
20.1.2.1. Mass .....	276
20.1.2.2. Location .....	276
20.1.2.3. Coordinate Frame .....	277
20.1.2.4. Mass Moment of Inertia .....	277
20.1.3. Dynamics Tab .....	277
20.1.3.1. External Force Definitions .....	278
20.1.3.2. External Torque Definitions .....	278
20.1.3.3. Degrees of Freedom .....	279
20.1.3.3.1. Translational Degrees of Freedom .....	279
20.1.3.3.2. Rotational Degrees of Freedom .....	279
20.1.3.4. Gravity .....	280
20.1.4. Initial Conditions Tab .....	280
20.1.4.1. Center of Mass .....	280
20.1.4.2. Linear Velocity .....	281

20.1.4.3. Angular Velocity .....	281
20.1.4.4. Linear Acceleration .....	281
20.1.4.5. Angular Acceleration .....	281
<b>21. Units and Dimensions .....</b>	<b>283</b>
21.1. Units Syntax .....	283
21.2. Using Units in CFX-Pre .....	284
21.2.1. Units Commonly Used in CFX .....	284
21.2.2. Defining Your Own Units .....	287
21.3. Setting the Solution Units .....	288
<b>22. Solver Control .....</b>	<b>289</b>
22.1. Basic Settings Tab .....	289
22.1.1. Basic Settings: Common .....	289
22.1.1.1. Advection Scheme .....	289
22.1.1.2. Transient Scheme .....	289
22.1.1.3. Turbulence Numerics .....	289
22.1.1.4. Convergence Criteria .....	290
22.1.1.5. Elapsed Wall Clock Time Control .....	290
22.1.1.6. Interrupt Control .....	290
22.1.1.6.1. List Box .....	291
22.1.1.6.1.1. [Interrupt Condition Name] .....	291
22.1.1.7. Junction Box Routine .....	292
22.1.2. Basic Settings for Steady-State Simulations and for Transient Blade Row Cases that use the Harmonic Balance Transient Method .....	292
22.1.2.1. Convergence Control: Min. Iterations .....	292
22.1.2.2. Convergence Control: Max. Iterations .....	292
22.1.2.3. Convergence Control: Fluid Timescale Control .....	292
22.1.2.4. Convergence Control: Solid Timescale Control .....	292
22.1.3. Basic Settings for Transient Simulations .....	293
22.1.3.1. Convergence Control .....	293
22.1.3.1.1. Min. Coeff. Loops .....	293
22.1.3.1.2. Max. Coeff. Loops .....	293
22.1.3.1.3. Fluid Timescale Control .....	293
22.1.3.1.4. Solid Timescale Control .....	293
22.1.4. Immersed Solid Control .....	293
22.2. Equation Class Settings Tab .....	295
22.3. Particle Control Tab .....	295
22.4. Rigid Body Control Tab .....	295
22.5. Advanced Options Tab .....	299
<b>23. Output Control .....</b>	<b>309</b>
23.1. User Interface .....	309
23.1.1. Results Tab .....	309
23.1.1.1. Option .....	309
23.1.1.2. File Compression .....	310
23.1.1.3. Output Variables List .....	310
23.1.1.4. Include Mesh Check Box .....	310
23.1.1.5. File Format Check Box .....	310
23.1.1.6. Output Equation Residuals Check Box .....	310
23.1.1.7. Output Boundary Flows Check Box .....	310
23.1.1.8. Output Variable Operators Check Box .....	310
23.1.1.9. Extra Output Variables List .....	310
23.1.1.10. Output Particle Boundary Vertex Fields Check Box .....	310

23.1.2. Backup Tab .....	310
23.1.2.1. List Box .....	310
23.1.2.2. [Backup Results Name] .....	311
23.1.2.2.1. Option .....	311
23.1.2.2.2. File Compression .....	311
23.1.2.2.3. Output Variables List .....	311
23.1.2.2.4. Include Mesh Check Box .....	311
23.1.2.2.5. Output Equation Residuals Check Box .....	311
23.1.2.2.6. Output Boundary Flows Check Box .....	311
23.1.2.2.7. Output Variable Operators Check Box .....	311
23.1.2.2.8. Extra Output Variables List .....	311
23.1.2.2.9. Output Particle Boundary Vertex Fields Check Box .....	311
23.1.2.2.10. Include Tracks of One-way Coupled Particles Check Box .....	311
23.1.2.2.11. Output Frequency: Option .....	312
23.1.2.3. Backup Data Retention .....	312
23.1.2.3.1. Option .....	312
23.1.3. Transient Results Tab .....	312
23.1.3.1. List Box .....	312
23.1.3.2. [Transient Results Name] .....	312
23.1.3.2.1. Option .....	312
23.1.3.2.2. File Compression .....	312
23.1.3.2.3. Output Variables List .....	313
23.1.3.2.4. Include Mesh Check Box .....	313
23.1.3.2.5. File Format Check Box .....	313
23.1.3.2.6. Output Equation Residuals Check Box .....	313
23.1.3.2.7. Output Boundary Flows Check Box .....	313
23.1.3.2.8. Output Variable Operators Check Box .....	313
23.1.3.2.9. Extra Output Variables List .....	313
23.1.3.2.10. Output Particle Boundary Vertex Fields Check Box .....	313
23.1.3.2.11. Output Frequency .....	313
23.1.3.3. Transient Blade Row Results .....	313
23.1.4. Transient Statistics Tab .....	314
23.1.4.1. List Box .....	314
23.1.4.2. [Transient Statistics Name] .....	314
23.1.4.2.1. Option .....	314
23.1.4.2.2. Output Variables List .....	315
23.1.4.2.3. Start Iteration List Check Box .....	315
23.1.4.2.3.1. Start Iteration List Check Box: Start Iteration List .....	315
23.1.4.2.4. Stop Iteration List Check Box .....	315
23.1.4.2.4.1. Stop Iteration List Check Box: Stop Iteration List .....	315
23.1.5. Monitor Tab .....	316
23.1.5.1. Monitor Objects Check Box .....	316
23.1.5.1.1. Monitor Coeff. Loop Convergence .....	316
23.1.5.1.2. Monitor Balances: Option .....	317
23.1.5.1.3. Monitor Forces: Option .....	317
23.1.5.1.4. Monitor Residuals: Option .....	317
23.1.5.1.5. Monitor Totals: Option .....	317
23.1.5.1.6. Monitor Particles: Option .....	317
23.1.5.1.7. Efficiency Output Check Box .....	318
23.1.5.1.7.1. Inflow Boundary .....	318
23.1.5.1.7.2. Outflow Boundary .....	318

23.1.5.1.7.3. Efficiency Type .....	318
23.1.5.1.7.4. Efficiency Calculation Method .....	318
23.1.5.1.8. Monitor Points And Expressions .....	318
23.1.5.1.8.1. Monitor Points And Expressions: List Box .....	318
23.1.5.1.8.2. [Monitor Name]: Option .....	318
23.1.5.1.8.3. [Monitor Name]: Output Variables List .....	319
23.1.5.1.8.4. [Monitor Name]: Cartesian Coordinates .....	319
23.1.5.1.8.5. [Monitor Name]: Cylindrical Coordinates .....	319
23.1.5.1.8.6. [Monitor Name]: Expression Value .....	320
23.1.5.1.8.7. [Monitor Name]: Coordinate Frame .....	320
23.1.5.1.8.8. [Monitor Name]: Monitor Location Control .....	320
23.1.5.1.8.9. [Monitor Name]: Monitor Statistics .....	322
23.1.5.1.9. Aerodynamic Damping .....	324
23.1.5.1.9.1. Aerodynamic Damping: List Box .....	324
23.1.5.1.9.2. [Aerodynamic Damping Name]: Option .....	324
23.1.5.1.9.3. [Aerodynamic Damping Name]: Normalization .....	325
23.1.5.1.9.4. [Aerodynamic Damping Name]: Location Type .....	325
23.1.5.1.9.5. [Aerodynamic Damping Name]: Integration Multiplier .....	325
23.1.5.1.9.6. [Aerodynamic Damping Name]: Integration Interval .....	325
23.1.5.1.10. Monitor GT-SUITE .....	325
23.1.5.1.11. Radiometer .....	325
23.1.5.1.11.1. Radiometer: List Box .....	326
23.1.5.1.11.2. [Radiometer Name]: Option .....	326
23.1.5.1.11.3. [Radiometer Name]: Cartesian Coordinates .....	326
23.1.5.1.11.4. [Radiometer Name]: Temperature .....	326
23.1.5.1.11.5. [Radiometer Name]: Quadrature Points .....	326
23.1.5.1.11.6. [Radiometer Name]: Coordinate Frame Check Box .....	327
23.1.5.1.11.7. [Radiometer Name]: Coordinate Frame Check Box: Coordinate Frame .....	327
23.1.5.1.11.8. [Radiometer Name]: Diagnostic Output Level Check Box .....	327
23.1.5.1.11.9. [Radiometer Name]: Diagnostic Output Level Check Box: Diagnostic Output Level .....	327
23.1.5.1.11.10. [Radiometer Name]: Direction: Option .....	327
23.1.5.1.11.11. [Radiometer Name]: Direction: X Component, Y Component, Z Component .....	327
23.1.5.1.12. Monitor Surfaces .....	327
23.1.5.1.12.1. Monitor Surfaces: List Box .....	328
23.1.5.1.12.2. [Monitor Surface Name]: Option .....	328
23.1.5.1.12.3. [Monitor Surface Name]: Coordinate Frame .....	328
23.1.5.1.12.4. [Monitor Surface Name]: Output Variables List .....	328
23.1.5.1.12.5. [Monitor Surface Name]: Output Location List .....	328
23.1.5.1.12.6. [Monitor Surface Name]: File Compression Level .....	329
23.1.5.1.12.7. [Monitor Surface Name]: Output Frequency .....	329
23.1.5.1.12.8. Monitor Location Data Retention .....	329
23.1.6. Particles Tab .....	329
23.1.6.1. Particle Track File Check Box .....	329
23.1.6.1.1. Option .....	329
23.1.6.1.2. Track Positions Check Box .....	330
23.1.6.1.3. Track Positions Check Box: Track Positions .....	330
23.1.6.1.4. Interval .....	330
23.1.6.1.5. Track Distance Spacing .....	330
23.1.6.1.6. Track Time Spacing .....	331

23.1.6.1.7. Track Printing Interval Check Box .....	331
23.1.6.1.8. Track Printing Interval Check Box: Interval .....	331
23.1.6.1.9. Keep Track File Check Box .....	331
23.1.6.1.10. Track File Format Check Box .....	331
23.1.6.1.11. Track File Format Check Box: Track File Format .....	331
23.1.6.2. Particle Histogram .....	332
23.1.6.2.1. Histogram Type .....	332
23.1.6.2.2. Number of Bins .....	332
23.1.6.2.3. Particle Definition .....	333
23.1.6.2.4. x Variable .....	333
23.1.6.2.5. y Variable .....	333
23.1.6.2.6. x Var. Lower Bound .....	333
23.1.6.2.7. x Var. Upper Bound .....	333
23.1.6.2.8. Sample Location .....	333
23.1.6.2.9. Export Format .....	333
23.1.6.2.10. Export Frequency .....	334
23.1.6.3. Transient Particle Diagnostics .....	334
23.1.6.4. Transient Particle Diagnostics: List Box .....	334
23.1.6.5. Transient Particle Diagnostics: [Transient Particle Diagnostics Name] .....	334
23.1.6.5.1. Option .....	334
23.1.6.5.2. Particles List .....	334
23.1.6.5.3. Spray Mass Frac. ....	334
23.1.6.5.4. Penetration Origin and Direction: Option .....	334
23.1.6.5.5. Penetration Origin and Direction: Injection Center .....	335
23.1.6.5.6. Penetration Origin and Direction: Injection Direction .....	335
23.1.6.5.6.1. Option .....	335
23.1.6.5.7. Axial Penetration: Option .....	335
23.1.6.5.8. Radial Penetration: Option .....	335
23.1.6.5.9. Normal Penetration: Option .....	335
23.1.6.5.10. Spray Angle: Option .....	335
23.1.6.5.10.1. Spray Angle: Spray Radius at Penetration Origin Check Box .....	335
23.1.6.5.10.2. Spray Angle: Spray Radius at Penetration Origin Check Box: Spray Radi- us .....	336
23.1.7. Export Tab .....	336
23.1.7.1. List Box .....	336
23.1.7.2. [Export Name]: Option .....	336
23.1.7.3. [Export Name]: Export Format .....	336
23.1.7.3.1. Filename Prefix Check Box .....	336
23.1.7.3.2. Filename Prefix Check Box: Filename Prefix .....	336
23.1.7.3.3. Include Topology Check Box .....	336
23.1.7.3.4. Include File Info Header Check Box .....	337
23.1.7.3.5. Precision Check Box .....	337
23.1.7.3.6. Precision Check Box: Precision .....	337
23.1.7.4. [Export Name]: Export Frequency .....	337
23.1.7.4.1. Option .....	337
23.1.7.5. [Export Name]: Export Surface .....	338
23.1.7.5.1. List Box .....	338
23.1.7.5.2. [Export Surface Name]: Option .....	338
23.1.7.5.3. [Export Surface Name]: Particle Definition List .....	338
23.1.7.5.4. [Export Surface Name]: Output Boundary List .....	338
23.1.7.5.5. [Export Surface Name]: Location Type .....	338

23.1.7.5.6. [Export Surface Name]: Output Variables List .....	339
23.1.7.5.7. [Export Surface Name]: Output Nodal Area Vectors Check Box .....	339
23.1.7.5.8. [Export Surface Name]: Excitation Frequency .....	339
23.1.7.5.9. [Export Surface Name]: Accumulation Period .....	339
23.1.8. Common Settings .....	340
23.1.8.1. Option .....	340
23.1.8.2. File Compression .....	341
23.1.8.3. Output Variables List .....	341
23.1.8.4. Include Mesh Check Box .....	341
23.1.8.5. File Format Check Box .....	341
23.1.8.6. Output Equation Residuals Check Box .....	342
23.1.8.7. Output Boundary Flows Check Box .....	343
23.1.8.8. Output Variable Operators Check Box .....	343
23.1.8.9. Output Particle Boundary Vertex Fields Check Box .....	343
23.1.8.10. Output Frequency Options .....	344
23.1.8.10.1. Timestep Interval .....	344
23.1.8.10.2. Timestep List .....	344
23.1.8.10.3. Time Interval .....	344
23.1.8.10.4. Time List .....	344
23.1.8.10.5. Every Timestep .....	344
23.1.8.10.6. Every Iteration .....	344
23.1.8.10.7. Iteration Interval .....	344
23.1.8.10.8. Iteration List .....	344
23.1.8.10.9. Wall Clock Time Interval .....	344
23.1.8.10.10. None .....	345
23.2. Working with Output Control .....	345
23.2.1. Working with Transient Statistics .....	345
23.2.1.1. Statistic Initialization and Accumulation .....	345
23.2.1.2. Statistics as Variable Operators .....	346
23.2.1.3. Using Statistics with Transient Rotor-Stator Cases .....	346
23.2.2. Working with Monitors .....	347
23.2.2.1. Setting up Monitors .....	347
23.2.2.2. Transient/ Mesh Deformation Runs .....	348
23.2.2.3. Output Information .....	348
23.2.2.4. Expression .....	348
23.2.2.5. Viewing Monitor Values during a Run .....	348
23.2.2.6. Viewing Monitor Point Values after a Run .....	349
23.2.2.7. Using Monitors in CEL .....	349
23.2.3. Working with Export Results .....	350
23.2.3.1. File Naming Conventions .....	350
23.2.3.2. Propagating Older Time Step Values .....	351
23.2.3.3. Output Boundary List and Output Region Naming .....	351
23.2.3.4. Output Variables List .....	351
<b>24. Transient Blade Row Models .....</b>	<b>353</b>
24.1. Inserting a New Transient Blade Row Models Object .....	353
24.2. Transient Blade Row Models Details View .....	353
24.2.1. Transient Blade Row Model Settings .....	353
24.2.1.1. Time Transformation Disturbance Settings .....	354
24.2.1.2. Fourier Transformation Disturbance Settings .....	356
24.2.1.2.1. Basic Settings Tab .....	356
24.2.1.2.2. Advanced Options Tab .....	358

24.2.1.2.2.1. Frequency Filtering .....	358
24.2.2. Transient Method .....	359
24.2.2.1. Time Integration .....	359
24.2.2.2. Harmonic Balance .....	360
<b>25. Mesh Adaption .....</b>	<b>361</b>
25.1. Overview .....	361
25.2. Setting Up Mesh Adaption .....	363
25.3. The Details View for Mesh Adaption .....	364
25.3.1. Basic Settings Tab .....	364
25.3.1.1. Region List .....	364
25.3.1.2. Save Intermediate Files .....	364
25.3.1.3. Adaption Criteria .....	364
25.3.1.3.1. Variables List .....	364
25.3.1.3.2. Max. Num. Steps .....	365
25.3.1.3.3. Option .....	365
25.3.1.4. Adaption Method .....	365
25.3.1.4.1. Option .....	365
25.3.1.4.2. Minimum Edge Length .....	366
25.3.1.5. Adaption Convergence Criteria .....	366
25.3.2. Advanced Options Tab .....	366
25.3.2.1. Node Alloc. Parameter .....	366
25.3.2.2. Number of Levels .....	367
25.4. Advanced Topic: Adaption with 2D Meshes .....	367
<b>26. Expert Control Parameters .....</b>	<b>369</b>
26.1. Modifying Expert Control Parameters .....	369
<b>27. Coordinate Frames .....</b>	<b>371</b>
27.1. Creating a New Coordinate Frame .....	371
27.2. Coordinate Frame Basic Settings Tab .....	371
27.2.1. Coordinate Frame: Option .....	372
27.2.1.1. Point and Normal .....	372
27.2.1.2. Axis Points .....	372
27.2.2. Coordinate Frame: Centroid .....	372
27.2.2.1. Location .....	372
27.2.2.2. Centroid Type .....	373
27.2.3. Coordinate Frame: Direction .....	373
27.2.3.1. Invert Normal Axis Direction Check Box .....	373
27.2.3.2. Point on Reference Plane 1-3 Check Box .....	373
27.2.3.3. Point on Reference Plane 1-3 Check Box: Coordinate .....	374
27.2.4. Coord Frame Type .....	374
27.2.5. Reference Coord Frame .....	374
27.2.6. Origin .....	374
27.2.7. Z-Axis Point .....	374
27.2.8. X-Z Plane Point .....	375
27.2.9. Frame Motion Settings .....	375
27.2.10. Visibility Check Box .....	375
<b>28. User Locations .....</b>	<b>377</b>
28.1. User Surface .....	377
28.2. User Line .....	378
28.3. User Line Cloud .....	380
28.4. User Point .....	384
28.5. User Point Cloud .....	385

<b>29. Materials and Reactions</b> .....	387
29.1. Materials .....	387
29.1.1. Library Materials .....	387
29.1.2. Material Details View: Common Settings .....	388
29.1.2.1. Option .....	388
29.1.2.2. Material Group .....	389
29.1.2.2.1. User .....	389
29.1.2.2.2. Air Data .....	389
29.1.2.2.3. CHT Solids .....	389
29.1.2.2.4. Calorically Perfect Ideal Gases .....	389
29.1.2.2.5. Constant Property Gases / Liquids .....	389
29.1.2.2.6. Dry/Wet Redlich Kwong .....	389
29.1.2.2.7. Dry/Wet Redlich Kwong RGP .....	389
29.1.2.2.8. Dry/Wet Peng Robinson RGP .....	390
29.1.2.2.9. Dry/Wet Soave Redlich Kwong RGP .....	390
29.1.2.2.10. Dry/Wet Steam .....	390
29.1.2.2.11. Gas Phase Combustion .....	390
29.1.2.2.12. Interphase Mass Transfer .....	390
29.1.2.2.13. Particle Solids .....	390
29.1.2.2.14. Soot .....	391
29.1.2.2.15. Water Data .....	391
29.1.2.3. Material Description .....	391
29.1.2.4. Thermodynamic State .....	391
29.1.2.5. Coordinate Frame .....	391
29.1.3. Material Details View: Pure Substance .....	391
29.1.3.1. Basic Settings Tab .....	391
29.1.3.2. Material Properties Tab .....	392
29.1.3.2.1. General Material .....	392
29.1.3.2.1.1. Equation of State - Value .....	392
29.1.3.2.1.2. Equation of State - Ideal Gas .....	393
29.1.3.2.1.3. Equation of State - Real Gas .....	393
29.1.3.2.2. Table .....	394
29.1.3.2.3. Table Generation .....	394
29.1.3.2.3.1. Minimum and Maximum Temperature .....	394
29.1.3.2.3.2. Minimum and Maximum Absolute Pressure .....	394
29.1.3.2.3.3. Error Tolerance .....	394
29.1.3.2.3.4. Maximum Points .....	395
29.1.3.2.3.5. Pressure/Temperature Extrapolation .....	395
29.1.4. Material Details View: Fixed Composition Mixture .....	395
29.1.4.1. Basic Settings Tab .....	395
29.1.4.2. Mixture Properties Tab .....	396
29.1.5. Material Details View: Variable Composition Mixture .....	396
29.1.5.1. Basic Settings Tab .....	396
29.1.5.2. Mixture Properties Tab .....	396
29.1.6. Material Details View: Homogeneous Binary Mixture .....	398
29.1.6.1. Basic Settings Tab .....	398
29.1.6.2. Saturation Properties Tab .....	398
29.1.6.2.1. General .....	398
29.1.6.2.2. Table .....	398
29.1.6.2.3. Real Gas .....	399
29.1.6.2.4. Table Generation .....	399



29.1.7. Material Details View: Reacting Mixture .....	399
29.1.7.1. Basic Settings Tab .....	399
29.1.7.2. Mixture Properties Tab .....	399
29.1.8. Material Details View: Hydrocarbon Fuel .....	400
29.1.8.1. Basic Settings Tab .....	400
29.1.8.2. Proximate/ Ultimate Analysis Tab .....	400
29.1.8.3. Mixture Materials Tab .....	400
29.2. Reactions .....	401
29.2.1. Basic Settings Tab .....	401
29.2.2. Single Step .....	401
29.2.2.1. Single Step: Basic Settings .....	401
29.2.2.2. Single Step: Reactants .....	401
29.2.2.3. Single Step: Products .....	402
29.2.2.4. Single Step: Reaction Rates .....	402
29.2.3. Multiple Step .....	402
29.2.4. Flamelet Library .....	403
29.2.5. Multiphase .....	403
29.2.5.1. Multiphase: Basic Settings .....	403
29.2.5.2. Multiphase: Reactants .....	403
29.2.5.3. Multiphase: Products .....	404
29.2.5.4. Multiphase: Multiphase Reactions .....	404
<b>30. Additional Variables .....</b>	<b>405</b>
30.1. User Interface .....	405
30.1.1. Insert Additional Variable Dialog Box .....	405
30.1.2. Basic Settings Tab for Additional Variable Objects .....	405
30.1.2.1. Variable Type .....	406
30.1.2.2. Units .....	406
30.1.2.3. Tensor Type .....	406
30.1.3. Fluid Models and Fluid Specific Models Tabs for Domain Objects .....	406
30.1.3.1. Additional Variable Details: List Box .....	406
30.1.3.2. Additional Variable Details: [Additional Variable Name] Check Box .....	406
30.1.3.2.1. Option .....	407
30.1.3.2.1.1. Transport Equation .....	407
30.1.3.2.1.2. Diffusive Transport Equation .....	407
30.1.3.2.1.3. Homogeneous Transport Equation .....	407
30.1.3.2.1.4. Homogeneous Diffusive Transport Equation .....	407
30.1.3.2.1.5. Poisson Equation .....	407
30.1.3.2.1.6. Homogeneous Poisson Equation .....	407
30.1.3.2.1.7. Fluid Dependent .....	407
30.1.3.2.1.8. Algebraic Equation .....	408
30.1.3.2.1.9. Vector Algebraic Equation .....	408
30.1.3.2.2. Value .....	408
30.1.3.2.3. Kinematic Diffusivity Check Box .....	408
30.1.3.2.4. Kinematic Diffusivity Check Box: Kinematic Diffusivity .....	408
30.1.3.2.5. AV Properties for Fluid: Frame Overview .....	408
30.1.3.2.6. AV Properties for Fluid: List Box .....	409
30.1.3.2.7. AV Properties for Fluid: [Fluid Name] Check Box .....	409
30.1.3.2.8. AV Properties for Fluid: [Fluid Name] Check Box: Kinematic Diffusivity .....	409
30.1.3.2.9. Vector xValue, Vector yValue, and Vector zValue .....	409
30.1.4. Boundary Details and Fluid Values Tabs for Boundary Condition Objects .....	409
30.1.4.1. Additional Variables: List Box .....	410

30.1.4.2. Additional Variables: [Name] .....	410
30.1.4.2.1. Option .....	410
30.1.4.2.2. Value .....	410
30.1.4.2.3. Flux .....	410
30.1.4.2.4. Transfer Coefficient .....	410
30.2. Creating an Additional Variable .....	410
<b>31. Expressions</b> .....	413
31.1. Expressions Workspace .....	413
31.1.1. Definition .....	414
31.1.2. Plot .....	415
31.1.3. Evaluate .....	415
31.2. Creating an Expression .....	415
31.3. Modifying an Expression .....	416
31.4. Importing or Exporting an Expression .....	416
31.4.1. Importing CCL Expressions .....	416
31.4.2. Exporting CCL Expressions .....	417
<b>32. User Functions</b> .....	419
32.1. Interpolation (Data Input) User Functions .....	420
32.1.1. One Dimensional Interpolation .....	420
32.1.1.1. Extended Minimum and Maximum .....	421
32.1.2. Three Dimensional Interpolation .....	422
32.1.3. Importing Data from a File .....	422
32.1.4. Using Interpolation (Data Input) Functions in CEL Expressions .....	423
32.2. Interpolation (From File) User Functions (Profile Functions) .....	423
32.2.1. Establishing Profile Data and Creating Profile Functions .....	424
32.2.1.1. Profile Data Format .....	424
32.2.1.2. Profile Function Details View .....	426
32.2.1.2.1. Associating Profile Data with CEL Parameters .....	426
32.2.1.2.2. Coordinate System Definition .....	426
32.2.1.2.3. Visualization Options for Profile Data .....	427
32.2.2. Using Interpolation (From File) Functions in CEL Expressions .....	428
32.3. Table User Functions .....	428
32.3.1. Establishing Table Data and Creating Table Functions .....	429
32.3.1.1. Table Data Format .....	430
32.3.1.2. Table Editor .....	430
32.3.1.2.1. Auto Fill Values Dialog Box .....	432
32.3.1.3. Table Function Details View .....	432
32.3.2. Using Table Data User Functions in CEL Expressions .....	433
32.4. Functional Mockup Unit Instance .....	433
32.5. User CEL Functions .....	434
32.5.1. Function Name .....	435
32.5.1.1. Defining Quantities with User CEL Functions .....	435
32.5.2. Argument Units .....	435
32.5.3. Result Units .....	435
32.6. GT-SUITE Patch Functions .....	435
32.7. GT-SUITE Turboshaft Functions .....	436
<b>33. User Routines</b> .....	439
33.1. User CEL Routines .....	439
33.1.1. Calling Name .....	440
33.1.2. Library Name and Library Path .....	440
33.2. Junction Box Routines .....	441

33.3. Particle User Routines .....	441
<b>34. GT-SUITE Model Object .....</b>	<b>443</b>
34.1. GT-SUITE Model Details View .....	443
<b>35. Simulation Control .....</b>	<b>445</b>
<b>36. Operating Points .....</b>	<b>447</b>
<b>37. Execution and Termination Control .....</b>	<b>449</b>
37.1. Execution Control .....	449
37.1.1. Overview of Defining CFX-Solver Startup .....	449
37.1.2. The Details View for Execution Control .....	450
37.1.2.1. Run Definition Tab .....	450
37.1.2.1.1. Parallel Environment .....	451
37.1.2.2. Operating Points Tab .....	451
37.1.2.3. Initial Values Tab .....	451
37.1.2.4. Partitioner Tab .....	453
37.1.2.4.1. Executable Settings .....	453
37.1.2.4.2. Partitioning Detail .....	453
37.1.2.4.2.1. Partition Type .....	453
37.1.2.4.2.2. Partition Weighting .....	454
37.1.2.4.2.3. Multidomain Option .....	454
37.1.2.4.2.4. Multipass Partitioning .....	455
37.1.2.4.3. Partition Smoothing .....	455
37.1.2.4.4. Partitioner Memory .....	455
37.1.2.5. Solver Tab .....	455
37.1.2.6. Interpolator Tab .....	457
37.2. Termination Control .....	457
37.2.1. Overview of Configuration Termination .....	458
37.2.2. Details View for Termination Control .....	458
<b>38. Configurations .....</b>	<b>461</b>
38.1. Overview of Defining a Configuration .....	461
38.2. The Details View for Configuration .....	462
38.2.1. General Settings Tab .....	462
38.2.2. Remeshing Tab .....	463
38.2.2.1. User Defined Remeshing .....	464
38.2.2.2. Ansys ICEM CFD Replay Remeshing .....	465
38.2.2.2.1. ICEM CFD Geometry Control .....	466
38.2.2.2.2. ICEM CFD Mesh Control .....	466
38.2.2.2.3. Scalar Parameter .....	467
38.2.3. Run Definition Tab .....	467
38.2.4. Initial Values Tab .....	467
38.2.5. Partitioner Tab .....	468
38.2.6. Solver Tab .....	468
38.2.7. Interpolator Tab .....	468
<b>39. Quick Setup Mode .....</b>	<b>469</b>
39.1. Starting a New Case in Quick Setup Mode .....	469
39.2. Simulation Definition Tab .....	469
39.2.1. Simulation Data .....	470
39.2.1.1. Problem Type .....	470
39.2.2. Working Fluid .....	470
39.2.3. Mesh Data .....	470
39.3. Physics Definition .....	471
39.3.1. Analysis Type .....	471

39.3.2. Model Data .....	471
39.4. Boundary Definition .....	471
39.5. Final Operations .....	472
<b>40. Turbomachinery Mode .....</b>	<b>473</b>
40.1. Starting a New Case in Turbo Mode .....	474
40.2. Navigation through Turbo Mode .....	474
40.3. Basic Settings .....	474
40.3.1. Machine Type .....	474
40.3.2. Axes .....	474
40.3.2.1. Coordinate Frame .....	475
40.3.2.2. Rotational Axis .....	475
40.3.2.3. Axis Visibility .....	475
40.3.3. Analysis Type .....	475
40.4. Component Definition .....	475
40.4.1. Component Selector .....	475
40.4.2. Component Type .....	476
40.4.3. Mesh File .....	476
40.4.4. Passages and Alignment .....	476
40.4.4.1. Passages/Mesh .....	476
40.4.4.2. Passages to Model .....	477
40.4.4.3. Passages in 360 .....	477
40.4.4.4. Theta Offset .....	477
40.4.5. Available Volumes .....	477
40.4.6. Region Information .....	477
40.4.7. Wall Configuration .....	477
40.5. Physics Definition .....	478
40.5.1. Fluid .....	478
40.5.2. Model Data .....	478
40.5.2.1. Reference Pressure .....	478
40.5.2.2. Heat Transfer .....	478
40.5.2.3. Turbulence .....	478
40.5.2.4. Mass Flow .....	478
40.5.3. Boundary Templates .....	479
40.5.4. Interface .....	479
40.5.5. Solver Parameters .....	479
40.5.5.1. Advection Scheme .....	479
40.5.5.2. Convergence Control .....	479
40.5.5.3. Time Scale Option .....	479
40.5.5.4. Max. Coeff. Loops .....	479
40.6. Interface Definition .....	479
40.6.1. Type .....	480
40.7. Disturbance Definition .....	481
40.7.1. Type .....	481
40.8. Boundary Definition .....	481
40.8.1. Boundary Data .....	481
40.8.2. Flow Specification/Wall Influence on Flow .....	482
40.9. Final Operations .....	482
<b>41. Library Objects .....</b>	<b>483</b>
41.1. Boiling Water .....	483
41.2. Cavitating Water .....	484
41.3. Coal Combustion .....	484

41.4. Comfort Factors .....	484
41.5. Multigray Radiation .....	485
<b>42. Command Editor Dialog Box .....</b>	<b>487</b>
42.1. Using the Command Editor .....	487
42.2. Performing Command Actions .....	488
42.3. Using Power Syntax .....	488

---

# List of Figures

- 1.1. The New Case Dialog Box ..... 35
- 1.2. Sample CFX-Pre Interface ..... 36
- 1.3. Sample CFX-Pre Details View ..... 44
- 2.1. Mouse Mapping using Workbench Defaults ..... 58
- 2.2. Viewport Control ..... 62
- 18.1. Jet from a Wall into a Crossflow ..... 260
- 18.2. Local Hole Coordinates ..... 261
- 18.3. Local Hole Coordinates and a Jet Cross-section showing Cross-sectional Coordinates ..... 263
- 25.1. Mesh Adaption Process ..... 362
- 28.1. Line Cloud Illustration ..... 383
- 31.1. Sample Expressions Workspace ..... 414

---

---

## List of Tables

2.1. Mouse Operations and Shortcuts .....	58
13.1. Domain Motion Options .....	172
13.2. Advanced Turbulence Parameters .....	180
38.1. Reload Options .....	463
38.2. Scalar Parameters .....	467





---

# Chapter 1: CFX-Pre Basics

---

CFX-Pre is the physics-definition preprocessor for Ansys CFX. You import meshes (which can be produced in a variety of mesh generation software packages) into CFX-Pre and select physical models<sup>[1]</sup> to be used in the CFD simulation. Files produced by CFX-Pre are sent to CFX-Solver.

This chapter describes:

- 1.1. Starting CFX-Pre
- 1.2. CFX-Pre Modes of Operation
- 1.3. Working with the CFX-Pre Interface
- 1.4. CFX-Pre File Types

## 1.1. Starting CFX-Pre

---

When starting CFX-Pre for the first time, the default system font is obtained and, if it is deemed inappropriate for CFX-Pre, a dialog box appears that enables you to choose a new font. When a new font is selected, it is stored for future sessions. For details, see [Appearance \(p. 88\)](#).

CFX-Pre can be started in different ways:

- From within Ansys Workbench choose **Fluid Flow (CFX)** from **Toolbox > Analysis Systems** or **CFX** from **Toolbox > Component Systems**. In the **Project Schematic**, right-click the **Setup** cell and select **Edit**.
- From the Ansys CFX Launcher: set the working directory and then click **CFX-Pre 2021 R2**.
- From the command line. The basic command is:

```
<CFXROOT>/bin/cfx5pre
```

The command line options are described in [Starting CFX-Pre from the Command Line \(p. 33\)](#).

## Starting CFX-Pre from the Command Line

To start CFX-Pre from a command line, you must either:

- Specify the full path to the executable (<CFXROOT>/bin/cfx5pre)
- Append the path to the Ansys CFX executables (<CFXROOT>/bin/) to your PATH.

---

[1] For details on physical models, see [Physical Models in the CFX-Solver Modeling Guide](#).

- On Windows, right-click the **My Computer** icon and select **Properties**. Click the **Advanced** tab, then click **Environment Variables**. In the **System variables** section, edit `PATH` to include the path to the Ansys CFX executables; typically this will be something like:

```
C:\Program Files\ANSYS Inc\v212\CFX\bin;
```

- On Linux, edit your `.<window_manager>rc` file to include the path to the Ansys CFX executables.

Once the `PATH` has been updated, the basic command is:

```
cfx5pre
```

- Run the executable from the launcher **Tools > Command Line** (which has the path to the Ansys CFX executables built-in).

There are a number of optional command line flags, some of which are summarized in the following table:

Argument	Alternative Form	Usage
<code>-batch &lt;file name&gt;.pre</code>		Starts CFX-Pre in batch mode, running the session file you enter as an argument.
<code>-display &lt;display&gt;</code>	<code>-d</code>	Displays the graphical user interface on the X11 server <code>&lt;display&gt;</code> instead of using the X11 server defined by the <code>DISPLAY</code> environment variable.
<code>-gui</code>		Starts CFX-Pre in graphical user interface (GUI) mode. This is the default mode.
<code>-line</code>		Starts CFX-Pre in line interface mode.
<code>-graphics ogl</code> <code>-graphics mesa</code>	<code>-gr ogl</code> <code>-gr mesa</code>	For Linux only. Specify the graphics system: options are <code>ogl</code> (default) and <code>mesa</code> .  If you are running CFX-Pre/CFD-Post/TurboGrid on a Linux machine that does not have a compatible OpenGL driver installed, you can try starting the CFX application with software rendering enabled by adding the command line parameter: <code>-gr mesa</code> .
<code>-def &lt;file&gt;</code>		Loads the named CFX-Solver input file after starting.
<code>-session &lt;file&gt;</code>	<code>-s</code>	Plays the named session file after starting.
<code>-cfx &lt;file&gt;</code>		Loads the named case file after starting.
<code>-verbose</code>	<code>-v</code>	Specifying this option may result in additional output being sent to the standard output.

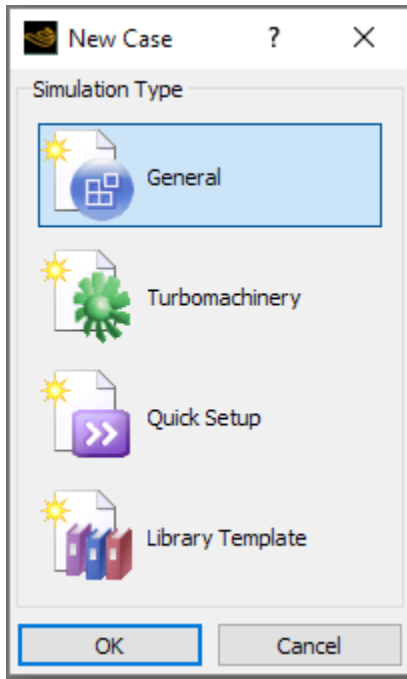
To view a full list of command-line flags, execute:

```
cfx5pre -help
```

## 1.2. CFX-Pre Modes of Operation

When you select **File > New Case** to create a new simulation, CFX-Pre presents four different modes of operations:

**Figure 1.1: The New Case Dialog Box**

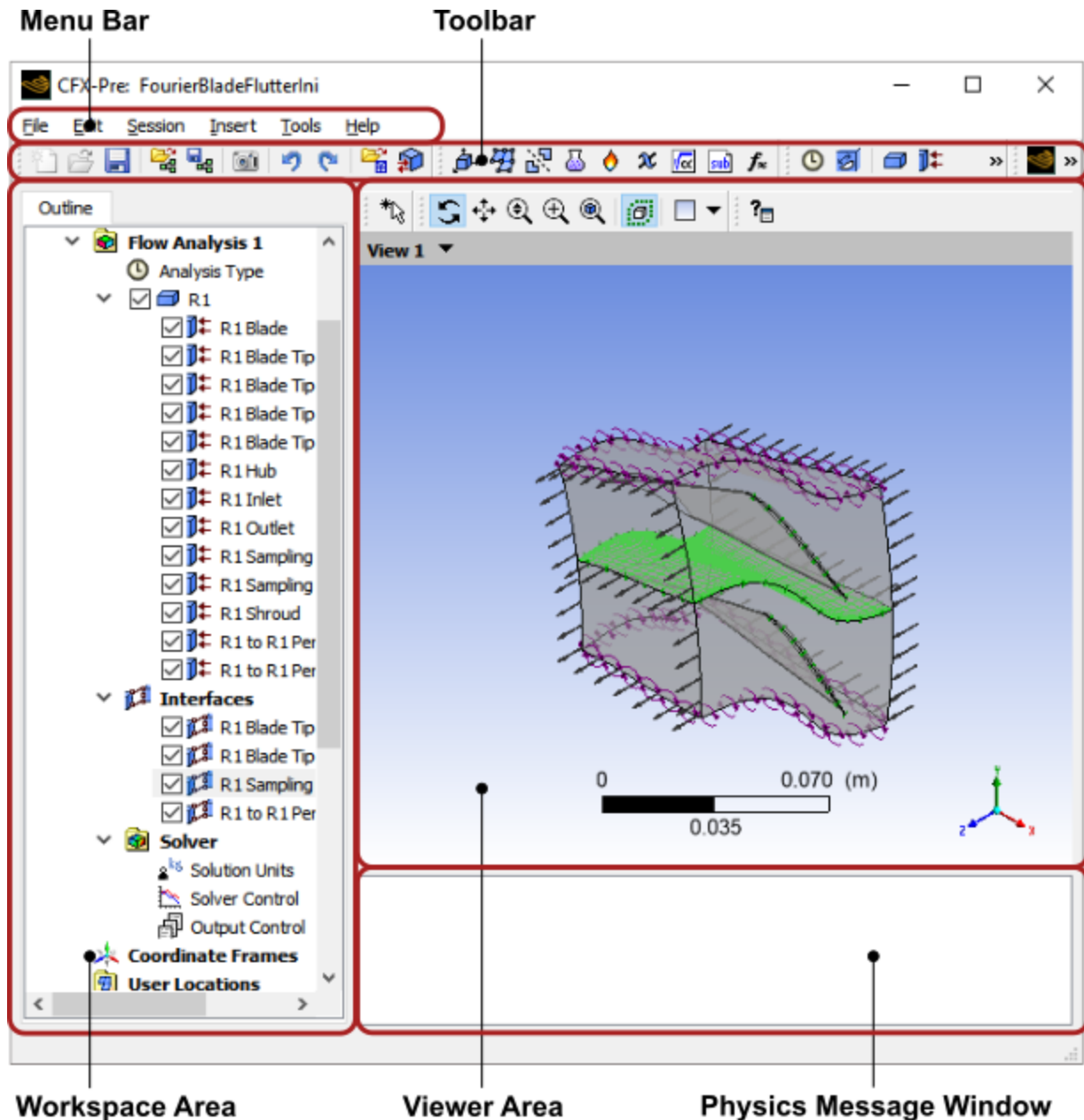


- *General* mode is the general-purpose mode for defining all types of CFD simulation. This mode uses the general CFX-Pre interface, which is described in [Working with the CFX-Pre Interface \(p. 35\)](#).
- *Turbomachinery* mode is a customized mode for defining turbomachinery simulations. For details, see [Turbomachinery Mode \(p. 473\)](#).
- *Quick Setup* mode greatly simplifies the physics setup for a simulation. Quick Setup mode is limited to a single-domain and single-phase problems; more complex physics, such as multiphase, combustion, radiation, advanced turbulence models, and so on are not available. You can, however, use Quick Setup mode to get started, and then add more physics details later. For details, see [Quick Setup Mode \(p. 469\)](#).
- *Library Template* mode provides a set of library files that are available with templates for specific physical problem definitions. In this mode you can easily define a complex physics problem by loading a template file, importing a mesh, and defining specific problem data. For details, see [Library Objects \(p. 483\)](#).

## 1.3. Working with the CFX-Pre Interface

The CFX-Pre interface enables the easy definition of a simulation. The main components are the viewer (to display and manipulate meshes), the workspaces (to define different aspects of the physics setup), the physics message window, and the menus and tool bars (access to extra tools and utilities).

Figure 1.2: Sample CFX-Pre Interface



### 1.3.1. Viewer

The viewer displays imported geometries and meshes and enables manipulations and transformations to be viewed. Information about boundary conditions, domains, point sources, and so on, is also displayed, and items can be picked directly from the Viewer.

CFX-Pre uses the same viewer as CFD-Post. Information on the generic CFX-Pre/CFD-Post viewer is available in [CFX-Pre 3D Viewer](#) (p. 51). Many aspects of the viewer appearance can be customized, as described in [Options](#) (p. 78).

## 1.3.2. CFX-Pre Workspace

The CFX-Pre workspace contains a tree view as well as various details views that are used during the specification of mesh import, mesh transformation, physics, regions, materials, and expressions.

A powerful feature of CFX-Pre is automatic physics checking. Objects that contain inconsistent or incorrect settings are highlighted in red. Detailed error messages are shown in the physics validation summary window. For details, see [Physics Message Window \(p. 45\)](#).

### 1.3.2.1. Outline Tree View

The **Outline** tree view displays a summary of the physics that have been defined for the simulation. The tree view window initially contains a default list of objects in a tree format.

The following topics are discussed in this section:

- [General Considerations \(p. 37\)](#)
- [Outline Tree View Structure \(p. 37\)](#)
- [Outline Tree View Shortcut Menu Commands \(p. 40\)](#)

---


#### Tip:

Typing **Ctrl+F** activates the search facility, which can be used to quickly locate an item in the tree. Note that the search is case-sensitive and that the text box disappears after a few seconds of inactivity.

---

#### 1.3.2.1.1. General Considerations

When working with the tree view, note the following:

- New objects are displayed in the tree as they are created.
- Clicking any object that is applied to a region will highlight that region in the viewer when highlighting is enabled (that is, when the *Highlighting* icon  is selected in the **3D Viewer** toolbar). For details, see [3D Viewer Toolbar \(p. 52\)](#).
- Objects shown in red contain incorrect physics definitions.
- Right-click an object (or group selection of objects) to display the shortcut menu.

For details, see [Outline Tree View Shortcut Menu Commands \(p. 40\)](#).

#### 1.3.2.1.2. Outline Tree View Structure

The **Outline** tab displays the tree view, which shows a summary of the current physics definition for a simulation. The tree structure displayed reflects the structure used in the CFX Command Language (CCL) for physics definition. You can select any object in the tree and double-click to gain direct access to the appropriate tab to edit its settings. You can also right-click an object

and display the CCL definition of an object in the **Command Editor** dialog box, where it can be edited.

The remainder of this section describes the main areas in the **Outline** tree view.

## Mesh

Provides access to all mesh operations in CFX-Pre. This includes mesh import, mesh transformations, and the render/visibility properties of meshes in the viewer. Meshes generated in many other mesh generation packages can be imported into CFX-Pre. For details, see [Importing and Transforming Meshes \(p. 121\)](#).

Meshes that have been glued together are listed under **Connectivity** in the tree view.

## Simulation

Enables you to define the one or more analyses of the simulation.

Optionally, you can open a copy of the Simulation branch in a separate tab.

## Analysis

Enables you to define and edit an analysis:

### Analysis Type

Enables the specification of an analysis as steady-state or transient, and whether it requires coupling to an external solver. For details, see [Analysis Type \(p. 161\)](#).

### Domains

Enables you to define and edit the type, properties and region of the fluid, porous or solid. For details, see [Domains \(p. 165\)](#), [Boundary Conditions \(p. 217\)](#), [Subdomains \(p. 247\)](#) and [Source Points \(p. 271\)](#).

### Domain Interfaces

Enables you to define and edit the method of connecting meshes or domains together. For details, see [Domain Interfaces \(p. 205\)](#).

### Global Initialization

Enables you to set global initial conditions (across all domains). Domain specific initialization is set through the domain forms. For details, see [Initialization \(p. 237\)](#).

## Solver

Enables the defining and editing of [Setting the Solution Units \(p. 288\)](#), [Solver Control \(p. 289\)](#), [Solver: Expert Parameter \(p. 98\)](#), [Output Control \(p. 309\)](#) and [Mesh Adaption \(p. 361\)](#).

## Coordinate Frames

Creates and edits coordinate frames. A Cartesian coordinate frame exists by default, but other Cartesian coordinate frames can be made. For details, see [Coordinate Frames in the CFX-Solver Modeling Guide](#) and [Coordinate Frames \(p. 371\)](#).

## User Locations

Creates user locations, including user line, user point, user point cloud, and user surface. For details, see [User Locations \(p. 377\)](#).

## Transformations

Creates and edits transformations that can be used for positioning data from a results file as part of initialization. For details, see [Interpolation Mapping in the CFX-Solver Modeling Guide](#).

## Materials / Reactions

Creates, edits, and displays materials and reactions. Many different material types can be defined, edited or imported. Specialist materials and reactions can be imported from external files, such as the RGP (Real Gas Properties) file and Flamelet reaction files. For details, see [Materials and Reactions \(p. 387\)](#).

## Expressions, Functions, and Variables

Used to create, edit, and plot: Additional Variables, expressions, user functions, user routines, GT-SUITE models. For details, refer to the following sections:

- [Additional Variables \(p. 405\)](#)
- [Expressions \(p. 413\)](#)
- [User Functions \(p. 419\)](#)
- [User Routines \(p. 439\)](#)
- [GT-SUITE Model Object \(p. 443\)](#)

## Simulation Control

Enables you to set up the control of analyses in the simulation. This control is facilitated by defining and editing one or more configurations as well as global solver execution control. For more information, see [Solve \(p. 87\)](#).

## Case Options

The **Graphics Style, Labels and Markers**, and **General** options enable you to override the defaults for the current simulation only — they will not be retained for future simulations. The default settings for CFX-Pre are set in the **Edit > Options** dialog box — these settings are retained and take effect when a new case is started. See [CFX-Pre Options \(p. 78\)](#) for a description of these settings.



## Extensions

Enables you to access to any customized extensions available to CFX-Pre. For details, see [CFX-Pre Extensions Menu \(p. 117\)](#).


### 1.3.2.1.2.1. Outline Tree View Shortcut Menu Commands

Right-clicking any object in the tree view displays a shortcut menu. Double-clicking an object performs the default action for that object. Shortcut menu command descriptions follow:

Command	Description
Configuration	<b>Simulation Control &gt; Configurations &gt; Insert &gt; Configuration</b> opens the <b>Configuration Editor</b> .
Copy	The <b>Copy</b> command is usually combined with <b>Paste</b> to quickly replicate objects.
Define Connection	<b>Mesh &gt; Define Connection</b> opens the <b>Mesh Connections Editor</b> .
Delete	Deletes the selected object. The physics for the simulation are checked after objects are deleted. Objects containing invalid parameters are highlighted in red in the tree view.
Delete All Mesh	Deletes the mesh, but not the named areas in the <b>Outline</b> tree view. When this happens, the physics message window will show errors that say the named objects cannot be found. If you then import a new mesh that uses the same names for objects, the names will be resolved and the errors will disappear.
Duplicate	Copies the definition of the selected object to a new object. You will be required to enter a name for the duplicated object, which will then be created at the same level (that is, for a boundary condition, the new boundary will be created in the same domain as the initial object).
Edit	Opens the relevant tab where new parameters for the object can be entered. In most cases <sup>[a]</sup> , you can also edit an object by double-clicking it in the tree view.
Edit In Command Editor	Opens the <b>Command Editor</b> dialog box and displays the CCL definition for the highlighted object. You can edit the CCL directly to change the object definition <sup>[a]</sup> . For details, see <a href="#">Command Editor Dialog Box (p. 487)</a> .
Expand/Collapse Sub-Branches	Provides a fast way to navigate the tree view.
Export CCL	Opens the <b>Export CCL</b> dialog box, which is similar to the dialog box described in <b>Export Region Data</b> , below.
Export CSV	Opens the <b>Export CSV</b> dialog box. For details, see <a href="#">Export CSV Command (p. 72)</a> .
Export Region Data	Opens the <b>Export Region CCL</b> dialog box, used to save the region data to a <code>.ccl</code> file.
Glue Adjacent Meshes	If there are multiple mesh assemblies that have matched meshes, you can use this option to try to glue them together. Select the assemblies in the tree view (while holding down the <b>Ctrl</b> key).

Command	Description
	Gluing can be useful to avoid setting up a GGI interface within a domain, but does require that the meshes match exactly on the surfaces that are to be glued. When you transform or copy multiple assemblies, each copy is not only glued to its original assembly or to other copies, but also to any other assemblies that are transformed or created. For more information, see <a href="#">Gluing Meshes Together (p. 147)</a> .
Hide	Makes the active object invisible in the viewer. <b>Hide</b> has the same effect as clearing the check box next to an object in the tree view.
Hide Interface Boundaries	Hides the interface boundaries in the <b>Outline</b> tree view.
Import CCL	Opens the Import CCL dialog box, which is similar to the dialog box described in <b>Import Region Data</b> , below.
Import Mesh	Opens the <b>Import Mesh</b> dialog box. This is used to import a new mesh using an appropriate file. For details, see <a href="#">Importing Meshes (p. 121)</a> .
Import Library Data	This command, available from the <b>Materials</b> branch in the tree view, is used to add a new material to the simulation. Examples of such a material include <b>Methanol CH4O</b> , <b>Rubber</b> , <b>Water at 25 C</b> , and many more.
Import Region Data	Opens the <b>Import Region CCL</b> dialog box. This is used to load region data from a <b>.ccl</b> file.
Insert	Various objects are available for insertion, depending on which object is highlighted. All of the options available from this menu can also be accessed from the <b>Insert</b> menu. For details, see <a href="#">CFX-Pre Insert Menu (p. 95)</a> .
Mesh Statistics	<p>Opens the <b>Mesh Statistics</b> dialog box and provides a detailed information about the active mesh. The Mesh Statistics dialog box can be invoked for one or more assemblies and/or primitive 3D/2D regions. The data displayed includes the number of nodes, elements, the number of each element type, and physical extents of the mesh. The <b>Maximum Edge Length Ratio</b> is also calculated.</p> <p>Note that for multiple selections, or for selections that consist of composite regions containing multiple primitive regions, the number of nodes and elements displayed is the sum of the number of nodes and elements in each of the underlying primitive regions. The sum takes no account of whether any nodes in the underlying primitive regions are common to two or more of the primitive regions in the selection. As a result, the number of nodes displayed may be higher than the number of nodes in the final mesh that is written to the CFX-Solver input file. In general, the number of nodes in the final mesh will depend on the physics definition, which affects whether nodes that are common to multiple primitive regions can be considered to be the same node.</p>
Paste	The <b>Paste</b> command is available when you have already used the <b>Copy</b> command on an object.

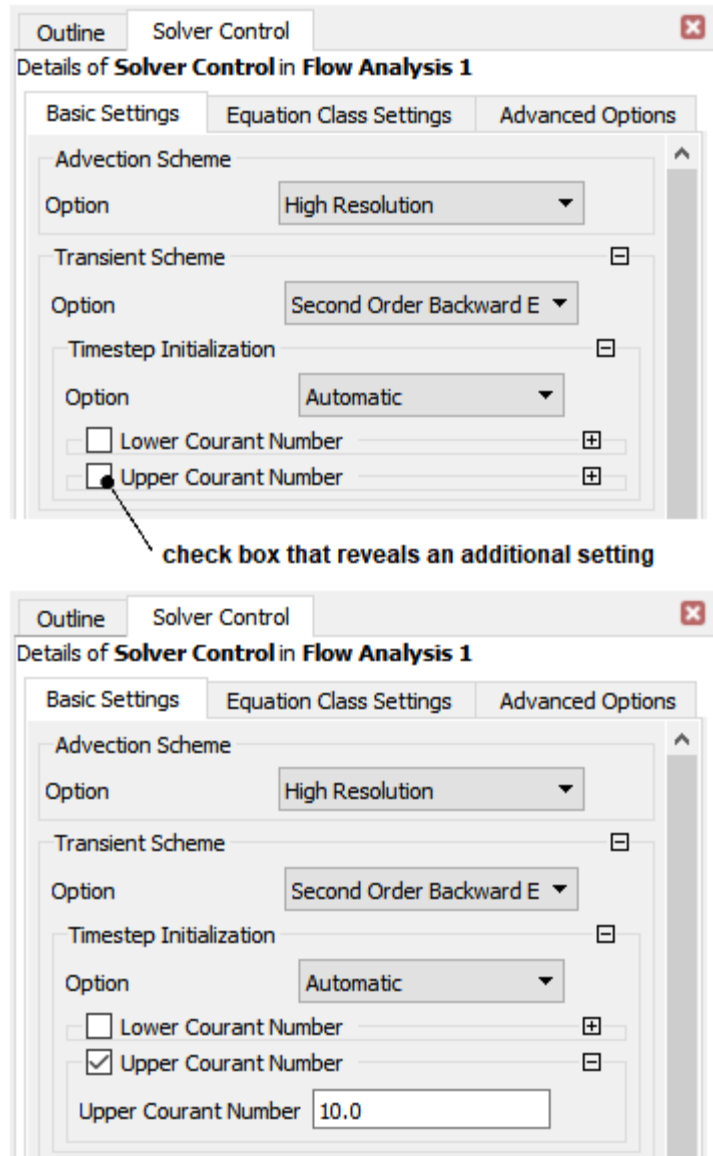
Command	Description
	<p>To avoid producing objects with the same name, you are prompted to provide a name when you paste the new object. For objects that contain a location parameter (such as domains and boundary conditions), you will usually need to edit the new object after pasting it to avoid multiple objects that reference the same location.</p> <p>If you are pasting a domain object, then you will need to edit each child object in the domain that references a location. For example, you will need to change the locations that boundary conditions reference so that they point to locations in the new domain. You can simply delete a default domain boundary in this situation; this will enable CFX-Pre to create a new default boundary for the domain that references the correct locations.</p>
Reload Mesh Files	<p>If any of the mesh regions become corrupted or are accidentally deleted, selecting <b>Reload Mesh Files</b> reloads all mesh files used in the simulation. This command cannot be used to insert a new mesh; do to so, select <b>Import Mesh</b>. For details, see <a href="#">Reload Mesh Files Command (p. 69)</a>.</p> <hr/> <p><b>Note:</b></p> <p>This command is not required (and is not available) in CFX-Pre launched from Ansys Workbench.</p> <hr/>
Rename	Changes the selected object's name.
Render	Enables you to change the appearance of almost any object. For example, a boundary condition or domain interface can be displayed with a solid color, the transparency of a domain can be altered, and so on.
Report Interface Summary	Invokes a message box that shows a summary of the interfaces and their types. For details, see <a href="#">Mesh Connection Options in the CFX-Solver Modeling Guide</a> .
Show/Hide	Makes the active object either visible ( <b>Show</b> ) or invisible ( <b>Hide</b> ) in the viewer. <b>Show</b> and <b>Hide</b> have the same respective effects as selecting and clearing the check box next to a specific object in the tree view.
Show Interface Boundaries	Shows the interface boundaries in the <b>Outline</b> tree view.
Start Solver	Enables you to access the <b>Define Run</b> , <b>Run Solver</b> , and <b>Run Solver and Monitor</b> commands. These commands are also available from the main toolbar.
Transform Mesh	Opens the <b>Mesh Transformation Editor</b> dialog box, enabling you to modify the location of the active mesh through rotation, translation, or reflection. The mesh can also be resized using a scaling method. For details, see <a href="#">Transform Mesh Command (p. 140)</a> .
Use as Workbench Input Parameter	Available when an expression is selected, this command allows the expression to be used as a workbench input parameter.

Command	Description
View By	This command, available for the <code>Mesh</code> object, opens a new tab that presents a detailed mesh information in one of two ways. Selecting <b>View By &gt; Source File</b> displays the mesh regions based on the mesh file provided, whereas <b>View By &gt; Region Type</b> organizes the areas of the mesh based on the defined 2D regions.
View in New Tab	<b>Simulation &gt; View in New Tab</b> enables you to view a copy of the contents of the <b>Simulation</b> branch in a separate tab.
View in CFD-Post	Prompts you to save a CFX-Solver Input file, then automatically starts CFD-Post with that file loaded.
Write Solver Input File	Has the same effect as clicking <i>Write Solver Input File</i>  or selecting <b>Tools &gt; Solve &gt; Write Solver Input File</b> from the menu bar. For details, see <a href="#">Write Solver Input File Command (p. 115)</a> .

<sup>[a]</sup> An expression that is set as an input parameter in Ansys Workbench cannot be edited in CFX-Pre or CFD-Post (as the results of such edits are not passed to Ansys Workbench) and will be grayed out. However, the expression can be declared to no longer be an input parameter or it can be deleted.

### 1.3.2.2. Details View

*Details view* is a generic term for the editor pane that opens when you edit an object of the **Outline** tree view. A details view appears on a tab beside the **Outline** tab. Each details view presents the settings and controls that define an object.

**Figure 1.3: Sample CFX-Pre Details View**

Some check boxes reveal additional settings when selected. In the example above, selecting the **Upper Courant Number** check box makes it possible to see and override the default value; the white background of the field indicates that you can edit the value shown.

Most CFX-Pre settings have default values that will enable you to define an object or set a control as easily as possible. If there is a setting that requires you to set a value, basic physics checking occurs when you click **OK** or **Apply** on a details view and most missing settings are detected then. Complete physics checking takes place when you attempt to write a solver file and all missing settings are detected and reported at that time.

You use the details view to define the properties of an object. The details view contains one or more tabs, depending on the type of object being defined.

Many properties can be set via a CEL expression. To enter an expression:



1. Click in the field for a property.

2. Click the Enter Expression icon that appears beside the field. This enables the field to accept an expression name.
3. Either enter an expression definition directly, or type the name of an existing expression. You must ensure that the expression evaluates to a value having appropriate units for the property that uses the expression.

For details on CEL expressions, see [Expressions \(p. 413\)](#).

For CFX components in Ansys Workbench, any CEL expression can be made into a parameterized CEL expression by defining it as a Workbench input parameter. You can do this by creating an expression and parameterize it by right-clicking it in the Expression editor. You can then use that expression as the value of a property.

You can stop a property from being specified as a Workbench input parameter. However, the corresponding CEL expression persists and can be managed by the Expression editor.

A property that accepts only an integer value (but not a CEL expression) is also capable of being controlled by a Workbench input parameter. For such a property, the *Set to Workbench input parameter*  and *Set to value*  icons are used to toggle the value between being a Workbench input parameter or not.

### 1.3.3. Physics Message Window

As you work through your simulation, CFX-Pre continually checks the physics definitions you have specified. Whenever an action is carried out, the physics validator runs a check on the CCL definitions of all the objects created up to that point. Physics checking is carried out by comparing the current CCL data against library files such as `RULES`, `VARIABLES` and `PHYSICS`, which are known to contain only valid physics specifications. If an inconsistency is found in the physics, the object with associated error(s) is highlighted in red text in the tree view.

In addition to object name highlighting, the physics validation window displays all error types in the simulation: global errors, physics errors and expression errors. The output in this window gives an explanation of each of the detected errors. Double-clicking on a red item or a maroon item (an expression error) in the physics validation window will take you to the correct place in order to edit the object.

Global errors apply to the entire simulation and show errors that are not specific physics errors. Often these errors show required objects that need to be defined to complete the simulation (for example, initial conditions or a domain). They also show invalid referencing of regions in a simulation. In some cases, the global errors offer a suggestion rather than being a definite error. For example, if you have created two valid boundary conditions on one region, a global error will be shown (despite the fact that the physics for both boundary conditions may be correct) because you cannot specify more than one boundary condition on any given surface.

Physics errors (highlighted in red) involve an incorrect application of physics.

- Global errors appear in blue text.
- Specific physics errors appear in red text. You can double-click these to edit the object containing the error.

There are two common situations when you are likely to encounter physics errors:

1. CFX-Pre defines some objects, such as the **Solver Control** settings, by default. If you create a new object that is not compatible with the default objects settings, the physics validation summary window will show errors in the default object. This occurs when creating a solid domain because the default **Solver Control** settings do not contain a solid time scale. These errors will disappear when you define the **Solver Control** settings.
2. When changing the physics of an existing model. There are many instances where you might want to change the description of your simulation. One particular situation is when you want to use the values in a results file as the initial field for another run with different physics.

When a domain is modified, perhaps with new model options, you will receive errors or warnings in the physics validation summary window if you need to revisit and update existing boundary conditions, initialization, solver controls, and so on. This happens, for example, when the turbulence model is changed from the laminar model to the  $k-\epsilon$  model and the boundary conditions for the laminar case do not contain turbulence data (for example, at an Inlet). You should fix any such errors before writing a CFX-Solver input file.

You should update boundary conditions if the number of Additional Variables has been increased, or if the units for Additional Variable specifications have been changed.

If the simulation is set up correctly, there will not be any physics errors when you are ready to write the CFX-Solver input file.

### 1.3.3.1. Physics Errors from Old .def/.res Files

When you load CFX-Solver input/results files from previous versions of Ansys CFX, you may receive error messages, despite the fact that the files can be run in the CFX-Solver. This is due to differences in the previous CFX-Solver input files. In CFX-Pre, a more strict approach to CCL structure and content has been implemented to ensure the integrity of the CCL made available to the CFX-Solver.

CFX-Pre performs some automatic updates when opening CFX-Solver input or results files from previous versions of Ansys CFX.

### 1.3.3.2. Physics Message Window Shortcut Menu Commands

Right-click a message in the physics message window to perform the following functions:

#### **Copy**

Enables copying of the selected message.

#### **Edit**

Enables you to use the Details View to edit the object generating the error.

#### **Auto Fix Physics**

Enables you to attempt to correct inconsistent physics automatically. In many cases, you will find that this fixes the problem without a need to change any settings on the form. Alternatively, you can use the Details View to edit the object generating the error.

Viewing the type of error before performing auto fix is strongly recommended. For example, auto fix cannot fix a domain with an incorrectly specified location. In effect, auto fix opens the default layout of the panel and performs an apply. If you are unsure about auto fix, you should subsequently open the form and verify that the settings are still valid for your problem. You should fix all physics validation errors to ensure that the CFX-Solver input file runs in the solver. If any errors are found when you attempt to write the CFX-Solver input file, a warning message is displayed giving you the option to write the file anyway or cancel the operation.

### **Auto Fix All**

Runs auto-fix on all objects that have physics validation errors.

### **Suppress this message**

Suppress the selected message; a message summary is displayed instead.

### **Suppress all messages**

Suppresses all messages; message summaries are displayed instead. Note that messages generated subsequently will not be suppressed.

### **Unsuppress all messages**

Unsuppress all messages.

## **1.3.4. Menu Bar**

The menu bar provides access to CFX-Pre functions. Some of these functions are also available from the [Toolbar \(p. 48\)](#).

### **File Menu**

Provides access to file operations including opening and saving simulations, as well as importing or exporting CCL. For details, see [CFX-Pre File Menu \(p. 65\)](#).

### **Edit Menu**

Enables you to change the default options used by Ansys CFX and undo/redo actions. For details, see [CFX-Pre Edit Menu \(p. 77\)](#).

---

#### **Note:**

Some options can be overridden for the current simulation; see *Case Options* in [Outline Tree View Structure \(p. 37\)](#) for details.

---

### **Session Menu**

Controls the recording and playing of session files. Session files are used to record a set of operations. You can then play back a session file to quickly reproduce the same operations. For details, see [CFX-Pre Session Menu \(p. 91\)](#).



### Insert Menu

Enables you to create new objects such as domains or boundary conditions, or edit existing objects of that type. For details, see [CFX-Pre Insert Menu \(p. 95\)](#).

### Tools Menu

Provides access to tools such as command editor, macro calculator as well as quick setup and turbo modes. For details, see [CFX-Pre Tools Menu \(p. 99\)](#).

### Extensions Menu

Provides access to any customized extensions available to CFX-Pre. For details, see [CFX-Pre Extensions Menu \(p. 117\)](#).

### Help Menu

For details, see [CFX-Pre Help Menu \(p. 119\)](#).

## 1.3.5. Toolbar

The toolbar provides quick access to commonly used functions. The toolbar contains the most common menu items and viewer controls. Holding the mouse pointer over a toolbar icon for a short period of time displays the icon's function.

## 1.4. CFX-Pre File Types

---

The following file types are used by or produced by CFX-Pre:

### Case Files (.cfx)

The case file contains the physics data, region definitions, and mesh information for the simulation and is used by CFX-Pre as the 'database' for the simulation setup. The case file is generated when you save a simulation in CFX-Pre. To re-open a simulation, select **File > Open Case** and pick a case file to open.

When you import a mesh into CFX-Pre, it passes through an import filter and is stored as part of the case file. Therefore, once a mesh has been imported, the original mesh file is not required by CFX-Pre. Additional information on importing meshes is available in [Importing Meshes \(p. 121\)](#).

The case file is a binary file and cannot be edited directly.

You can open cases on any supported platform, regardless of the platform on which they were created.

### Mesh Files

There are many types of mesh files that can be imported into CFX-Pre. For details, see [Supported Mesh File Types \(p. 123\)](#).

## CFX-Solver Input Files (.def, .mdef)

A CFX-Solver input file is created by CFX-Pre. The input file for a single configuration simulation (.def) contains all physics and mesh data; the input file for a multi-configuration or operating point simulation (.mdef) contains global physics data only (that is, Library and Simulation Control CFX Command Language specifications). An .mdef input file for a multi-configuration simulation is supplemented by Configuration Definition (.cfg) files that:

- Are located in a subdirectory that is named according to the base name of the input file
- Contain local physics and mesh data.

---

### Note:

Use the `-norun` command line option (described in [Command-Line Options and Keywords for cfx5solve in the CFX-Solver Manager User's Guide](#)) to merge global information into the configuration definition files, and produce a CFX-Solver input file (.def) that can be run by the CFX-Solver.

---

You can load a CFX-Solver input file back into CFX-Pre to recreate a simulation. CFX-Solver input files from previous releases of Ansys CFX can be loaded into CFX-Pre, although the physics definition may have to be updated for such files. For details, see [Physics Errors from Old .def/.res Files \(p. 46\)](#).

## CFX-Solver Results Files (.res, .mres, .trn, .bak)

Intermediate and final results files are created by the CFX-Solver:

- Intermediate results files, which include transient and backup files (.trn and .bak, respectively) are created while running an analysis.
- Final results files for single-configuration simulations (.res), multi-configuration simulations (.mres), and operating point simulations (.mres) are written at the end of the simulation's execution. For a multi-configuration simulation, a configuration result file (.res) is also created at the end of each configuration's execution. For an operating point simulation, a result file (.res) is also created at the end of each operating point job.

Each results file contains the following information as of the iteration or time step at which it is written:

- The physics data (that is, the CFX Command Language specifications)
- All or a subset of the mesh and solution data.

## CFX-Solver Backup Results Files (.bak)

A backup file (.bak) is created at your request, either by configuring the settings on the **Backup** tab in **Output Control** in CFX-Pre, or by choosing to write a backup file while the run is in progress in the CFX-Solver Manager.

## CFX-Solver Transient Results Files (.trn)

A transient results file (.trn) is created at your request, by configuring the settings on the **Output Control > Trn Results** tab in CFX-Pre.

### CFX-Solver Error Results Files (.err)

An error results file (.err) is created when the CFX-Solver detects a failure and stops executing an analysis. The .err file can be loaded into CFD-Post and treated the same way as a .bak file, but if the CFX-Solver encounters another failure while writing the .err file, it may become corrupted and accurate solutions cannot be guaranteed.

### Session Files (.pre)

Session files are used by CFX-Pre to record CFX Command Language (CCL) commands executed during a session. The commands can be played back at a later date to reproduce the session. These files are in ASCII format and can be edited or written in a text editor. For details, see [New Session Command \(p. 91\)](#).

### CCL Files (.ccl)

CFX CCL files are used by CFX-Pre to save CFX Command Language (CCL) statements. CCL files differ from session files in that only a snapshot of the current state is saved to a file. These files are in ASCII format and can be edited or written in a text editor. The CCL statements stored in these files replace or append the existing CCL data, depending on the option chosen. For details, see:

- [Import CCL Command \(p. 70\)](#)
- [Append or Replace \(p. 70\)](#).

An overview of the files used throughout Ansys CFX is available in [Ansys CFX File Types in the CFX Introduction](#).

---

## Chapter 2: CFX-Pre 3D Viewer

---

In CFX-Pre, the **3D Viewer** is visible whenever a partial or complete case is loaded. After importing a mesh into CFX-Pre, you can see a visual representation of the geometry in the **3D Viewer**. You can create various other objects that can be viewed in the **3D Viewer**; for details, see [CFX-Pre Insert Menu \(p. 95\)](#). The visibility of each object can be turned on and off using the check boxes in the tree view; for details, see [Object Visibility \(p. 51\)](#).

Descriptions of the various viewing modes and **3D Viewer** commands, including toolbars, shortcut menus, and hotkeys, are given in [3D Viewer Modes and Commands \(p. 51\)](#).

You can switch between four adjustable "views" that each remember the camera angle and state of visibility of all objects.

The **3D Viewer** can display multiple viewports at a time. The viewport arrangement is controlled from the viewer toolbar.

This chapter describes:

- 2.1. [Object Visibility](#)
- 2.2. [3D Viewer Modes and Commands](#)
- 2.3. [Views and Figures](#)
- 2.4. [Stereo Viewer](#)

### 2.1. Object Visibility

---

The visibility of each object can be turned on and off using the check boxes in the tree view, as described in [Object Visibility](#). You can also hide objects by right-clicking them and selecting **Hide**.

Once an object has been hidden, you can show it again by selecting that object's check box in the **Outline** tree view.

### 2.2. 3D Viewer Modes and Commands

---

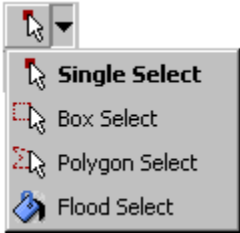





The topics in this section include:











- [3D Viewer Toolbar \(p. 52\)](#)
- [Shortcut Menus \(p. 54\)](#)
- [Viewer Hotkeys \(p. 56\)](#)
- [Mouse Button Mapping \(p. 57\)](#)
- [Picking Mode \(p. 59\)](#)




- [Labels and Markers \(p. 60\)](#)

## 2.2.1. 3D Viewer Toolbar

The **3D Viewer** toolbar has the following tools:

Tool	Description
	<p>Makes one of the picking tools active; see <a href="#">Picking Mode (p. 59)</a> for details.</p>
	<p>Selects objects. When a number of objects overlap, the one closest to the camera is picked. In CFX-Pre, when selecting regions for boundaries, if more than one region is under the mouse pointer at the location that you click, those regions will be listed in a list box, enabling you to select the intended region.</p>
	<p>Selects objects using a box. Drag a box around the objects you want to select.</p>
	<p>Selects objects using an enclosed polygon. Click to drop points around the objects. Double-click to complete the selection.</p> <hr/> <p><b>Note:</b></p> <p>Polygon Select mode will not enable you to create an invalid region, such as would occur if you attempted to move a point such that the resulting line would cross an existing line in the polygon.</p> <hr/>
	<p>The paint function selects the all the mesh elements on the face. The "counter" widget enables you to change the crease angle in degrees used to decide where the flood-pick algorithm will stop.</p>
	<p>This feature controls which objects you can select.</p> <p><b>Visible Entities</b> treats the contents of the Viewer as opaque and enables you to select only the top objects at any point.</p> <p><b>All Entities</b> treats the contents of the Viewer as "transparent" and enables you to select any of the objects that you would encounter if you drilled through the object at a given point. You use this option with the depth indicator in the bottom-left of the Viewer.</p>

Tool	Description
 <p><b>Choose Fully Enclosed</b> Choose Enclosed and Touching</p>	<p>This feature controls which objects you can select with a box or enclosed polygon.</p> <p><b>Choose Fully Enclosed</b> selects only the objects that have boundaries that are completely within the box or polygon you draw.</p> <p><b>Choose Enclosed and Touching</b> selects both the objects that are completely within the box or polygon you draw as well as any objects of which any part is within that area.</p>
 <p><b>All Regions</b> Primitive Regions Only Composite Regions Only</p>	<p>These features control whether primitives are chosen over composites.</p>
	<p>Rotates the view as you drag with the mouse. Alternatively, hold down the middle mouse button to rotate the view.</p>
	<p>Pans the view as you drag with the mouse. Alternatively, you can pan the view by holding down <b>Ctrl</b> and the middle mouse button.</p>
	<p>Adjusts the zoom level as you drag with the mouse vertically. Alternatively, you can zoom the view by holding down <b>Shift</b> and the middle mouse button.</p>
	<p>Zooms to the area enclosed in a box that you create by dragging with the mouse. Alternatively, you can drag and zoom the view by holding down the right mouse button.</p>
	<p>Centers all visible objects in the viewer.</p>
	<p>Toggles highlighting according to the highlighting preferences (select <b>Edit &gt; Options &gt; CFX-Pre &gt; Graphics Style &gt; Object Highlighting &gt; Type</b>). Highlighting is active only when the viewer is set to Picking Mode. For details, see <a href="#">Picking Mode (p. 59)</a>.</p>
	<p>Enables you to select mesh nodes. When picking a point from the viewer to populate a widget that defines a coordinate, the point can either be a point in space or a mesh "node". This tool enables you to select the mesh node nearest to the location you click.</p>
	<p>Displays the <b>Labels and Markers</b> dialog box that is used to select/clear the display of named regions and markers in the viewer. For details, see <a href="#">Labels and Markers (p. 60)</a>.</p>

Tool	Description
	<p>Selects the viewport arrangement. You can perform Independent zoom, rotation and translate options in each viewport.</p>
	<p>Toggles between locking and unlocking the views of all viewports. When the views are locked, the camera orientation and zoom level of the non-selected viewports are continuously synchronized with the selected viewport. Locking the view for the viewports in this way can be a useful technique for comparing different sets of visible objects between the viewports. This tool is available only when all viewports are using the Cartesian (X-Y-Z) transformation.</p>
	<p>Displays the <b>Viewer Key Mapping</b> dialog box. See <a href="#">Viewer Hotkeys (p. 56)</a> for details.</p>

## 2.2.2. Shortcut Menus

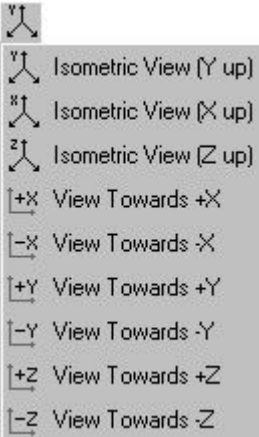

You can access the shortcut menu by right-clicking anywhere on the viewer. The shortcut menu is different depending on where you right-click.

### 2.2.2.1. CFX-Pre 3D Viewer Shortcut Menu

#### 2.2.2.1.1. Shortcuts for CFX-Pre (Viewer Background)

The following commands are available in CFX-Pre when you right-click the viewer background:

Command	Description
Import Library Data	<p>Opens the <b>Select Library Data to Import</b> dialog box so that you can add a new material to the simulation. Examples of such materials include Methanol CH<sub>4</sub>O, Rubber, Water at 25 C, and many more.</p> <p>This option is the same as right-clicking on <b>Materials</b> in the tree view and selecting <b>Import Library Data</b>.</p>
Start Solver	<ul style="list-style-type: none"> <li>• <b>Define Run</b> writes the CFX-Solver input file and starts the CFX-Solver Manager.</li> <li>• <b>Run Solver</b> writes the CFX-Solver input file and starts the CFX-Solver.</li> <li>• <b>Run Solver and Monitor</b> writes the CFX-Solver input file and starts both the CFX-Solver and the CFX-Solver Manager.</li> </ul>
View in CFD-Post	Writes the CFX-Solver input file and start CFD-Post

Command	Description
Write Solver Input File	The same as selecting <b>Tools &gt; Solve &gt; Write Solver Input File</b> . For details, see <a href="#">Write Solver Input File Command (p. 115)</a> .
Create New View ...	Creates a new view. The new view will become the current view. For more information about views, see <a href="#">Views and Figures (p. 61)</a> .
Delete View	Delete the current view.
Predefined Camera	Displays different views by changing the camera angle to a preset direction. 
Fit View	Centers all visible objects in the viewer. This is equivalent to clicking the  icon.
Projection	Switches between perspective and orthographic camera angles.
Default Legend	Shows or hides the default legend object.
Axis	Shows or hides the axis orientation indicator (triad) in the bottom-right corner of the viewer.
Ruler	Shows or hides the ruler on the bottom of the viewer.
Labels	Controls the display of labels. For more information, see <a href="#">Labels and Markers (p. 60)</a> .
Markers	Controls the display and properties of boundary markers. For more information, see <a href="#">Labels and Markers (p. 60)</a> .
Save Picture	Same as selecting <b>File &gt; Save Picture</b> . For details, see <a href="#">Save Picture Command (p. 72)</a> .
Viewer Options	Opens the <b>Options</b> dialog box with the viewer options displayed. For details, see <a href="#">Graphics Style (p. 83)</a> .

### 2.2.2.1.2. Shortcuts for CFX-Pre (Viewer Object)

The following commands are available in CFX-Pre when you right-click an object in the viewer:

Command	Description
Edit, Edit Definition, Edit Mesh	Opens the details view for the selected object so that you can edit its properties.
Mesh Statistics	Opens the <b>Mesh Statistics</b> dialog box and provides a detailed information about the active mesh. The Mesh Statistics dialog box can



Command	Description
	<p>be invoked for one or more assemblies and/or primitive 3D/2D regions. The data displayed includes the number of nodes, elements, the number of each element type, and physical extents of the mesh. The <code>Maximum Edge Length Ratio</code> is also calculated.</p> <p>Note that for multiple selections, or for selections that consist of composite regions containing multiple primitive regions, the number of nodes and elements displayed is the sum of the number of nodes and elements in each of the underlying primitive regions. The sum takes no account of whether any nodes in the underlying primitive regions are common to two or more of the primitive regions in the selection. As a result, the number of nodes displayed may be higher than the number of nodes in the final mesh that is written to the CFX-Solver input file. In general, the number of nodes in the final mesh will depend on the physics definition, which affects whether nodes that are common to multiple primitive regions can be considered to be the same node.</p>
Insert	<p>Enables you to insert a boundary, interface, subdomain, injection region, or source point. For details, see <a href="#">Boundary Conditions (p. 217)</a>, <a href="#">Domain Interfaces (p. 205)</a>, <a href="#">Subdomains (p. 247)</a>, <a href="#">Injection Regions (p. 253)</a>, or <a href="#">Source Points (p. 271)</a>.</p>
Edit in Command Editor	<p>Opens the <b>Command Editor</b> dialog box, displaying the CEL for the selected object. For details, see <a href="#">Using the Command Editor (p. 487)</a>.</p>
Render	<p>Displays the following render options:</p> <p><b>Color</b> enables you to choose a color for the selected object.</p> <p><b>Lines</b> enables you to select to <b>Show Wireframe</b>, <b>Show Mesh</b> or <b>No Lines</b>.</p> <p><b>Transparency</b> enables you to set the transparency levels of the domain. The choices are <b>Opaque</b>, <b>0.25</b>, <b>0.5</b>, <b>0.75</b>, or <b>Fully Transparent</b>.</p> <p><b>Properties</b> invokes the <b>Render Options</b> dialog box. For details, see <a href="#">Render Options (p. 148)</a>.</p>
Show	<p>Shows the object in the viewer.</p>
Hide	<p>Hides the selected object in the <b>3D Viewer</b>.</p>
Delete	<p>Deletes the selected object.</p>
Rename	<p>Changes the selected object's name.</p>
Alternatives	<p>When you right-click a location in the viewer, CFX-Pre presents a shortcut menu for one object at that location. Shortcut menus for the other objects at the same location are accessible as submenus under the <b>Alternatives</b> heading.</p>

### 2.2.3. Viewer Hotkeys

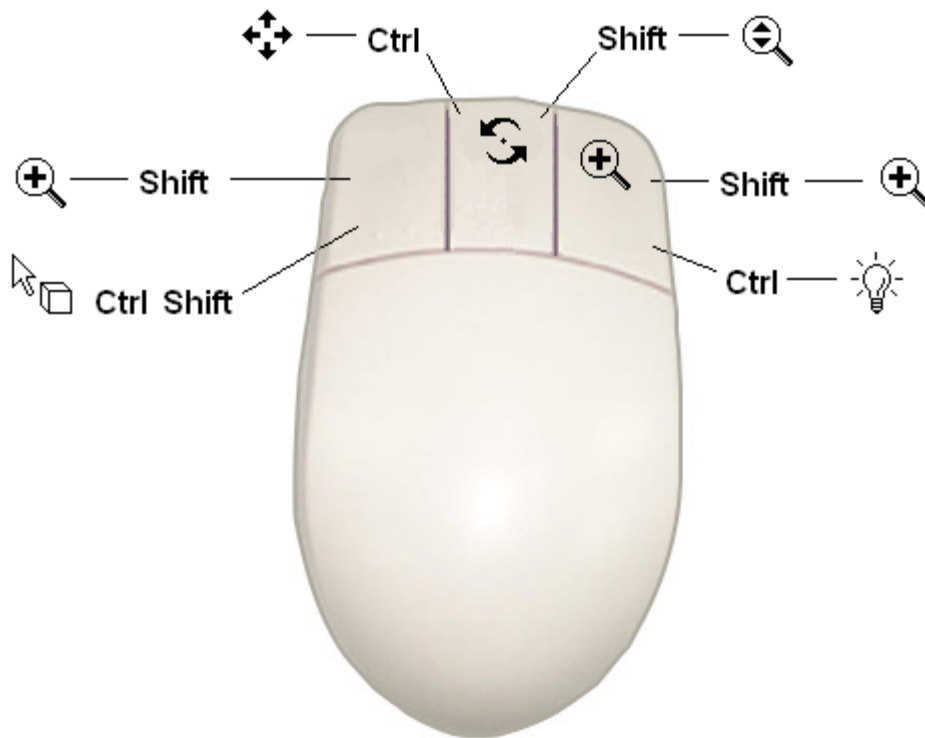
A number of shortcut keys are available to carry out common viewer tasks. These can be carried out by clicking in the viewer window and pressing the associated key.

Key	Action
<b>space</b>	Toggles between picking and viewing mode.
arrow keys	Rotates about horizontal and vertical axes.
<b>Ctrl</b> + up/down arrow keys	Rotates about an axis normal to the screen.
<b>Shift</b> + arrow keys	Moves the light source.
<b>1</b>	Switches to one viewport.
<b>2</b>	Switches to two viewports.
<b>3</b>	Switches to three viewports.
<b>4</b>	Switches to four viewports.
<b>C</b>	Centers the graphic object in the viewer window.
<b>N</b>	Toggles the projection between orthographic and perspective.
<b>R</b>	Resets the view to the initial orientation.
<b>S</b>	Toggles the level of detail between auto, off, and on.
<b>U</b>	Undoes transformation.
<b>Shift+U</b>	Redoes transformation.
<b>X</b>	Sets view from +X axis.
<b>Shift+X</b>	Sets view from -X axis.
<b>Y</b>	Sets view from +Y axis.
<b>Shift+Y</b>	Sets view from -Y axis.
<b>Z</b>	Sets view from +Z axis.
<b>Shift+Z</b>	Sets view from -Z axis.

The information in this table is accessible by clicking the *Show Help Dialog*  toolbar icon in the **3D Viewer** toolbar.

## 2.2.4. Mouse Button Mapping




The mouse mapping options enable you to assign viewer actions to mouse clicks and keyboard/mouse combined clicks. To adjust or view the mouse mapping options, select **Edit > Options**, then **Common > Viewer Setup > Mouse Mapping**.

**Figure 2.1: Mouse Mapping using Workbench Defaults****Table 2.1: Mouse Operations and Shortcuts**

Operation	Description	Workbench Mode Shortcuts	CFX Mode Shortcuts
Zoom Object Zoom Camera Zoom	To zoom out, drag the pointer up; to zoom in, drag the pointer down.	<b>Shift</b> + middle mouse button	Middle mouse button  <b>Shift</b> + middle mouse button zooms in a step.  <b>Shift</b> + right mouse button zooms out a step.
Translate	Drag the object across the viewer.	<b>Ctrl</b> + middle mouse button	Right mouse button
Zoom Box	Draw a rectangle around the area of interest, starting from one corner and ending at the opposite corner. The selected area fills the viewer when the mouse button is released.	Right mouse button  <b>Shift</b> + left mouse button  <b>Shift</b> + right mouse button	<b>Shift</b> + left mouse button

Operation	Description	Workbench Mode Shortcuts	CFX Mode Shortcuts
Rotate	Rotate the view about the pivot point (if no pivot point is visible, the rotation point will be the center of the object).	Middle mouse button	
Set Pivot Point	Set the point about which the Rotate actions pivot. The point selected must be on an object in the <b>3D Viewer</b> . When you set the pivot point, it appears as a small red sphere that moves (along with the point on the image where you clicked) to the center of the <b>3D Viewer</b> . To hide the red dot that represents the pivot point, click a blank area in the <b>3D Viewer</b> .	Left mouse button when in rotate, pan, zoom, or zoom box mode (as set by the icons in the viewer's toolbar).	<b>Ctrl</b> + middle mouse button
Move Light	Move the lighting angle for the <b>3D Viewer</b> . Drag the mouse left or right to move the horizontal lighting source and up or down to move the vertical lighting source. The light angle hold two angular values between 0 - 180.	<b>Ctrl</b> + right mouse button	<b>Ctrl</b> + right mouse button
Picking Mode	Select an object in the viewer.	<b>Ctrl</b> + <b>Shift</b> + left mouse button	<b>Ctrl</b> + <b>Shift</b> + left mouse button

## 2.2.5. Picking Mode

Picking mode is used to select and drag objects in the viewer. The mesh faces must be visible on an object or region to enable it to be picked. Enter picking mode by selecting the *Single Select*  tool in a drop-down menu of the viewer toolbar. If the *Single Select*  icon is already visible, you can simply click the *New Selection*  icon.

You can also pick objects while still in viewing mode by holding down **Ctrl+Shift** as you click in the viewer.

There are several types of entities that you can select in the viewer, and which type you use depends on the context of what you are doing:




- When you insert a primitive region, you need to use picking to select the mesh elements to put into the new primitive region.
- When you add any sort of location (such as domain, sub-domain, or boundary condition), you can also use picking to select the 2D or 3D regions for these locations. To select a 3D region, you have to select all the 2D regions that are part of it. (CFX-Pre does not plot 3D regions directly; it plots only the associated 2D regions.)

- When you need to select Cartesian coordinates for locations such as monitor points or source points, you can use picking to select the coordinates. Note that the coordinates selected always correspond to the location of the point in the global coordinate frame, Coord 0.

### 2.2.5.1. Selecting Objects

Use the mouse to select objects (for example, points and boundaries) from the viewer. When a number of objects overlap, the one closest to the camera is picked.

You can change the picking mode by selecting one of the toolbar icons:

-  **Single Select**
-  **Box Select**
-  **Polygon Select**

For details on the operation of the toolbar icons, see [3D Viewer Toolbar \(p. 52\)](#).

### 2.2.6. Labels and Markers

Click `Case Options > Labels and Markers` to invoke the **Labels and Markers Options** details view, used to select/clear the display of named regions and markers in the viewer as well as change the appearance of the markers.

---

**Note:**

You can set the defaults for these settings. For details, see [Labels and Markers \(p. 85\)](#).

---

Also see [Boundary Condition Visualization \(p. 235\)](#) for more details.

#### 2.2.6.1. Labels

Select options under **Show Labels** to show various types of labels. To disable all labels, clear the **Show Labels** option. The first three options refer to primitive and composite regions. For details, see [Assemblies, Primitive Regions, and Composite Regions \(p. 152\)](#).

#### 2.2.6.2. Boundary Markers

The **Show Boundary Markers** option turns on boundary condition symbols such as arrows indicating flow direction at an inlet.

The **Marker Quantity** slider controls the number of markers displayed. Moving the slider to the right increases the number.

The **Marker Length** slider controls the size of the markers displayed. Moving the slider to the right increases the size.

### 2.2.6.3. Boundary Vectors

The **Vector Quantity** slider controls the number of vectors displayed. Moving the slider to the right increases the number.

The **Vector Length** slider controls the size of the vectors displayed. Moving the slider to the right increases the size.

See [Boundary Plot Options Tab \(p. 234\)](#) for a discussion of displaying boundary vectors.

### 2.2.6.4. Profile Vectors

The **Vector Length** slider controls the size of the vectors displayed. Moving the slider to the right increases the size.

See [Visualization Options for Profile Data \(p. 427\)](#) for a discussion of displaying profile vectors

### 2.2.6.5. Injection Region Vectors

The **Vector Length** slider controls the size of the vectors displayed. Moving the slider to the right increases the size.

See [Injection Region Vector \(p. 267\)](#) for a discussion of displaying injection region vectors.

### 2.2.6.6. Particle Injection Regions

The **Rendering** option controls how many symbols are drawn for each particle injection region.

The options are:

- Full

This option causes multiple symbols to be drawn per particle injection region in order to represent the Injection Method settings (see [Injection Method \(p. 200\)](#)).

- Reduced

This option causes exactly one symbol to be drawn per particle injection region.

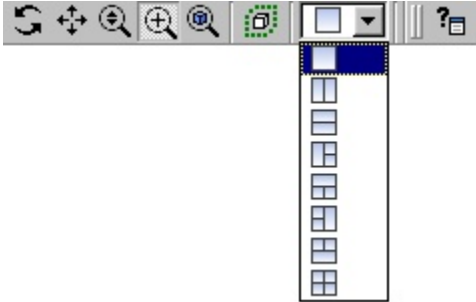
- None

This option turns off visual representation of particle injection regions.

## 2.3. Views and Figures

---

The **3D Viewer** opens with a single *viewport*; you can increase the number of viewports to four by using the viewport icon:

**Figure 2.2: Viewport Control**

The contents of a viewport are a *view*, which is a CCL object that contains the camera angle, zoom level, lighting, and visibility setting of each object in the tree view.

Each viewport contains a different, independent view. By default, four views exist: **View 1**, **View 2**, **View 3**, **View 4**.

When you select an object in the tree view, its information is applied to the active viewport. When you manipulate an object in the viewport, the view's CCL is updated immediately. However if the focus is on that viewport, you can press **u** to revert your change.

### 2.3.1. Switching to a View or Figure

To switch to a view or figure, do one of the following:

- Use the drop-down menu in the upper-left corner of the viewport.
- For figures only: Double-click the figure in the tree view (under the `Report` object).
- For figures only: Right-click the figure in the tree view (under the `Report` object), then select **Edit** from the shortcut menu.

### 2.3.2. Changing the Definition of a View or Figure

To change a view or figure:

1. Switch to the view or figure that you want to change.

For details, see [Switching to a View or Figure \(p. 62\)](#).

2. Change the view or figure (for example, rotate the view).

View and figure objects are saved automatically when you switch to a different view or figure.

## 2.4. Stereo Viewer

If you:

1. Have a standard stereo display
2. Have a graphics card that supports quad buffering OpenGL output

3. Have set your graphics card to "Stereo"
4. Have set your view to Perspective mode (right-click in the Viewer and select **Projection > Perspective**)

...you can view output in stereo. To enable this functionality:

1. Select **Edit > Options**.
2. In the **Options** dialog box, select **CFX-Pre > Viewer**.
3. On the **Viewer** panel:
  - a. Set **Stereo > Mode** to Stereo.
  - b. Set the **Stereo Effect**. The value of the "stereo effect" that is required is related to the distance between the observer and the display. If the stereo effect is too strong, either move away from the display, or move the slider towards **Weaker**.
4. Click **OK** to save the settings.





---

## Chapter 3: CFX-Pre File Menu

---

There are a number of basic functions available in CFX-Pre, such as opening and saving an existing case. These are described in detail in this chapter:

- 3.1. New Case Command
- 3.2. Open Case Command
- 3.3. Close Command
- 3.4. Save Case Command
- 3.5. Save Project Command
- 3.6. Refresh Command (Ansys Workbench only)
- 3.7. Save Case As Command
- 3.8. Reload Mesh Files Command
- 3.9. Import Mesh Command
- 3.10. Import CCL Command
- 3.11. Export CCL Command
- 3.12. Export CSV Command
- 3.13. Save Picture Command
- 3.14. Recent Case Files Submenu
- 3.15. Recent CCL Files Submenu
- 3.16. Recent Session Files Submenu
- 3.17. Quit Command

### 3.1. New Case Command

---

**Note:**

If a case is open, **New Case** is not available. To create new cases, ensure all open cases are saved (if required) and closed.

---

1. Select **File > New Case**.

The **New Case** dialog box appears.

2. Select a case type.
  - **General** makes use of all features in CFX-Pre. This is the most common mode of operation.
  - **Turbomachinery** is used specifically for turbomachinery applications and enables quick setup in such cases. For details, see [Turbomachinery Mode \(p. 473\)](#).

- **Quick Setup** provides fewer model options and is suitable for simple physics setup. It is useful as a tool to learn the basic paradigms of CFX-Pre before using General mode. For details, see [Quick Setup Mode \(p. 469\)](#).
- **Library Template** enables a CCL physics definition to be imported for use on a mesh. For details, see [Library Objects \(p. 483\)](#).

## 3.2. Open Case Command

---

The **Open Case** command can be used to open existing CFX-Pre case files (.cfx), as well as implicitly start a new case by opening a \*.def, \*.mdef, \*.res, .ccl, "full" transient results file (\*.trn), or backup file (\*.bak). Other supported file types include: Mesh or Simulation Database file (\*.cmdb or \*.dsdb), and GTM Database file (.gtm).

---

### Note:

If a case is already open, **Open Case** is not available. To open cases, ensure that all open cases are saved (if required) and closed.

---

1. Select **File > Open Case**.

The **Load Case File** dialog box appears.

2. Select a location to open the file from.
  3. Under **Files of type**, select the type of file to open.
    - Case files can be selected. CFX case files (\*.cfx) contain all of the physics, region, and mesh information for your case. For details, see [Opening Case \(.cfx\) Files \(p. 67\)](#).
    - CFX-Solver input or result files can be selected. For details, see [Opening CFX-Solver Input \(.def, .mdef\), Results \(.res\), Transient \(.trn\) or Backup \(.bak\) Files \(p. 67\)](#).
    - CCL files can be selected. For details, see [Opening CCL \(.ccl\) Files \(p. 68\)](#).
    - Mesh or Simulation Database files can be selected. For details, see [Opening Meshing \(.cmdb or .dsdb\) Files \(p. 68\)](#).
    - GTM Database files can be selected. For details, see [Opening CFX-Mesh \(.gtm\) Files \(p. 68\)](#).
  4. Select the file to open and click **Open**.
- 

### Note:

When CFX-Solver input or results files from a previous release of CFX are opened in CFX-Pre, physics errors are highlighted in red in the message area. If these errors are ignored, a case can still run in the CFX-Solver in many cases, but it is recommended that the errors be fixed. This ensures CCL is updated to the current version. These errors are usually fixed easily by right-clicking on the object and selecting **Auto Fix Physics**. Also, double-clicking on the error

in the message area opens the details view in which the error was made. For details, see [Physics Errors from Old .def/.res Files \(p. 46\)](#). Also, the **Command Editor** can be used to correct CCL. For details, see [Command Editor Dialog Box \(p. 487\)](#).

---

### 3.2.1. Recover Original Session

When opening a CFX-Solver input file, the option **Recover Original Session** can be used to find the location of the original CFX Case file (that was used to generate the CFX-Solver input file) and load it. Using this option enables CFX-Pre to access more information (such as composite regions, unused materials and meshes, layouts, and views) and then write that information to the CFX-Solver input file.

When the **Recover Original Session** option is selected, the **Replace Flow Data** option is available. This will extract the CCL from the CFX-Solver input file and replace the existing Case file data. This is useful to recover the problem definition when it has been modified outside of CFX-Pre during the run.

### 3.2.2. Opening Case (.cfx) Files

When opening an existing case file, CFX-Pre opens the case in the state in which it was last saved, including the mesh.

### 3.2.3. Opening CFX-Solver Input (.def, .mdef), Results (.res), Transient (.trn) or Backup (.bak) Files

CFX-Solver input and results files from the current and previous releases of CFX can be opened. When opening these files, a new case file is created. The mesh and physics are imported into the new case. All preprocessing information in these files is imported into CFX-Pre and is edited in the same way as in other case files.

CFX-Pre can also load "full" transient results file (\*.trn) or backup file (\*.bak) by typing \*.trn or \*.bak, respectively, as the **File Name** in the **Load Case File** dialog box. Using the \* character returns a list of available files of type \*.trn or \*.bak. The selected file is imported as a CFX-Solver input file.

---

#### Note:

- If a Release 11.0 .def file containing automatically generated interfaces is loaded into CFX-Pre, and these interfaces were generated as a result of 'contact' information in the original .cldb file, these interfaces may be removed by CFX-Pre. This is a problem only when loading Release 11.0 .def files, and will occur only in a small percentage of cases. Loading a .cfx file will work correctly.
  - It is not possible to load an .mres file into CFX-Pre.
-

### 3.2.4. Opening CCL (.ccl) Files

Opening a CCL file creates a new case file. Any physics, material and expression information is imported into CFX-Pre and can be edited in the same way as for case files. CCL files do not contain any mesh data, so it is necessary to import a mesh before assigning locations to domains and boundary conditions.

### 3.2.5. Opening Meshing (.cmdb or .dsdb) Files

Opening a `.cmdb` or `.dsdb` file loads the mesh and creates an initial physics state, in the same manner as creating a new case.

---

**Important:**

`.cmdb` and `.dsdb` files require the `cfxacmo` library, which is supplied with Ansys Workbench. If you are unable to load such files into CFX-Pre, one solution is to install Ansys Workbench to make those library files available.

---

### 3.2.6. Opening CFX-Mesh (.gtm) Files

Opening a `.gtm` file loads the mesh and creates an initial physics state, in the same manner as creating a new case.

## 3.3. Close Command

---

Closes the existing case, prompting to save if appropriate.

---

**Note:**

On Linux machines with an NVIDIA graphics card, processes related to CFX-Pre (in particular, `PreGui_ogl.exe`) do not always terminate automatically when CFX-Pre is closed. These processes may need to be terminated manually.

---

## 3.4. Save Case Command

---

When CFX-Pre is started from the Ansys CFX Launcher, the Save Case command writes the current state to the case file. You should save a case before closing it to be able to reopen it at a later date; all data is lost if CFX-Pre is closed without saving the case.

When CFX-Pre is started from Ansys Workbench, the Save Project command writes the current state of the project.

## 3.5. Save Project Command

---

When CFX-Pre is started from Ansys Workbench, the Save Project command writes the current state of the project.

---

## 3.6. Refresh Command (Ansys Workbench only)

---

Reads the upstream data, but does not perform any long-running operation.

## 3.7. Save Case As Command

---

When using **Save As**, the previous case files are closed and remain unchanged from the last time they were explicitly saved.

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

The **Save Case** dialog box appears.

2. Select a location where the file will be saved.
3. Under **File name**, type the name to save the file as.
4. Click **Save**.

A new file is saved and is kept open in CFX-Pre.

## 3.8. Reload Mesh Files Command

---

It is possible to import and manipulate many meshes within a CFX-Pre case. In some cases, this can result in a complex set of operations (for example, a single 'blade' mesh may have been imported, and then 30 copies may have been made). In some cases, it is desirable to swap this mesh for one that is much finer, or of better quality. The process of deleting all existing meshes, re-importing the new mesh and then applying the transformations again could be costly. Hence, the mesh reload function enables one or more mesh files to be replaced in a fraction of the time.

1. Select **File > Reload Mesh Files**.

The **Reload Mesh Files** dialog box appears.

2. Select or clear the mesh files to replace the ones that were previously imported.
3. Click **OK**.

---

### Note:

This feature is not required (and is not available) in CFX-Pre launched from Ansys Workbench.

---

## 3.9. Import Mesh Command

---

Numerous options are available when importing a mesh. For details, see [Importing Meshes \(p. 121\)](#).

## 3.10. Import CCL Command

---

CFX Command Language (CCL) consists of commands used to carry out actions in CFX-Pre, the CFX-Solver Manager and CFD-Post. All of the steps carried out in CFX-Pre are executed as CCL commands in the software's engine, and these commands can be exported and imported to other cases.

---

### Tip:

- You can also import expressions and regions using the Import CCL command.
  - To import composite region definitions from older versions of CCL, use the **Import CCL** command found in the File menu, rather than the import command found in the Regions workspace.
- 

A useful application of importing CCL is to apply the same preprocessing data to a number of different meshes. In such a case, the following general workflow may be ensued:

1. Import the new mesh.
2. Import the CCL data.
3. Assign mesh locations to the domains and boundary conditions, if required.
4. Write the CFX-Solver input file for the CFX-Solver.

The benefit of using this workflow is that there would be no need to specify all of the preprocessing data again.

Importing a set of commonly used customized material or reaction definitions is also possible by importing a CCL file. A useful application of the import CCL feature is demonstrated when using Library Mode. For details, see [Library Objects \(p. 483\)](#).

### 3.10.1. Append or Replace

#### 3.10.1.1. Append

This option never deletes existing objects such as domains, boundary conditions, and initialization. Objects with a different name than existing objects are added. If an object of the same name and type already exists, parameters within the object that are unique to the imported CCL file are added to the existing object. When the imported CCL file contains parameter definitions that already exist within existing objects, they will replace the existing definitions.

#### 3.10.1.2. Replace

This option overwrites, in full, existing objects of the same name and type. Because boundary conditions, subdomains, and so on are defined within a domain, if that domain is replaced, these objects are lost if not defined in the imported CCL file. Objects with a unique name are added to the existing case.

**Replace** only replaces any object listed in the new file that is a subobject of FLOW with the new definition in the file. However, it does not remove objects that are not listed in the new file. To remove objects not mentioned in the replacement CCL, you can either delete FLOW (and all its contents) before importing the new CCL or import the new CCL and then remove the objects that are no longer required.

### 3.10.1.2.1. Auto-load materials

When **Replace** is selected, the **Auto-load materials** check box is made available. When a file is imported with the **Auto-load materials** check box selected, any materials and reactions that are missing from the problem setup being imported, and that are not defined in the case already, will be loaded automatically. These added materials and reactions can be found in the standard materials and reactions library files.

## 3.11. Export CCL Command

---

Using the **Export CCL** command, you can export a file containing some or all CCL definitions.

1. Select **File > Export > CCL**.

The **Export CCL** dialog box appears.

2. Select or clear **Save All Objects**.

A list of all existing CCL objects is available. To export particular objects, clear **Save All Objects** and select only the objects to export. For details, see [Save All Objects \(p. 71\)](#).

3. Select a location to export to.
4. Enter a name for the exported file.
5. Click **Save**.

To export the physics definition for a problem, select all the FLOW objects. Additional Variables, CEL expressions, User Functions and material definitions are stored in LIBRARY objects; these will need to be included if you want to export these objects.

### 3.11.1. Save All Objects

When **Save All Objects** is selected, all CCL object definitions are written to the CCL file. To export only a sub-set of CCL objects, clear this and select only the required CCL objects.

#### 3.11.1.1. Sample of Saving CEL Expressions

This sample is specifically for the export of expressions.

1. Select **File > Export > CCL**.
2. Clear **Save All Objects** and expand LIBRARY.
3. Expand CEL.



4. Select `EXPRESSIONS`.
5. Select a location to export to.
6. Enter a name for the exported file.
7. Click **Save**.

## 3.12. Export CSV Command

---

Using the **Export CSV** command, you can export a file containing data from one or more objects. For a given exported file, the exported objects must be all of the same type. The eligible types are:

- User Location
- Profile function
- Table function

Furthermore, in order to be eligible for export, any objects of type User Location must be entirely known by CFX-Pre; they must not be generated only by the CFX-Solver.

1. Select **File > Export > CSV**. Alternatively, right-click an object (for example, a profile function) from the **Outline** tree view and select **Export CSV**.

The **Export CSV** dialog box appears.


2. In the list of objects, select the objects to export.

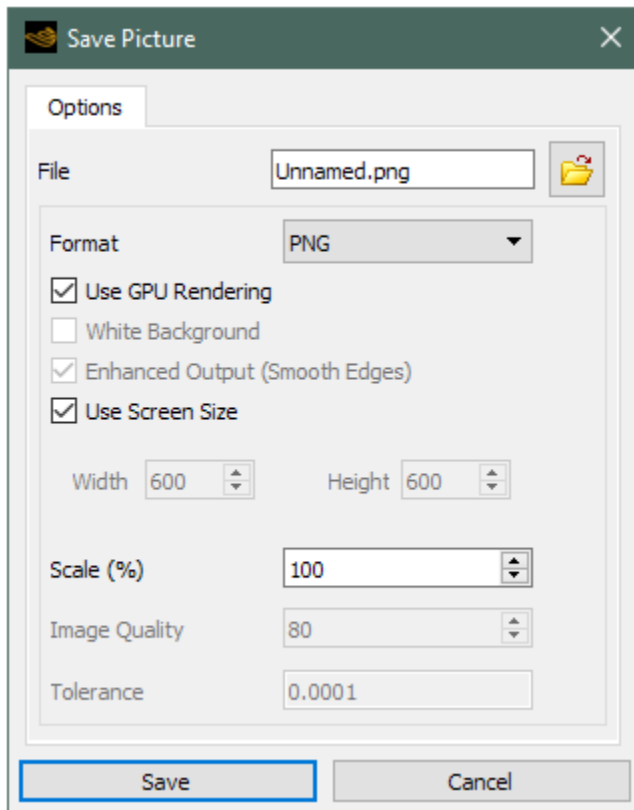
Note that the selections must be all User Locations, or all functions of type "Profile Data", or all functions of type "Table".

3. Enter a name for the exported file.
4. Click **Save**.

## 3.13. Save Picture Command

---


To save the current contents of the viewer window to a file, select **File > Save Picture** from the main menu or click *Save Picture* . The **Save Picture** dialog box is displayed.



## Options Tab


The **Options** tab has the following settings:

### File

Enables you to specify the filename of the file. You may enter the filename and path into the **File** field, or click the *Browse*  icon and search for the directory in which the file is to be saved.

Files are always saved with the file extension corresponding to the selected graphics format.

### Format

To choose a file format, click . When creating a new image file, the file format you choose affects the quality of the image:

#### PNG

Portable Network Graphics is a raster file format (\*.png) that supports lossless image compression.

#### AVZ (3D)

Ansyes Viewer Format is a file format (\*.avz) used to present interactive three-dimensional views. It can be displayed using the Ansyes Viewer.

**JPEG**

A compressed file format (\*.jpg) developed for compressing raw digital information. File sizes are relatively small. Due to compression artifacts, this format is not recommended for line drawings.

**Windows Bitmap**

A file type (\*.bmp) that is usually large and does not adjust well to resizing or editing. This file type does retain all of the quality of the original image and can be easily converted to other formats.

**PPM**

Portable Pixel Map is a file format (\*.ppm) similar to a Windows Bitmap. It is an uncompressed format and is not recommended for large images.

**PostScript and Encapsulated PS**

PostScript (\*.ps) and Encapsulated PS (\*.eps) are generally recommended for output to a printer or line drawings. However, some graphics objects and features can cause the PS/EPS to output as a very large bitmap file, in which case a PNG file would be a more efficient alternative. Note that the Ansys logo and the axis do not cause the PS/EPS output to become a bitmap.

**VRML (3D)**

Virtual Reality Modeling Language is a file format (\*.vrl) used to present interactive three-dimensional views. The output is VRML 2.0.

---

**Note:**

Existing CFD-Viewer State (\*.cvf) files previously generated can be converted by using an application supplied as part of the CFX, TurboGrid and CFD-Post installations:

```
C:\Program Files\ANSYS Inc\v$ANSYS\CFXRELEASE\CFX\bin\cfx5cvfconvert <cvf-file>
```

---

**Use GPU Rendering Check Box**

If your graphics hardware is compatible with GPU rendering, selecting this option is strongly recommended. You can set the default value for this option using the following preference: **Edit > Options > Common > Viewer Setup > Use GPU Rendering for Printing**.

If **Use GPU Rendering** is selected, GPU rendering is used unless compatible GPU hardware is not found, in which case all saved pictures are instead software rendered (by the CPU).

When you save a picture with GPU rendering, your graphics hardware renders an image that closely matches what is shown in the viewer.

If GPU rendering is not used, software rendering is used instead. Software rendering is relatively slow and does not always render as shown in the viewer. One benefit of software rendering is that it has no graphics hardware requirements.

### White Background Check Box

You can save the current image with a white background by selecting **White Background**.

When the White Background check box is selected, certain white objects may be colored black and certain black objects may be colored white in the image file. Objects that are not affected can usually be manually colored by editing them.

### Enhanced Output (Smooth Edges) Check Box

When **Enhanced Output (Smooth Edges)** is selected, the image is processed by antialiasing.

### Use Screen Size Check Box

When **Use Screen Size** is selected, the output has the same width and height, measured in pixels, as shown in the viewer. You can clear the check box to specify the width and height manually.

### Width/Height

You can specify the width and height of the image in pixels by entering values for **Width** and **Height**. In order to use these settings, the **Use Screen Size** check box must be cleared.

### Scale (%)

**Scale (%)** is used to scale the size of bitmap images to a fraction (in percent) of the current viewer window size. This option is available when **Use Screen Size** is selected.

### Image Quality

**Image Quality** is available only for the JPEG format. A value of 100 specifies the highest image quality; a value of 1 specifies the lowest image quality.

### Tolerance

**Tolerance** is a non-dimensional value used in face sorting when generating pictures. Larger values result in faster generation times, but may cause defects in the resulting output.

Click **Save** to save the current viewer contents to an image file.

## 3.14. Recent Case Files Submenu

---

CFX-Pre saves the file paths of the last five case files (.cfx) opened. To open one of these case files, select **File > Recent Case Files**.

## 3.15. Recent CCL Files Submenu

---

CFX-Pre saves the file paths of the last five CCL files opened. To open one of these CCL files, select **File > Recent CCL Files**.

## 3.16. Recent Session Files Submenu

---

CFX-Pre saves the file paths of the last five session files opened. To open one of these session files (\*.pre), select **File > Recent Session Files**.

## 3.17. Quit Command

---

To quit CFX-Pre, select **File > Quit**. In stand-alone mode, if the case is not already saved, there will be a prompt as to whether a save should be done.

---

# Chapter 4: CFX-Pre Edit Menu

---

This chapter describes:

4.1. Undo and Redo

4.2. Options

**Undo** and **Redo** commands are available in the **Edit** menu. Additionally, there are a variety of options that can be set to customize the software.

## 4.1. Undo and Redo

---

The undo and redo capability is limited by the amount of available memory.

In stand-alone mode, the undo stack is cleared whenever a **New**, **Open**, or **Close** action occurs. Similarly, when using CFX-Pre/CFD-Post from within Ansys Workbench, the undo stack is cleared in CFX-Pre/CFD-Post after the application receives commands from Ansys Workbench.

Issue the **Undo** command by doing any of the following:

- Select **Edit > Undo**.
- Click *Undo*  on the toolbar.
- Press **Ctrl + Z**

---

### Note:

- You can repeatedly issue the **Undo** command.
- Some viewer manipulations cannot be reverted using the **Undo** command.
- Some commands that you issue have multiple components. For example, when you create some objects the software creates the object and sets the visibility of the object on (in two separate operations). Thus, when you perform an undo operation in such a situation, you are setting the visibility of the object off; you must choose undo a second time to "uncreate" the object.
- **Undo** cannot be used when recording session files.

---

The redo feature is used to do an action that you have just undone using the **Undo** command. Issue the **Redo** command by doing any of the following:

- Select **Edit > Redo**.

- Click *Redo*  on the toolbar.
- Press **Ctrl + Y**

## 4.2. Options

---

The **Options** dialog box enables you to set various general preferences. Settings are retained per user.

1. Select **Edit > Options**.

The **Options** dialog box appears.

2. Set options as required. For descriptions of the available options, see:
  - [CFX-Pre Options \(p. 78\)](#)
  - [Common Options \(p. 87\)](#)

If desired, you can use the **CFX Defaults** or the **Workbench Defaults** buttons at the bottom of the dialog box to quickly set CFX-Pre, CFX-Solver Manager, and CFD-Post to have the standard appearance and operation of CFX or Workbench respectively. The only CFX-Pre settings that are affected by these buttons are:

- **CFX-Pre > Graphics Style > Background > Color Type**
- **CFX-Pre > Graphics Style > Background > Color**
- **CFX-Pre > Colors > Labels**
- **CFX-Pre > Colors > Legend Text**
- **CFX-Pre > Colors > Turbo Axis**
- **Common > Viewer Setup > Mouse Mapping**

3. Click **OK**.

---

### Note:

Changes made under the **Options** dialog box do not take effect until a new case is opened.

When changing some user preferences, it will be necessary to restart the application for the setting to take effect. For example, changes to the highlighting mode, whether made in the **Edit > Options** panel or from the **Outline** tree view **Case Options > General > General Options** panel, do not take place until the application is restarted.

---

### 4.2.1. CFX-Pre Options

The **CFX-Pre** options are.

- **Record Default Session File**

When selected, a session file named `cfx.xx.pre` will be recorded automatically each time CFX-Pre is started (where 'xx' is the next available number).

- **Default User Mode** can be set to `General`, `Turbo`, or `Quick Setup`.

This determines the default mode that CFX-Pre will use when creating a simulation. For details on Turbo mode, see [Turbomachinery Mode \(p. 473\)](#). For details on Quick Setup mode, see [Quick Setup Mode \(p. 469\)](#).

- **Report CCL Update Summary** produces an information window when you load a file that contains CCL from a previous version of CFX-Pre. This window describes the updates that were made to the CCL to make it compatible with the current software release.

The following sections describe the options under **CFX-Pre**:

[4.2.1.1. General](#)

[4.2.1.2. Graphics Style](#)

[4.2.1.3. Render](#)

[4.2.1.4. Mesh](#)

[4.2.1.5. Turbo](#)

[4.2.1.6. Labels and Markers](#)

[4.2.1.7. User Interface](#)

[4.2.1.8. Extensions](#)

[4.2.1.9. Customization](#)

[4.2.1.10. Solve](#)

[4.2.1.11. Viewer](#)

## 4.2.1.1. General

The **General** options can be accessed in two ways:

- Edit Menu

Select **Edit > Options** to open the **Options** dialog box, then select **CFX-Pre > General** in the tree.

These settings persist until you change them, and can be used to control the behavior of CFX-Pre for new cases.

- Case Options

Go to the **Outline** tree view and edit **Case Options > General**.



These settings apply only for the current case. The corresponding **General** settings of the **Options** dialog box are used to initialize the case options, but are overridden by modifications made to the case options.

---

**Note:**

Some of the options related to Beta features cannot be set in the case options.

---

#### 4.2.1.1.1. Beta Options

The Beta options are:

[4.2.1.1.1.1. Physics Beta Features](#)

[4.2.1.1.1.2. Topology Beta Features](#)

##### 4.2.1.1.1.1. Physics Beta Features

The physics Beta feature options are:

- **Enable Beta Features**

Some Beta features are hidden in the user interface. You can select this option to "unhide" those features. When selected, such features are identified by "(Beta)" in the user interface. Note that Beta features are unofficial and not well tested.

- **Constant Domain Physics**

---

**Note:**

Physics Beta features are normally indicated by a message in the physics message window. The physics message window is shown in [Figure 1.2: Sample CFX-Pre Interface \(p. 36\)](#).

---

##### 4.2.1.1.1.2. Topology Beta Features

The topology Beta feature options are:

- **Enable Simplification On Load and Import**
- **Preserve Direct Connections On Load and Import** (applies only when **Enable Simplification On Load and Import** is selected)
- **Combine Unreferenced Primitives On Load and Import** (applies only when **Enable Simplification On Load and Import** is selected)
- **Enable Simplification On Write**

##### 4.2.1.1.2. Auto Generation

- **Automatic Default Domain**

When this option is selected, a domain with the name `Default Domain` will be created upon importing a mesh.

To toggle default domain generation on or off for a session, without affecting the user preference setting, you can right-click the `Simulation` object in the tree view and select **Automatic Default Domain** from the shortcut menu.

If you manually delete a default domain, the default domain mechanism will be disabled, and a warning message will appear in the physics message window.

If you create a domain that uses the same region(s) as the default domain, the latter will be redefined with the remaining locations, or deleted if all the regions are referenced by user-defined domains.

If you modify the location of the default domain, the name will change to `Default Domain Modified` and no additional default domain will be generated.

When loading an existing case (`cfx` file or `def` file), if there are any mesh volumes that are not assigned to a domain, the default domain generation will be disabled. It can be re-activated as described previously.

- **Automatic Default Interfaces**

When selected, CFX-Pre will attempt to create domain interfaces when a domain is created or modified.

To toggle default interface generation on or off for a session, without affecting the user preference setting, you can right-click the `Simulation` object in the tree view and select **Automatic Default Interfaces** from the shortcut menu.

Domain interface generation is always deactivated when loading an existing simulation.

- **Interface Method**

When **Automatic Default Interfaces** has been selected, the **Interface Method** can be set to one of the following to control how interfaces are automatically generated between domains where regions are found to be connected:

- `One per Interface Type`

This method groups as many domains into as few interfaces as possible.

- `One per Domain Pair`

An interface is generated for each pair of domains.

- **Default Boundary** can be set to one of the following:

- `Standard`

A default boundary condition is created that covers all primitive regions that are not assigned to any boundary condition in the current domain. The default boundary is modified dynamically when other boundary conditions are subsequently added or deleted such that it includes all regions not assigned to any other boundary condition.

- One per Relevant Region

A default boundary condition on each relevant region not assigned to any boundary condition is created. In this context, 'relevant' means every composite 2D region, plus any 2D primitive regions that are not referenced by a composite 2D region. If boundary conditions are subsequently deleted, causing some regions to be unassigned, a single default boundary condition will include all such regions.

- One per Primitive Region

A default boundary condition on each individual 2D primitive region not assigned to any boundary condition is created. If boundary conditions are subsequently deleted, causing some regions to be unassigned, a single default boundary condition will include all such regions.

- Disabled

- **Strict Domain Type Checking for Automatic Interfaces** option checks that the domain type is consistent between all parts of an interface before the automatic domain interfaces are created.

#### 4.2.1.1.3. Physics

- **Disable Physics Validation**

This option prevents CFX-Pre from issuing messages in the physics message window. For details, see [Physics Message Window](#) (p. 45).

- **Filter Messages**

Three types of messages are displayed in the physics message window: information messages, warning messages, and error messages.

**Filter Messages** can be set to one of the following to control the types of messages that are displayed:

- Suppress None

Upon applying this setting, all information messages, warning messages, and error messages are displayed.

- Suppress Information

Upon applying this setting, all information messages<sup>[1]</sup> are suppressed, and all warning messages and error messages are displayed.

- Suppress Information/Warning

Upon applying this setting, all information messages<sup>[1]</sup> and warning messages are suppressed, and all error messages are displayed.

---

[1] When you have suppressed messages, an information message appears summarizing the number of information messages, warning messages, and error messages that are suppressed. This message cannot be suppressed with the **Filter Messages** setting.

- Suppress All

Upon applying this setting, all information messages<sup>[1]</sup>, warning messages, and error messages are suppressed.

When you change the **Filter Messages** setting, it overrides any messages that have previously been individually suppressed.

---

**Note:**

An information message is displayed in the physics message window when Beta Features are enabled. This message cannot be suppressed with the **Filter Messages** setting.

---

- **Automatic Physics Update**

If this option is selected and you change settings in the simulation definition, CFX-Pre will, for certain settings, respond by changing other settings automatically in an attempt to make problem specification consistent. This incurs an overhead, so for large problems you may want to disable this feature.

- **Show Interface Boundaries in Outline Tree**

Shows the interface boundaries in the **Outline** tree view.

## 4.2.1.2. Graphics Style

Settings made here set the default operation for CFX-Pre; however, you can override these settings for your current simulation by going to the **Outline** tree view and editing **Case Options > Graphics Style**.

### 4.2.1.2.1. Object Highlighting

Controls how an object that is generated after a change to the setting of this option is highlighted in the viewer. Such highlighting occurs when in picking mode, when selecting a region in a list, or when selecting items in the tree view.

Under **Type**, select one of the following:

- **Surface Mesh**: Displays the surface mesh for selected regions using lines.
- **Face Highlight**: Displays the selected regions using faces.
- **Wireframe**: Traces objects that contain surfaces with green lines.

- **Bounding Box:** Highlights the selected objects with a green box.

---

**Note:**

When you load a case, the highlighting is dictated by the setting that is stored in the case, rather than by the current preferences setting.

---

#### 4.2.1.2.2. Background

Set **Mode** to **Color** or **Image**.

##### 4.2.1.2.2.1. Color

Use **Color Type** to set either a solid color or a gradient of colors; use **Color** to set the color (and **Color 2** for gradients).

##### 4.2.1.2.2.2. Image

Select one of a list of predefined images or a custom image.

If selecting a custom image, choose an image file and a type of mapping. Image types that are supported include \*.bmp (24-bit BMP only), \*.jpg, \*.png, and \*.ppm. Mapping options are **Flat** and **Spherical**. Flat maps are stationary while spherical maps surround the virtual environment and rotate with the objects in the viewer.

Custom images have some restrictions: all background images and textures sent to the viewer must be square and must have dimensions that are powers of 2 (for example, 512 x 512 or 1024 x 1024).

If the dimensions of your background image is not a power of 2, the viewer sizes the image to be a power of 2 by doing bicubic resampling.


To make the background image square, transparent pixels are added to the smaller dimension to make it the same as the larger dimension. The transparent pixels enable you to see the regular viewer background, which gives you control over what fill color your background has.

#### 4.2.1.2.3. Colors

##### 4.2.1.2.3.1. Labels

Set the labels to be bright or dark.

##### 4.2.1.2.3.2. Legend Text and Turbo Axis

Select a color by clicking in the box, or clicking the *Ellipsis*  icon.

#### 4.2.1.2.4. Visibility

##### 4.2.1.2.4.1. Axis and Ruler Visibility

Select or clear **Axis Visibility** or **Ruler Visibility** to show or hide the axis indicator or ruler in the viewer.

### 4.2.1.3. Render

These settings are used to control the display properties of faces and lines. For details, see [Render Options](#) (p. 148).

### 4.2.1.4. Mesh

**Mesh Match Tolerance** is used when creating domain interfaces. It is used to determine whether a one-to-one connection can be made at a domain interface. The tolerance is relative to the local mesh length scale; the default value is 0.005 (or 0.5%) of the local edge length on the first side of the interface. A node on the second side must be within this tolerance to a node on the first side for the two to be considered coincident.

#### 4.2.1.4.1. Mesh Import Options

**Source Format** specifies which type of mesh file is the general default. For details, see [Supported Mesh File Types](#) (p. 123). **Source Directory** specifies the default directory from which meshes are imported upon selecting the **Import Mesh** command. It is also possible to set other general options (such as mesh units) and specific advanced options on a per-mesh format basis.

### 4.2.1.5. Turbo

These settings are used in the recognition of turbo regions when importing a mesh using Turbo mode.

### 4.2.1.6. Labels and Markers

The settings under this category control whether labels and boundaries appear in the cases displayed in the **3D Viewer**. Settings made here set the default operation for CFX-Pre; however, you can override these settings for your current simulation by going to the **Outline** tree view and editing **Case Options > Labels and Markers**.

#### 4.2.1.6.1. Labels

The **Show Labels** option controls whether any labels are displayed; when selected, the remaining options control whether particular types of labels are displayed.

#### 4.2.1.6.2. Boundary Markers

When **Show Boundary Markers** is selected, the check boxes in that panel control which markers are displayed.

The **Marker Quantity** slider controls the number of markers displayed. Moving the slider to the right increases the number.

The **Marker Length** slider controls the size of the markers displayed. Moving the slider to the right increases the size.

#### 4.2.1.6.3. Boundary Vectors

The **Vector Quantity** slider controls the number of vectors displayed. Moving the slider to the right increases the number.

The **Vector Length** slider controls the size of the vectors displayed. Moving the slider to the right increases the size.

See [Boundary Plot Options Tab \(p. 234\)](#) for a discussion of displaying boundary vectors.

#### 4.2.1.6.4. Profile Vectors

The **Vector Length** slider controls the size of the vectors displayed. Moving the slider to the right increases the size.

See [Visualization Options for Profile Data \(p. 427\)](#) for a discussion of displaying profile vectors

#### 4.2.1.6.5. Injection Region Vectors

The **Vector Length** slider controls the size of the vectors displayed. Moving the slider to the right increases the size.

See [Injection Region Vector \(p. 267\)](#) for a discussion of displaying injection region vectors.

#### 4.2.1.6.6. Particle Injection Regions

The **Rendering** option controls how many symbols are drawn for each particle injection region.

The options are:

- Full

This option causes multiple symbols to be drawn per particle injection region in order to represent the Injection Method settings (see [Injection Method \(p. 200\)](#)).

- Reduced

This option causes exactly one symbol to be drawn per particle injection region.

- None

This option turns off visual representation of particle injection regions.

#### 4.2.1.7. User Interface

Set **GT-SUITE Initialization > Selectable Versions** to a comma-delimited list of GT-SUITE versions, each version having the same name as the corresponding version-specific subdirectory under the GT-SUITE installation directory. For example, you could set **Selectable Versions** to "v2019 , v2020" (excluding the quotes). This enables you to make additional GT-SUITE versions selectable in CFX-Pre if necessary. However, you should refer to [Supported Capabilities and Limitations in the CFX-Solver Modeling Guide](#) for a list of GT-SUITE versions that are officially supported.

The versions that you specify become options for the following settings:

- The **GT-SUITE Version** setting in the **Initialize GT-SUITE Coupling** dialog box. For details, see [Initialize GT-SUITE Coupling \(p. 109\)](#).
- The **Version** setting in the `GT-SUITE model` object, which is listed in the **Outline** tree view. For details, see [GT-SUITE Model Details View \(p. 443\)](#).

For details about using GT-SUITE with CFX, see [Coupling CFX to an External Solver: GT-SUITE Coupling Simulations in the CFX-Solver Modeling Guide](#).

#### 4.2.1.8. Extensions

When **Include Installed Extension Files** is selected, you have the option of creating a comma-separated list of files to exclude.

#### 4.2.1.9. Customization

The **Use Custom Files** setting enables the creation of special-purpose interfaces that extend the functionality of CFX-Pre for your environment. Contact your Customer Support representative for more information.

The **Force generation of rules files** an advanced setting used to maintain synchronization of customized RULES files. This option is useful during the development of customized RULES files and is available only when **Use Custom Files** is selected.

#### 4.2.1.10. Solve

The **Definition File Timeout** setting controls how long CFX-Pre will wait in seconds while attempting to obtain enough data from the CFX-Solver in order to spawn a CFX-Solver Manager to monitor an existing batch run. This parameter is used when employing the **Simulation Control > Start Solver > Run Solver and Monitor** command to start the CFX-Solver Manager. See [Simulation Control in Outline Tree View Structure \(p. 37\)](#) for details on monitoring a running solver batch run.

#### 4.2.1.11. Viewer

For details on **Stereo** settings, see [Stereo Viewer \(p. 62\)](#).

### 4.2.2. Common Options

The **Common** options are:

#### Auto Save

Select the time between automatic saves.

To turn off automatic saves, set **Auto Save** to `Never`.

---


#### Note:

This option affects more than one CFX product.

---



## Temporary directory

To set a temporary directory, click *Browse*  to find a convenient directory where the autosave feature will save state files.

The following sections describe the options under **Common**:

### 4.2.2.1. Appearance

### 4.2.2.2. Viewer Setup

### 4.2.2.3. Units

## 4.2.2.1. Appearance

The appearance of the user interface can be controlled from the **Appearance** options. The default user interface style will be set to that of your machine. For example, on Windows, the user interface has a Windows look to it. If, for example, a Motif appearance to the user interface is preferred, select to use this instead of the Windows style.

1. Under **GUI Style**, select the user interface style to use.
2. For **Font** and **Formatted Font**, specify the fonts to use in the application.

---

### Note:

It is important not to set the font size too high (over 24 pt. is not recommended) or the dialog boxes may become difficult to read. Setting the font size too small may cause some portions of the text to not be visible on monitors set at low resolutions. It is also important not to set the font to a family such as Webdings, Wingdings, Symbols, or similar type faces, or the dialog boxes become illegible.

---

## 4.2.2.2. Viewer Setup

1. If you have complicated simulations that feature many overlapping lines, you can specify a **Picking Tolerance** that will increase the resolution for picking operations. Values must be between 1 (low resolution) and 0 (very high resolution); the default value is 0.1. Note that increasing the resolution will slow printing times.
2. Select **Double Buffering** to use two color buffers for improved visualization.

Double Buffering is a feature supported by most OpenGL implementations. It provides two complete color buffers that swap between each other to animate graphics smoothly. If your implementation of OpenGL does not support double buffering, you can clear this check box.

3. Select or clear **Unlimited Zoom**.

By default, zoom is restricted to prevent graphics problems related to depth sorting. Selecting **Unlimited Zoom** allows an unrestricted zoom.

4. Select or clear **Use GPU Rendering for Printing**.

If your graphics hardware is compatible with GPU rendering, selecting this preference is strongly recommended.

When selected, this preference causes saved pictures in CFD-Post to use GPU rendering and saved pictures in CFX-Pre and TurboGrid to (by controlling the default value of a setting in the **Save Picture** dialog box) use GPU rendering by default.

When you save a picture with GPU rendering, your graphics hardware renders an image that closely matches what is shown in the viewer.

If GPU rendering is not set or cannot be used (for example, if compatible GPU hardware is not found), software rendering is used instead. Software rendering is relatively slow and does not always render as shown in the viewer. One benefit of software rendering is that it has no graphics hardware requirements.

---

**Note:**

Batch mode printing in CFX-Pre and TurboGrid does not support GPU rendering.

---

#### 4.2.2.2.1. Mouse Mapping

The mouse-mapping options enable you to assign viewer actions to mouse clicks and keyboard/mouse combinations. These options are available when running in stand-alone mode. To adjust or view the mouse mapping options, select **Edit > Options**, then **Common > Viewer Setup > Mouse Mapping**. For details, see [Mouse Button Mapping](#).

#### 4.2.2.3. Units

1. Under **System**, select the unit system to use. Unit systems are sets of quantity types for mass, length, time, and so on.

The options under **System** include SI, CGS, English Engineering, British Technical, US Customary, US Engineering, or Custom. Only Custom enables you to redefine a quantity type (for example, to use inches for the dimensions in a file that otherwise used SI units).

The most common quantity types appear in the main **Options** dialog box; to see *all* quantity types, click **More Units**.

2. Select or clear **Always convert units to Preferred Units**.

If **Always convert units to Preferred Units** is selected, the units of entered quantities are immediately converted to those set in this dialog box.

For example, if you have set **Velocity** to  $[m\ s^{-1}]$  in this dialog box to make that the preferred velocity unit, and elsewhere you enter 20  $[mile\ hr^{-1}]$  for a velocity quantity, the entered value is immediately converted and displayed as 8.94078  $[m\ s^{-1}]$ .

The two sets of units are:

- The units presented on this dialog box, which control the default units presented in the user interface as well as the units used for mesh transformation.
- The solution units. For details, see [Setting the Solution Units \(p. 288\)](#).

#### **4.2.2.3.1. Additional Help on Units**

For additional information about units, see [Mesh Units \(p. 122\)](#).

---

# Chapter 5: CFX-Pre Session Menu

---

A *session file* is a record of the actions performed during a CFX-Pre session, saved in a file as commands. The actions that cause commands to be written to a session file include:

- Viewer manipulation performed using the commands available by right-clicking in the viewer window.
- All actions available from the **File** and **Edit** menus.
- Creation of expressions.
- Creation of new objects and changes to an object committed by clicking **OK** or **Apply** on any of the panels available from the **Tools** and **Insert** menus/toolbars.
- Commands issued in the **Command Editor** dialog box.
- Mesh import, delete, and transformation operations.

This chapter describes:

- 5.1. [New Session Command](#)
- 5.2. [Start Recording and Stop Recording Commands](#)
- 5.3. [Play Session Command](#)

## 5.1. New Session Command

---

To record your actions in CFX-Pre in a session file:

1. Select **Session > New Session**. This opens the **Set Session File** dialog box, where you can enter a filename for your session file. Once you have saved the file, it becomes the current session file.

---

**Note:**

This command is available only when a session file is not currently being recorded.

---

2. Browse to the directory in which you want to create the session file, and then enter a name for the file ending with a `.pre` (CFX-Pre) extension.
3. Click **Save** to create the file.
4. To start recording to the session file, select **Session > Start Recording**.

- To stop recording to the session file, select **Session > Stop Recording**.

---

**Important:**

Session files must not contain > undo commands. These commands would produce errors when playing back the session file.

---

If you create more than one session file during a CFX-Pre session, the most recently created file is the current session file by default. You can set a different file to be the current session file by selecting an existing file from the **New Session > Set Session File** window and then clicking **Save**. Because the file exists, a warning dialog box appears:

- If you select **Overwrite**, the existing session file is deleted and a new file is created in its place.
  - If you select **Append**, commands will be added to the end of the existing session file when recording begins.
- 

**Note:**

By default, CFX-Pre does not continuously write commands to a session file while you are working on your simulation. You can change a setting in **Edit > Options** so that a session file is recorded by default. If a session file is being recorded by CFX-Pre, whether by default or intentionally, a new session file cannot be recorded.

---

## 5.2. Start Recording and Stop Recording Commands

---

The **Start Recording** action activates recording of CCL commands issued to the current session file. A session file must first be set before you can start recording (see [New Session Command \(p. 91\)](#)).

**Stop Recording** terminates writing of CCL commands to the current session file. You can start and stop recording to a session file as many times as necessary.

## 5.3. Play Session Command

---

A session file is a record of the actions performed during a CFX-Pre session, saved to a file when you have defined a session filename and have clicked **Session > Start Recording**. By default, this file is stored in your working directory.

After you have recorded a session file, you can select **Session > Play Session**, which opens the **Play Session File** dialog box in which you can select the session file to play. The commands listed in the selected session file are then executed.

---

**Important:**

If a session file is played while a current simulation is open, existing data will be lost in the following situations:

- If the session file starts a new simulation (that is, if it contains a `>load` command), then the current simulation is closed without saving.
  - If the session file does not contain a `>load` command, the behavior is the same as importing a CCL file using the **Append** option. For details, see [Append or Replace \(p. 70\)](#). Existing objects with the same name as objects defined in the session file are replaced by those in the session file.
- 

To play a session file:

1. From the menu bar, select **Session > Play Session**.
  2. Browse to the directory containing the session file and select the file you want to play.
  3. Click **Open** to play the session file.
- 

**Note:**

You can play session files in stand-alone CFX-Pre, but not in CFX-Pre in Ansys Workbench.

---



---

# Chapter 6: CFX-Pre Insert Menu

---

The **Insert** menu enables you to create new objects, such as domains or boundary conditions, or to edit existing objects.

---

## Tip:

You are also able to create objects from shortcut menus in some contexts.

---

The settings specified in the various **Insert** menu panels correspond to all the data displayed in the **Outline** tree view. In many cases, the name of the new object can be specified. This name must be no more than 80 characters in length.

## Valid Syntax for Named Objects

Any of the following characters are allowed in names of objects in CFX-Pre: A-Z a-z 0-9 <space> (however, the first character must be A-Z or a-z). Multiple spaces are treated as a single space character, and spaces at the end of a name are ignored.

In general, object names must be unique within the physics setup.

## Analysis

Creates a new **Flow Analysis** in the **Outline** tree view under **Simulation**. This enables you to define a steady-state analysis or a transient analysis.

## Analysis Type

Specifies a steady-state or a transient analysis (in the analysis you select, when multiple analyses are available). Steady-state analyses are used to model flows that do not change over time, while transient analyses model flows that are time-dependent. For details, see [Analysis Type \(p. 161\)](#).

## Domain

Creates new fluid and solid domains (in the analysis you select, when multiple analyses are available). These are the bounding volumes within which your CFD analysis is performed. You can create many domains in CFX-Pre and each can be stationary or rotate at its own rate, using different mesh element types. For details, see [Domains \(p. 165\)](#).

## Boundary

Sets the conditions on the external boundaries of a specified domain in a selected analysis. In CFX-Pre, boundary conditions are applied to existing 2D mesh regions. For details, see [Boundary Conditions \(p. 217\)](#).



**Subdomain**

Creates subdomains, which are volumes within a specified domain in a selected analysis that are used to create volumetric sources. For details, see [Subdomains \(p. 247\)](#).

**Injection Region**

Creates fluid mass sources or sinks at a set of positions on selected surfaces or within the volume of a selected domain. For details, see [Injection Regions \(p. 253\)](#).

**Source Point**

Creates sources of quantities at a point location within a specified domain in a selected analysis. For details, see [Source Points \(p. 271\)](#).

**Domain Interface**

Connects fluid domains together (in the analysis you select, when multiple analyses are available). If a frame change occurs across the interface, you have the choice of using a frozen rotor, stage, or transient rotor-stator model to account for the frame change. You can also take advantage of domain interfaces to produce periodic connections between dissimilar meshes. For details, see [Domain Interfaces \(p. 205\)](#).

**Global Initialization**

Sets values or expressions for the global initial conditions (across all domains in the analysis you select, when multiple analyses are available). Domain specific initialization is set through the domain forms. In CFX-Pre, you can set linearly varying conditions from inlet to outlet using the initialization forms. For details, see [Initialization \(p. 237\)](#).

**Coordinate Frame**

Creates and edits coordinate frames. A Cartesian coordinate frame exists by default, but other Cartesian frames can be made. For details, see [Coordinate Frames in the CFX-Solver Modeling Guide](#) and [Coordinate Frames \(p. 371\)](#).

**Transformation**

Creates transformations that can be used for positioning data from a results file as part of initialization. For details, see [Interpolation Mapping in the CFX-Solver Modeling Guide](#).

**Material / Reaction**

Creates and modifies materials and reactions. For details, see [Materials and Reactions \(p. 387\)](#).

**Regions: Composite Region / Primitive Region**

Composite regions can be created from basic primitive regions that are imported with a mesh. The Regions details view supports union and alias operations. This enables you to manipulate existing 2D and 3D regions without returning to the mesh generation software package. The creation of new regions is limited by the topology of the existing primitive regions; therefore, you must still create appropriate regions in the mesh-generation software package.

You can specify physics on either a primitive region, a composite region, or a mixture of both.

---

For details, see [Regions \(p. 155\)](#).

### **User Locations: User Line, User Point, User Point Cloud, User Surface**

Defines a User Location, which represents a set of one or more points and, depending on the type, connectivity data. User Locations can be used for:

- Monitoring and postprocessing

For example, solution data can be written using the User Surface instead of every vertex of the mesh, which reduces the amount of data present in results files. For details, see [Monitor Surfaces \(p. 327\)](#).

- Specifying injection locations

For details, see [Injection Positions \(p. 255\)](#).

For details, see [User Locations \(p. 377\)](#).

### **Expressions, Functions and Variables: Additional Variable**

Creates and modifies Additional Variables. For details, see [Additional Variables \(p. 405\)](#).

### **Expressions, Functions and Variables: Expression**

Creates and generates expressions using the CFX Expression Language (CEL). For details, see [Expressions \(p. 413\)](#).

### **Expressions, Functions and Variables: User Functions**

Creates 1D and cloud of points interpolation functions. The interpolation functions are typically used to set boundary and initialization values in addition to profile data interpolation functions. For details, see [User Functions \(p. 419\)](#).

### **Expressions, Functions and Variables: User Routines**

Creates User CEL, Junction Box, and Particle User Routines. For details, see [User Routines \(p. 439\)](#).

### **Solver: Solution Units**

Sets the solution units used by the CFX-Solver (in the analysis you select, when multiple analyses are available). These are the units that your results will appear in. For details, see [Units and Dimensions \(p. 283\)](#).

### **Solver: Solver Control**

Controls the execution of the CFX-Solver (in the analysis you select, when multiple analyses are available). This includes timestep and convergence details, as well as the choice of advection scheme. For details, see [Solver Control \(p. 289\)](#).

### **Solver: Output Control**

Controls output from the CFX-Solver, including backup and transient results file creation (in the analysis you select, when multiple analyses are available). For details, see [Output Control \(p. 309\)](#).

**Solver: Mesh Adaption**

Controls if and how the mesh will be automatically refined during the solution (in the analysis you select, when multiple analyses are available). This technique can be used to refine the mesh to a particular flow feature whose location is unknown prior to starting the analysis, such as a shock wave. For details, see [Mesh Adaption \(p. 361\)](#).

**Solver: Expert Parameter**

Provides advanced control of the CFX-Solver (in the analysis you select, when multiple analyses are available). For most analyses, you do not need to use expert parameters. For details, see [Expert Control Parameters \(p. 369\)](#).

**Solver: Operating Points**

Enables you to define a set of operating points for an operating point case.

For modeling information, see [Operating Maps and Operating Point Cases in the CFX-Solver Modeling Guide](#).

**Solver: Execution Control**

Enables you to define how the CFX-Solver is to be started for a simulation. See [Execution Control \(p. 449\)](#) for details.

**Configurations: Configuration, Termination Control**

Simulation controls enable you to define the execution of analyses and related tasks such as remeshing in the simulation. Specific controls include definitions of global execution and termination controls for one or more configurations. See [Configurations \(p. 461\)](#) for additional information.

---

# Chapter 7: CFX-Pre Tools Menu

---

The **Tools** menu provides access to the following:

- 7.1. Command Editor
- 7.2. Edit Profile Data
- 7.3. Initialize Profile Data
- 7.4. Initialize FMU Instance
- 7.5. Initialize GT-SUITE Coupling
- 7.6. Macro Calculator
- 7.7. Solve
- 7.8. Applications
- 7.9. Quick Setup Mode
- 7.10. Turbo Mode

## 7.1. Command Editor

---

Displays and edits the CCL definition of objects, and as well issues commands directly to CFX-Pre. For details, see [Command Editor Dialog Box \(p. 487\)](#).

## 7.2. Edit Profile Data

---


When setting up a boundary profile for a turbomachinery case (for example, involving an inlet disturbance or blade flutter), it is sometimes necessary or convenient to perform various transformations of the profile data. As an example, you might expand a profile around the machine axis so that it can be used for a boundary condition that contains all passages of interest (more passages than in the original profile). As another example, you might want to map a profile to a new location.

To edit profile data:


1. Select **Tools > Edit Profile Data**.

The **Edit Profile Data** dialog box appears.

2. Set **Source Profile** to the name of the original profile file.

(You may use the file browser for this purpose, by clicking *Browse* .)

3. Set **Write to Profile** to the name of the file that will hold the edited profile.

(You may use the file browser for this purpose, by clicking *Browse* .)

4. To have the written profile data automatically initialized (the equivalent of selecting **Tools > Initialize Profile Data** and then opening the written profile using the **Initialize Profile Data** dialog box), leave **Initialize New Profile After Writing** selected.
5. Add one or more transformations as needed.

The following sections provide details on the profile transformation options.

6. Click **OK** to begin the operation.
7. When finished, click **Close** to close the dialog box.

The **Apply** button begins the operation but leaves the **Edit Profile Data** dialog box opened.

Note that if the path or filename of the source profile (as it appears in the **Source Profile** field) is altered, the **OK** and **Apply** buttons become unavailable. You must then click **Reload** to re-read the specified source profile.

The following topics are discussed:

- 7.2.1. Profile Data Expansion (Option = Expansion)
- 7.2.2. Profile Data Mapping (Option = Map to Mesh Regions)
- 7.2.3. Profile Data Expansion with Mapping (Option = Mapped Expansion)
- 7.2.4. Mapping Profile Data to a File
- 7.2.5. Modifying Spatial Fields in Profile Data
- 7.2.6. Other Profile Transformation Options

### 7.2.1. Profile Data Expansion (Option = Expansion)

Given an existing profile data file that describes, in Cartesian coordinates, a section that possesses rotational periodicity around a rotation axis, you can use an Expansion transformation to obtain a new profile with multiple "copies" of the section, rotated around the axis, extending to as many passages as required (up to a full wheel). The input profile is "replicated" about the specified axis in the positive rotational direction (according to the right-hand rule). The number of replications required is either specified directly or, if **Expansion Option** is set to `Expand to Full Circle`, given by the ratio of the specified values for **Passages in 360** and **Passages in Profile**.

In the expanded profile, a sector number is added to account for the number of passages that must be covered by the expansion. Sector numbering starts from 1. In the case of a single passage per sector, the sector number corresponds to the passage number. Note that sector numbers are not added to profiles that contain gaps within the part being expanded.

The procedure for expanding profile data involves the following steps:

1. Set the profile transformation option to `Expansion`.
2. If the original profile contains variables that require phase shifting during expansion (for example, Mode Shapes from Ansys Mechanical cyclic symmetry analysis), the **Use Profile Instancing** option is available (and selected by default). This option is available only for a profile that has been exported by Ansys Mechanical and that contains a complex mode shape, including the Harmonic Index parameter.

Expanding the profile using instancing takes proper account of the phase shift between the blade passages.

3. Specify the axis about which the profile must rotate during the expansion process.

If you select the `Rotation Axis` option, enter two Cartesian points on the axis to indicate the axis origin and the direction of increasing axial coordinate. These points can be entered manually or selected in the viewer by clicking any coordinate box and then clicking in the viewer. The working units are assumed to apply to the point coordinates.

4. Set **Passages in Profile** to the number of passages that are modeled by the original profile.
5. Set **Passages in 360** to the number of passages in a complete 360° revolution.
6. By default, **Expansion Option** is set to `Expand to Full Circle`, causing the profile to expand such that it wraps completely around the machine axis. If this is not wanted, then set **Expansion Option** to `Spec. Number of Passages` and specify the required number of passages for the expansion (**Pass. in Expansion**).
7. Specify a non-zero **Theta Offset** value to carry out the expansion starting from a rotated version of the original profile.

### 7.2.1.1. Notes and Limitations of Profile Data Expansion

Profile Expansion has the following limitations:

- If a non-integer number of replications is needed to obtain a 360° profile, then the data for the last replicated portion is truncated and a gap is left. The gap is positioned to be as far away from the original profile as possible.
- Only files that have spatial data consisting of three spatial Cartesian coordinates (X, Y, Z) are supported.
- Variables in the profile file that are identified as vector components in cylindrical coordinates will be transformed so that their axial, circumferential, and radial components remain constant in each replication. It is assumed that vector variables in cylindrical coordinates (such as `Velocity Axial`, `Velocity Radial`, and `Velocity Circumferential`) are relative to the same axis of rotation as used for the profile expansion.
- Variables are identified as components of Cartesian vectors if they are standard CFX Cartesian vector components or if they have named with a **X**, **Y**, or **Z** suffix (upper or lower case). If all three components of a vector are present, then the three components are treated as components of a Cartesian vector, and the vector is rotated appropriately when the profile is replicated. If not all three components of the vector are present, then the components are ignored and will not appear in the expanded profile file. It is assumed that all three components of a vector have consistent units.
- Variables that are standard CFX tensor components (for example, components of Reynolds Stress) will be ignored and will not appear in the expanded profile file.
- Variables from the profile file that are not identified as vector and tensor components will be assumed to be scalar variables and will take the same value in each replication, leading to a rotationally-periodic variable definition in the expanded profile.

- It is advisable to check an expanded profile file by initializing it and then coloring the resulting User Function, to ensure that vector variables have been transformed as intended.
- A new variable, **Sector Tag**, is written to the expanded profile file only if an integer number of replications is needed to obtain a 360° profile. This identifies each replication by a unique integer.

## 7.2.2. Profile Data Mapping (Option = Map to Mesh Regions)

When the points in a boundary condition profile do not exactly match the underlying CFX mesh, the CEL functions that evaluate the associated expressions use a "cloud of points" method, using the closest three points, to interpolate the profile data onto the CFX mesh (see [Three Dimensional Interpolation \(p. 422\)](#)). This can be inaccurate for some profiles and meshes, particularly those situations where the profile is coarse and the mesh is fine, or where relatively thin structures (like blades) are present.

To improve accuracy in these situations, the profile mapping operation maps the profile data onto the selected CFX mesh regions explicitly, using a more accurate mapping algorithm to interpolate the profile data onto the CFX mesh.

After the mapping process is complete, you are presented with the diagnostics for the mapping operation. If any of the "Overlap face/area/node" statistics are significantly below 100% then you should check the resulting profile carefully to ensure that the mapping setup has worked as expected.

The procedure for mapping profile data involves the following steps:

1. Set the profile transformation option to `Mapped Expansion`.
2. Specify the **Mesh Regions** onto which the profile data needs to be mapped.
3. Leave **Add Node Mapping Status Variable** selected to create variable `Node Mapping Status`, which has a value of 1 for a mapped node and 0 for an unmapped node. This variable can be visualized in CFX-Pre (by coloring by a profile variable) or by importing the profile into CFD-Post.

### 7.2.2.1. Notes and Limitations of Profile Data Mapping

The following limitations apply to profile data mapping:

- A coordinate frame defined in the profile is ignored; the nodal coordinates, when passed to the mapper, and the resulting mapped profile, are assumed to be in the global frame. However, the coordinate frame definition is still included in the mapped file.
- Failures in the mapping process might occur if the selected mesh regions include regions from both sides of a periodic boundary. If such a failure occurs then you can map onto subsets of the mesh regions and combine the resulting profile files as necessary.
- Mapping is performed in a way that preserves profiles for solution variables; flow quantities are not necessarily conserved.

## 7.2.3. Profile Data Expansion with Mapping (Option = Mapped Expansion)

The `Mapped Expansion` option enables you to streamline your workflow by combining into one operation:

- Profile mapping to mesh regions
- Profile expansion

When you select this option, you are presented with all of the settings used by both the `Expansion` and `Map to Mesh Regions` options.

When the profile transformation is applied, the following operations take place:

1. An initial profile expansion takes place, using the expansion-related settings, except that the `Expand to Full Circle` option is used.

The initial expansion ensures that the original profile covers the whole of the regions of interest, even though the location of the periodic interfaces may differ between the original profile and the CFX mesh. It is assumed that the position of the profile already aligns with the CFX mesh (that is, the blades/hub/shroud are coincident); otherwise, other profile transformations must first be set up in order to align the profile correctly.

2. The profile is mapped onto the selected mesh regions.
3. A second expansion takes place, using your expansion settings. Note that it is assumed that the selected mesh regions are all from the same passage; this is important for ensuring that the `Sector Tag` variable (added in the Expansion process) correctly numbers the passages in the expanded profile — all of the selected mesh regions are given a `Sector Tag` variable set to 1.

For very large meshes or profiles, the initial full-wheel expansion in step 1 could be computationally expensive, since the mapping process needs to match the full-wheel profile with the selected mesh regions. If the initial full-wheel expansion is not necessary in order to ensure that the selected mesh regions are completely covered by the initial profile, you should consider setting up the three individual transformations separately.

---

#### Note:

Note: The mapping statistics displayed for a `Mapped Expansion` transformation refer to the profile immediately before the mapping operation (that is, after the initial profile expansion) and the profile immediately after the mapping operation (that is, before the second expansion). This means that the reported overlap percentages will normally be significantly less than 100%, and the variable ranges may be quite different before and after the mapping.

---

## 7.2.4. Mapping Profile Data to a File

The `Map to File` transformation option enables you export CFX results onto the mesh of a profile file, writing the mapped results to a new profile file. This enables you to send CFX results to another application with results already interpolated to the mesh coordinates used by the other application.

To use this option:

1. Set **Source Profile** to the profile file that contains the results to export. For example, if you want to export CFX results to another application, select a profile file that contains exported CFX results.



2. Set **Write to Profile** to the name of the profile file that will result from this mapping process; this profile file will contain the results of **Source Profile** on the mesh of **Target Profile**.
3. Set **Mapping Definition > Target Profile** to the name of the profile file that defines the mesh to which to map the profile data; the mesh must include face connectivity data. If you are writing, for example, a mapped profile for input to Ansys Mechanical, then the **Target Profile** file contains a definition of the Mechanical mesh. The **Target Profile** file is not overwritten.
4. Leave **Add Node Mapping Status Variable** selected to create variable `Node Mapping Status`, which has a value of 1 for a mapped node and 0 for an unmapped node. This variable can be visualized in CFX-Pre (by coloring by a profile variable) or by importing the profile into CFD-Post.

The alignment setting, **Norm. Vec. Alignment** (normal vector alignment), controls how the source mesh is mapped to the target mesh. In the mapping process, an attempt is made to map each mesh face in the target mesh to one or more mesh faces in the source mesh. The alignment settings provide a way of excluding certain source faces from this mapping process to avoid spurious matches, for example, at corners, or mapping to the wrong side of a thin structure such as a blade.

For each source face and target face, the surface normal can be calculated, and the angle between each source/target face pair can be calculated – the "face normal angle". Source faces are excluded from the mapping for a particular target face if this face normal angle falls outside a specific range, depending on the **Norm. Vec. Alignment** option.

Suppose the **Alignment Tolerance** is `theta`. The default for `theta` is 45 degrees.

- With **Norm. Vec. Alignment** set to `Aligned`, a source face would only be mapped when the face normal angle is between `0 [degree]` and `theta`.
- With **Norm. Vec. Alignment** set to `Opposite`, a source face would only be mapped when the face normal angle is between `180 [degree]` and `( 180 [degree] - theta )`.
- With **Norm. Vec. Alignment** set to `None`, the face normal angle is only used to exclude spurious matches at corners; that is, a source face would only be excluded from mapping when the face normal angle is between 85 and 95 degrees (independent of the **Alignment Tolerance** setting).

The `cfx5profile` script enables you to map profile data without having to use CFX-Pre, for example as part of a batch process. Command line help is available by providing the `-h` argument.

## 7.2.5. Modifying Spatial Fields in Profile Data

You can modify the spatial fields of profile data using the **Modify Spatial Fields** transformation option. For example, you can transform the spatial fields from cylindrical to Cartesian coordinates.

**Spatial Fields Definition > Option** can be set to:

- `Convert Spatial Fields`

This option can be used to:

- Convert the profile data spatial coordinates from cylindrical to Cartesian.

To do this, set **Source Coord. Sys.** to `Cylindrical`. The source profile must have three cylindrical spatial fields (for example, "r", "theta", "z"). The Cartesian coordinates "X", "Y", "Z" in the target profile are calculated from these cylindrical fields as "r cos(theta)", "r sin(theta)", and "z".

---

#### Note:

In CFD-Post, the "Theta" variable is calculated to be dependent on the position of the geometry and is usually offset from theta calculated directly from X- and Y-coordinates ([Theta in the CFD-Post User's Guide](#)). If the source profile has been exported from CFD-Post and uses "Theta" as one of the spatial fields, the calculated Cartesian coordinates might not be what was intended.

---

- Convert the profile data spatial coordinates using a transformation derived from a profile function.

To do this, set **Source Coord. Sys.** to `Coordinate System From Profile` and set **Profile Name** to the name of a profile function.

The profile data being edited must have non-Cartesian spatial fields, and the profile function specified by **Profile Name** must define a mapping from these non-Cartesian fields to Cartesian fields using its **Coordinate System Definition** settings.

For details on the **Coordinate System Definition** settings of a profile function, see [Coordinate System Definition \(p. 426\)](#).

If some points in the edited profile data cannot be successfully mapped to Cartesian coordinates, then the **Ignore Unmapped Points** setting determines the behavior. If the setting is selected, then unmapped points are removed from the target profile data. Otherwise, the transformation process stops with an error message.

**Target Coord. Sys.** can only be set to:

- `Cartesian Coordinates`

This means that the transformed spatial coordinates are always Cartesian. The original cylindrical or custom fields are retained in the profile data as new value fields.

- `Use Selected Variable Fields`

This option swaps the spatial coordinates with the profile variables that you select.

The original spatial coordinates become new profile variables and the selected profile variables become the spatial coordinates.

## 7.2.6. Other Profile Transformation Options

- **Rotation**

These settings work in the same way as the corresponding mesh transformation settings, described in [Transformation: Rotation in the CFX-Pre User's Guide \(p. 141\)](#), except that the `Full Circle` option is not available and that the global coordinate frame is assumed.

- **Translation**

These settings work in the same way as the corresponding mesh transformation settings, described in [Transformation: Translation in the CFX-Pre User's Guide \(p. 143\)](#), except that the global coordinate frame is assumed.

- **Scale**

These settings work in the same way as the corresponding mesh transformation settings, described in [Transformation: Scale in the CFX-Pre User's Guide \(p. 143\)](#), except that the global coordinate frame is assumed.

- **Reflection**

These settings work in the same way as the corresponding mesh transformation settings, described in [Transformation: Reflection in the CFX-Pre User's Guide \(p. 144\)](#), except that the global coordinate frame is assumed.

- **From Existing Transformation**

Select an existing transformation from under **Transformations** in the **Outline** tree view in CFX-Pre.

---

**Note:**

The profile is transformed assuming that the transformations are specified in the same coordinate frame as the profile coordinates.

---

## 7.3. Initialize Profile Data

---

The **Initialize Profile Data** command invokes the **Initialize Profile Data** dialog box, which can:

- Import data from a profile data file, creating a profile user function that can be used in applications such as profile boundary conditions or injection regions.
- Import data from a table data file, creating a table user function that can be used in applications such as operating point input parameter tables.

### To import profile or table data

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 the file containing your profile data.
4. Click **Open**.

The **Select Profile Data File** dialog box disappears. The profile data file is loaded. Basic information about the loaded profiles appears in the **Initialize Profile Data** dialog box.

Note that if the path or filename is altered by typing in **Data File**, the **OK** and **Apply** buttons become unavailable. You must then click **Reload** to re-read the specified file and update the displayed profile information.

5. Specify the **Profile Data Storage** setting.

The options are:

- Use External File

With this option, the simulation gets profile data directly from the profile data file. Any changes to numerical values in the file take effect when the simulation is run. More substantial changes (new or renamed columns, etc.) require re-initialization in CFX-Pre.

- Copy Profile Data Into Setup

With this option, the profile/table data is copied into the current case definition and the profile/table data file is no longer referenced. Any changes to the profile/table data file require re-initialization in CFX-Pre in order to take effect.

6. In the list of profiles that appears near the top of the **Initialize Profile Data** dialog box, select the check boxes for the profiles that you want to initialize.

The Function Status column indicates whether or not initialization would produce a new function or update an existing function for the applicable profile.

In the list, whichever profile is selected, not via the check box but via highlighting the entire list item, the following basic data appears: index names; parameter names and their units; field names and their units.

7. For a profile that has three Cartesian spatial coordinates, you can select the **Visibility** check box to display the profile geometry, enabling you to visually confirm that the profile geometry is correctly aligned and scaled with the boundaries of the CFD model.

When you select a profile's **Visibility** check box in the **Initialize Profile Data** dialog box, or make the corresponding user function (in the **Outline** tree view) visible, you can control the rendering properties of the profile geometry. The appearance of the profile shape will depend on the specified face connectivity.

- If a profile has face connectivity specified, in addition to nodal values, then the profile geometry appears as faces.
- If a profile has no face connectivity, then the profile geometry appears as points based on the nodes.

For details, see [Render Options in the CFX-Pre User's Guide \(p. 148\)](#).

8. Click **OK**.

In the **Outline** tree view, under **Simulation > Expressions, Functions and Variables**, a user function object is listed for each selected profile or table function. In the case of profile functions based on three Cartesian coordinates with visibility turned on, the profile geometry is displayed in the viewer.

---

**Note:**

If you use the `Copy Profile Data Into Setup` option, and there is already existing data for this function stored in the case, the existing data will be overwritten and cannot be restored using the **Undo** command.

---

---

**Note:**

If you use the `Copy Profile Data Into Setup` option, you can update the stored data by using the **Tools > Initialize Profile Data** command again. Alternatively, you can use the `cfx5dfile` script to update the profile data or table data stored within the case using an external data file.

If you use `cfx5cmds` to update CCL, and/or you use `cfx5dfile` to update the profile data or table data stored within the case, then you must, in addition to updating an `mdef` file, also perform the same update to any associated `.def` or `.cfg` files.

---

## 7.4. Initialize FMU Instance

---

Creates a new, or re-initializes an existing, user function of type Functional Mockup Unit Instance (see [Functional Mockup Unit Instance \(p. 433\)](#)). This type of user function provides two-way data transfer capabilities for running a Functional Mock-up Interface (FMI) co-simulation involving CFX and a solver binary provided in a Functional Mock-up Unit (FMU) file.

To create a new function or re-initialize an existing function:

1. Select **Tools > Initialize FMU Instance**.

The **Initialize FMU Instance** dialog box appears.

2. Select **Create new function** or **Re-initialize existing function** as appropriate.
3. If you are creating a new function, specify the **FMU file** and a unique **Function Name**. If you are instead re-initializing an existing function, set **Function Name** to the name of the function that requires re-initialization and ensure that **FMU File** is set appropriately.
4. If you are re-initializing an existing function and the FMU file has changed, or if you have specified a different FMU file, click **Reload** to read the file.
5. Review the variables listed under **FMU Input Variables (required)**. CFX will make the latest values of these variables available to the FMU during the co-simulation. Each variable holds a real number with applicable units.

6. Under **Select the FMU Parameter Variables to use**, hold the **Ctrl** or **Shift** key while you click to multi-select parameters (constants). CFX will pass these parameters to the FMU once: at the start of a co-simulation. Each parameter holds a real number with applicable units.
7. Under **Select the FMU Output Variables to use**, hold the **Ctrl** or **Shift** key while you click to multi-select FMU output variables. CFX will receive these variables from the FMU during the co-simulation. Each variable holds a real number with applicable units.
8. If you select **Edit CFX names for FMU variables with invalid CFX names** then, if any of the input variables, selected parameters, or selected output variables has an invalid name for CFX (either due to having an unsupported character or due to having a name already in use), then, upon clicking **OK** or **Apply**, the **Specify CFX Names For FMU Variables** dialog box will appear, enabling you to edit the names.
9. If you select **View and edit CFX names for all FMU variables** then, upon clicking **OK** or **Apply**, the **Specify CFX Names For FMU Variables** dialog box will appear, enabling you to edit the names.
10. Click **Apply** to apply the settings of the **Initialize FMU Instance** dialog box, or click **OK** to both apply the settings and close the dialog box.

The user function that was specified under **Function Name** is created or updated accordingly.

11. Configure the applicable function details view. For details, see [Functional Mockup Unit Instance \(p. 433\)](#).

For modeling details, see [Coupling CFX to an External Solver: Functional Mock-up Interface \(FMI\) Co-simulation in the CFX-Solver Modeling Guide](#).

## 7.5. Initialize GT-SUITE Coupling


You can make a new GT-SUITE Model object, or re-initialize an existing one, by selecting **Tools > Initialize GT-SUITE Coupling**. This opens the **Initialize GT-SUITE Coupling** dialog box, which has the following settings and controls:

- **Create new GT-SUITE model** and **Re-initialize existing GT-SUITE model**

If you choose to re-initialize an existing model (which currently appears in the **Outline** tree view, set **Model Name** to the name of the GT-SUITE Model object that you want to re-initialize.

- **Model File**

Specify the `.gtm` file that was produced by GT-SUITE.

You can optionally click *Browse*  to browse using the **Select GT-SUITE File** dialog box.

- **GT-SUITE Version**

Select the version of GT-SUITE that produced the `.gtm` file.

---

**Note:**

Selecting the wrong version of GT-SUITE will likely lead to an error message being displayed later when you click **Read Setup From File**.

---

The versions that are available under **GT-SUITE Version** are controlled by a preference. For details, see [User Interface \(p. 86\)](#).

- **Read setup from related .dat file if available**

If a data (`.dat`) file has already been written by GT-SUITE alongside the `.gtm` file, and the data file is up-to-date with the `.gtm` file, you can select this option to read the GT Model settings from the data file instead of from the `.gtm` file, usually saving time as a result.

- You can optionally select **Specify GT-SUITE Installation** and set **GT-SUITE Installation** to the name of the installation directory, which is typically similar to the following on a Windows system:

```
C:\Program Files (x86)\GTI
```

You can optionally click *Browse*  to browse using the **Select Directory** dialog box.

By specifying a path, you override the default path set by environment variable `GTIHOME`. The specified installation directory must contain the version-specific subdirectory indicated by the **GT-SUITE Version** setting.

- **Read Setup From File** button

When you click this button, CFX-Pre runs GT-SUITE to read the model settings from the `.gtm` (or data) file and then adds several tabs to the **Initialize GT-SUITE Coupling** dialog box that contain additional settings. These tabs are described in the subsections below.

- **OK, Apply, Close** buttons

- The **OK** button applies the settings and closes the dialog box.
- The **Apply** buttons applies the settings and leaves the dialog box opened.
- The **Close** button closes the dialog box, without retaining the setting configuration of the dialog box and without applying any settings (although the settings could already be applied if you have previously clicked **Apply**).

After you have configured all the applicable settings in the dialog box, including its tabs, click **OK** or **Apply** to generate the following objects, which appear in the **Outline** tree view:

- Functions of type `GT-SUITE Patch`.

For details, see [GT-SUITE Patch Functions \(p. 435\)](#).

- A GT-SUITE Model object for each initialized or re-initialized model. For details, see [GT-SUITE Model Details View \(p. 443\)](#).

For details on GT-SUITE modeling, see [Coupling CFX to an External Solver: GT-SUITE Coupling Simulations in the CFX-Solver Modeling Guide](#).

The following topics are discussed:

[7.5.1. Boundary Association Tab](#)

[7.5.2. Material Association Tab](#)

[7.5.3. Turboshaft Setup Tab](#)

[7.5.4. Monitoring Tab](#)

[7.5.5. Function Names Tab](#)

## 7.5.1. Boundary Association Tab

On this tab, you set up a mapping between GT-SUITE interfaces and CFX boundaries.

You can set **Option** to one of the following:

- `Specify Boundary Names`

With this option, you can map GT-SUITE interfaces to CFX boundaries by selecting and/or entering CFX boundary names in the CFX Boundary column of the two-column mapping table that appears on the **Boundary Association** tab.

- `No Boundary Association`

With this option, no GT-SUITE interfaces are mapped to CFX boundaries as part of the GT-SUITE Coupling initialization process. In this case, you must set up the mapping manually later. Mappings can be set up later by editing each GT-SUITE Patch function associated with the GT-SUITE model, to specify which CFX boundary will be used to provide quantities to the corresponding GT-SUITE interface. For details, see [GT-SUITE Patch Functions \(p. 435\)](#).

You can select **Allow new boundary names** to enable you to name non-existent CFX boundaries, facilitating the mapping process. If you do select this check box then you must create any such boundaries later, unless you also select the **Create missing boundaries** check box, in which case CFX-Pre, upon clicking **OK** or **Apply**, automatically creates any non-existent mapped boundaries. After clicking **OK** or **Apply**, you are given the chance (in the **Specify Locations for New Boundaries** dialog box) to specify the relevant domain for each new boundary that will be created, and, optionally, the location for the boundary. You should later check the settings of all automatically created CFX boundaries.

If you select **Modify boundaries to use GT-SUITE coupling** then CFX-Pre, upon clicking **OK** or **Apply**, automatically changes the settings of the mapped CFX boundaries so that each mapped boundary interacts with its associated GT-SUITE interface. You should later check the settings of all automatically modified CFX boundaries.

If **Allow new boundary names** is selected, the **Matching and Selection Options** settings allow you to choose which names are available for selection in the CFX Boundary column. If you click the **Reset matches and selections in table** button, CFX-Pre will disregard any CFX boundary names that are already specified and re-calculate the best matches from the currently specified list of boundary names and/or mesh region names.



## 7.5.2. Material Association Tab

On this tab, you set up a mapping between GT-SUITE material species and CFX material species.

You can set **Option** to one of the following:

- Create Material From Species

This option enables you to create a new Variable Composition mixture in CFX that incorporates all of the listed CFX species.

1. Click **Import Species From File** and use the **Select Library Data to Import** dialog box to specify a CCL material library file that contains materials for CFX.
2. Set **Material Name** to a unique name for the new mixture.
3. If the names of the imported materials do not match the names of the GT-SUITE CFD species (in the **GT-SUITE Species** column), you must manually select the corresponding **CFX Component** names.

- From Existing Material

This option is similar to the `Create Material From Species` option, except that you set **Material** to the name of an existing CFX mixture, and the species of that mixture then become available for you to manually select as the corresponding **CFX Component** names.

- Specify Component Names

This option does not create a new, or reference any existing, CFX mixture. This option allows you to manually specify component names. The corresponding material and components must be created separately (if they do not already exist) to complete the setup.

## 7.5.3. Turboshaft Setup Tab

If the selected GT-SUITE Model contains a turboshaft, you can use the **Turboshaft Setup** tab to set up the transfer of torque to GT-SUITE and the transfer of shaft speed from GT-SUITE.

The **Turboshaft Setup** tab has the following settings:

- List

The list contains the turboshafts defined in the GT-SUITE setup. The settings below apply to the selected turboshaft.

- **Axis Definition**

Select the axis for the shaft rotation.

**Option** can be set to:

- `Coordinate Axis`

Set **Rotation Axis** to one of the coordinate axes.

---

- Two Points

Specify two points using Cartesian coordinates. **Rotation Axis From** serves as the origin. The direction of the rotation axis is derived from the relative position of **Rotation Axis To**. The direction of positive Theta is implied by the right hand rule.

- **Torque** check box

You can select this check box to set up torque details for the selected turboshaft as part of the GT-SUITE Coupling initialization process. If you do not select this check box, or if the check box is unavailable (because the necessary domain or boundaries do not exist), you can set up the torque details later, by editing the related GT-SUITE Turboshaft function.

**Option** can be set to:

- Boundary List

Set **Boundary List** to a comma-delimited list of CFX boundaries. The net torque on these boundaries is sent to the GT-SUITE solver during a coupled run.

- Expression

Set **Expression** to a CEL expression for torque. The computed value of torque is sent to the GT-SUITE solver during a coupled run.

- **Modify domain motion to use GT-SUITE Coupling** check box

You can select this check box to specify a comma-delimited list of domains that should use the shaft speed received from GT-SUITE. During the GT-SUITE Coupling initialization process, each of the listed domains will be updated to have its domain motion set to use GT-SUITE Coupling. If you do not select this check box, you can set up domain motion later.

Set **Domain List** to a comma-delimited list of domains intended to receive shaft speed data from GT-SUITE.

## 7.5.4. Monitoring Tab

On this tab, you can optionally specify solution monitors for GT-SUITE interfaces and/or, if applicable, GT-SUITE Turboshaft functions. The values of these monitors are the variable values supplied to CFX by GT-SUITE.

These monitors are created as part of the GT-SUITE initialization process and appear in the CFX-Solver Manager as the solution proceeds.

---

**Note:**

You can alternatively or also add solution monitors after GT-SUITE initialization; for details, see [Working with Monitors](#) (p. 347).

---

The **Monitoring** tab has the following settings:

- **Add monitors for GT-SUITE Interfaces**

Selecting this check box makes settings available for specifying the GT-SUITE interfaces and the GT-SUITE variables to monitor.

- **Add monitors for Turboshaft**

Selecting this check box makes settings available for specifying the GT-SUITE Turboshaft functions and the GT-SUITE variables to monitor.

### 7.5.5. Function Names Tab

On this tab, you provide names for special CFX functions, known as *GT-SUITE Patch functions* and *GT-SUITE Turboshaft functions*, that will be created as part of the GT-SUITE Coupling initialization process.

Each GT-SUITE Patch function provides CEL access to values on the corresponding GT-SUITE interface. In this context, a GT-SUITE interface is also known as a *GT-SUITE patch*.

A GT-SUITE Patch function can be used to retrieve values from a GT-SUITE interface and apply those values to a setting of the corresponding CFX boundary.

Each GT-SUITE Patch function also serves to send boundary information from CFX to the applicable GT-SUITE patch, according to the **Boundary** and **GT-SUITE Patch** specifications, respectively, in the function settings.

For details on GT-SUITE Patch function settings, see [GT-SUITE Patch Functions \(p. 435\)](#).

Each GT-SUITE Turboshaft function provides CEL access to the torque transferred to, and the shaft speed received from, the corresponding GT-SUITE turboshaft.

A GT-SUITE Turboshaft function can be used to retrieve shaft speed from a GT-SUITE turboshaft and, using the rotation axis specified in the function settings, apply angular velocity to the motion of the corresponding CFX domain.

Each GT-SUITE Turboshaft function also serves to send the net torque on the shaft (whether from forces on boundaries or from a CEL expression) from CFX to the applicable GT-SUITE turboshaft, according to the **GT-SUITE Torque** specification in the function settings.

For details on GT-SUITE Turboshaft function settings, see [GT-SUITE Turboshaft Functions \(p. 436\)](#).

## 7.6. Macro Calculator

The macro calculator in CFX-Pre is very similar to the one in CFD-Post. For details, see [Macro Calculator in the CFD-Post User's Guide](#). There are some minor differences between the two, however. For instance, an additional widget type, `Location`, is available in the CFX-Pre macro calculator. This enables the selection of mesh regions within the macro. An example of how to use this widget type is:

```
# Macro GUI begin
#
# macro name = StaticMixer
# macro subroutine = test
# macro report file = test_report.html
#
# macro parameter = Domain Location
# type = Location
# value list = 3d composites, 3d primitives
#
```

A number of standard lists are available for this widget. The valid `value list` entries are as follows:

- `2d primitives / 3d primitives`: all primitive 2D and 3D regions for the model
- `internal 2d primitives`: all primitive 2D regions that are internal to the model
- `composites`: all composite regions
- `2d composites / 3d composites`: all 2D and 3D composite regions
- `assemblies`: all assemblies

Also, predefined macros are not supplied for CFX-Pre the way they are in CFD-Post. For details, see [Predefined Macros in the CFD-Post User's Guide](#).

## 7.7. Solve

Available in stand-alone mode for the current definition of the case, you can use the **Solve** option to do the following:

- From **Start Solver**, you can:
  - Select **Define Run** to write the CFX-Solver input file and start the CFX-Solver Manager
  - Select **Run Solver** to write the CFX-Solver input file and start the CFX-Solver
  - Select **Run Solver and Monitor** to write the CFX-Solver input file and start both the CFX-Solver and the CFX-Solver Manager.
- From **View in CFD-Post**, you can write the CFX-Solver input file and start CFD-Post.
- From **Write Solver Input File**, you can write the CFX-Solver input file.

### 7.7.1. Write Solver Input File Command

A CFX-Solver input file contains information (such as physics, mesh) required to execute a case in CFX-Solver to solve physics.

1. Select **Tools > Solve > Write Solver Input File** from the menu bar or click *Write Input Solver File*



The **Write Solver Input File** dialog box appears.

2. Select a location to which to save the file.
3. Under **File name**, type the name of the file.
4. Click **Save**.

If the filename assigned is the same as an existing filename in the same location, select **Overwrite** to overwrite the original file, **Re-select** to specify a new filename, or **Cancel** to cancel the writing of the `.def` file.

## 7.8. Applications

---

Available in stand-alone mode, these commands immediately load CFX-Solver Manager or CFD-Post.

## 7.9. Quick Setup Mode

---

Quick Setup Mode is used to quickly specify cases that involve simple physics. For details, see [Quick Setup Mode \(p. 469\)](#).

## 7.10. Turbo Mode

---

Set up certain turbomachinery cases quickly and easily using Turbo mode. For details, see [Turbomachinery Mode \(p. 473\)](#).

---

## Chapter 8: CFX-Pre Extensions Menu

---

The **Extensions** menu provides access to any customized extensions available to CFX-Pre.



---

## Chapter 9: CFX-Pre Help Menu

---

The **Help** menu has the following commands:

### **CFX-Pre**

Opens [CFX-Pre User's Guide \(p. 1\)](#).

### **Contents**

Opens a page that lists various help resources associated with this product.

### **[various help resources]**

Each of these commands goes directly to a particular help resource.

### **Ansys Product Improvement Program**

Provides a brief description of, and enables you to control participation in, the Ansys Product Improvement Program.

### **About CFX-Pre**

This gives the point releases and software patches that are installed.

### **Help on Help**

Opens documentation about the help system: [Help On Help in the CFX Introduction](#).





---

# Chapter 10: Importing and Transforming Meshes

---

CFX-Pre can import meshes from a wide range of sources. Once imported, you can position and scale each mesh as required (as described in [Transform Mesh Command \(p. 140\)](#)).

You can import more than one mesh per CFX-Pre simulation. After you have imported all your meshes and created all your domains, the domains should be joined together, either by gluing them together, or by using domain interfaces. For details, see [Gluing Meshes Together \(p. 147\)](#) and [Domain Interfaces \(p. 205\)](#).

This chapter describes:


- 10.1. Importing Meshes
- 10.2. Mesh Tree View
- 10.3. Deleting Meshes and Mesh Components from the Tree View
- 10.4. Transform Mesh Command
- 10.5. Gluing Meshes Together
- 10.6. Mesh Editor
- 10.7. Render Options
- 10.8. Mesh Topology in CFX-Pre
- 10.9. Advanced Topic: cfx5gtmconv Application

Additional information on assemblies, primitive regions, composite regions, and the regions that are created when importing meshes is available in [Mesh Topology in CFX-Pre \(p. 152\)](#).

## 10.1. Importing Meshes

---

Meshes are imported via the **Import Mesh** dialog box, which is accessible in several ways:

- By selecting **File > Import > Mesh**
- By right-clicking the `Mesh` branch in the tree view and selecting **Import Mesh** from the shortcut menu
- By selecting *Browse*  when setting the filename for a mesh (for example, in Turbomachinery mode).

You can multi-select mesh files by holding the **Ctrl** key while you click the filenames.

Import options may appear on the **Import Mesh** dialog box, depending on the type of mesh being imported. Some common import options are described next. Other options that are specific to particular mesh formats are discussed in [Supported Mesh File Types \(p. 123\)](#).

### 10.1.1. Importing Multiple Meshes

It is possible in CFX-Pre to import multiple mesh file to construct an appropriate model for your simulation. Each mesh imported is represented in the **Mesh** part of the **Outline** tree view by a unique identifier based on the name of the mesh file imported.

In general, a mesh file is represented by the filename of the file imported without any preceding path (for example, if you imported `C:\Directory\File.cmdb`, this will be represented in the tree as `File.cmdb`). If after transforming the filename in this way the transformed name is already present in the tree, either because this is an earlier import of the same file or another file with the same name has been imported from a different directory, the new file will be labeled with a suffix, such as `File.cmdb(1)` for example.

If multiple mesh files are transformed in such a way that the result of the transformation glues the two files together or the files are explicitly glued together, the original mesh file entries will no longer appear under the **Mesh** entry as file names, but the resulting Principal 3D regions will appear under a **Merged Meshes** item under **Mesh**.

### 10.1.2. Common Import Options

#### 10.1.2.1. Mesh Units

This option is displayed depending on the file type selected. The units selected on the **Import Mesh** dialog box are the units used to import the mesh and are the default units for transforming mesh assemblies using the **Mesh Transformation Editor** dialog box. These units are local to the mesh import and transformation options and do not affect either the solution units or the units set under **Edit > Options**. For details, see:

- [Setting the Solution Units \(p. 288\)](#)
- [Units \(p. 89\)](#).

CFX-Pre will attempt to determine the units used in a mesh file and convert them to the specified units during import. For example, a mesh of 1000 units long, with units in the mesh file of mm, will appear in CFX-Pre as 1 m long, if units of m are set on the **Import Mesh** dialog box. If CFX-Pre cannot determine the units used in the mesh file, then in this example the mesh would appear as 1000 m long.

#### 10.1.2.2. Assembly Prefix

This is the name used to prefix the assemblies that are created when the mesh is imported. A number suffix is added to the second, and any subsequent meshes, using the same assembly prefix, so that each assembly is named uniquely.

#### 10.1.2.3. Primitive Strategy

This setting enables you to control the names of split regions.

The following options are available:

**Standard** - Select this option so that the name of each split region starts with "Primitive 2D" or "Primitive 3D". For example, this option splits "My Region Name" into "Primitive 2D A" and "Primitive 2D B".

**Derived** - Select this option so that the name of each split region is derived from the name of the region that is being split. For example, this option splits "My Region Name" into "My Region Name A" and "My Region Name B".

#### 10.1.2.4. Ignore Invalid Degenerate Elements

If your mesh import fails because of invalid degenerate elements, then you can enable this toggle. However, your mesh may not be valid for use in the CFX-Solver. You may have to fix or remove the degenerate elements in the software used to generate the mesh.

#### 10.1.2.5. Duplicate Node Checking

Duplicate Node Checking is off by default and, in general, need not be selected.

Nodes within the specified relative tolerance are equivalenced into a single node (duplicate node removal). The default tolerance of 1e-04 is sensible and you should not change it. The relative tolerance is based on the **local** mesh length scale, so by default nodes within 0.001% of the average mesh edge length of all edges connected to a node will be equivalenced.

### 10.1.3. Supported Mesh File Types

Mesh file types supported by CFX-Pre are:

10.1.3.1. Ansys Meshing Files

10.1.3.2. CFX-Mesh Files

10.1.3.3. CFX-Solver Input files

10.1.3.4. ICEM CFD Files

10.1.3.5. Ansys Files

10.1.3.6. Fluent Files

10.1.3.7. CGNS Files

10.1.3.8. CFX-TASCflow Files

10.1.3.9. CFX-4 Grid Files

10.1.3.10. CFX-BladeGenPlus Files

10.1.3.11. PATRAN Neutral Files

10.1.3.12. IDEAS Universal Files

10.1.3.13. GridPro/az3000 Grid Files

10.1.3.14. NASTRAN Files

10.1.3.15. Pointwise Gridgen Files

### 10.1.3.16. User Import

---

**Note:**

Users of the DesignModeler, Meshing application, and Ansys CFX products should refer to [Named Selections and Regions for Ansys CFX in the \*Meshing User's Guide\*](#) for important information about region definitions.

---

### 10.1.3.1. Ansys Meshing Files

Ansys Meshing files of the form `.cmdb` and `.dsdb` can be imported.

---

**Note:**

- You must have Ansys Workbench installed in order to import Ansys Meshing files (`.cmdb` and `.dsdb`) into CFX-Pre or CFD-Post.
  - CFX-Pre does not support importing meshes from `.cmdb` files generated by the Meshing application prior to Release 11.0.
  - When CFX-Pre is run outside of Ansys Workbench, and when importing or loading `.cmdb` or `.dsdb` files into CFX-Pre, meshes generated with the "Assembly Mesh" method cannot be read.
- 

You can specify an assembly prefix. For details, see [Common Import Options \(p. 122\)](#).

There are import settings that are specific to Ansys Meshing files.

The **Model(s) To Read** setting defaults to `ALL`, which specifies that all models are to be imported from the Ansys Meshing file. However, if you load a `cmdb/dsdb` file that has multiple models in it, you can specify which models to load.

---

**Note:**

You must click  next to the **Model(s) To Read** setting before the other models will appear.

---

#### 10.1.3.1.1. Named Selections

Named selections are aliases for collections of regions. When importing a mesh from an Ansys Meshing file, you can preserve these named selections based on where they were created:

- **CFX Mesh Names** for regions defined in CFX-Mesh.
- **Simulation Names** for named selections generated in the Mechanical application and Ansys Workbench Meshing.
- **Symmetry Names** for named selections of 2D symmetry and periodic regions generated in the Mechanical application and Ansys Workbench Meshing.

- **Part Manager Names** for named selections generated in DesignModeler or other CAD systems that are not written to the `.cddb` file by the meshing application.
- **Fall Back to Part Manager Names** for using named selections generated directly by DesignModeler or other CAD systems as a fall back if no CFX-Mesh, Simulation or Symmetry named selections are found.

When importing a mesh from an Ansys Meshing file, you can generate pairs of new named selections, each pair representing the two sides of certain existing named selections.

To generate new named selections:

1. Select **Generate New Named Selections**.
2. To generate new named selections from direct connection sides, select **From Direct Connection Sides**.
3. To generate new named selections from contact sides, select **From Contact Sides**.
4. If you want to specify the name suffixes for each side, select **Specify Names** and provide the suffixes for side 1 and side 2. The user interface ignores any leading whitespace. To add an initial space to each suffix, select **Add Space Before Suffix**.

To form the names of the new named selections, the specified suffixes are each appended to the name of the existing named selection.

If you do not specify suffixes, naming is automatic.

A named selection is not generated if its planned name is already in use.

In order for an existing named selection to be eligible for use in generating new named selections, the existing named selection must consist of exactly two primitive regions. The two new named selections correspond to the two primitive regions.

Also see [Named Selections Involving Interior Faces in the Meshing User's Guide](#).

### 10.1.3.1.2. Contact Detection Settings

The **Contact** check box, when selected, makes contact detection settings available.

When importing Ansys Mesh files (`.cddb` / `.dsdb` files), it is possible to select **Detection Method > Read** to read contact information from the file or to select **Detection Method > Detect** to use the contact detection methods to determine whether regions within the mesh are "in contact" with each other. CFX-Pre uses the Mechanical application contact detection methods to determine which mesh volumes should be placed within each mesh assembly and which 2D regions are connected.

The **Read Method** setting can be set to `Modify Topology` meaning that CFX-Pre will create connections between pairs of regions according to the contact information read from the Ansys Meshing file. The topology is modified as part of this process, which can result in two or more bodies becoming part of a single assembly. When using the `Create Regions` option, the contact information read from the file will result in each contact being represented as composite regions on each side of the contact definition allowing easy identification of the contact regions. Using the `Create Regions` option will not modify the underlying topology of the mesh.

The **Detection Between** setting can be set to **Bodies** or **All Contact**. When using the **Bodies** option, 2D regions will be matched between different bodies. This is the default option and should result in bodies that are "close" to one another being placed in the same mesh assembly. If automatic domain interface generation is selected, interfaces will be generated between such regions. When using the **All Contact** option, CFX-Pre will still recognize contact between discrete bodies, but in addition, it will look for contact between 2D regions within the same "body", or "volume". This can result in unexpected behavior, such as adjacent surfaces being considered "in contact" and hence this is not the default option, but in some cases, where there are non-matched 2D mesh regions within a mesh volume, it can help generate "internal" interfaces.

The tolerance that is used in detecting contact can be altered and it is possible to define it relative to the local geometry size, or as an absolute spatial value.

If CFX-Pre is set to read contact information from the file, then it will import only connections that connect two single regions. Connections connecting multiple regions to a single region, or multiple regions to multiple regions, will be ignored. Also, contact detection works only within a single file; CFX-Pre will not read or detect contact between meshes that are imported from different files.

If the Meshing application is set to generate connections automatically, you can set the **Global Contact Setting** option to **Group By > None** to generate only single region to single region connections. For more details, see [Generation of Contact Elements in the Meshing User's Guide](#).

### 10.1.3.2. CFX-Mesh Files

The CFX-Mesh (.gtm, .cfx) files are native for CFX-Pre; therefore, all information in such a file is read in by the import process. There are no options needed to control the reading of these files.

---

**Note:**

Only .cfx files that are version 11.0 or newer are supported.

---

### 10.1.3.3. CFX-Solver Input files

CFX-Solver files include CFX-Solver input (.def), results (.res), transient results (.trn), and backup (.bak) results files. There are no options specific to importing CFX Def/Res, files but the general advanced options are described in [Common Import Options \(p. 122\)](#).

For additional information on the regions created in CFX-Pre when CFX-Solver files are imported, see [Mesh Topology in CFX-Pre \(p. 152\)](#).

### 10.1.3.4. ICEM CFD Files

ICEM CFD files are of the form .cfx, .cfx5, .msh. There are no import options specific to ICEM CFD files; however the [Common Import Options \(p. 122\)](#) apply.

### 10.1.3.5. Ansys Files

Ansys files are of the form .cdb or .inp. There are no import options specific to Ansys files; however the [Common Import Options \(p. 122\)](#) apply.

Only `.cdb` files can be imported into CFX-Pre. If you have an Ansys `.db` file, you can convert it to a `.cdb` file in Ansys by:

1. Opening the Ansys database in Ansys.
2. Issuing the `ALLSEL` command to select everything.
3. Issuing the `CDWRITE, DB` command to write the `.cdb` file.

To get a list of all element types (`ET`) / `keyops` (`KEYOP`) that are supported by mesh import, you can run the following from the operating system command line:

```
<CFXROOT>/bin/<OS>/ImportANSYS.exe -S
```

---

### Note:

Before executing the `CDWRITE` command, verify that the data base has a separate named component of 2D MESH200 elements for each surface that will require a boundary condition. Delete any MESH200 elements that are not members of named components. To define specific 3D regions, create a 3D named component of 3D elements. The component names will appear in CFX-Pre as defined regions.

---

## 10.1.3.6. Fluent Files

Fluent files of the form `.cas` and `.msh` can be imported.

---

### Note:

When importing meshes from Fluent files in CFX-Pre, Release 12.0 (or later), the topology and naming of regions may not be the same as those generated by importing these meshes into previous releases. As a result, session files generated in CFX-Pre Release, 11.0 (or earlier) that import meshes from Fluent files may generate errors when loaded into CFX-Pre, Release 12.0 (or later).

---

### 10.1.3.6.1. Override Default 2D Mesh Settings

#### 10.1.3.6.1.1. Interpret 2D Mesh as

##### 10.1.3.6.1.1.1. Axisymmetric

This option enables you to create a 3D geometry by extruding a 2D geometry through a specified rotation angle in the third dimension.

#### Number of Planes:

This value enables you to create additional planes, arranged in the extruded direction, to create a 3D problem. This will increase the number of elements in the extruded direction, but does not change the enclosed angle of the mesh.



**Angle (deg):**

This is the angle through which the original 2D mesh is extruded.

**Remove Duplicate Nodes at Axis:**

This check box enables you to choose to have the duplicate node removed from the axis of an axisymmetric case upon import.

**10.1.3.6.1.1.2. Planar**

This option enables you to create a 3D geometry by linearly extruding a 2D geometry in the third dimension.

**Extrude Distance:**

This is the distance through which the geometry is extruded in the third direction.

For further advice on how to model 2D problems in CFX, refer to [Modeling 2D Problems in the CFX-Solver Modeling Guide](#).

**10.1.3.7. CGNS Files**

CGNS files are of the form: `.cgns`. Applicable import options are:

- [Ignore Invalid Degenerate Elements \(p. 123\)](#)
- [Duplicate Node Checking \(p. 123\)](#)

**10.1.3.7.1. Importing CGNS files into CFX****10.1.3.7.1.1. Method**

After you create a new case or open an existing case in CFX-Pre, you can import meshes from CGNS (CFD General Notation System) files of version 2.4 or 3.X (up to 3.3). Extensions for such files are typically `.cgns` or `.cgn`. To import a CGNS file, select the file and, if necessary, change the options used to import the mesh under the **Advanced Options** section.

**10.1.3.7.1.2. Base (Base\_t)**

The top-level object in a CGNS file is a container called a base, a CGNS file that can contain multiple bases. What a base contains is user defined so that CFX-Pre allows all bases to be read by one import, or single bases to be read by separate imports.

**10.1.3.7.1.3. Zone (Zone\_t)**

Each base contains one or more zones. For each base read, the import process reads all zones, provided they are 3D dimensional (structured or unstructured zones are supported).

- Grids can be read in single or double precision.
- Zones may be specified in Cartesian or Cylindrical coordinates. Other coordinate systems are not currently supported.

#### 10.1.3.7.1.4. Elements (ElementSection\_t)

Element sections can be imported as regions of interest or ignored. How this is done is controlled by the user interface - you must understand which behavior you want to see. It may be useful to import the element sections, for example, if the file has been written with all faces (2D elements) in a boundary patch as a separate element section, which could be useful for setting up the problem in CFX-Pre. Similar scenarios can be imagined in 3D element sections or even mixed element sections.

#### 10.1.3.7.1.5. Element Types Supported

Supported 3D elements (TETRA\_4, PYRA\_5, PENTA\_6 and HEXA\_8). Other 3D elements can be read but are reduced to the lower order elements (that is, TETRA\_10 is translated to TETRA\_4 and then this is imported).

Supported 2D elements (TRI\_3 and QUAD\_4). Other 2D elements can be read but are reduced to the lower order elements (that is, TRI\_6 is translated to TRI\_3 and then is imported).

The vertices of 2D elements should ideally be based on the node indices as are used for to define the 3D elements.

It is preferable to define 2D elements with parent information so that mapping from 2D elements to 3D elements does not have to be determined by the process, therefore, reducing import times.

#### 10.1.3.7.1.6. Boundary Conditions (BC\_t)

Boundary conditions are processed but physical setup information (for example, equations) is ignored. The facility for importing the CGNS files into CFX (CFX-Pre) is a mesh (grid) importer, not a physics importer.

No physics information is imported. Boundary condition locations are read because the collections (regions) of mesh elements the condition is defined upon are required for ease of use and correct physics setup in CFX.

It is quicker to read boundary conditions when they are defined as a range of elements (ElementRange) or a list of elements (ElementList), rather than a range of nodes (PointRange) or a list of nodes (PointList). The latter may also be read, but the nodes referenced must also be used by higher-dimension elements (for example, 3D elements) for correct interpretation.

#### 10.1.3.7.1.7. Families (Family\_t, FamilyBC\_t, FamilyName\_t)

Families are read and, in general, imported as composite regions (groupings) of underlying primitive regions.

#### 10.1.3.7.1.8. Grid Connectivity (GridConnectivity\_t and GridConnectivity1to1\_t)

Grid connectivity can be read but with certain restrictions.

- If the interface is read from a GridConnectivity1to1\_t node or is a read from a GridConnectivity\_t node and is of type Abutting1to1, importing of the node mapping is attempted.

- If the node mapping cannot be established or you request that the two sides of the interface are imported as separate regions.

Other interface types are always imported as two separate regions.

#### **10.1.3.7.1.9. CGNS Data Ignored**

The CGNS Mid Level Library Documentation Page ([http://www.grc.nasa.gov/WWW/cgns/CGNS\\_docs\\_current/midlevel/index.html](http://www.grc.nasa.gov/WWW/cgns/CGNS_docs_current/midlevel/index.html)) details the interface used for reading CGNS files within CFX-Pre. The following high level headings used within the document are ignored.

- Simulation Type
- Descriptors
- Physical Data
- Location and Position<sup>[1]</sup>
- Auxiliary Data
- Solution Data
- Equation specification
- Time Dependent Data

#### **10.1.3.7.2. Prefix regions with zone name**

This check box determines whether or not each imported region is prefixed with the name of the zone within which it is defined.

#### **10.1.3.7.3. Create Regions From: Element Sections**

Each element section that specifies the topology of elements within the CGNS file may or may not imply a grouping of these elements that is important. If the grouping of elements within each element section is important, this option should be selected so the grouping is preserved within CFX-Pre.

Element sections can be 2D or 3D or a mixture of both, and as such can form 3D regions or 2D regions in CFX-Pre.

The way they are grouped depends on vendor interpretation of the CGNS standard.

#### **10.1.3.7.4. Create Regions From: Boundary Conditions**

This check box determines whether or not to import boundary conditions as regions.

---

[1] Rind Data is processed but not imported.

### 10.1.3.7.5. Create Regions From: Families

This check box determines whether or not to import families of elements, or faces as regions.

### 10.1.3.7.6. Create Regions From: Connectivity Mappings

This check box determines whether or not to import zone interfaces (that is, 1-to-1 and GGI connections) as regions.

### 10.1.3.7.7. Example of Create Regions From

Consider a CGNS file with one zone, *Zone 1*, made up of four element sections (*ES1* and *ES2* defining the 3D elements, and *ES3* and *ES4* defining the 2D elements). It also contains 2D boundary conditions *BC1* and *BC2*.

These element sections, *ES1* and *ES2*, could consist of, for example, hexahedral elements in *ES1* and tetrahedral elements in *ES2*. In this case, the groupings of elements into the first two element sections appears to be due to their topological identity. However, this may or may not be the case and you must decide as to whether importing these groupings is important.

In this case, it may be that *ES1* and *ES2* should be combined by clearing the **Create Regions From: Element Sections** option. Another possibility is that *ES1* may be a subregion of mesh that should be kept separate (that is, it will be set up as a subdomain). If that were the case, **Element Sections** should be selected.


If *BC1* is defined on all the faces in *ES3* and *BC2* is defined on all the faces in *ES4*, then it will probably not be necessary to select **Boundary Conditions** if **Element Sections** is selected, as this would introduce complexity in the region definitions (that is, composites would be defined). However if the groupings of *ES3* and *ES4* are different from the groupings in the boundary conditions then **Create Regions From: Boundary Conditions** should be selected.

### 10.1.3.7.8. Read Only One CGNS Base

When this toggle is selected, a mesh is read from a single base specified by the **CGNS base to read** number. If your CGNS file contains only a single base, you should leave the number set to 1. If it contains more than one base, you should specify the base number from which to read. If the base number specified does not exist, an error will be raised. If it does not contain a valid mesh then a mesh will not be imported.

---

#### Note:

You must click  before you can specify which base to read.

---

If you disable the **Read Only One CGNS Base** toggle, then CFX-Pre will look for meshes in all bases and import them. If multiple assemblies are imported and they overlap, then the mesh will be invalid within CFX-Pre unless assemblies are transformed in some way.

For details, see [SplitCGNS.exe in the CFX Reference Guide](#). This is a program that splits a CGNS file into multiple problem files.

### 10.1.3.8. CFX-TASCflow Files

CFX-TASCflow mesh files are of the form `.grd` or are simply named `grd`. You may receive warning messages when importing a CFX-TASCflow mesh file: these will usually tell you which regions have not been imported. The sections below indicate the situations when a warning message may occur.

- If **Convert 3D Region Labels to Regions** is selected, then the 3D Region labels in the `.grd` file are imported as individual 3D Regions. The default setting omits all 3D Region labels.
- If **Ignore One-to-One Connections** is selected, then one-to-one contiguous grid connections are deleted on import. You would then have to recreate the connections in CFX-Pre. There are very few cases when you would want to enable this toggle.
- Select the file type for the imported mesh from the **GRD File Format Type** drop-down. You can select from `Formatted`, `Unformatted` or `Unknown`. If you select `Unknown`, CFX attempts to determine the file format before importing the mesh.
- If **Retain Block Off** is selected, then "user defined" elements that are blocked off in the mesh file are not imported into CFX-Pre. If not selected, then "user defined" objects are ignored and the elements are included in the imported mesh (rarely desired).

Additional information is available in:

- [Ignore Invalid Degenerate Elements \(p. 123\)](#)
- [Duplicate Node Checking \(p. 123\)](#).

#### 10.1.3.8.1. Convert 3D Region Labels to Regions

This toggle controls 3D region import from the `.grd` file only. When selected, 3D regions in the `.grd` file will be imported into separate 3D primitives in CFX-Pre. If you do not select this option, all mesh elements will be imported into a single 3D primitive that is uniquely named by the import process. 3D regions defined in the `.gci` and `.bcf` files are always imported.

#### 10.1.3.8.2. Grid Connections Processed (in the `.grd` file)

When importing CFX-TASCflow meshes, the only grid connections that are imported automatically are "many-to-one" contiguous topology connections that are specified as one-to-one node pairings.

"Many-to-one" contiguous topology connections that involve any number of many-to-one node groupings are ignored and a warning message is issued; however, the two sides of the connection are preserved as a pair of 2D regions on which a GGI Connection can be defined. You should recreate the connection in CFX-Pre using a Fluid-Fluid Domain Interface. For details, see [Creating and Editing a Domain Interface \(p. 205\)](#). In some cases, if you have not created regions in CFX-TASCflow on each side of an interface, you will not be able to recreate it in CFX-Pre because there

will be no region available for selection. If this occurs, you should explicitly create regions in CFX-TASCflow before importing the mesh into CFX-Pre.

---

### Important:

Some Ansys TurboGrid grids contain many-to-one node groupings. These will not be imported into CFX-Pre. You need to know if your grid contains these connections and then recreate them in CFX-Pre using Fluid-Fluid Domain Interfaces.

---

"Many-to-one" periodic topology connections are always removed with a warning message issued. You should recreate the connections using a periodic domain interface. For details, see [Creating and Editing a Domain Interface \(p. 205\)](#).

The regions associated with periodic boundary conditions are imported, but you will need to assign the regions to a periodic domain interface.

#### 10.1.3.8.3. Grid Embedding

Embedded grids, along with the parent grid, are automatically imported into separate assemblies in CFX-Pre. The many-to-one topology connections on the interface between the embedded grid and the parent grid will be removed and a warning issued. You will need to create fluid-fluid domain interfaces between the embedded grid and the parent grid. For details, see [Creating and Editing a Domain Interface \(p. 205\)](#).

#### 10.1.3.8.4. Retain Block-off

The **Retain BlockOff** toggle is selected by default. There is no harm in leaving this on, but it is not required unless user defined block-off is defined in the `.bcf` file, and you want it to remain blocked-off (ignored).

Porous and CHT objects in the `.bcf` file are ignored, and must be manually created in CFX-Pre after importing the grid. You should make sure that a 3D volume region was defined in the `grd` file for the porous or CHT object location prior to import.

By default, CFX-Pre will look in the same directory as the `.grd` file to locate the `.bcf` file. If the `.bcf` file is located elsewhere, you can browse and select the file.

#### 10.1.3.8.5. Regions in the `.grd` file

You should delete any regions from the `.grd` file that are not needed.

If necessary, you can force all "user defined" regions to be included in a `.grd` file by executing the following command at the TASCtool command prompt:

```
TASCtool{}: write grd all_regions_to_grd=on
```

This is usually not needed because you can import regions from the `.gci` file directly (see below).

Note that when faces are referenced by more than one named region, the import process will resolve this conflict such that faces are not referenced by more than one region.

### 10.1.3.8.6. Boundary Conditions in .bcf File

The regions associated with the boundary conditions defined in the `.bcf` file are imported into CFX-Pre. The boundary condition physics definitions are ignored and must be defined in CFX-Pre.

The CFX-TASCflow symmetry/slip boundary condition should be recreated as either:

- A symmetry boundary condition for flat surfaces.
- A wall boundary condition using the **Free Slip** option for curved surfaces.

### 10.1.3.8.7. Regions in the .gci File

Regions in the `.gci` file defined in (i, j, k) coordinates (such as boundary conditions) are imported if the **Use GCI file** toggle is selected on the **Advanced Options** tab. By default, CFX-Pre looks in the same directory as the `.grd` file for the location of the `.gci` file. You should select the location of the `.gci` file by clicking on the browse icon if it is located elsewhere.

Regions defined in physical space (x, y, z coordinates) are always ignored.

An alternative method for reading the `.gci` file is to force all regions to be included in the `.grd` file. For details, see [Regions in the .grd file \(p. 133\)](#).

### 10.1.3.8.8. Importing CFX-TASCflow TurboPre MFR Grids

You can create multiple domains from a single `.grd` file if it contains multiple 3D regions or GGI connections. For an MFR grid, a separate assembly will be created for each noncontinuous grid region. This enables a multiple frame of reference (MFR) case to be easily recreated in CFX-Pre from a single mesh import.

Grids from CFX-TASCflow TurboPre usually contain many named regions that may not be required to set up the problem in CFX-Pre. You might want to remove some of these regions before importing the grid to speed up the import of the mesh and simplify the imported mesh.

In CFX-TASCflow TurboPre, you can create multiple copies of blade passages. The 'open ends' of the machine section will use a periodic connection. These must be recreated in CFX-Pre using a periodic domain interface. For details, see [Creating and Editing a Domain Interface \(p. 205\)](#). The internal connection between blade passages can be connected in CFX-TASCflow TurboPre using an automatic periodic boundary condition. If such a connection is used you will have to manually reconnect each passage in CFX-Pre. You might therefore want to define a many-to-one topology connection for one-to-one grid connections so that passages are connected by CFX-TASCflow TurboPre as topology connections (which import immediately). For details, see [Grid Connections Processed \(in the .grd file\) \(p. 132\)](#).

### 10.1.3.8.9. Parameter File

CFX-TASCflow does not have units checking, whereas CFX-Pre does. Grid numbers will be imported using the units specified on the **Import Mesh** dialog box. You should convert all units in the properties and parameter files within TASCflow into SI units (kg, meter, second) prior to import.

## 10.1.3.9. CFX-4 Grid Files

CFX grid files are of the form `.geo`.

- Select **Split Symmetry Planes** to split symmetry planes that exist in more than one region. For details, see [Split Symmetry Planes \(p. 135\)](#).
- Select **Import from Cylindrical Coordinates** to transform a problem defined in cylindrical coordinates into Cartesian coordinates for use in CFX-Pre. It should be selected for all CFX-4 problems that use cylindrical coordinates. For details, see [Import from Cylindrical Coordinates \(p. 135\)](#).
- Select **Block Interfaces** to create 2D regions in CFX-Pre on block interfaces. For details, see [Create 2D Regions on \(p. 135\)](#).
- Import 2D axisymmetric mesh. For details, see [Import 2D Axisymmetric Mesh \(p. 136\)](#).

Other available options are:

- [Ignore Invalid Degenerate Elements \(p. 123\)](#)
- [Duplicate Node Checking \(p. 123\)](#)

#### 10.1.3.9.1. Split Symmetry Planes

The **Split Symmetry Planes** option is on by default. Symmetry planes that are defined by more than one CFX-4 region will be split so that each definition is imported. For example, a symmetry plane that is defined on two sides of a 3D region will be split into regions named `<regionname>1` and `<regionname>2`, and so on, where `<regionname>` is the original name of the symmetry plane in the CFX-4 file.

#### 10.1.3.9.2. Import from Cylindrical Coordinates

CFX-Pre can import problems defined in Cylindrical Coordinate  $(x, r, \theta)$  form from CFX-4. The problem is converted to Cartesian Coordinates  $(x, y, z)$  by the import process. The resulting CFX-Solver input file will not be written in cylindrical coordinates. You must select the **Import from Cylindrical Coordinates** option to successfully import a CFX-4 cylindrical coordinate problem.

---

**Note:**

This is *not* the same as an axisymmetric problem. For details, see [Import 2D Axisymmetric Mesh \(p. 136\)](#).

---

#### 10.1.3.9.3. Create 2D Regions on

##### Block Interfaces

When this option is selected, named regions will be created on the interfaces between mesh blocks. This can produce many regions in CFX-Pre, so it is usually better to define all the regions you require as patches in CFX-4.

#### 10.1.3.9.4. Create 3D Regions on

##### Fluid Regions (USER3D, POROUS)

In CFX-4, most 3D regions are classified as USER3D patches. Porous regions are treated in the same way as USER3D regions when importing them into CFX-Pre. When the **Fluid Regions**



**(USER3D, POROUS)** toggle is not selected, these regions are not imported. When the toggle is selected, they are imported as separate 3D regions. This toggle should be selected if you need the USER3D regions to create domains and subdomains. You should disable it to simplify the regions created in CFX-Pre. If, in CFX-4, you have created a USER3D region for the purpose of creating thin surfaces, you do not need to import the USER3D region in CFX-Pre because thin surfaces can be defined without the need for a separate subdomain.

#### 10.1.3.9.5. Blocked Off Regions (SOLIDs)

- If **Fluid Regions (USER3D, POROUS)** is not selected, and **Blocked Off Regions (SOLIDs)** is not selected, then SOLID regions are blocked-off (that is, this part of the mesh is not imported).
- If **Fluid Regions (USER3D, POROUS)** is not selected, and **Blocked Off Regions (SOLIDs)** is selected, then SOLID regions are imported into the default 3D region created by the import process.
- If **Fluid Regions (USER3D, POROUS)** is selected, and **Import SOLID regions** is toggled OFF, then SOLID regions become blocked-off (that is, this part of the mesh is not imported).
- If **Fluid Regions (USER3D, POROUS)** is selected, and **Import SOLID regions** is toggled ON, then SOLID regions are imported as separate 3D regions (which can be useful for CHT simulations).

#### 10.1.3.9.6. Conducting Solid Regions (SOLCONs)

- These are regions defined as conducting solid regions in CFX-4. There is no way to completely ignore SOLCON regions, they are always imported as either a separate region or as part of the parent region. If you want to ignore these regions (that is, so that there is no flow), then they should be removed from the CFX-4 mesh file using CFX-4 or with manual editing. Alternatively they can be imported but simply not used to define a subdomain in CFX-Pre. The import behavior is described below:
  - If **Fluid Regions (USER3D, POROUS)** is not selected, and **Conducting Solid Regions (SOLCONs)** is not selected, then SOLCON regions are imported as part of the "Assembly 3D" region.
  - If **Fluid Regions (USER3D, POROUS)** is not selected, and **Conducting Solid Regions (SOLCONs)** is selected, then SOLCON regions are imported as separate 3D regions.
  - If **Fluid Regions (USER3D, POROUS)** is selected, and **Conducting Solid Regions (SOLCONs)** is not selected, then SOLCON regions are imported as part of the regions in which they appear.
  - If **Fluid Regions (USER3D, POROUS)** is selected, and **Conducting Solid Regions (SOLCONs)** is selected, then SOLCON regions are imported as separate 3D regions and will be cut out of the parent regions.

#### 10.1.3.9.7. Import 2D Axisymmetric Mesh

You can enable this toggle if you want to import a mesh created as a 2D mesh on an axisymmetric section in CFX-4. This is different to a mesh defined using cylindrical coordinates; however, it can also use an  $(x, r, \theta)$  coordinate system. The CFX-4 mesh must be only 1 element thick in the  $k$  direction to use this option.

The **Number of Planes** value enables you to create additional planes in the  $\theta$  direction within the original 2D mesh to create a 3D problem. This will increase the number of elements in the  $k$  direction, but does not change the extent of the mesh.

The **Angle** value should be the  $\theta$  angle of the mesh section in degrees. Because the mesh is only one element thick, then  $\theta$  is the same for all nodes.

#### 10.1.3.9.8. Importing MFR Grids

If you have a CFX-4 MFR case, it can easily be imported into CFX-Pre and recreated.

- Each noncontinuous mesh section will be imported into a separate assembly.
- Each USER3D region will be imported into a separate 3D primitive.
- Both assemblies and 3D primitives can be used to create separate domains.

#### 10.1.3.10. CFX-BladeGenPlus Files

CFX-BladeGenPlus files are of the form `.bg+`. There are no options specific to importing CFX-BladeGenPlus files. For details, see [Common Import Options \(p. 122\)](#).

#### 10.1.3.11. PATRAN Neutral Files

PATRAN Neutral files are of the form `.out`.

- Select **Import Distributed Loads as 2D Regions** to convert predefined distributed loads as 2D primitives within CFX-Pre.

For details, see [Common Import Options \(p. 122\)](#).

#### 10.1.3.12. IDEAS Universal Files

IDEAS Universal files are of the form `.unv`.

- Select the entities, under **IDEAS Universal Specific Options**, to import from **Permanent Groups**.

For details, see [Common Import Options \(p. 122\)](#).

IDEAS mesh files contain groups of nodes, faces and/or elements. The groups can be normal groups or permanent groups. The normal groups are imported into CFX-Pre as up to three separate regions, depending on the information available in the mesh file. These regions will be named:

- `<groupName>_Nodes`
- `<groupName>_Faces`
- `<groupName>_Elements`

Only permanent groups of the selected types are imported into CFX-Pre. If overlapping regions are imported, CFX-Pre will split them into distinct regions; therefore, you may not want to import all permanent group types.

### 10.1.3.13. GridPro/az3000 Grid Files

GridPro/az3000 'grid' files are of the form `.grid`.

- Select **Include Periodic Regions** to convert predefined periodic boundaries into 2D primitives on import.
- Select **Ignore Connectivity** to import grid blocks as unconnected 3D primitives. Ignoring connectivity does not equivalence nodes at grid block interfaces.
- Select **Import Grid Blocks as Subdomains** so that for each predefined grid block, a separate 3D primitive is created.
- Selecting **Ignore Properties** causes data in the properties file to be ignored. This includes boundary conditions, 2D and 3D regions, and other data.

Additional information is available in:

- [Ignore Invalid Degenerate Elements \(p. 123\)](#)
- [Duplicate Node Checking \(p. 123\)](#).

### 10.1.3.14. NASTRAN Files

NASTRAN files can be imported.

- When **Include Subdomains** is cleared, all mesh elements are merged into a single 3D primitive.
- "Distributed loads" are pressure boundaries that, if imported, are used to generate 2D primitives in CFX-Pre. Select **Import Loads as 2D Regions** to import distributed loads.

Additional information is available in:

- [Ignore Invalid Degenerate Elements \(p. 123\)](#)
- [Duplicate Node Checking \(p. 123\)](#).

### 10.1.3.15. Pointwise Gridgen Files

Pointwise Gridgen files can be imported. There are no options available specific to the Pointwise Gridgen format. For details, see [Common Import Options \(p. 122\)](#).


### 10.1.3.16. User Import

If you should require facilities for importing a mesh other than those available through the standard **Mesh Import** forms, you can create your own customized mesh import program and make it available through the **Import Mesh** forms. For details, see [Volume Mesh Import API in the CFX Reference Guide](#). If you have created your own mesh import program, it must be run from within CFX-Pre; one way of doing this is by using the **Import Mesh** dialog box.

To run a custom import program using the **Import Mesh** dialog box:

1. Open the **Import Mesh** dialog box.

For details, see [Importing Meshes \(p. 121\)](#).

2. Set **Files of Type** to `User Import (*)`.
3. Select the file containing the mesh to import.
4. Click *Browse*  to browse to the location of the user executable file or enter its name under **Exec Location**.
5. Under **Exec Arguments**, enter the command-line arguments that should be passed to the import program.
6. Set advanced options as required.

For details, see:

- [Ignore Invalid Degenerate Elements \(p. 123\)](#)
- [Duplicate Node Checking \(p. 123\)](#)

7. Click **Open**.

CFX-Pre calls the custom import program with a command line that has the following form:

```
<user import executable> <executable arguments> <mesh file>
```

It is important therefore that the executable handles any arguments that are specified.

If you usually use a particular import program, you can set it as the default import program by any one of the following methods:

- Specify the full pathname of the import program, and other settings, in the **Options** dialog box.
- Add the following line to the `.cfx5rc` file:

```
CFX_IMPORT_EXEC="<executable_path>"
```

where *<executable\_path>* is the full path and name of your executable.

For details, see [Resources Set in cfx5rc Files in the CFX Introduction](#).

- Set `CFX_IMPORT_EXEC` in the system environment.

## 10.2. Mesh Tree View

The **Mesh** branch of the main tree view shows the regions of the imported meshes arranged in a hierarchy for each loaded mesh file. You can also view regions arranged in a hierarchy based on composite regions and mesh assemblies. To view either hierarchy in a separate Mesh tree view, right-click **Mesh** (from the **Mesh** branch of the main tree view) and select one of the **View by** submenu commands.

When viewing the file-based hierarchy, each imported mesh forms one or more assemblies at the first level of the tree. The second level of the tree shows all 3D primitives and the third level shows

- 2D primitives bounding each 3D primitive
- some composite regions.


### 10.2.1. Shortcut Menu Commands for Meshes and Regions

Right-clicking on a region, mesh assembly, 3D primitive, or 2D primitive in the tree view displays a shortcut menu containing various options depending on what is selected. For details about the generic shortcut menu commands, see [Outline Tree View Shortcut Menu Commands \(p. 40\)](#). Details about the mesh-related shortcut menu items are provided in:

- [Importing Meshes \(p. 121\)](#)
- [Deleting Meshes and Mesh Components from the Tree View \(p. 140\)](#)
- [Transform Mesh Command \(p. 140\)](#)
- [Gluing Meshes Together \(p. 147\)](#)
- [Mesh Editor \(p. 148\)](#)
- [Render Options \(p. 148\)](#)

---

**Note:**

If *Highlighting*  is selected (from the viewer toolbar), mesh entities will be highlighted in the viewer when you select them in the tree view.

---

## 10.3. Deleting Meshes and Mesh Components from the Tree View

---

There are several options for deleting meshes and mesh components and composite regions using shortcut menu items available in the **Outline** tree view in CFX-Pre. These options are:

- **Delete All Mesh:** Available when you right-click **Mesh**. When selected this option deletes all meshes currently present.
- **Delete Mesh:** Available when you right-click individual meshes themselves or Composite 3D Regions or Primitive 3D Regions that map directly and entirely to one or more assemblies. When selected, this option deletes all mesh associated with the selected assemblies.
- **Delete Definition:** Available when you right-click a composite region. When selected this option deletes the definition of the composite region name but not the underlying mesh.

## 10.4. Transform Mesh Command

---

You can transform meshes using the **Mesh Transformation Editor** dialog box. To access this dialog box, right-click a mesh file or selection of one or more 3D meshes in the tree view, then select the **Transform Mesh** command from the shortcut menu. There are four basic transformations: *Rotation*, *Translation*, *Scale*, and *Reflection*. More complex transformations can be achieved by successive

application of the basic types. You can also copy meshes, either by retaining the original mesh, or by creating multiple copies.

When picking points from the viewer, the **Show Faces** render option must be selected to enable a point on a region to be picked. It may also be useful to have **Snap to Node** selected (on by default in the viewer toolbar).


The values entered on this form use the units defined on the **Edit > Options > Common > Units** form. For details, see [Units \(p. 89\)](#).

The topics in this section include:

- [Target Location \(p. 141\)](#)
- [Reference Coordinate Frame \(p. 141\)](#)
- [Transformation: Rotation \(p. 141\)](#)
- [Transformation: Translation \(p. 143\)](#)
- [Transformation: Scale \(p. 143\)](#)
- [Transformation: Reflection \(p. 144\)](#)
- [Transformation: Turbo Rotation \(p. 145\)](#)
- [Multiple Copies \(p. 145\)](#)
- [Advanced Options \(p. 146\)](#)
- [Automatic Transformation Preview \(p. 147\)](#)

### 10.4.1. Target Location

Select the assemblies and/or other regions to transform from the **Target Assemblies** drop-down box.

Click the  icon to access the full list of available regions.

Not all regions are transformable. For example, 2D regions or 3D regions not resolving to at least one assembly are not transformable.

### 10.4.2. Reference Coordinate Frame

Toggle on **Reference Coordinate Frame** to specify if the transformation is defined in another coordinate frame. Select the reference coordinate frame from the **Coordinate Frame** list.

### 10.4.3. Transformation: Rotation

Use the `Rotation` transformation to rotate an assembly about an axis defined by two points or a principal axis.

### 10.4.3.1. Rotation Option: Principal Axis

This **Rotation Option** uses the X, Y, or Z axis as the axis of rotation. Select one of the principal axes, under **Axis**, to be the axis of rotation.

### 10.4.3.2. Rotation Option: Rotation Axis

This **Rotation Option** uses a user-defined axis as the axis of rotation for the transformation. This axis is defined by two Cartesian points, **From** and **To**. These points can be entered manually or selected in the viewer by clicking any coordinate box and then clicking in the viewer. The working units are assumed to apply to the point coordinates.

### 10.4.3.3. Rotation Angle Option

The rotation angle options are **Specified**, **Full Circle**, and **Two Points**.

#### 10.4.3.3.1. Specified

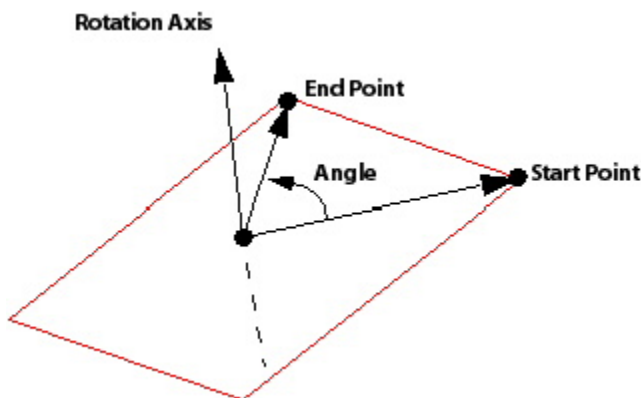
The **Specified** option simply rotates the assembly by the specified angle. When looking from the start point to the end point of the axis, a positive angle will produce a rotation in the clockwise direction.

#### 10.4.3.3.2. Full Circle

The **Full Circle** option should be used in conjunction with **Multiple Copy**, otherwise, the assembly will simply be transformed back to its original position. The effect this has is described more fully in **Multiple Copy**.

#### 10.4.3.3.3. Two Points

The **Two Points** option calculates an angle using the axis of rotation and the two points specified, as shown in the following figure. The two points and the start point of the axis define a plane with a normal direction pointing towards the end point of the axis. The angle proceeds in the clockwise direction from the **Start** to the **End** point when looking from the start point to the end point of the axis. When picking points from the viewer, the Show Faces render option must be selected to enable a point on a region to be picked. It may also be useful to have **Snap to Node** selected (on by default in the viewer toolbar).



## 10.4.4. Transformation: Translation

Use the translation transformation to move an assembly in the X, Y, and Z directions.

### 10.4.4.1. Method: Deltas

The `Deltas` method moves the mesh by the **Dx, Dy, Dz** values entered. Enter the **Dx, Dy, Dz** values with which to translate the mesh. This is equivalent to a displacement vector translation, using the origin as the start point of the vector and the point entered as the end point. A point can be entered manually or selected in the viewer after clicking any coordinate box. The working units are assumed to apply to the displacement vector components.

### 10.4.4.2. Method: Vector

The `Vector` option moves the assembly by the vector described by the **From** and **To** points.

Enter **From** and **To** points to describe the translation. These points can be entered manually or selected in the viewer after clicking any coordinate box.

## 10.4.5. Transformation: Scale

The `Scale` method is used to scale an assembly by a scale factor.

### 10.4.5.1. Method: Uniform

The `Uniform` option uses the same scale factor for all coordinate directions, therefore scaling the size of the assembly while maintaining the same aspect ratio. Specify the scale factor by entering a value for **Uniform Scale** (which must be greater than zero).

### 10.4.5.2. Method: Non Uniform

The `Non Uniform` option scales the assembly independently in each coordinate direction.

Enter scale factors **Sx, Sy,** and **Sz** to scale the X, Y, and Z coordinates of the assembly, respectively.

### 10.4.5.3. Scale Origin

Scaling is achieved by multiplying the location of each mesh node relative to the **Scale Origin** by the scaling factor.

Enter the **Scale Origin** as a Cartesian coordinate (for example, [0 0 0]), or use Picking mode by clicking any Cartesian coordinate box then picking a point from the viewer. The working units are assumed to apply to the point coordinates. When you are in Picking mode, the Cartesian coordinate boxes turn yellow. To manipulate the object in the viewer while in this state you have to click the viewer icons (rotate, pan, zoom) in the toolbar. You can turn off Picking mode by changing the keyboard focus (by clicking on another field, for example).



#### 10.4.5.4. Apply Scale To

This setting controls whether the transformation is applied to the original mesh or to a copy of the mesh. If you have set up physics locations on the original mesh, such locations are retained after the transformation.

The following options are available:

**Original (No Copy)**

Transforms the original mesh without making a copy.

**Copy (Keep Original)**

Copies the original mesh before applying the transformation. In this case, the original mesh remains in its current location.

#### 10.4.6. Transformation: Reflection

The **Reflection** method is used to mirror a mesh in a specified plane. Apart from using the principal planes (for example, the XY plane), arbitrary planes can be created with the Three Points and the Point and Normal methods. These are the same plane definition methods that are available in CFD-Post.

##### 10.4.6.1. Method

The options available are YZ Plane, XZ Plane, XY Plane, Three Points and Point and Normal.

When using the YZ Plane, XZ Plane, or XY Plane method, an offset can be applied by entering values in the **X, Y, Z** offset box.

If you use the Three Points or Point and Normal method, the points can be manually entered or selected in the viewer after you click in any coordinate field.

##### 10.4.6.2. Apply Reflection To

This setting controls whether the transformation is applied to the original mesh or to a copy of the mesh. If you have set up physics locations on the original mesh, such locations are retained after the transformation.

The following options are available:

**Original (No Copy)**

Transforms the original mesh without making a copy.

**Copy (Keep Original)**

Copies the original mesh before applying the transformation. In this case, the original mesh remains in its current location.

## 10.4.7. Transformation: Turbo Rotation

Use the **Turbo Rotation** transformation to rotate an assembly about an axis defined by the rotation axis or a principal axis.

### 10.4.7.1. Rotation Option: Principal Axis

This **Rotation Option** uses the X, Y or Z axis as the axis of rotation. Select one of the principal axis, under **Axis**, to be the axis of rotation.

### 10.4.7.2. Rotation Option: Rotation Axis

The **Rotation Option** uses a user-defined axis as the axis of rotation for the transformation. This axis is defined by two Cartesian points, **From** and **To**. These points can be entered manually or selected in the viewer by clicking any coordinate box and then clicking in the viewer.

### 10.4.7.3. Rotation Axis Options

In addition to the **From** and **To** points, you can select the following options:

#### Passages per Mesh

An indication of the number of blade passages that exist in the selected mesh file. The value will normally be 1.


#### Passages to Model

An optional parameter that is used to specify the number of passages in the section being modeled. This value is used in CFD-Post.

#### Passages in 360

An optional parameter that is used to specify the number of passages in the machine. This value is used in CFD-Post.

#### Theta Offset

Rotates the selected mesh, about the rotational axis, through an angle Theta. The offset can be a single value or set to an expression by clicking .

## 10.4.8. Multiple Copies

When the **Multiple Copies** toggle is disabled, then the assembly is simply transformed to the new location, without retaining a copy of the assembly at the original location. You can enable the **Multiple Copies** toggle to enable multiple copies of an assembly to be made during the transformation. It should be noted that this section is not available for **Scale Transformations**.

In general, the multiple copies will be evenly spaced throughout the transformation. For rotational transformations copies will appear at evenly spaced angles, while for translational transformations copies will appear at evenly spaced intervals along the vector describing the translation. For example, if you have a mesh for a single blade passage, you can make copies of it using the rotation transform-

ation. If your full machine has 60 blades and you want to reproduce the full geometry, you should use the **Full Circle** option for the **Angle** and select to make 59 copies (the original copy is the 60<sup>th</sup>).

### 10.4.8.1. # of Copies

Enter the number of copies for the assembly to make. This number does not include the original copy.

### 10.4.8.2. Delete Original

Controls whether the original copy is retained or deleted after the transformation. Composite regions associated with the original mesh are not deleted during this operation.

## 10.4.9. Advanced Options

The **Advanced Options** control your mesh-gluing strategy as described below.

### 10.4.9.1. Glue Adjacent Meshes

If you enable this toggle, CFX-Pre will attempt to automatically glue each copy of the assembly together. This means that CFX-Pre will try to create a continuous mesh contained in a single assembly from the multiple copies. For the glue to be successful, physically matching boundaries with one-to-one node pairings must be found between the copy (or copies) and the original. The multiple copies will then be treated as a single continuous mesh in a single assembly with multiple 3D regions. A single domain can be created for the entire assembly without the need to create domain or periodic interfaces between each copy. If multiple domains are created, automatic domain interfaces can be created. For details, see [Automatic Creation and Treatment of Domain Interfaces in the CFX-Solver Modeling Guide](#).

If boundaries do not physically match or one-to-one node pairings do not exist, then each copy will form a new assembly, which will require the creation of domain interfaces to connect them together.

When **Delete Original** is used in conjunction with **Glue Matching Meshes**, the original is deleted only if the gluing operation is successful.

For more information on gluing, see [Gluing Meshes Together \(p. 147\)](#).

You can set the following advanced options:

#### Glue Strategy

Choose the strategy that CFX-Pre will use in deciding how mesh selections being transformed are glued with each other and with other areas of mesh:

- **Location and Transformed** causes CFX-Pre to try to create connections automatically between the selected location being transformed and any copies that are made.
- **Location and Transformed and Touching** requests that CFX-Pre tries to glue the transformed locations with any copies made and also with any other mesh locations that are in contact with the transformed location or transformed copies.

## Keep Assembly Names

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

When transforming a location, existing assemblies can be modified or created by removing connections between 3D regions or can be merged by creating connections between 3D regions. The setting of **Keep Assembly Names** can be altered to indicate whether CFX-Pre should attempt to preserve assembly names that were present in the problem before the transformation took place, therefore ensuring that the locations used by physics objects are not invalidated:

- **None:** no attempt is made by CFX-Pre to retain existing assembly names
- **Existing:** assembly names specified before the transformation takes place are preserved
- **Existing and Intermediate:** the names of assemblies prior to the transformation and also any intermediate assembly names created during the transformation process will be preserved.

### 10.4.10. Automatic Transformation Preview

This toggle enables you to see the transformation before you click **Apply**. After you click **Apply**, the preview toggle clears automatically.

## 10.5. Gluing Meshes Together

If there are multiple mesh assemblies that have matched meshes, you can try to glue these together by selecting the assemblies in the tree view (using the **Ctrl** key), right-clicking, and selecting **Glue Regions** from the shortcut menu. If the meshes match exactly, a 1:1 connection is made; otherwise a GGI connection is used. In either case, the connection appears as a mapping of two regions under **Connectivity** in the tree view (for example, F3 . B1 . P3 <-> F3 . B2 . P4). Note that you can select **Connectivity > Hide 1:1 Connections** so that the list of GGI connections is easier to work with.

---

### Important:


Merging/connecting meshes in CFX-Pre is not supported when those meshes are provided by Ansys Workbench during a Workbench session. Doing so may result in extra copies of these meshes appearing when refreshing meshes.

---

### Tip:

Another way to glue two meshes together is to select **Connectivity > Define Connection**

from the tree view. In the **Mesh Connections Editor** that appears, click  to browse the

**Selection Dialog** for the regions to choose for **Side One** and **Side Two**. If  is selected, the regions are highlighted in the viewer as you highlight regions in the **Selection Dialog**.

---

If a pair of meshes cannot be glued together, you can use a domain interface instead. For details, see [Domain Interfaces \(p. 205\)](#).

---

**Note:**

- There is limited checking of the validity of GGI connections created by gluing meshes together.
  - When you transform or copy multiple assemblies, it is possible to have each copy glued to its original assembly or to other copies made. For details, see [Advanced Options \(p. 146\)](#) in the [Transform Mesh Command \(p. 140\)](#) section.
  - For more information on mesh connection types, see [Mesh Connection Options in the CFX-Solver Modeling Guide](#).
- 

## 10.6. Mesh Editor

---

The mesh editor is described in [Editing Regions in CFX-Pre \(p. 156\)](#).

## 10.7. Render Options

---

The **Render Options** dialog box controls how 2D objects will appear in the viewer, such as visibility, line width, line color, and so on. Rendering for individual 2D objects, or any composite regions that resolve to only one 2D object, is set on the **Render Options** dialog box for 2D objects. The **Render Options** dialog box can be used to control the properties of both, 2D primitives and profile geometry. For details, see [Render Options Dialog Box \(p. 148\)](#).

**Render Options** for any regions that are made up of more than one 2D object (such as a 3D region or a composite 2D region consisting of more than one 2D object) can be set on a global basis for all 2D objects within the particular region. For details, see [Render Options - Multiple 2D Regions \(p. 151\)](#).

Profile geometries can be rendered in different ways depending upon the underlying profile file. For details on profile rendering, see [Render Options - Profile Geometry \(p. 151\)](#).

You can access the **Render Options** dialog box by right-clicking on a region or profile geometry in the tree view and then selecting **Render > Properties** from the shortcut menu.

### 10.7.1. Render Options Dialog Box

When the **Render Options** dialog box is accessed by right-clicking on regions, the controls apply only to those regions selected.


#### 10.7.1.1. Symbols

Symbols are displayed to represent a profile geometry for which there is no face connectivity. You can change the appearance of the symbols by modifying their type and size.

### 10.7.1.1.1. Symbol

Enables you to change the appearance of the symbol.

### 10.7.1.1.2. Symbol Color

Pick a **Symbol Color** by clicking on the color box to cycle through common colors or click  to select a custom symbol color.


### 10.7.1.1.3. Symbol Size

The symbol size can be changed by entering a value for **Symbol Size**.

## 10.7.1.2. Draw Faces

Shows the faces of the mesh elements on 2D primitives or the faces of a profile geometry if faces are available. **Show Faces** should be selected if the effect of changing the face options is to be seen.

### 10.7.1.3. Face Color

The color used for the faces drawn on the 2D objects. Pick a **Face Color** by clicking on the color box to cycle through common colors or click  to select a custom face color.

### 10.7.1.4. Transparency

Select a **Transparency** level from 0 to 1, where 0 is opaque and 1 is transparent.

## 10.7.1.5. Draw Mode/Surface Drawing

Controls the shading property applied to faces on 2D objects.

### 10.7.1.5.1. Flat Shading

Each element is colored a constant color. Color interpolation is not used across or between elements.

### 10.7.1.5.2. Smooth Shading

Color interpolation is applied, which results in color variation across an element based on the color of surrounding elements.

### 10.7.1.6. Face Culling

This controls the visibility for element faces of objects that either face the viewer or point away from the viewer. Domain boundaries always have a normal vector that points out of the domain. The two sides of a thin surface have normal vectors that point towards each other.

---

**Note:**

**Face Culling** affects printouts performed using the **Screen Capture** method only.

---

#### 10.7.1.6.1. Front Faces

Clears visibility for all outward-facing element faces (the faces on the same side as the normal vector).

#### 10.7.1.6.2. Back Faces

Clears visibility for inward-facing element faces (the faces on the opposite side to the normal vector).

#### 10.7.1.6.3. No Culling

Shows element faces when viewed from either side.

### 10.7.1.7. Lighting

Toggle the lighting source on or off.

### 10.7.1.8. Specular

When selected, treats the object as a reflector of light.

### 10.7.1.9. Draw Lines

Shows the lines of the surface mesh elements on 2D primitives, or the face edges of a profile geometry if faces are available.


### 10.7.1.10. Edge Angle/Render Edge Angle

To change how much of the line or wireframe is drawn, you can change the edge angle. The **Edge Angle** is the angle between one edge of a face and its neighboring face. Setting an **Edge Angle** will define a minimum angle for drawing edges of the surface mesh or faces. If you want to see more of the surface mesh, reduce the edge angle.

### 10.7.1.11. Line Width

The line width can be changed by entering a value in the **Line Width** field corresponding to the pixel width of the line. When the box is active, the up and down arrow keys on your keyboard can be used to increment the value.

### 10.7.1.12. Line Color

Pick a **Line Color** by clicking on the color box to cycle through common colors or click  to select a custom line color.

### 10.7.1.13. Visibility

Set the visibility for the primitives in the viewer. Clearing the visibility may improve the viewer performance for complex meshes.

## 10.7.2. Render Options - Multiple 2D Regions

The **Render Options** dialog box for multiple regions is both an indicator of consistency of the render options of the regions selected, and a tool to set render options for all selected regions. The presence of a check mark in the check box for an option indicates whether consistent settings exist for that option across all of the selected regions.

For example if the **Shading** option is selected and set to `Flat Shading` this means that all the regions selected have their **Shading** options set to `Flat Shading`. By contrast, if **Face Color** is cleared this indicates that at least one region has a different color. You can still apply a color to all regions by enabling the check box next to **Face Color** and selecting a color. After any changes, CFX-Pre will recheck all objects for consistency, and update the form accordingly. The options themselves are the same as for individual regions.

## 10.7.3. Render Options - Profile Geometry

The **Render Options** dialog box can be used to control the appearance of profile geometries imported from external profile data files. When you enable the **Visibility** option in the **Initialize Profile Data** dialog box, or make the corresponding User Function visible, to visualize the geometry of the profile, you have the ability to modify the render options for the profile geometry. The appearance of the profile geometry will differ, depending on the specified face connectivity information in the profile file.

- If a profile file has face connectivity specified, in addition to nodal values, then the profile geometry will appear as faces.
- If a profile file has no face connectivity, then the profile geometry will appear as symbols based on the nodes.

Right-click the profile geometry to modify color, wireframe appearance, and transparency options.

### 10.7.3.1. Color

Use the **Color** option to pick a color for the profile geometry symbol or faces. Click the color box to cycle through common colors or select a custom color.

### 10.7.3.2. Lines

Use the **Lines** option to show or hide the wireframe and/or face edges on profile geometry faces.



### 10.7.3.3. Transparency

Use the **Transparency** option to set a transparency level for the profile geometry faces.

### 10.7.3.4. Properties

Launches Render Options dialog box for additional rendering options. For details, see [Render Options Dialog Box \(p. 148\)](#).

---

**Note:**

Not all rendering options are applicable for profile geometries displayed as symbols.

---

## 10.8. Mesh Topology in CFX-Pre

---

The mesh topology in CFX-Pre is largely determined by the primitive regions imported with the mesh. You must consider the requirements of the physics being simulated when generating the geometry and mesh outside of CFX-Pre.

### 10.8.1. Assemblies, Primitive Regions, and Composite Regions

Each mesh is imported into one or more assemblies. An assembly represents a connected mesh. A mesh containing one-to-one node connections is considered to be connected and is imported into a single assembly.

Each assembly contains one or more 3D primitives (mesh regions), and each 3D primitive is bounded by one or more 2D primitive mesh regions. Each 3D primitive may also contain 2D mesh primitives that are located within the interior of the mesh. A primitive is the lowest level of region information available in a mesh file.

Primitives could be regions that were explicitly created in the mesh generation software. However, in some mesh files, the mesh references underlying CAD faces, in which case these will be the primitive regions. GTM files are an example of this; a 2D primitive region will resolve to the CAD face `Solid 1.2`, for example. If CAD face data is available in the mesh file, then regions explicitly created in the mesh generation software, or in CFX-Pre, will reference the CAD faces and, therefore, themselves will not be the lowest level of region data. These regions are known as composite regions because they are composed of one or more primitive regions.

---

**Note:**

Because CFX-Pre can recognize underlying CAD surfaces from CFX GTM Files, it is not necessary to create composite regions, although it will often make selecting locations easier in CFX-Pre. Other mesh types may or may not require the definition of composite regions within CFX-Pre.

---

New composite regions can be created in CFX-Pre using the **Regions** details view. However, the topology of the existing primitives limits the scope of composite region creation and it is not possible to create any new primitives in CFX-Pre. For details, see [Defining and Editing Composite Regions \(p. 158\)](#).

The number and location of 2D primitives and 3D primitives is defined by the software that generated the mesh. You should consider your domain, boundary condition, domain interface and subdomain requirements when creating the mesh and create appropriate regions that can be used in CFX-Pre. You will need to create each region explicitly in the mesh generation software if your mesh file does not contain data that references the underlying CAD faces.

If primitives reference the underlying CAD faces, it does *not* mean that the exact CAD geometry is recovered. The mesh simply references all the CAD faces and makes the mesh associated with them available in CFX-Pre.

In CFX-Pre 3D primitives are always distinct, as such a mesh element is always contained in a single 3D primitive. All regions in the mesh file that define a set of 3D elements are imported into CFX-Pre. If any element exists in more than one grouping of elements, the import process will split the groupings so that each element is contained within a single 3D primitive. Composite regions will be defined that group the 3D primitives into the topology that the original mesh file represented. Depending on your mesh file, this could include 3D subregions, solid regions, block-off regions, user defined 3D regions, porous regions, and so on.

If a 2D primitive spans more than one 3D primitive, it will be split into multiple 2D primitives on import, so that each 2D primitive is part of only one 3D primitive. All overlapping 2D primitives are also split into distinct primitives upon import and composite regions are created to represent the original regions read from the mesh file. When a 2D primitive forms a boundary between 3D primitives, it will be split into two sides, such that a 2D primitive is associated with each 3D primitive. When a 2D primitive is split, a suffix is added to the name so that the resulting 2D primitives are named uniquely. For example, a 2D primitive called `Solid 1.2` would be split into `Solid 1.2A` and `Solid 1.2B`.

### 10.8.1.1. Composite Regions

Composite regions are defined as combinations of one or more 2D primitive, 3D primitive or other composite regions. New composite regions created in CFX-Pre must therefore be defined by a combination of at least one other region, however it is possible that a composite region can be defined that resolves to nothing.

Composite regions that are specified in the original mesh file imported into CFX-Pre will be imported into the application if the import format can be translated into one that CFX-Pre can use. The composite regions imported into CFX-Pre can be selected, modified and deleted in the same way as composite regions defined in the application.

Additional information on primitive and composite regions is available in [Assemblies, Primitive Regions, and Composite Regions](#) (p. 152).

For details about creating regions, see [Regions](#) (p. 155).

#### 10.8.1.1.1. Applications of the Composite Regions

For details, see [Applications of Composite Regions](#) (p. 159).

## 10.8.2. Domain and Subdomain Locations

Domains are created from a list of 3D primitives, and subdomains from list of 3D primitives that are also contained in a domain. Assemblies and 3D composites can also be used as locations for domains

and subdomains. In this case, all 3D primitives contained within the assembly or 3D composite are included in the domain.

Assemblies and 3D primitives not included in a domain are not used in the simulation. A 3D primitive may be implicitly included if it forms part of a 3D composite or assembly that is used in a domain.

The domains in a multi-domain simulation must be continuous or connected via domain interfaces - you cannot have separate isolated domains.

### 10.8.3. Boundary Condition and Domain Interface Locations

2D primitives and 2D composite regions can be used as locations to create boundary conditions and domain interfaces.

If your assembly has more than one 3D primitive and they share a common boundary, then at least one pair of 2D primitives will exist at the common boundary. One 2D primitive of each pair will bind one of the 3D primitives that shares the common boundary.

It is not possible for a region to span more than one domain in a single boundary condition.

### 10.8.4. Importing Multi-domain Cases

Meshes intended or previously used for multi-domain simulations can be imported into CFX-Pre. You will still be able to set up a multi-domain simulation from a single mesh import.

If the imported mesh is not connected, a separate assembly will be created for each connected section. Each assembly can be used to create a separate domain. If the mesh is connected, then a single assembly will be created but 3D primitives will be created for each 3D region defined in the mesh file. Each 3D primitive can be used to create a separate domain, even if it is contained in a single assembly.

## 10.9. Advanced Topic: cfx5gtmconv Application

---

The `cfx5gtmconv` application is a command line executable that can be used to convert between a number of mesh file formats. It can be used to perform import of a mesh into a GTM database or to convert a GTM database into a CFX-Solver input file that can be viewed in CFD-Post. If appropriate physics CCL is available, it can also be used to create a definition that can be run in a solver. In this case, the exported mesh obeys the constraints imposed on it by the solver and the physics model.

Full details can be found by entering:

```
<CFXROOT>/bin/cfx5gtmconv -help
```

at the command line, where `<CFXROOT>` is the path to your installation of CFX-Pre.

---

# Chapter 11: Regions

---

CFX-Pre has two types of regions:

- Primitive
- Composite

This chapter describes:

- 11.1. Primitive Regions
- 11.2. Composite Regions
- 11.3. Using Regions in CFX-Pre
- 11.4. Editing Regions in CFX-Pre
- 11.5. Applications of Composite Regions

Additional information on primitive and composite regions is available. For details, see [Mesh Topology in CFX-Pre \(p. 152\)](#).

## 11.1. Primitive Regions

---

*Primitive regions* are a unique selection of 2D faces or 3D elements that define a location in the model.

A model containing a mesh will have at least one 2D primitive region and one 3D primitive region.

It is not possible for a primitive region to contain 2D faces and 3D elements.

## 11.2. Composite Regions

---

*Composite regions* are regions defined in terms of other regions. For example:

- A named Region "A" may be an alias for another Region "B"

A composite region that is an alias can directly reference only one other region, but may reference more than one region if the region it references is itself another composite region.

- Region "C" may be a union (that is, all) of Region "D" and Region "E".

A composite region that is a union will reference one or more other regions directly and may indirectly reference many other regions if the regions it references themselves reference other regions.

Composite regions ultimately resolve to primitive regions.

The tree view and the Region details view are used to select, create, rename, modify, and delete composite regions.

If any of the primitive regions to which a composite region resolves does not exist in the model, the composite region is said to be *unresolved*.

Composite regions can be defined in terms of 2D and 3D primitive regions. If the composite region resolves to both 2D and 3D primitive regions, the composite region is known as a *mixed dimensionality composite*.

An *Assembly* is a special case of a mixed-dimensionality composite region. It can be used in the same way, but its composition implies connectivity within the mesh. All 3D mesh volumes within an Assembly 'know' about their connections to each other. This information is used by CFX-Pre when calculating interfaces between domains.

Composite regions that are specified in the original mesh file imported into CFX-Pre will be imported into the application if the import format can be translated into one that CFX-Pre can use. The composite regions imported into CFX-Pre can be selected, modified, and deleted in the same way as composite regions defined in CFX-Pre.

### 11.3. Using Regions in CFX-Pre

---

A composite 2D region may be used in exactly the same way as a primitive 2D region to define the location of a Boundary Condition, Domain Interface, and so on, in the model.

A composite 3D region may be used in exactly the same way as a primitive 3D region to define the location of a Domain or Sub-domain in the model.

Mixed-dimensionality composite regions can be used as locators, but only the primitive regions of appropriate dimensionality are used in the location. For example, a mixed-dimensionality region used as the location of a boundary condition will mean that the boundary condition is defined only on the 2D components.

### 11.4. Editing Regions in CFX-Pre

---

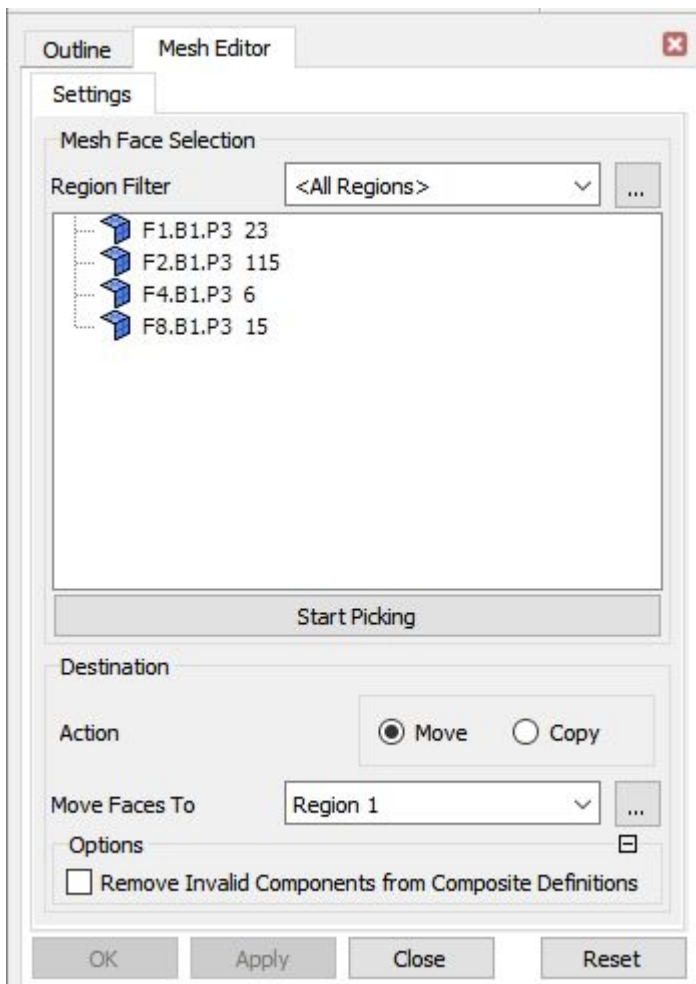
As there are two types of regions in CFX-Pre, there are two editors used for defining and modifying these regions. For details, see:

- [Defining and Editing Primitive Regions \(p. 156\)](#)
- [Defining and Editing Composite Regions \(p. 158\)](#)

#### 11.4.1. Defining and Editing Primitive Regions

Currently only definition and modification of 2D primitive regions is supported.

To define a primitive region, select **Regions** from the **Insert > Primitive Region** on the main menu or from the shortcut menu available from **Mesh** in the Outline or Mesh trees. To edit an existing primitive 2D region when on the name of the region in the Outline or Mesh trees, right-click and select **Edit Mesh** from the shortcut menu.







Faces can be moved or copied from one or more 2D primitive regions into a new or an existing 2D primitive region.

The **Region Filter** enables you to modify the source from which faces will be picked. Select **All Regions** from the drop-down list if faces are to be selected from anywhere in the model, or any number of regions if you want to restrict your source regions. (Note: If you have entered the editor by selecting **Edit Mesh**, the region filter will be set to the regions selected in the tree by default. You are able to change this selection if required.)

Initially no faces will be selected in the viewer and the dialog box will indicate this.

Click **Start Picking** and use one of the toolbar buttons on the **3D Viewer**, for details, see [3D Viewer Toolbar \(p. 52\)](#):

-  To set the pick mode to single face selection, click this button. Clicking in the viewer will select the first face to move.
-  To flood fill an area, click this button and then click in the viewer. Changing the crease angle will control how far the flood will extend. The angle indicates that any face that bounds the face first selected and has a normal within the angle will be selected. The same angle is then used again on any faces selected by the algorithm until no more faces can be reached using this method.

-  To select all faces within a rectangle, click this button and then click in the viewer and drag the box to perform the selection. The option to the left of **Pick All** indicates whether the selection only includes fully enclosed faces or any touching or enclosed faces.
-  To select all faces within a polygon, click this button and then click multiple times in the viewer finishing with a double-click to perform the selection. The option to the left of **Pick All** indicates whether the selection only includes fully enclosed faces or any touching or enclosed faces.

Appending further faces to the current selection is performed in the same way as above, but by using **Ctrl** and click to pick the faces in the viewer. All operations can use this method.

The names of the 2D primitive regions from which faces have been selected are shown in the **Mesh Face Selection** tree. The number of faces selected from each 2D primitive is also shown. The set of faces associated with a single 2D primitive can be removed by right-clicking the 2D primitive in the tree.

Faces are moved or copied to a destination region; the action can be selected from those shown in **Destination** box.

You can select the destination for the faces from the list to the right of the **Move Faces To** field, or you can type a new name into the field.

---

**Note:**

Unexpected results may occur if the topology of the current model is altered in some way during the course of the edit. For example adding a new composite region or deleting an existing one or importing or deleting a mesh may alter how the editor acts. In a similar way, performing an **Undo** or **Redo** when faces are selected may change the topology. If any of these operations are performed, click **Reset** and re-pick the faces as required.

---

### 11.4.1.1. Advanced Options

At the bottom of the form, the **Options** box can be expanded and **Remove Invalid Components from Composite Definitions** can be selected. Selecting this option will remove any references to primitives that are completely removed by the operation. If this is not done, composite regions that reference a removed primitive region will become unresolved.

### 11.4.2. Defining and Editing Composite Regions

The **Regions** details view is used to create new, and edit existing composite regions. To create a new region, select **Composite Region** from either the **Insert > Regions** menu or the shortcut menu available from **Mesh** in the Outline or Mesh trees. To edit an existing composite region, right-click it in the Outline or Mesh tree view, then select **Edit Definition** from the shortcut menu.

A composite region is defined by specifying a list of regions and a method for combining them. The **Regions** details view can be used to create or modify a region by selecting a method of combination from the **Combination** list and a selection of regions from the **Region List**.

---

**Note:**

The **Regions** details view enables you to restrict the regions available for selection by limiting them by **Dimension(Filter)**. Selecting 2D will cause the **Region List** to only display 2D regions and selecting 3D will cause the **Region List** to only select 3D regions. Regions of mixed dimensionality are always available.

---

### 11.4.2.1. Union

A **Combination** setting of **Union** combines the area or volume of the selected regions to create a new region. The new region will include all the regions from which it is constructed. For example, two or more 3D regions can be combined to create a new region, which can then be used as the location for a domain.

### 11.4.2.2. Alias

A **Combination** setting of **Alias** is used to produce a composite region that when resolved is based upon the same set of primitive regions as the region it is defined on. A composite region with a **Combination** of **Alias** may only reference a single region (this may be a composite or primitive region). The new composite region may, however, resolve to more than one primitive region. This feature is useful to assign recognizable names to regions with non-intuitive names.

## 11.5. Applications of Composite Regions

---

Composite Regions that are defined in the simulation can be used as locations for domains, sub-domains, boundary conditions and domain interfaces. For example, a composite region with a combination method of **Union** can be used to group two separate 3D regions. A domain that spans both 3D regions can then be created using the single composite 3D region. Domain interfaces should still be created to connect the two assemblies together if the composite region does not form a continuous mesh and flow is to pass between the two assemblies. For details, see [Mesh Topology in CFX-Pre \(p. 152\)](#).

Another application of composite regions is to set up a consistent set of locations that can be applied to a number of different simulations that use the same physics definition. By referencing the composite regions in the physics definitions, the need to edit the definitions for each mesh is avoided and if differences in the mesh topology do exist this can be coped with by editing the composite regions used to locate the physics relatively simply. In this way for every problem in which the physics is to be applied, the mesh, region CCL, and physics CCL can be imported. Locations of boundary conditions, domains and subdomains should all match provided that the composite regions can all be resolved as expected.





---

# Chapter 12: Analysis Type

---

The **Analysis Type** details view is used to specify whether the analysis is steady-state, transient, or transient blade row. A discussion on steady-state and transient flows is presented in [Steady State and Transient Flows in the CFX-Solver Modeling Guide](#).

## 12.1. Basic Settings Tab

---

The following topic is discussed in this section:

- [Analysis Type Settings \(p. 161\)](#)

### 12.1.1. Analysis Type Settings

The **Analysis Type** settings enable you to specify an analysis as being `Steady State`, `Transient`, or `Transient Blade Row`. Steady-state analyses are used to model flows that do not change over time; transient analyses model flows that are time-dependent. Transient blade row analyses are transient analyses that have special handling for turbomachinery cases.

The **Analysis Type** options are described in the following sections:

- [Steady State \(p. 161\)](#)
- [Transient \(p. 161\)](#)
- [Transient Blade Row \(p. 163\)](#)

#### 12.1.1.1. Steady State

No further settings are required for the `Steady State` option. Modeling advice for setting the time scale for steady-state simulations is provided in [Steady-state Time Scale Control in the CFX-Solver Modeling Guide](#).

#### 12.1.1.2. Transient

##### 12.1.1.2.1. Time Duration

Set **Option** to determine the length of the transient analysis:

- Total Time
- Time per run
- Maximum Number of Timesteps

- Number of Timesteps per Run
- Coupling Time Duration

For details, see [Time Duration in the CFX-Solver Modeling Guide](#).

### 12.1.1.2.2. Time Steps

Set **Option** to determine the size of timesteps for the run:

- Timesteps
- Timesteps for the Run
- Adaptive
- Coupling Timesteps

The **Timesteps** and **Timesteps for the Run** parameters can accept a single value or lists. If a list is entered, it should be comma separated, for example, 2, 1.2, 2.4. If an expression is used, you must associate units with each item in the list, for example, 2 [s], 1.2 [s], 2.4 [s]. In addition, it is possible to define multiples of a timestep value in the user interface *when not using the expression method*. For example, you could enter 5\*0.1, 2\*0.5, 10\*1 as a list of values, and set the units to [s] separately. The corresponding CCL that would be generated would be:

```
0.1 [s], 0.1 [s], 0.1 [s], 0.1 [s], 0.1 [s], 0.5 [s], 0.5 [s], 1 [s], 1 [s], 1 [s], 1 [s], 1 [s], 1 [s], 1 [s], 1 [s], 1 [s], 1 [s]
```

If you accidentally enter 5\*0.1 [s], 2\*0.5 [s], 10\*1 [s] as an expression, the multiplication would be carried out, and the corresponding CCL that would be generated would be:

```
0.5 [s], 1.0 [s], 10.0 [s]
```

For details, see [Transient Timestep Control in the CFX-Solver Modeling Guide](#).

When Adaptive time is selected, set one of the following three conditions for **Timestep Adaption** to automate the calculation of timestep size:

- Number of Coefficient Loops
- RMS Courant Number
- MAX Courant Number

For details, see [Transient Timestep Control in the CFX-Solver Modeling Guide](#).

### 12.1.1.2.3. Initial Time

Set the **Option** to specify the **Initial Time** for a transient analysis.

- Automatic
- Automatic with Value

- Value
- Coupling Initial Time

For details, see [Initial Time in the CFX-Solver Modeling Guide](#).

### 12.1.1.3. Transient Blade Row

The `Transient Blade Row` option is required in order to access the Transient Blade Row models:

- None
- Profile Transformation
- Time Transformation
- Fourier Transformation

The **Initial Time** settings must be specified in the **Analysis Type** settings. For details on these settings, see [Initial Time \(p. 162\)](#).

For instructions on setting up and using Transient Blade Row models, see [Transient Blade Row Modeling in the CFX-Solver Modeling Guide](#).



---

# Chapter 13: Domains

---

This chapter describes:

[13.1. Creating New Domains](#)

[13.2. The Details View for Domain Objects](#)

[13.3. Using Multiple Domains](#)

[13.4. User Interface](#)

CFX-Pre uses the concept of domains to define the type, properties, and region of the fluid, porous, or solid. Domains are regions of space in which the equations of fluid flow or heat transfer are solved. This section describes how to use the domain details view to define the physics of fluid, porous or solid domains in your simulation. This includes selecting the 3D bounding regions and choosing appropriate physical models.

A list of the physical models available in CFX, as well as additional information on the physical meaning of the models used, is available. For details, see [Physical Models in the CFX-Solver Modeling Guide](#).

Domains are created from a list of Assemblies, 3D primitive regions and/or 3D composite regions that are associated with a volume of an imported mesh. A discussion of these objects can be found in CFX-Pre. For details, see [Mesh Topology in CFX-Pre \(p. 152\)](#).


In some cases, separate domains will need to be connected via a domain interface, while in other cases, no interface is required or a default interface is created and is suitable. For details, see [Domain Interfaces \(p. 205\)](#).

Within fluid, porous, and solid domains, internal 3D regions can be assigned to a subdomain. These are used to create volumetric sources of mass, momentum, energy, and so on. For details, see [Subdomains \(p. 247\)](#).

Boundary conditions can be applied to any bounding surface of a 3D primitive that is included in a domain (that is, including internal surfaces). For details, see [Boundary Conditions \(p. 217\)](#).

## 13.1. Creating New Domains

---

New domains are created by selecting **Insert > Domain** or clicking the *Domains*  icon. Note that creation of domains from the menu bar or toolbar may subsequently require selection of the appropriate analysis type. Domains can also be created by right-clicking the appropriate analysis type in the **Outline** tree view.

Creating a new domain will present a dialog box where a unique name for the domain should be entered.

Additional information on valid names is available in [Valid Syntax for Named Objects \(p. 95\)](#). Existing domains may be edited by double-clicking the domain in the **Outline** tree view, or by right-clicking the domain and selecting **Edit**. For details, see [Outline Tree View \(p. 37\)](#).

## 13.2. The Details View for Domain Objects

---

After entering a name for the domain, or selecting a domain to edit, the domain details view appears in the workspace. In this view, you should complete each of the following tabs in turn, proceeding from left to right across the tabs. The tabs shown depends on your simulation, but could be:

- **Basic Settings:** Sets the location and type of domains, as well as the fluid, porous or solid, used in the domain. The reference pressure, buoyancy options and domain motion are also set here. For details, see [Basic Settings Tab \(p. 168\)](#).
- **Porosity Settings:** Only available for porous domains. Set the general description of a porous domain.
- **Fluid Models:** Only available for fluid domains. Sets the physical models that apply to all domain fluids. For details, see [Fluid Models Tab \(p. 175\)](#).
- **Fluid Specific Models** (for example, Water at RTP): Only available for fluid or porous domains when more than one fluid is selected, or for a single phase case when particles are included. A separate tab is used for each fluid in the domain and uses the fluid name as the name for the tab. This sets physical model options that are specific to each domain fluid. For details, see [Fluid Specific Models Tab \(p. 186\)](#).
- **Fluid Pair Models:** Only available for fluid domains using multiple fluids or when particles are included. This sets options that depend on the interaction between fluid pairs, such as transfer options. For details, see [Fluid Pair Models Tab \(p. 189\)](#).
- **Solid Models:** Only available for solid domains. Sets the physical models that apply to the solid. For details, see [Solid Models Tab \(p. 195\)](#).
- **Particle Injection Regions:** When a particle tracking simulation is used, custom injection regions can be created using this tab. For details, see [Particle Injection Regions Tab \(p. 199\)](#).
- **Initialization:** Sets initial conditions on a domain basis. For details, see [Initialization Tab \(p. 203\)](#). This is optional since global initialization can also be performed, but is essential for solid domains.
- **Solver Control:** Sets solver control settings on a domain basis. For details, see [Solver Control Tab \(p. 204\)](#).

## 13.3. Using Multiple Domains

---

For any given CFD problem, more than one domain may be defined. By default, the physical models used in each domain must be consistent; therefore, each time you create or edit a domain, the physical models (fluid lists, heat transfer models, and so on) are applied across all domains of the same type (such as fluid or solid), possibly overwriting models chosen earlier for other domains.

Note the following:

- Some exceptions exist when using fluid and solid domains together and also to enable MFR (multiple frame of reference) simulations to be defined.
- If a domain interface is required, refer to [Using Domain Interfaces in the CFX-Solver Modeling Guide](#) for information on the correct use of interfaces.

### 13.3.1. Multiple Fluid Domains

When consistent physics has been enforced, all settings are copied across *all* fluid domains when *any* fluid domain is edited with the following exceptions:

- **Location:** The location of each domain must obviously be different.
- **Coordinate Frame:** Each domain can use a different reference local coordinate frame.
- **Domain Motion:** Each domain can be independently stationary or rotating. For rotating domains, the angular velocity and axis of rotating can be different for each domain. This enables MFR simulations to be set up.

Note that these parameters are all set on the **Basic Settings** tab on the **Domains** form.

Fluid and Solid Domains: Settings are *not* copied between fluid and solid domains with the exception of **Thermal Radiation Model**. If any solid domain uses the Monte Carlo radiation model, then all fluid domains must also model radiation and must use the Monte Carlo model. If no solid domain has radiation modeling (that is, **Option** = `None`), then the fluid domains can use any radiation model.

### 13.3.2. Multiple Solid Domains

Settings are *not* copied between solid domains. Each solid domain can be made from a different material and can mix the Monte Carlo radiation model and no radiation model.

## 13.4. User Interface

---

The following topics will be discussed:

- 13.4.1. Basic Settings Tab
- 13.4.2. Fluid Models Tab
- 13.4.3. Polydispersed Fluid Tab
- 13.4.4. Fluid Specific Models Tab
- 13.4.5. Fluid Pair Models Tab
- 13.4.6. Solid Models Tab
- 13.4.7. Porosity Settings Tab
- 13.4.8. Particle Injection Regions Tab
- 13.4.9. Boundary Models Tab
- 13.4.10. Initialization Tab
- 13.4.11. Solver Control Tab




## 13.4.1. Basic Settings Tab

The basic settings apply to the whole of the domain. When you create a new domain, the **Basic Settings** tab is initially shown.

### 13.4.1.1. Location and Type

#### 13.4.1.1.1. Location

The **Location** is the list of assemblies, 3D primitive regions and/or 3D composite regions that define the volume of the domain. For details, see [Mesh Topology in CFX-Pre \(p. 152\)](#). Using an assembly or a composite region in the Location list implicitly includes all 3D primitives contained within the object. You can use more than one location by using the **Shift** or **Ctrl** keys to pick

multiple entries from the drop-down list. The  icon to the right of the drop-down list can be used to pick locations from an expanded list. Alternatively, clicking a location in the viewer displays a small box containing the available locations.

For details, see [Domain and Subdomain Locations \(p. 153\)](#).

#### 13.4.1.1.2. Domain Type

The **Domain Type** setting can be set to one of the following:

- Fluid Domain

Fluid domains are used to model one fluid or a combination of fluids, with a wide range of modeling options. It is possible to deform the mesh to simulate movement of the boundaries of the domain; for details, see [Mesh Deformation \(p. 174\)](#).

- Solid Domain

Solid domains are used to model regions that contain no fluid or porous flow. Several modeling options are available, including heat transfer (see [Conjugate Heat Transfer in the CFX-Solver Modeling Guide](#)), radiation (see [Radiation Modeling in the CFX-Solver Modeling Guide](#)), and Additional Variables (see [Additional Variables \(p. 405\)](#) and [Additional Variables in the CFX-Solver Modeling Guide](#)). In addition, you can model the motion of a solid that moves relative to its reference frame; for details, see [Solid Motion \(p. 196\)](#).

- Porous Domain

Porous domains are similar to fluid domains, but are used to model flows where the geometry is too complex to resolve with a grid. For details, see [Flow in Porous Media in the CFX-Solver Theory Guide](#).

- Immersed Solid

Immersed Solid domains can be used in transient simulations to model rigid solid objects that move through fluid domains; for details, see [Domain Motion \(p. 172\)](#) and [Immersed Solids in the CFX-Solver Modeling Guide](#).





### 13.4.1.1.3. Coordinate Frame

By default in a fluid domain, **Coordinate Frame** is set to the default Cartesian frame, `Coord 0`, but you can select any predefined coordinate frame. To create a new coordinate frame, select **Insert > Coordinate Frame** from the menu bar. For details, see [Coordinate Frames \(p. 371\)](#) and [Coordinate Frames in the CFX-Solver Modeling Guide](#).

The coordinate frame set for a domain is local to only that domain and is used to interpret all x, y and z component values set in the domain details view. This includes the gravity components in a buoyant flow and the rotation axis definition in a rotating domain. The coordinate frame set here has no influence on boundary conditions for the domain. For details, see [Global Coordinate Frame \(Coord 0\) in the CFX-Solver Modeling Guide](#).

### 13.4.1.2. Fluid and Particle Definitions and Solid Definitions

To define a fluid, particle or solid (including the solid portion of a porous domain):

1. If required, click *Add new item*  to the right of the definition list, type a name for the definition and click **OK**. For multiphase simulation, more than one fluid is required. For details, see [Multiphase Flow Modeling in the CFX-Solver Modeling Guide](#).
2. For the definition **Option** select `Material Library` (the default) to enable choosing a material from a supplied or user defined library or `Material Definition` for Reacting Mixtures.
3. For the definition **Material** select from the drop-down list for some commonly used materials or click *Select from extended list*  to access a complete list of materials.
4. After clicking  you may also choose to select *Import Library Data*  to load library data from a file.

The specification of material properties (for example, density and viscosity) and the creation of custom materials is performed in the **Materials** details view. For details, see [Materials \(p. 387\)](#). New materials are added to the relevant drop-down list.

A solid domain must be made from a single solid material.

#### 13.4.1.2.1. Morphology

Which morphology options are available depends on whether you are setting fluid-specific details for an Eulerian phase or for a particle phase. For Eulerian phases, the options are:

- Continuous Fluid
- Dispersed Fluid
- Dispersed Solid
- Droplets (Phase Change)
- Polydispersed Fluid

For details, see [Morphology in the CFX-Solver Modeling Guide](#).

For a particle phase, the options are:

- Particle Transport Fluid
- Particle Transport Solid

For details, see [Particle Morphology Options in the CFX-Solver Modeling Guide](#).

#### 13.4.1.2.1.1. Mean Diameter

For Dispersed Fluid and Dispersed Solid phases, a mean diameter is required. For details, see [Mean Diameter in the CFX-Solver Modeling Guide](#).

#### 13.4.1.2.1.2. Minimum Volume Fraction

This is available for dispersed phases, but you will not usually need to set a value. For details, see [Minimum Volume Fraction in the CFX-Solver Modeling Guide](#).

#### 13.4.1.2.1.3. Maximum Packing

This is available for the Dispersed Fluid and Dispersed Solid phases. For details, see [Maximum Packing in the CFX-Solver Modeling Guide](#).

#### 13.4.1.2.1.4. Restitution Coefficient

This restitution coefficient setting holds a value from 0 to 1 that indicates the degree of elasticity of a collision between a pair of particles. For such a collision, the restitution coefficient is the ratio of separation speed to closing speed. This restitution coefficient setting is used only for the kinetic theory model. For details, see [Kinetic Theory Models for Solids Pressure in the CFX-Solver Theory Guide](#).

#### 13.4.1.2.1.5. Particle Diameter Distribution

This is available for particle phases. For details, see [Particle Diameter Distribution in the CFX-Solver Modeling Guide](#).

#### 13.4.1.2.1.6. Particle Shape Factors

This is available for particle phases. For details, see [Particle Shape Factors in the CFX-Solver Modeling Guide](#).

#### 13.4.1.2.1.7. Particle Diameter Change

This option is available when multiphase reactions have been enabled with particle tracking. When **Particle Diameter Change** is selected choose either Mass Equivalent or Swelling Model.

##### 13.4.1.2.1.7.1. Swelling Model

Select a reference material from the list. Enter a **Swelling Factor** greater than or equal to zero; a value of zero indicates no swelling, and CEL expressions are permitted. For details, see [Particle Diameter Change Due to Swelling in the CFX-Solver Modeling Guide](#).

### 13.4.1.3. Particle Tracking

To include particles in the domain, define a particle in **Fluid and Particle Definitions...**, select the particle material and select the **Particle Transport Fluid** or **Particle Transport Solid** option for **Fluid and Particle Definitions...** > <particle definition> > **Morphology** on the **Basic Settings** tab. For details, see [Particle Transport Modeling in the CFX-Solver Modeling Guide](#).

### 13.4.1.4. Domain Models

The **Domain Models** settings are described next:

13.4.1.4.1. GT-SUITE Domain Models

13.4.1.4.2. Pressure: Reference Pressure

13.4.1.4.3. Buoyancy: Option

13.4.1.4.4. Domain Motion

13.4.1.4.5. Mesh Deformation

13.4.1.4.6. Passage Definition

#### 13.4.1.4.1. GT-SUITE Domain Models

The **GT-SUITE Domain Models** settings are available if the simulation is set up to be coupled with GT-SUITE with turboshaft coupling enabled. For details, see [Coupling CFX to an External Solver: GT-SUITE Coupling Simulations in the CFX-Solver Modeling Guide](#).

Domains can be set up to receive angular velocity from a GT-SUITE Turboshaft function, with the shaft speed coming from GT-SUITE.

As prerequisites:

- There must be at least one GT-SUITE Model object listed in the **Outline** tree view under **Simulation** > **Expressions, Functions and Variables** > **GT-SUITE Models**. The corresponding GT-SUITE model must include a turboshaft.
- There must be at least one GT-SUITE Turboshaft function listed in the **Outline** tree view under **Simulation** > **Expressions, Functions and Variables** > **User Functions**.

To have the domain receive input from GT-SUITE:

1. Select **GT-SUITE Domain Models** > **Use GT-SUITE Coupling** to enable coupling of the domain with GT-SUITE in a coupled simulation.
2. Under **Use GT-SUITE Coupling**, set **GT-SUITE Setup** > **GT SUITE Model** to the name of an existing GT-SUITE Model object intended to send shaft speed data to the domain.
3. Under **Domain Models** > **Domain Motion**, set **Option** to **GT-SUITE Coupling** and set **Turboshaft Name** to the name of a GT-SUITE Turboshaft function that corresponds to the domain.

For details about using GT-SUITE with CFX, see [Coupling CFX to an External Solver: GT-SUITE Coupling Simulations in the CFX-Solver Modeling Guide](#).

### 13.4.1.4.2. Pressure: Reference Pressure

This sets the absolute pressure level to which all other relative pressure set in a simulation are measured. For details, see [Setting a Reference Pressure in the CFX-Solver Modeling Guide](#).

### 13.4.1.4.3. Buoyancy: Option

For flows in which gravity is important, you should include the buoyancy term. Gravity components in the x, y and z directions should be entered; these are interpreted in the coordinate frame for the domain. For details, see [Coordinate Frames in the CFX-Solver Modeling Guide](#).

There are two different buoyancy models in CFX: the one used depends upon the properties of the selected fluid(s). Depending on the types of fluid selected, a **Buoyancy Reference Temperature** and / or a **Buoyancy Reference Density** must be set. This is because different fluids use either the full or Boussinesq buoyancy model. In multiphase flows, the reference density can have a significant effect.

The **Buoyancy Reference Location** can be set automatically, or to a specific location with X/Y/Z coordinates. For details, see:

- [Buoyancy in the CFX-Solver Modeling Guide](#)
- [Buoyancy in the CFX-Solver Theory Guide](#).

### 13.4.1.4.4. Domain Motion

The available Domain Motion options depend on the type of domain, and are described in [Table 13.1: Domain Motion Options \(p. 172\)](#):

**Table 13.1: Domain Motion Options**

Option	Eligible Domain Types	Description
Stationary	All domain types	The domain remains stationary in the absolute frame of reference.
Rotating	All domain types	The domain rotates with a specified angular velocity about the given axis.  For fluid, porous, and solid domains, a <b>Rotational Offset</b> setting exists.  For fluid and porous domains, an <b>Alternate Rotation Model</b> option exists.
Speed and Direction	Immersed solid domain	The domain translates at the specified speed in the specified direction. The translation direction can be specified by Cartesian components, or by a coordinate axis.
Specified Displacement	Immersed solid domain	The domain is displaced according to the specified Cartesian components. For example, you could use CEL expressions that are functions of time to move the domain.

Option	Eligible Domain Types	Description
General Motion	Immersed solid domain	<p>Specify a reference origin that is considered to be attached to the domain. Then specify a motion for that origin, and a rotation of the domain about that origin.</p> <p>The reference origin location is specified by the <b>Reference Location</b> settings.</p> <p>The motion of the reference origin is specified by <b>Origin Motion</b> settings that are similar to those for the <b>Domain Motion</b> options (other than <code>General Motion</code>).</p> <p>The rotation of the domain about the reference origin is specified by the <b>Body Rotation</b> settings.</p>
Rigid Body Solution	Immersed solid domain	<p>Specify a mass, moment of inertia, and various dynamics settings. The dynamics settings include external forces and torques, translational and rotational degrees of freedom, and gravity. All of these settings are analogous to the settings of the rigid body object, which is described in <a href="#">Rigid Bodies (p. 275)</a>. Note that, unlike for a rigid body object, you cannot specify initialization values for the rigid body solution that applies to an immersed solid domain; these initialization values (such as angular velocity and angular acceleration) are effectively initialized with values of magnitude zero.</p> <p>For additional information on modeling rigid bodies, see <a href="#">Rigid Body Modeling in the CFX-Solver Modeling Guide</a>.</p>

Details of some of the settings mentioned in [Table 13.1: Domain Motion Options \(p. 172\)](#):

- **Angular Velocity:** The angular velocity gives the rotation rate of the domain, which can be a function of time.
- **Axis Definition:** The axis of rotation can be a coordinate axis of the local coordinate frame or a local cylindrical axis defined by two points.
  - If `Coordinate Axis` is selected, the available axes are all local and global coordinate axes. `Coord 0` is the global coordinate frame, and its axes are referred to as `Global X`, `Global Y` and `Global Z`. A local coordinate frame's axes are referred to as `myCoord. 1`, `myCoord. 2`, `myCoord. 3` where 1,2,3 represent the local X,Y,Z directions.
  - If `Two Points` is selected, **Rotation Axis From** and **Rotation Axis To** must be set. The points are interpreted in the coordinate frame for the domain. If the coordinate frame is cylindrical, then the components correspond to the  $r, \theta, z$  directions. For details, see [Coordinate Frames in the CFX-Solver Modeling Guide](#).

- **Rotational Offset:** This setting transforms the domain by the specified rotation angle. The rotation axis used for this transformation is specified by the **Axis Definition** settings.
- **Alternate Rotation Model:** For details, see [Alternate Rotation Model in the CFX-Solver Modeling Guide](#).
- **Reference Location:** The reference location is an origin point that should be defined to conveniently describe the body rotation of the immersed solid domain. When **Body Rotation > Option** is set to `None`, the reference location will be neglected. Specify the reference location by choosing an existing coordinate frame origin, or by specifying Cartesian coordinates.

A solver run that starts from a previous run should have the same domain motion options in the immersed solids domains, and must have identical reference location specifications if supplied.

- **Origin Motion:** The origin motion can be specified in any of the ways that the domain motion can be specified (not counting the `General Motion` option for domain motion), and by `Specified Velocity`, which accepts Cartesian components of velocity.
- **Body Rotation:** The body rotation options are:
  - `None`
  - `Rotating`

Specify an angular velocity and the instantaneous axis of rotation.
  - `Specified Angular Velocity`

Use Cartesian components to define a vector. The rotation axis passes through the reference location in the direction of the specified vector. The angular velocity is the magnitude of the specified vector.

---

**Note:**

- CEL expressions used to define domain motion can be functions of time only.
- If you create two or more fluid domains and modify a model setting of one of the domains, that setting is generally copied to all other fluid domains in the simulation. An exception to this is that if you edit the **Domain Motion** settings of a domain, those settings are not copied to any other domains; this enables each domain to rotate or remain stationary independently of the other domains.

---

#### 13.4.1.4.5. Mesh Deformation

Mesh deformation can be used to model flows with varying geometry, for both transient and steady-state simulations. The following mesh deformation model options are available for a domain:

- `None`

- **Regions of Motion Specified:** permits wall boundaries and subdomains to move, and makes mesh motion settings available. These include a mesh motion model option and mesh stiffness settings. For details, see [Regions of Motion Specified in the CFX-Solver Modeling Guide](#).
- **Periodic Regions of Motion:** permits wall boundaries to move, and makes mesh motion settings available. These include a **Periodic Model** setting, a mesh motion model option and mesh stiffness settings. The **Periodic Regions of Motion** option can save computational time compared to the **Regions of Motion Specified** option by taking advantage of the nature of periodic motion and solving the mesh motion equations only at the start of the simulation. However, the **Periodic Regions of Motion** mesh deformation model implementation uses a linear approximation and has some limitations compared to the **Regions of Motion Specified** option. For details, see [Periodic Regions of Motion in the CFX-Solver Modeling Guide](#).
- **Junction Box Routine:** reads mesh coordinate datasets from a file into the CFX-Solver as the solution proceeds. This step requires the specification of a series of meshes and User Fortran routine(s). For details, see [Junction Box Routine in the CFX-Solver Modeling Guide](#).

#### 13.4.1.4.6. Passage Definition

For transient blade row cases, specify the number of passages in 360° and the number of passages per component for the domain. This information may be used in the automatic calculation of time step size, depending on the **Transient Method** settings (which are described in [Transient Method \(p. 359\)](#)).

### 13.4.2. Fluid Models Tab

The **Fluid Models** tab is where models are chosen, which apply to all Eulerian fluids in the simulation. By default, the fluids models must be consistent between all fluid domains in a multidomain simulation, but CFX supports inconsistent physics through the setting of an environment variable. For details, see:

- [Using Multiple Domains \(p. 166\)](#)
- [Solid Models Tab \(p. 195\)](#).

In a multiphase simulation, the options that are allowed to vary between fluids will appear on the **Fluid Specific Models** tab instead. For details, see [Fluid Specific Models Tab \(p. 186\)](#).

Some fluid models can apply to all fluids or can be set on a fluid-specific basis, these models will appear on the **Fluid Models** section with a **Fluid Dependent** option. If this is selected, then the model appears on the **Fluid Specific Models** tab.

The options available on the **Fluid Specific Models** tab depends on the simulation set up (including the type and number of fluids used in the simulation (such as single or multicomponent, single or multiphase, reacting or non-reacting)) and whether **Additional Variables** have been created.

All details related to **Particle Tracking** are set on the **General Settings** tab and the models chosen on the **Fluid Models** tab do not apply to the particle phase.

Radiation with multiphase is not supported. However, it is allowed for single Eulerian particle tracking cases on the **Fluid Specific Models** tab.



The available settings depend on the physical models chosen in your simulation.

### 13.4.2.1. Multiphase Options

These options are only applicable to multiphase simulations.

#### 13.4.2.1.1. Homogeneous Model

Inhomogeneous is the general case of multiphase flow, where each fluid has its own velocity field, turbulence field, and so on. You can select the **Homogeneous Model** check box to switch to this model, where all fluids share a velocity field, turbulence field, and so on. For details, see [The Homogeneous and Inhomogeneous Models in the CFX-Solver Modeling Guide](#). Both the inhomogeneous and homogeneous models have a **Free Surface Model** option.

#### 13.4.2.1.2. Free Surface Model

You can select the `Standard` free surface model if you are modeling multiphase flow with a distinct interface between the fluids. For details, see [Free Surface Flow in the CFX-Solver Modeling Guide](#).

#### 13.4.2.1.3. Multiphase Reactions

**Multiphase Reactions** are available when any reactions have been defined with type `Multiphase`. For details, see [Multiphase: Basic Settings \(p. 403\)](#). Any reactions that are to be included in the simulation should be selected from the drop-down list. For details, see [Multiphase Reactions in the CFX-Solver Modeling Guide](#).

### 13.4.2.2. Heat Transfer

#### 13.4.2.2.1. Homogeneous Model

For details, see [Homogeneous Heat Transfer in Multiphase Flow in the CFX-Solver Modeling Guide](#).

#### 13.4.2.2.2. Heat Transfer: Option

Depending on your simulation, the following heat transfer options are possible. For details, see [Heat Transfer in the CFX-Solver Modeling Guide](#).

- `None`: Not available for compressible fluids, since a temperature is required at which to evaluate the fluid properties.
- `Isothermal`: Not available for reacting fluids.
- `Thermal Energy`: Models the transport of enthalpy through the fluid and is suitable for modeling heat transfer in low-speed flows. For details, see [The Thermal Energy Equation in the CFX-Solver Theory Guide](#).

The effective **Turbulent Prandtl Number** may be customized by selecting the **Turbulent Flux Closure for Heat Transfer** check box in the **Turbulence** settings. For details, see [Turbulent Flux Closure for Heat Transfer \(p. 179\)](#).

- **Total Energy:** Includes high-speed energy effects. You should include the viscous work term in the energy equation (select **Heat Transfer > Incl. Viscous Work Term**). For details, see [The Total Energy Equation in the CFX-Solver Theory Guide](#).

The effective **Turbulent Prandtl Number** may be customized by selecting the **Turbulent Flux Closure for Heat Transfer** check box in the **Turbulence** settings. For details, see [Turbulent Flux Closure for Heat Transfer \(p. 179\)](#).

- **Fluid Dependent:** Is used to set different heat transfer models for each fluid in a multiphase simulation. A heat transfer model is then set for each fluid on the **Fluid Specific Models** tab. This option cannot be used when **Homogeneous Model** is selected for the heat transfer model.

---

### Important:

If a compressible transient flow is undertaken with only one iteration per time step, then the solution can be incorrect if the **Heat Transfer** option is not set to **Total Energy**, or if heat transfer is not included in the simulation. This is due to the CFX-Solver not extrapolating the pressure at the start of the time step in these circumstances. This means that density is not extrapolated, and so the solver cannot calculate an accurate value for the time derivative of density on the first iteration. The workaround for this problem is to either run with at least two iterations per time step, or to use the **Total Energy** option for **Heat Transfer**.

---

## 13.4.2.3. Turbulence

Advice on which turbulence model is appropriate for your simulation and a description of each model can be reviewed. For details, see:

- [Turbulence and Near-Wall Modeling in the CFX-Solver Modeling Guide](#)
- [Turbulence Modeling in Multiphase Flow in the CFX-Solver Modeling Guide](#)
- [Turbulence Models in the CFX-Solver Theory Guide](#).

The following topics are discussed:

- 13.4.2.3.1. Homogeneous Model
- 13.4.2.3.2. Turbulence: Option
- 13.4.2.3.3. High Speed (compressible) Wall Heat Transfer Model
- 13.4.2.3.4. Buoyancy Turbulence
- 13.4.2.3.5. Wall Function
- 13.4.2.3.6. Turbulent Flux Closure for Heat Transfer
- 13.4.2.3.7. Transitional Turbulence
- 13.4.2.3.8. Advanced Turbulence Control

### 13.4.2.3.1. Homogeneous Model

If you have not selected **Homogeneous Model** under **Multiphase Options**, then **Homogeneous Model** under **Turbulence** frame will be available.

If selected, this will solve a single turbulence field for an inhomogeneous simulation. There will be no fluid-specific turbulence data to set. For details, see [Homogeneous Turbulence in Inhomogeneous Flow in the CFX-Solver Modeling Guide](#).

If you do not select this check box, then you will usually select `Fluid Dependent` and specify turbulence data on the fluid-specific tabs. Alternatively, the Laminar model can be picked to apply to all fluids (this is not homogeneous turbulence).


Homogeneous multiphase flow always uses homogeneous turbulence; therefore, you only need select the turbulence model to use.

### 13.4.2.3.2. Turbulence: Option

You can select one of the following turbulence models:

---

#### Note:

The  icon to the right of the drop-down list can be used to select a turbulence model from an expanded list.

---

- `None (Laminar)`: Turbulence is not modeled. This should only be used for laminar flow. Of the combustion models, only Finite Rate Chemistry is available for laminar flow. For details, see [The Laminar Model in the CFX-Solver Modeling Guide](#).
- `k-Epsilon`: A standard fluid model that is suitable for a wide range of simulations. For details, see [The k-epsilon Model in the CFX-Solver Modeling Guide](#).
- `Fluid Dependent`: Allows you to set different turbulence models for each fluid in the domain. If this option is selected, the turbulence model for each fluid is set in the **Fluid Specific Models** tab. This is only available for multiphase simulations when **Homogeneous Model** is not selected.
- `Shear Stress Transport`: Recommended for accurate boundary layer simulations. For details, see [The k-omega and SST Models in the CFX-Solver Modeling Guide](#).
- `GEKO`: A generalized k-Omega model that, depending on how it is tuned, can make predictions over a wide range of flow scenarios. For details, see [GEKO model in the CFX-Solver Modeling Guide](#).
- `Omega Reynolds Stress / BSL Reynolds Stress`: For details, see [Omega-Based Reynolds Stress Models in the CFX-Solver Modeling Guide](#).
- `QI / SSG / LRR Reynolds Stress`: Provides high accuracy for some complex flows. For details, see [Reynolds Stress Turbulence Models in the CFX-Solver Theory Guide](#).
- `Zero Equation`: Only the Finite Rate Chemistry combustion model is available when using the zero equation turbulence model. For details, see [The Zero Equation Model in the CFX-Solver Modeling Guide](#).
- `RNG k-Epsilon`: A variation of the k-epsilon model.
- `k-Omega / BSL`: The SST model is often preferred to this model.

- `k epsilon EARSM / BSL EARSM`: These models are a simplified version of the Reynolds stress models with application to problems with secondary flows as well as flows with streamline curvature and/or system rotation. For details, see [Explicit Algebraic Reynolds Stress Model in the CFX-Solver Theory Guide](#)
- `LES Smagorinsky / LES WALE / LES Dynamic Model`: Available for transient simulation only. For details, see [The Large Eddy Simulation Model \(LES\) in the CFX-Solver Modeling Guide](#).
- `Detached Eddy Simulation`: Available for transient simulation only. For details, see [The Detached Eddy Simulation Model \(DES\) in the CFX-Solver Modeling Guide](#).
- `Stress Blended Eddy Simulation`: Available for transient simulation only. This is a refinement of Detached Eddy Simulation. For details, see [The Stress-Blended Eddy Simulation \(SBES\) Model in the CFX-Solver Modeling Guide](#).
- `SAS SST`: Available for transient simulation only. For details see, [The Scale-Adaptive Simulation \(SAS\) in the CFX-Solver Modeling Guide](#).
- `Eddy Viscosity Transport Equation`: A one-equation variation of the k-epsilon model. For details, see [The Eddy Viscosity Transport Model in the CFX-Solver Theory Guide](#).

#### 13.4.2.3.3. High Speed (compressible) Wall Heat Transfer Model

For details on the **High Speed (compressible) Wall Heat Transfer Model** setting, see [Treatment of Compressibility Effects in the CFX-Solver Theory Guide](#).

#### 13.4.2.3.4. Buoyancy Turbulence

Buoyancy Turbulence is available for two (or more) equation turbulence models. For details, see [Buoyancy Turbulence in the CFX-Solver Modeling Guide](#).

#### 13.4.2.3.5. Wall Function

The wall function is automatically set depending on the turbulence model selected. Therefore, you will not need to change this setting. For multiphase flow, if the fluid dependent turbulence model option is selected, the wall function option appears on the fluid- specific tabs. The Laminar and zero equation turbulence models do not use wall functions. For details, see [Modeling Flow Near the Wall in the CFX-Solver Modeling Guide](#).

#### 13.4.2.3.6. Turbulent Flux Closure for Heat Transfer

**Turbulent Flux Closure for Heat Transfer** is available for turbulent flow with the Thermal Energy or Total Energy heat transfer model.

Here, the options are:

- `Eddy Diffusivity`

For the **Eddy Diffusivity** option, you specify a value for the turbulent Prandtl number (**Turb. Prandtl Num.**).

- `Anisotropic Diffusion` (only available for some Reynolds Stress turbulence models)

For the `Anisotropic Diffusion` option, there are two parameters: anisotropic diffusion coefficient (**Ani. Diffusion Coeff.**) and cross derivative weighting (**Cross Deriv. Coeff.**).

For details, see [Heat Transfer in the CFX-Solver Modeling Guide](#), [Turbulent Flux Closure in the CFX-Solver Modeling Guide](#), and [Turbulent Flux Closure for Heat Transfer in the CFX-Solver Theory Guide](#).

### 13.4.2.3.7. Transitional Turbulence

For details on the **Transitional Turbulence** settings, see [Ansys CFX Laminar-Turbulent Transition Models in the CFX-Solver Modeling Guide](#).

### 13.4.2.3.8. Advanced Turbulence Control

The available **Advanced Turbulence Control** settings for turbulence modeling depend on the turbulence model. The settings can be used to specify the coefficients for the selected turbulence model. These coefficients are described in [Table 13.2: Advanced Turbulence Parameters \(p. 180\)](#):

**Table 13.2: Advanced Turbulence Parameters**

Parameter	Description
SST Reattachment Modification	Available only when <b>Turbulence &gt; Option</b> is set to <code>Shear Stress Transport</code> . For details, see <a href="#">The Reattachment Modification (RM) Model in the CFX-Solver Theory Guide</a> .
Eddy Viscosity	Enables you to specify a value that will be picked up by the specified turbulence model; in this case the internal algorithm will not be used.
BC TKI Factor	A factor for turbulence intensity used in turbulence boundary conditions based on "Autocompute Length Scale", such as "Default Intensity and Autocompute Length Scale" (default = 1000).  For more details, see <a href="#">Inlet (Subsonic) in the CFX-Solver Theory Guide</a> .
BetaStar Coefficient	Model coefficient; appears when an omega-based turbulence model is selected (default = 0.09).  Can appear in several equations. For example, see <a href="#">Equation 2.36 in the CFX-Solver Theory Guide</a> or <a href="#">Equation 2.51 in the CFX-Solver Theory Guide</a> .
Cmu Coefficient	Model coefficient; appears when an epsilon-based turbulence model is selected (default = 0.09).  See <a href="#">Equation 2.22 in the CFX-Solver Theory Guide</a> .
Compressible Production	For details, see <a href="#">The k-epsilon Model in Ansys CFX in the CFX-Solver Theory Guide</a> .
A1 Coefficient	Model coefficient of the SST turbulence model (default = 0.31).  See <a href="#">Equation 2.60 in the CFX-Solver Theory Guide</a> .

Curvature Correction	<p>For details, see:</p> <ul style="list-style-type: none"> <li>• <a href="#">Curvature Correction for Two-Equation Models in the CFX-Solver Modeling Guide</a></li> <li>• <a href="#">Explicit Algebraic Reynolds Stress Model in the CFX-Solver Theory Guide</a></li> </ul>
Corner Coefficient	<p>Model coefficient of the GEKO turbulence model.</p> <p>For details, see <a href="#">Corner Correction in the CFX-Solver Modeling Guide</a>.</p>
High Lift Modification	<p>For improved modeling of high-lift devices (for example, airfoils and wings). Leads to stronger flow separation and reduced maximum lift.</p> <p>Available only when <b>Turbulence &gt; Option</b> is set to <code>Shear Stress Transport</code>.</p> <p>The High Lift Correction option has an adjustable <b>CHL Coefficient</b>, which has a range from 0.5 to 1.0 and a default value of 0.9. For details, see <a href="#">Modifications to SST for High Lift Devices in the CFX-Solver Theory Guide</a>.</p>
K Coefficients	<p>For details, see:</p> <ul style="list-style-type: none"> <li>• <a href="#">The k-epsilon Model in Ansys CFX in the CFX-Solver Theory Guide</a></li> <li>• <a href="#">The k-omega Models in Ansys CFX in the CFX-Solver Theory Guide</a></li> </ul>
k-Omega Regime	<p>For details, see:</p> <ul style="list-style-type: none"> <li>• <a href="#">The Baseline (BSL) k-omega Model in the CFX-Solver Theory Guide</a></li> <li>• <a href="#">The Shear Stress Transport (SST) Model in the CFX-Solver Theory Guide</a></li> </ul>
k-Epsilon Regime	<p>For details, see:</p> <ul style="list-style-type: none"> <li>• <a href="#">The Baseline (BSL) k-omega Model in the CFX-Solver Theory Guide</a></li> <li>• <a href="#">The Shear Stress Transport (SST) Model in the CFX-Solver Theory Guide</a></li> </ul>
Geko Coefficients	<p>For details, see <a href="#">The GEKO Model in the CFX-Solver Theory Guide</a></p>

Epsilon Coefficients	For details, see <a href="#">The k-epsilon Model in Ansys CFX in the CFX-Solver Theory Guide</a> .
Production Limiter	For details, see <a href="#">Production Limiters in the CFX-Solver Theory Guide</a> .
DES Coefficients	Model parameters for the Detached Eddy Simulation model. For details, see: <ul style="list-style-type: none"> <li>• <a href="#">Detached Eddy Simulation Theory in the CFX-Solver Theory Guide</a></li> <li>• <a href="#">The Detached Eddy Simulation Model (DES) in the CFX-Solver Modeling Guide</a></li> </ul>
SBES Coefficients	Model parameters for the Stress-Blended Eddy Simulation model. For details, see: <ul style="list-style-type: none"> <li>• <a href="#">The Stress-Blended Eddy Simulation (SBES) Model in the CFX-Solver Modeling Guide</a></li> <li>• <a href="#">Stress-Blended Eddy Simulation (SBES) in the CFX-Solver Theory Guide</a></li> </ul>
Omega Coefficients	For details, see <a href="#">The k-omega Models in Ansys CFX in the CFX-Solver Theory Guide</a> .
High Wave Number Damping	Enables you to specify the limiter coefficient ( $C_G$ ) in <a href="#">Equation 2.215 in the CFX-Solver Theory Guide</a> for the SAS SST model (default = 0.11).

### 13.4.2.4. Reaction or Combustion Model

If the fluid material is defined as **Option** is `Material Definition` and **Composition Option** is `Reacting Mixture`, or if a reacting mixture from the material library has been selected as the material for one of the domain fluids, then you can select a combustion model as:

- Eddy Dissipation
- Finite Rate Chemistry
- Finite Rate Chemistry and Eddy Dissipation
- PDF Flamelet
- BVM (Partially Premixed)
- Extended Coherent Flame Model
- Fluid Dependent (multiphase only)

Only `Finite Rate Chemistry` is available when `Laminar` or `Zero Equation` turbulence model is used.

In multiphase simulations, when `Fluid Dependent` is selected, a different combustion model can be used for each reacting fluid in the simulation. If the homogeneous multiphase model is used, all fluids must be reacting mixtures that include reactions to enable a combustion to be modeled.

If the fluid material is defined as a reacting mixture from the material library, then the available combustion models are filtered in order to be compatible with the reactions specified in the reacting material.

If the fluid material is defined as **Option** is `Material Definition` and **Composition Option** is `Reacting Mixture`, then the complete list of combustion models is presented and the reactions list for the mixture has to be specified. Only those reactions from the material library will be available that are compatible with the selected combustion model.

Depending on the selected combustion model, additional options (such as `Autoignition Model`, `NO Model`, and `Chemistry Post-Processing`) and parameters may be available. For details, see [Combustion Modeling in the CFX-Solver Modeling Guide](#).

#### 13.4.2.4.1. Soot Model

When a combustion model is selected, you can optionally enable the Magnussen soot model to account for the formation of soot. In multiphase simulations, this model appears on the fluid-specific tab for each fluid that uses a combustion model.

A **Fuel** and **Soot Material** is required, and the following optional parameters can also be set:

- **Fuel Consumption Reaction**
- **Fuel Carbon Mass Fraction**
- **Soot Density**
- **Soot Particle Mean Diameter**

For details, see [Soot Model in the CFX-Solver Modeling Guide](#).

#### 13.4.2.5. Thermal Radiation Model



If a heat transfer model other than `None` has been selected, you can model thermal radiation. If a radiation model is selected, you must make sure that the radiation properties for that fluid have been set in the **Material** details view. For details, see [Material Properties Tab \(p. 392\)](#). Radiation is not supported for multiphase simulations in CFX.

The four radiation models available in CFX are:

- `Rosseland`
- `P1`
- `Discrete Transfer`
- `Monte Carlo`





A **Spectral Model** can be selected for all radiation models. If the `Multigray` or `Weighted Sum of Gray Gases` representation is selected for the **Spectral Model**, then you should create the required number of gray gases.

1. Click *Add new item*  to add a new gray gas. (You can click *Delete*  to delete a highlighted gray gas.)
2. Set the **Weight** and **Absorption Coefficient** for each gray gas.

For details, see [Multigray/Weighted Sum of Gray Gases in the CFX-Solver Modeling Guide](#).

Alternatively, if the **Multiband** representation is selected, you should create **Spectral Bands**:

1. Click *Add new item*  to add a new spectral band. (You can click *Delete*  to delete a highlighted spectral band.)
2. Set **Option** to either `Frequency`, `Wavelength` or `Wavenumber`.
3. Enter upper and lower limits for the option selected.

This defines the range of the spectral band. For details, see:

- [Multiband in the CFX-Solver Modeling Guide](#)
- [Spectral Model in the CFX-Solver Modeling Guide](#)
- [Radiation Modeling in the CFX-Solver Modeling Guide](#).

### 13.4.2.6. Electromagnetic Model

The **Electromagnetic Model** enables you to define:

#### Electric Field Model

**Option** can be set to `None`, `Electric Potential`, or `User Defined`. For details, see [Electric Field in the CFX-Solver Modeling Guide](#).

#### Magnetic Field Model

**Option** can be set to `None`, `Magnetic Vector Potential`, or `User Defined`.

If a user-defined model is selected, you must make sure that the electromagnetic properties have been set in the **Material** details view. For details, see [Material Properties Tab \(p. 392\)](#). Electromagnetic models are supported for multiphase simulations only if homogeneous.

For more information on electromagnetic theory, see [Electromagnetic Hydrodynamic Theory in the CFX-Solver Theory Guide](#).

### 13.4.2.7. Component Details

If your fluid contains more than one component (that is, you are using a variable composition or reacting mixture, or HCF fuel, created in the **Material** details view), then **Component Details** will need to be set on the **Fluid Models** tab. If using the Algebraic Slip Multiphase model (ASM), the settings are specified in this view as well. For details, see [Algebraic Slip Model \(ASM\) in the CFX-Solver Modeling Guide](#). When a non-ASM multiphase model is used, the **Component Details** form appears on the fluid-specific tabs.

- Select each component in turn and set the required option.
- Select the type of equation to solve for this component as `Automatic`, `Transport Equation`, `Constraint`, `Algebraic Equation` or `Algebraic Slip`. A description of the multiphase model is available in:
  - [Algebraic Slip Model \(ASM\) in the CFX-Solver Modeling Guide](#)
  - [Component Domain Settings in the CFX-Solver Modeling Guide](#).
- If you have chosen to solve a transport equation for the component, you can optionally enter a value for **Kinematic Diffusivity**. If you do not set **Kinematic Diffusivity**, then the **Bulk Viscosity** value is used.
- If you have chosen to solve a transport equation or an Algebraic Slip component, you can optionally set a component dependent **Turbulent Schmidt Number** by enabling the **Turbulent Flux Closure** check box. If you do not select **Turbulent Flux Closure**, the value from **Turbulence > Turbulent Flux Closure for Heat Transfer** will be used.

The **Component Details** specify the model used to calculate the mass fraction of each component throughout the domain. For details, see [Component Domain Settings in the CFX-Solver Modeling Guide](#).

### 13.4.2.8. Additional Variable Details

If you have defined any Additional Variables from the Additional Variable details view, then you can choose to include or exclude them here. An Additional Variable is included by selecting it from the **Additional Variables Details** list and then enabling the check box with the name of the Additional Variable. For details, see [Additional Variables \(p. 405\)](#).

If an Additional Variable is included, you must select how the Additional Variable level is calculated.

For single phase flows, the CFX-Solver can solve different variations of the conservation equations for the variable including `Transport Equation`, `Diffusive Transport Equation` or `Poisson Equation`.

For multiphase flows, the CFX-Solver can solve different variations of the conservation equations for the variable including `Homogeneous Transport Equation`, `Homogeneous Diffusive Transport Equation`, `Homogeneous Poisson Equation` or `Fluid Dependent`. When the `Fluid Dependent` option is selected, the Additional Variable model details can be set for each fluid on the **Fluid Specific Models** tab.

If a transport equation is being solved for an Additional Variable, the **Turbulent Flux Closure** may be optionally specified for turbulent flow. If you do not select **Turbulent Flux Closure** for the Ad-

ditional Variable, the default is **Option** is **Eddy Diffusivity** and the **Turb. Schmidt Num.** is set to **0.9**.

Alternatively, you can define the variable value algebraically using CEL by selecting the **Algebraic Equation** option. Note that the **Algebraic Equation** option is not available for homogeneous Additional Variables. In addition, only specific Additional Variables are permitted to be homogeneous. For details, see [Additional Variables in the CFX-Solver Modeling Guide](#).

### 13.4.3. Polydispersed Fluid Tab

The **Polydispersed Fluid** tab for a domain object contains settings that define the properties of polydispersed (MUSIG) fluids. It is accessible by selecting the **Polydispersed Fluid** option for **Fluid and Particle Definitions...** > <fluid definition> > **Morphology** on the **Basic Settings** tab.

For details, see [Polydispersed, Multiple Size Group \(MUSIG\) Model in the CFX-Solver Modeling Guide](#).

### 13.4.4. Fluid Specific Models Tab

The **Fluid Specific Models** tab contains settings for fluid-specific properties. It appears for multiphase simulations and when particles are included in the domain.

Which options are available depends on the simulation set up, including the type and number of fluids used in the simulation (for example, single or multicomponent, single or multiphase, reacting or non-reacting), and whether Additional Variables have been created.

#### 13.4.4.1. Fluid List Box

This list box is used to select a fluid (which can, in some cases, represent a solid). The rest of the tab contains settings for the selected fluid.

#### 13.4.4.2. Kinetic Theory

The **Kinetic Theory** settings control the solid particle collision model. When you set **Kinetic Theory** to **Kinetic Theory**, you should set the granular temperature model and radial distribution function. CFX-Pre will also set the **Solid Pressure Model**, **Solid Bulk Viscosity**, and the **Solid Shear Viscosity** settings to **Kinetic Theory**.

For details on these settings:	See:
Granular temperature model	<a href="#">Granular Temperature</a>
Radial distribution function	<a href="#">Kinetic Theory Models for Solids Pressure</a>
Solid pressure model	<a href="#">Solids Pressure</a>
Solid bulk viscosity	<a href="#">Solids Bulk Viscosity</a>
Solid shear viscosity	<a href="#">Solids Shear Viscosity</a>

For modeling information about solid particle collision models, see [Solid Particle Collision Models in the CFX-Solver Modeling Guide](#).

For theoretical information about solid particle collision models, see [Solid Particle Collision Models in the CFX-Solver Theory Guide](#).

### 13.4.4.3. Heat Transfer Model

If you have set **Heat Transfer** to `Fluid Dependent` on the **Fluid Models** tab, **Heat Transfer Model** options appear on the fluid-specific tabs for each Eulerian phase. If the heat transfer occurs between two fluids, then additional information must be entered on the **Fluid Pairs** tab. Information on heat transfer between phases is available in [Interphase Heat Transfer in the CFX-Solver Modeling Guide](#).

Some of the available **Heat Transfer Model** options are similar to those in the single-phase case. See [Heat Transfer \(p. 176\)](#) for details on the following options:

- None
- Isothermal
- Thermal Energy
- Total Energy

Additional **Heat Transfer Model** options include:

- `Particle Temperature`: Can only be used for particle transport fluids and solids. For details, see [Heat Transfer in the CFX-Solver Modeling Guide](#).
- `Saturation Temperature`: Can only be used for a fluid that is a component of a homogeneous binary mixture. This model is applicable only when the heat transfer modeling is inhomogeneous. For its most common use, one of the phases of a two-phase flow is set to use `Saturation Temperature` while the other uses a different **Heat Transfer Model** option, such as `Thermal Energy` or `Total Energy`. The `Saturation Temperature` option behaves similarly to the `Isothermal` option except that, instead of the fluid temperature being forced to be the specified **Fluid Temperature** value, the fluid temperature is forced to be the saturation temperature as computed from the saturation properties of the homogeneous binary mixture.
- `Small Droplet Temperature`: Can only be used for a fluid that has **Morphology > Option** set to `Droplets with Phase Change`. For details, see [Droplet Condensation Model in the CFX-Solver Modeling Guide](#).

### 13.4.4.4. Turbulence Model

If you have set **Turbulence** to `Fluid Dependent` on the **Fluid Models** tab, the **Turbulence Model** option appears on the fluid-specific tabs for each Eulerian phase. The models available are similar to those available in single-phase simulations, with the following exceptions:

- For dispersed fluid, or dispersed/polydispersed solid phases, only the `Dispersed Phase Zero Equation`, `Laminar` or `Zero Equation` models are available. The `Dispersed Phase Zero Equation` model is the recommended choice. For details, see [Phase-Dependent Turbulence Models in the CFX-Solver Modeling Guide](#).
- The `LES` and `DES` models are available for transient simulations for the continuous phase.

For details, see [Turbulence](#) (p. 177).

### 13.4.4.5. Turbulent Wall Functions

The turbulent wall functions are selected automatically, but apply only to the current fluid. For details, see [Wall Function](#) (p. 179).

### 13.4.4.6. Combustion Model

If you have set the reaction or combustion model to **Fluid Dependent** on the **Fluid Models** tab, the **Reaction or Combustion Model** option can appear on the **Fluid Specific Models** tab for each Eulerian phase. You will only be able to pick a combustion model for fluids that are reacting mixtures. The models available are similar to those available in single-phase simulations. For details, see:

- [Phasic Combustion in the CFX-Solver Modeling Guide](#)
- [Reaction or Combustion Model](#) (p. 182).

### 13.4.4.7. Erosion Model

The erosion properties specified on this form are applied to all wall boundaries. The wall boundaries can also have erosion properties set to override the global settings specified here. For details, see [Erosion in the CFX-Solver Modeling Guide](#).

### 13.4.4.8. Fluid Buoyancy Model

This option is available for multiphase buoyant flows and/or buoyant flows that include particles (set on the **Basic Settings** tab). For details, see [Buoyancy in Multiphase Flow in the CFX-Solver Modeling Guide](#).

### 13.4.4.9. Solid Pressure Model

This is available for `Dispersed Solid` Eulerian phases (phases with `Dispersed Solid` as the **Morphology** setting). For details, see [Solid Pressure Force Model in the CFX-Solver Modeling Guide](#).

### 13.4.4.10. Component Details

This is available for each Eulerian phase in the simulation that is a mixture of more than one component. It does not apply to fluids or solids using the particle tracking model. The options available are the same as those on the **Fluid Models** tab in a single-phase simulation. For details, see [Component Details](#) (p. 185).

If the component transfer occurs between two fluids, then additional information must be entered on the **Fluid Pairs** tab. This is only possible when more than one multicomponent fluid exists in a simulation. For details, see [Interphase Species Mass Transfer in the CFX-Solver Modeling Guide](#).

### 13.4.4.11. Additional Variable Models

This is available for each Eulerian phase in the simulation when Additional Variables have been created as well as selected and set to `Fluid Dependent` on the **Fluid Models** tab. The options

available are the same as those on the **Fluid Models** tab in a single-phase simulation. It does not apply to fluids or solids using the particle tracking model. For details, see [Additional Variable Details \(p. 185\)](#).

If the Additional Variable transfer occurs between two fluids, then additional information must be entered on the **Fluid Pairs** tab. This is possible only when more than one phase in a simulation includes Additional Variables. For details, see [Additional Variables in Multiphase Flow in the CFX-Solver Modeling Guide](#).

### 13.4.5. Fluid Pair Models Tab

The **Fluid Pairs** tab appears for multiphase simulations and/or when particles are included in the domain. It is used to specify how the fluids interact in a multiphase simulation and how particles interact with the fluids when particles are included.

For details, see [Fluid Pairs in the CFX-Solver Modeling Guide](#).

#### 13.4.5.1. Fluid Pair List box

The top of the **Fluid Pair Models** tab shows a list of all the phase pairs in the simulation. A phase pair will exist when the morphology of the pair is **Continuous Fluid | Continuous Fluid**, **Continuous Fluid | Dispersed Fluid** or **Continuous Fluid | Dispersed Solid**. If particles have also been included, then a pair will exist for each **Continuous Fluid | Particle** pair. You should select each pair in turn and set the appropriate options.

The options available will vary considerably depending on your simulation. Many options are not available when the homogeneous multiphase model is used. This is because the interphase transfer rates are assumed to be very large for the homogeneous model and do not require further correlations to model them.

#### 13.4.5.2. Particle Coupling

This only applies to **Continuous Fluid | Particle** pairs. For details, see [Particle Fluid Pair Coupling Options in the CFX-Solver Modeling Guide](#).

#### 13.4.5.3. Surface Tension Coefficient

You can optionally provide a **Surface Tension Coefficient**. This should be set in either of the following two cases:

- For a **Continuous Fluid | Dispersed Fluid** pair when you want to model the Drag Force using either the *Grace* or *Ishii Zuber* models. The flow must also be Buoyant to enable these models to be selected. For details, see [Interphase Drag for the Particle Model in the CFX-Solver Modeling Guide](#).
- When you want to use the surface tension model. This model is only available when *Standard* has been selected as the **Free Surface Model** on the **Fluid Models** tab.

You can set a **Surface Tension Coefficient** in other cases, but it will not be used in your simulation. It does not apply to **Continuous Fluid | Particle** pairs.

For details, see [Surface Tension in the CFX-Solver Modeling Guide](#).

#### 13.4.5.4. Surface Tension Force Model

You can model the surface tension force that exists at a free surface interface. This model applies to all morphology combinations for Eulerian | Eulerian pairs. You must also specify a **Surface Tension Coefficient** and select the **Primary Fluid**. For liquid-gas free surface flows, the primary fluid should be the liquid phase.

For details, see [Surface Tension in the CFX-Solver Modeling Guide](#).

#### 13.4.5.5. Interphase Transfer Model

This can be selected as one of the following:

##### 13.4.5.5.1. Particle Model

This model assumes a continuous phase fluid containing particles of a dispersed phase fluid or solid. It is available when the morphology of the pair is **Continuous Fluid | Dispersed Fluid** or **Continuous Fluid | Dispersed Solid**. For details, see [The Particle Model in the CFX-Solver Modeling Guide](#).

##### 13.4.5.5.2. Mixture Model

This model is only available when the morphology of the pair is **Continuous Fluid | Continuous Fluid**. An **Interface Length Scale** is required. It is usually used as a first approximation or combined with a custom interface transfer model. For details, see [The Mixture Model in the CFX-Solver Modeling Guide](#).

##### 13.4.5.5.3. Free Surface Model

This model is available when the free surface model is selected. For details, see [The Free Surface Model in the CFX-Solver Modeling Guide](#). For free surface flow, the particle model is also available if the phase pair is **Continuous Fluid | Dispersed Fluid**, and the mixture model is also available if the phase pair is **Continuous Fluid | Continuous Fluid**.

##### 13.4.5.5.4. None

For homogeneous multiphase flow in which there is no interphase transfer of any type, the interphase transfer model is not relevant and **None** may be selected.

#### 13.4.5.6. Momentum Transfer

There are a variety of momentum transfer that can be modeled, including the drag force and non-drag forces, which include lift force, virtual mass force, wall lubrication force and turbulent dispersion force.

##### 13.4.5.6.1. Drag Force

This option applies to all morphology pair combinations including **Continuous Fluid | Particle** pairs, but does not apply when the **Homogeneous** multiphase model is active.



There are many drag force models available in CFX, but most are only applicable to certain morphology combinations. For **Continuous Fluid | Particle** pairs, the available options are:

- The Schiller–Naumann drag model.

For details, see [Interphase Drag in the CFX-Solver Modeling Guide](#).

- The Drag Coefficient.

For details, see [Drag Force for Particles in the CFX-Solver Modeling Guide](#).

- The Ishii–Zuber drag model.

For details, see [Sparsely Distributed Fluid Particles: Ishii-Zuber Drag Model in the CFX-Solver Modeling Guide](#) and [Densely Distributed Fluid Particles: Ishii-Zuber Drag Model in the CFX-Solver Modeling Guide](#).

- The Grace drag model.

For details, see [Sparsely Distributed Fluid Particles: Grace Drag Model in the CFX-Solver Modeling Guide](#) and [Densely Distributed Fluid Particles: Grace Drag Model in the CFX-Solver Modeling Guide](#).

#### 13.4.5.6.2. Particle User Source

The **Particle User Source** check box is available when any User Routines of type Particle User Routines exist. For details, see:

- [Particle User Routines \(p. 441\)](#)
- [Particle User Sources in the CFX-Solver Modeling Guide](#).

#### 13.4.5.6.3. Lift Force

The lift force is only applicable to the `Particle Model`, which is active for **Continuous Fluid | Dispersed (Fluid, Solid)** and **Continuous Fluid | Polydispersed Fluid**. For details, see [Lift Force in the CFX-Solver Modeling Guide](#).

#### 13.4.5.6.4. Virtual Mass Force

This option applies to **Continuous Fluid | Dispersed Fluid** pairs using the `Particle Model`, and to **Continuous Fluid | Particle** pairs, but does not apply when the Homogeneous multiphase model is active. For details, see [Virtual Mass Force in the CFX-Solver Modeling Guide](#).

#### 13.4.5.6.5. Wall Lubrication Force

This option is only applicable to the `Particle Model`. For details, see [Wall Lubrication Force in the CFX-Solver Modeling Guide](#).

#### 13.4.5.6.6. Turbulent Dispersion Force

This applies to **Continuous Fluid | Dispersed Fluid**, **Continuous Fluid | Polydispersed Fluid** and **Continuous Fluid | Dispersed Solid** pair combinations for **Eulerian | Eulerian** pairs, but does not apply when the Homogeneous multiphase model is active. In these cases, the Lopez



de Bertodano model is used. For details, see [Interphase Turbulent Dispersion Force in the CFX-Solver Modeling Guide](#).

When particle tracking is used, the turbulent dispersion force also applies to **Continuous Fluid | Particle** pairs. In these cases, the **Particle Dispersion** models is used. For details, see [Turbulent Dispersion Force in the CFX-Solver Modeling Guide](#).

#### 13.4.5.6.7. Pressure Gradient Force

This option is only available for Particle Tracking simulations. For details, see [Pressure Gradient Force in the CFX-Solver Modeling Guide](#).

#### 13.4.5.7. Turbulence Transfer

This model is available for **Continuous Fluid | Dispersed Fluid**, **Continuous Fluid | Polydispersed Fluid** and **Continuous Fluid | Dispersed Solid** pair combinations for **Eulerian | Eulerian** pairs, but does not apply when the **Homogeneous** multiphase model is active and is not available for **Continuous Fluid | Particle** pairs. For details, see [Turbulence Enhancement in the CFX-Solver Modeling Guide](#).

#### 13.4.5.8. Heat Transfer

This applies to all morphology combinations for **Eulerian | Eulerian** and **Continuous Fluid | Particle** pairs, but does not apply when the **Homogeneous** multiphase model is active.

For details, see [Interphase Heat Transfer in the CFX-Solver Modeling Guide](#) for multiphase applications and [Interphase Heat Transfer in the CFX-Solver Modeling Guide](#) for particle transport modeling.

#### 13.4.5.9. Mass Transfer

Mass transfer can occur in homogeneous and inhomogeneous Eulerian multiphase flows. For such flows, you can set the **Mass Transfer** option to one of the following:

- None
- Specified Mass Transfer

This is an advanced option that allows you to define your own mass transfer sources. For details, see [User Specified Mass Transfer in the CFX-Solver Modeling Guide](#).

- Phase Change

This models mass transfer due to phase change, such as condensation, melting or solidification. For details, see [Thermal Phase Change Model in the CFX-Solver Modeling Guide](#).

Note that wall boiling settings are on the **Boundary Models** tab of the domain. For details, see [Boundary Models Tab \(p. 203\)](#).

- Cavitation

Vapor formation in low pressure regions of a liquid flow (cavitation) can be modeled using the Rayleigh Plesset model or, for advanced users, a user-defined model. For details, see [Cavitation Model in the CFX-Solver Modeling Guide](#).

### 13.4.5.10. Additional Variable Pairs

**Additional Variable Pairs** details describe the way in which Additional Variables interact between phases. It applies to all morphology combinations for **Eulerian | Eulerian** pairs, but does not apply when the **Homogeneous** multiphase model is active.

Only Additional Variable pairs where both are solved using the **Transport Equation** and have a **Kinematic Diffusivity** value set can be transferred between phases. These options are set on the fluid-specific tabs for each phase.

For example, consider two phases, Phase A and Phase B, and two Additional Variables, AV1 and AV2.

- AV1 uses a **Transport Equation** with diffusion in Phase A and is unused in Phase B.
- AV2 uses an **Algebraic Equation** in Phase A and uses a **Transport Equation** with diffusion in Phase B.

Additional Variable interphase transfer can only occur between Phase A / AV1 and Phase B / AV2

For details, see [Additional Variables in Multiphase Flow in the CFX-Solver Modeling Guide](#).

### 13.4.5.11. Component Pairs

#### 13.4.5.11.1. Eulerian | Eulerian Pairs

You can model transfer of components between phases for **Eulerian | Eulerian** pairs, when both fluids are multicomponent mixtures of any type (except fixed composition mixtures). Interphase species transfer (or component transfer) may be specified between any two components, A and B, in fluids,  $\alpha$  and  $\beta$ , respectively, subject to the following condition:

- The mass fractions of A and B must both be determined from transport equations.
- It is not possible to specify the species transfer of a component whose mass fraction is determined algebraically, or from the constraint equation.

For more information on implementing component pairs in CFX-Solver, see [Component Pairs in the CFX-Solver Modeling Guide](#).

Mixtures are created in the **Material** details view, which is described in [Material Details View: Variable Composition Mixture \(p. 396\)](#). Component transfer enables you to model processes such as evaporation, absorption, and dissolution.

To specify the component transfer model, you should select the component pair from the list on the **Fluid Pairs** tab and then select the associated toggle. The first component of the component pair corresponds to the first fluid in the fluid pairs list.

**Option** can be set to Two Resistance or Ranz Marshall. For details, see:

- [Two Resistance Model in the CFX-Solver Modeling Guide](#)
- [Ranz Marshall in the CFX-Solver Modeling Guide.](#)

The choice of interfacial equilibrium model depends on the process that you are modeling. For details, see [Interfacial Equilibrium Models in the CFX-Solver Modeling Guide](#).

The `Fluid1` and `Fluid2` `Species Mass Transfer` options are used to choose a correlation to model the mass transfer coefficient on each side on the interface. For details, see [Species Mass Transfer Coefficients in the CFX-Solver Modeling Guide](#).

#### 13.4.5.11.2. Continuous | Particle Pairs

Selecting the toggle enables mass transfer between the two phases.

The options for mass transfer are:

- `Ranz Marshall`. For details, see [Ranz Marshall in the CFX-Solver Modeling Guide](#).
- `Liquid Evaporation Model`. For details, see [Liquid Evaporation Model in the CFX-Solver Modeling Guide](#). For oil evaporation, the `Light Oil` check box should be selected. For details, see [Liquid Evaporation Model: Oil Evaporation/Combustion in the CFX-Solver Modeling Guide](#).
- `None`

For details on these options, see:

- [Latent Heat in the CFX-Solver Modeling Guide](#)
- [Particle User Source in the CFX-Solver Modeling Guide](#).

The drop-down list will contain any `User Particle Routines` you have created. For details, see [Particle User Routines \(p. 441\)](#).

Mass transfer between a species in a particle phase and a species in the continuous phase is possible. For example, consider liquid water from a particle evaporating into gaseous H<sub>2</sub>O in a continuous phase mixture. The particle can be a pure substance or variable composition mixture.

#### 13.4.5.12. Particle Breakup

The **Particle Breakup** models enable you to simulate the breakup of droplets due to external aerodynamic forces. The droplet breakup models are set on a per fluid-pair basis. By default, the **Use Liu Dynamic Drag Modification** option is activated for the `TAB`, `ETAB` and `CAB` breakup models, whereas the **Use Schmehl Dynamic Drag Law** option is activated for the `Schmehl` breakup model. See [Particle Breakup Model in the CFX-Solver Modeling Guide](#) for details on the available particle breakup models.

#### 13.4.5.13. Particle Collision

The particle collision model enables you to simulate dense gas-solid flows with high mass-loading while the particle volume fraction is still low. Select either `Sommerfeld Collision Model` or `User Defined` and specify values for the particle collision parameters outlined below:

## Sommerfeld Collision Model

- **Coefficient of Restitution:** Enter a numerical quantity or CEL based expression to specify the value of coefficient of restitution for inter-particle collisions. A value of '1.0' means a fully elastic collision, while a value of '0.0' would result in an inelastic collision.
- **Static Friction Coefficient and Kinetic Friction Coefficient:** Enter a numerical quantity or CEL based expression to specify values of coefficients of friction for inter-particle collisions.

See [Implementation Theory in the CFX-Solver Theory Guide](#) for more information on setting up Coefficient of Restitution, Static Friction Coefficient, and Kinetic Friction Coefficient.

## User Defined

This option is available only if you have created a particle user routine to set up the model. Specify the name of **Particle User Routine** and select input arguments and type of particle variables returned to the user routine from the **Arguments** and **Variable List** drop-down list, respectively. See [Particle User Routines in the CFX-Pre User's Guide \(p. 441\)](#) for information on setting up a particle user routine.

For additional information, see [Particle Collision Model in the CFX-Solver Modeling Guide](#) and the following topics available under [Particle Collision Model in the CFX-Solver Theory Guide](#):

- [Introduction to the Particle Collision Model](#)
- [Implementation of a Stochastic Particle-Particle Collision Model in Ansys CFX](#) (includes the discussion on the implementation theory, particle variables, and virtual collision partner)
- [Particle Collision Coefficients Used for Particle-Particle Collision Model](#)
- [Range of Applicability of Particle-Particle Collision Model](#)
- [Limitations of Particle-Particle Collision Model in Ansys CFX](#)

## 13.4.6. Solid Models Tab

The **Solid Models** tab sets the models that apply to solid domains and to the solid material in a porous domain. The models chosen can vary between each domain, but if radiation is modeled, then all fluid domains must also model radiation. For details, see [Using Multiple Domains \(p. 166\)](#).

### 13.4.6.1. Heat Transfer

The **Thermal Energy** model and **Isothermal** model are available for the solid domain. If you do not want to model heat transfer for the domain, then set **Heat Transfer > Option** to **None**.

For details, see [Conjugate Heat Transfer in the CFX-Solver Modeling Guide](#).

### 13.4.6.2. Thermal Radiation Model

You can use only the `Monte Carlo` option to model radiation in a solid domain. The options available are the same as for the `Monte Carlo` model in a fluid domain. For details, see [Thermal Radiation Model \(p. 183\)](#).

You cannot model radiation in a porous domain.

### 13.4.6.3. Electromagnetic Model

The **Electromagnetic Model** enables you to define:

#### Electric Field Model

**Option** can be set to `None`, `Electric Potential`, or `User Defined`.

For a `User Defined` setting, you have to specify the electric field strength for the X, Y, and Z directions.

#### Magnetic Field Model

**Option** can be set to `None`, `Magnetic Vector Potential`, or `User Defined`.

For the `Magnetic Vector Potential` option, you can specify **External Magnetic Field** settings using Cartesian or cylindrical components. Using the `User Defined` option will enable you to specify the induced magnetic field model in the X, Y, and Z directions.

If a user-defined model is selected, you must make sure that the electromagnetic properties have been set in the **Material** details view. For details, see [Material Properties Tab \(p. 392\)](#). Electromagnetic models are supported for multiphase simulations only if homogeneous.

For more information on electromagnetic theory, see [Electromagnetic Hydrodynamic Theory in the CFX-Solver Theory Guide](#).

### 13.4.6.4. Additional Variables Models

See [Additional Variables](#).

### 13.4.6.5. Solid Motion

You can model the motion of a solid that moves relative to its reference frame by selecting the **Solid Motion** option and specifying a velocity.

You can specify the velocity using one of the following methods:

- Cartesian velocity components

You must specify values for **U**, **V**, and **W**.

- Cylindrical velocity components

You must specify values for **Axial Component**, **Radial Component**, and **Theta Component**. You must also specify an **Axis Definition**.

- Rotating

Specify an **Angular Velocity** and an **Axis Definition**.

The velocity that you specify is interpreted as being relative to the domain motion which is, in turn, relative to the coordinate frame; both of these are specified on the **Basic Settings** tab for the domain.

The motion of the solid is simulated by adding an advection term to relevant transport equations in the solid (for example, Energy and User Variables). For more details, see [Conjugate Heat Transfer in the CFX-Solver Theory Guide](#).

On interfaces to other domains (fluid-solid or solid-solid interfaces) the solid must move only tangentially to its surface. On an external boundary, if the solid has a velocity component normal to the surface, then consider activating the advection term(s) on the boundary condition for that surface, by visiting the **Boundary Details** tab and selecting **Solid Motion > Boundary Advection**. For details on setting up boundary advection on a wall, see [Solid Motion: Wall \(p. 225\)](#).

---

**Note:**

- Most solid motion cases will involve setting either non-stationary domain motion (on the **Basic Settings** tab) or activating the **Solid Motion** setting (on the **Solid Models** tab) but *not* both.
- If you have a solid with **Solid Motion** activated that meets a fluid domain at a fluid-solid interface, then you must explicitly set the wall boundary condition applied to the fluid side of the interface to have a wall velocity corresponding to the solid motion, as required.

---

## 13.4.7. Porosity Settings Tab

The **Porosity Settings** tab is where the general description of a porous domain is specified for the simulation.

### 13.4.7.1. Area Porosity

**Area Porosity** represents the fraction of physical area that is available for the flow to go through. The default setting is *Isotropic*.

### 13.4.7.2. Volume Porosity

**Volume Porosity** is the local ratio of the volume of fluid to the total physical volume.

### 13.4.7.3. Loss Models

The porous loss options are:

- Isotropic Loss

For the isotropic loss model, the loss may be specified using either one of the following methods:

- Permeability And Loss Coefficient
- Linear And Quadratic Resistance Coefficients
- Directional Loss

Specify the streamwise direction via Cartesian or cylindrical components (in the coordinate frame specified on the **Basic Settings** tab).

The loss in the streamwise direction can be specified using one of the following options:

- Permeability And Loss Coefficient
- Linear And Quadratic Resistance Coefficients
- No Loss

The loss in the transverse directions can be specified using one of the following options:

- Streamwise Coefficient Multiplier
- Permeability and Loss Coefficient
- Linear and Quadratic Resistance Coefficients
- No Loss
- None

When specifying the loss coefficients, it is important to set the **Loss Velocity Type** properly. For details, see [Porous Momentum Loss Models in the CFX-Solver Theory Guide](#).

#### 13.4.7.4. Contact Area Model

This setting is available only when the porous domain has a solid phase.

When there is more than one fluid in a porous domain that has a solid phase, specify the contact area model to indicate how the contact area between a given fluid and the solid is to be calculated.

The options are:

- Use Volume Fraction

This option makes use of the volume fraction of each fluid to calculate the contact area between that fluid and the solid.

- Fluid Dependent

This option enables the direct specification of the contact area fraction for each fluid. The area fractions must sum to unity.

#### 13.4.7.5. Fluid Solid Area Density

This setting is available only when the porous domain has a solid phase.

Specify the fluid-solid area density, which is the contact area per unit volume, where the contact area is the surface area of the solid in contact with the fluid(s).

### 13.4.7.6. Fluid Solid Heat Transfer

This setting is available only when the porous domain has a solid phase.

Specify the overall heat transfer coefficient for heat transfer between the fluid(s) and the solid. This is analogous to the heat transfer coefficient between fluids in an inhomogeneous multiphase case, as described in [Inhomogeneous Interphase Heat Transfer Models](#).

### 13.4.7.7. Additional Variable Pair Details

This setting is available only when the porous domain has a solid phase.

Specify pairs of the same Additional Variable to enable the Additional Variable transfer between the solid and the fluid(s).

The options are:

- Additional Variable Transfer Coefficient

This option applies a bulk value of the Additional Variable transfer coefficient between the fluid(s) and the solid.

- Fluid Dependent

This option enables the direct specification of the Additional Variable transfer coefficient for each fluid.

The Additional Variable transfer coefficient is analogous to the transfer coefficient between fluids in an inhomogeneous multiphase case. For details, see [Additional Variable Interphase Transfer Models in the CFX-Solver Modeling Guide](#) and [Additional Variables in Multiphase Flow in the CFX-Solver Theory Guide](#).

## 13.4.8. Particle Injection Regions Tab

Injection regions are used to define locators anywhere within a domain, and can be set up as spheres, cones, or using a custom Fortran subroutine. For details, see [Particle Injection Regions in the CFX-Solver Modeling Guide](#).

### 13.4.8.1. Particle Injection Regions List Box

This list box is used to select **Particle Injection Regions** for editing or deletion. Particle Injection Regions can be created or deleted with the icons that appear beside the list box.

### 13.4.8.2. Coordinate Frame

In some cases it may be useful to specify injection quantities using coordinates from a coordinate frame other than the global default frame (COORD 0). To do this, choose a local coordinate frame from the drop-down list. You can then set the injection center and the injection direction relative



to the selected coordinate frame. In a typical application, it makes sense to use a coordinate frame that has a principal axis aligned with the general spray direction.

---

**Note:**

The coordinate frame of a particle injection region is independent of the coordinate frame of the domain.

---

**Note:**

In the case of user defined injection, the particle position and the injection velocity – if returned from the user routine – refer to the global coordinate frame `Coord 0` rather than to the one specified under **Coordinate Frame**.

---

### 13.4.8.3. Fluid: List Box

This list box is used to select a particle material in order to apply it to the injection region and define its properties for the injection region.

### 13.4.8.4. [fluid name] Check Box

This check box determines whether or not the particle is to be injected over the selected injection region.

For multicomponent particles, specify the mass fraction of each. Other quantities are optional and are the same as found on the **Fluid Values** tab. For details, see [Fluid Values for Inlets and Openings](#) (p. 229).

#### 13.4.8.4.1. Injection Method

The following table outlines various settings available on the **Particle Injection Regions** tab. The settings are marked as required or optional based on the type of injection method chosen.

Settings for Injection Method	Injection Method		
	Cone	Cone with Primary Breakup	Sphere
Injection Center	Required	Required	Required
Injection Velocity Magnitude <sup>[a]</sup>			Required
Radius of Injection Sphere <sup>[a]</sup>			Optional
Number of Positions <sup>[a]</sup>	Required	Required	Required
Particle Diameter Distribution	Optional		Optional
Particle Mass Flow Rate	Optional	Required	Optional
Cone Definition	Required		

Settings for Injection Method	Injection Method		
	Cone	Cone with Primary Breakup	Sphere
For details, see <a href="#">Settings for Cone Definition (p. 201)</a> .			
Injection Direction		Required	
Injection Velocity For details, see <a href="#">Settings for Injection Velocity (p. 201)</a> .	Required		
Particle Primary Breakup For details, see <a href="#">Settings for Particle Primary Breakup (p. 202)</a> .		Required	
Nozzle Definition <sup>[b]</sup>		Required	

<sup>[a]</sup> Enter a numerical quantity or CEL expression for the indicated parameter.

<sup>[b]</sup> **Nozzle Definition:** Select either `Ring Nozzle` or `Full Nozzle` and specify the values to define the nozzle.

For details, see the following topics in the [CFX-Solver Modeling Guide](#):

- [Particle Injection Regions](#) (includes description of various types of cone locators)
- [Number of Positions](#)
- [Particle Diameter Distribution](#)
- [Particle Mass Flow Rate](#)

#### 13.4.8.4.1.1. Settings for Cone Definition

Settings	Point Cone	Hollow Cone	Ring Cone	Full Cone
Cone Angle <sup>[a]</sup>	Required			
Dispersion Angle <sup>[a]</sup>	Optional	Optional		
Radius of Injection Plane <sup>[a]</sup>		Required		Required
Inner Radius Of Plane <sup>[a]</sup>			Required	
Outer Radius Of Plane <sup>[a]</sup>			Required	

<sup>[a]</sup> Enter a numerical quantity or CEL expression for the indicated parameter.

For details, see [Cone in the CFX-Solver Modeling Guide](#).

#### 13.4.8.4.1.2. Settings for Injection Velocity

The **Injection Velocity** options are:

- Velocity Magnitude

Specify the component of velocity normal to the 2D injection region. Note that the normal direction is specified by the **Injection Direction** settings.

Specify the cone angle, measured as the angle between the axis of the cone (which, for the Velocity Magnitude option, is normal to the 2D injection region) and one side of the cone.

- Cartesian Components

Specify the particle velocity components in Cartesian coordinates. These coordinates are in the coordinate frame specified for the particle injection region. The velocity components can be expressions.

- Cylindrical Components

Specify the particle velocity components in cylindrical coordinates. These coordinates are in the coordinate frame specified for the particle injection region. The velocity components can be expressions.

You can put swirl into the flow of injected particles by specifying a non-zero theta (circumferential) component of velocity.

- Zero Slip Velocity

Particles are injected at the local velocity of the continuous phase.

### 13.4.8.4.1.3. Settings for Particle Primary Breakup

Settings	Blob Method	Enhanced Blob Method	Lisa Model	Turbulence Induced Atomization
Cone Angle <sup>[a]</sup> <sup>[b]</sup>	Required	Required	Required	
Coefficient of Contraction <sup>[a]</sup>		Required		
Injection Total Pressure <sup>[a]</sup>		Required		
Injection Pressure Difference <sup>[a]</sup>			Required	Required
Pressure Probe Normal Distance <sup>[a]</sup>		Required		
Density Probe Normal Distance			Required	Required
Length/Diameter Ratio				Required
Particle Material Vapor Pressure		Required		

Settings	Blob Method	Enhanced Blob Method	Lisa Model	Turbulence Induced Atomization
Critical Weber Number			Optional	
Short Wave Ligament Factor			Optional	
Long Wave Ligament Factor			Optional	
Droplet Diameter Size Factor			Optional	
Swirl Definition			Optional	
Form Loss Coefficient				Optional
C1 Constant				Optional
C2 Constant				Optional
C3 Constant				Optional
C4 Constant				Optional
CA1 Constant				Optional
K1 Constant				Optional
Average Turbulent Energy Dissipation Factor				Optional
Turbulent Length Scale Power Factor				Optional
Nozzle Discharge Coefficient				Required

[a] Enter a numerical quantity or CEL expression for the indicated parameter.

[b] **Cone Angle:** Specify a fixed cone angle or select *Reitz* and *Bracco* option to set a correlation to compute the injection angle based on the nozzle geometry.

For details, see [Cone with Primary Breakup in the CFX-Solver Modeling Guide](#).

### 13.4.9. Boundary Models Tab

The settings on the **Boundary Models** tab relate to the wall boiling model for the domain. For details, see [Wall Boiling Model in the CFX-Solver Modeling Guide](#).

### 13.4.10. Initialization Tab

Initialization can be set on a domain or global basis; the available options are the same. For details, see [Initialization \(p. 237\)](#).

The **Initialization** tab for the domain sets domain initial conditions. These will override any settings made in the **Global Initialization** details view. Any domain for which initialization is not set will use the global initial conditions.

### 13.4.11. Solver Control Tab

For immersed solid domains, the **Solver Control** tab contains the **Immersed Solid Control** settings. For details, see [Immersed Solid Control](#) (p. 293).

For transient (and transient blade row) cases involving CHT (Conjugate Heat Transfer), the **Solver Control** tab contains the **Timescale Control** settings. For details, see [Timescale Control for Solid Domains \(Transient Cases; Transient Blade Row Cases with Harmonic Balance\)](#) (p. 204).

#### 13.4.11.1. Timescale Control for Solid Domains (Transient Cases; Transient Blade Row Cases with Harmonic Balance)

Due to a large disparity in time scales between fluid convection and solid diffusion, the transient modeling of a CHT (Conjugate Heat Transfer) problem is impractical, without special numerical treatment, because it can take many time steps for the temperature distribution in the solid to reach a steady state. For example, in a hot streak migration simulation, thousands of blade passing periods may have to be computed before the solution in the solid converges and reaches a steady-state temperature.

A good engineering approximation is to advance the simulation in time while altering the thermal response of the solid to effectively accelerate the solid thermal diffusion convergence. The thermal response in the solid can be altered by **Timescale Control** settings. Using the **Domain Solver Control > Timescale Control** settings, you can specify a value for **Timescale Factor**.

You must enter the solid **Timescale Factor**, which is defined as:

$$\text{Timescale Factor} = \text{solid time scale} / \Delta t$$

where  $\Delta t$  is the simulation time-step size necessary to resolve flow features of interest.

The solid time scale can be determined by the following formula:

$$\text{solid time scale} = L_{\text{solid}}^2 / \alpha_{\text{solid}}$$

where  $L_{\text{solid}}$  is the characteristic length of the solid (in the case of a turbine blade,  $L_{\text{solid}}$  can be the blade chord) and  $\alpha_{\text{solid}}$  is the solid material thermal diffusivity. You can alter the rate of the solid response by a slight modification of the **Timescale Factor**.

This modeling approximation increases the thermal response of the solid to the temperature variation in the fluid. As a result, the temperature at any given point in the solid will, during convergence, fluctuate slightly about an average value. It should be noted that the fluctuations in solid temperature are nonphysical. The time-averaged temperature (after convergence) is very close to the real physical temperature.

---

#### Note:

For steady-state cases, and for transient blade row cases that use the Harmonic Balance transient method, you can control the solid timescale (affecting all solid domains in a given flow analysis) using the Solid Timescale Control settings in the **Solver Control** details view. For details, see [Convergence Control: Solid Timescale Control](#) (p. 292).

---

---

# Chapter 14: Domain Interfaces

---

Domain interfaces have multiple purposes:

- Connecting domains or assemblies

Domain interfaces are required to connect multiple unmatched meshes within a domain (for example, when there is a hexahedral mesh volume and a tetrahedral mesh volume within a single domain) and to connect separate domains.

- Modeling changes in reference frame between domains

This occurs when you have a stationary and a rotating domain or domains rotating at different rates.

- Creating periodic interfaces between regions

This occurs when you are reducing the size of the computational domain by assuming periodicity in the simulation.

- Creating thin surfaces

Thin surfaces enable you to model physics such as heat transfer across a thin material or gap without needing to explicitly mesh the surface. For example, thin surfaces can be used to model contact resistance at a solid-solid interface, a thin film on a fluid-solid interface, or a thin baffle at a fluid-fluid interface.


Interface boundaries are created automatically for each domain interface. For details, see [Interface Boundary Conditions](#) (p. 234).

Additional information about domain interfaces is provided in [Overview of Domain Interfaces in the CFX-Solver Modeling Guide](#).

## 14.1. Creating and Editing a Domain Interface

---

To create a domain interface:

1. Select **Insert > Domain Interface** from the main menu or by clicking *Domain Interface*  on the main toolbar.
2. Enter a new name, if required, using the rules described in [Valid Syntax for Named Objects](#) (p. 95) and click **Apply**.

To edit an existing domain interface:

1. Right-click the domain interface's name in the **Outline** tree view.


2. Select **Edit**. The details view for the domain interface appears.

For more information on the edit command, see [Outline Tree View \(p. 37\)](#).

The details view describes the characteristics of a domain interface on a series of tabs:

- [Domain Interface: Basic Settings Tab \(p. 206\)](#)
- [Domain Interface: Additional Interface Models Tab \(p. 209\)](#).

### 14.1.1. Domain Interface: Basic Settings Tab

The **Basic Settings** tab is where you define the domain interface. It is accessible by clicking *Domain Interface* , or by selecting **Insert > Domain Interface**.

#### 14.1.1.1. Interface Type

- Fluid Fluid

Connects two fluid domains or makes a periodic connection between two regions in a fluid domain.

- Fluid Porous

Connects a fluid domain to a porous domain.

- Fluid Solid

Connects a fluid domain to a solid domain.

- Porous Porous

Connects two porous domains or makes a periodic connection between two regions in a porous domain.

- Solid Porous

Connects a solid domain to a porous domain.

- Solid Solid


Connects two solid domains or makes a periodic connection between two regions in a solid domain.

The interface type you select controls the domains that are available for **Interface Side 1/2**.

#### 14.1.1.2. Interface Side 1/2

##### 14.1.1.2.1. Domain (Filter)

The domain filter is used to filter out 2D regions that are of no interest. The drop-down list contains commonly used regions (all composite names and primitive names that are not referenced by

any composites) and the extended list (displayed by clicking the *Ellipsis*  icon) contains all regions in a domain.

#### 14.1.1.2.2. Region List

**Region List 1** and **Region List 2** enable the selection of regions that form each side of the interface.

#### 14.1.1.3. Interface Models

The interface model options (Translational Periodicity, Rotational Periodicity, and General Connection) each require that you specify a mesh connection method as well as specialized settings for some model options.

##### 14.1.1.3.1. Interface Model Option: Translational Periodicity

In the case of **Translational Periodicity**, the two sides of the interface must be parallel to each other such that a single translation transformation can be used to map Region List 1 to Region List 2. The Translational Periodicity model requires no specialized settings.

For details on the Translational Periodicity model, see [Translational Periodicity in the CFX-Solver Modeling Guide](#).

##### 14.1.1.3.2. Interface Model Option: Rotational Periodicity

In the case of **Rotational Periodicity**, the two sides of the periodic interface can be mapped by a single rotational transformation about an axis. This is the most common case of periodicity and is used, for example, in the analysis of a single blade passage in a rotating machine.

If a domain interface involves rotational periodicity, the axis for the rotational transformation must also be specified in the **Axis Definition** area.

##### 14.1.1.3.3. Interface Model Option: General Connection

In the case of a **General Connection**, more options apply. The settings are described below; for information about the General Connection model, see [General Connection in the CFX-Solver Modeling Guide](#).

###### 14.1.1.3.3.1. Frame Change/Mixing Model

###### 14.1.1.3.3.1.1. Option

- **None**
- **Frozen Rotor**
- **Stage (Mixing-Plane)**
- **Transient Rotor-Stator**

For details, see [Frame Change/Mixing Model in the CFX-Solver Modeling Guide](#).



#### 14.1.1.3.3.1.2. Frozen Rotor: Rotational Offset Check Box

This check box determines whether or not to apply a rotational offset for one side of the interface. For details, see [Rotational Offset in the CFX-Solver Modeling Guide](#).

When set, enter a **Rotational Offset** for one side of the interface.

#### 14.1.1.3.3.1.3. Stage: Pressure Profile Decay Check Box

This option affects solution stability. For details, see [Pressure Profile Decay in the CFX-Solver Modeling Guide](#).

When set, enter a **Pressure Profile Decay** numerical quantity or CEL expression that specifies the rate of decay of the pressure profile.

#### 14.1.1.3.3.1.4. Stage: Downstream Velocity Constraint

For details, see [Downstream Velocity Constraint in the CFX-Solver Modeling Guide](#).

#### 14.1.1.3.3.2. Pitch Change Options

The **Pitch Change** options are:

- **None**

A pitch change option of `None` cannot be used for a stage interface.

- **Automatic**

(applies only when **Interface Models: Frame Change/Mixing Model: Option** is *not* set to **None**)

- **Value**

(applies only when **Interface Models: Frame Change/Mixing Model: Option** is *not* set to **None**)

- **Specified Pitch Angles**

(applies only when **Interface Models: Frame Change/Mixing Model: Option** is *not* set to **None**)

For details, see [Pitch Change in the CFX-Solver Modeling Guide](#).

#### 14.1.1.3.3.2.1. Pitch Change: Value: Pitch Ratio

Enter the pitch ratio. For details, see [Value in the CFX-Solver Modeling Guide](#).

#### 14.1.1.3.3.2.2. Pitch Change: Specified Pitch Angles: Pitch Angle Side 1/2

Enter pitch angle for each side of the interface. For details, see [Specified Pitch Angles in the CFX-Solver Modeling Guide](#).

## 14.1.2. Domain Interface: Additional Interface Models Tab

The **Additional Interface Models** tab contains settings as described in the following sections:

- 14.1.2.1. Mass And Momentum
- 14.1.2.2. Heat Transfer
- 14.1.2.3. Electric Field
- 14.1.2.4. Additional Variable
- 14.1.2.5. Conditional Connection Control

### 14.1.2.1. Mass And Momentum

Allows the specification of additional models at the interface.

The **Mass And Momentum** options are described in the following sections:

- 14.1.2.1.1. Conservative Interface Flux
- 14.1.2.1.2. No Slip Wall
- 14.1.2.1.3. Free Slip Wall
- 14.1.2.1.4. Side Dependent

#### 14.1.2.1.1. Conservative Interface Flux

The `Conservative Interface Flux` **Mass And Momentum** option enables you to define the physics across a thin surface.

The **Interface Model** options are:

- `None`

The default behavior (conservative interface flux) is applied to the interface. No additional models are applied.

- `Pressure Change`

Enter a numerical quantity or CEL expression that specifies the pressure change across the interface from side 1 to side 2. If there is a pressure drop, the specified value should be negative.

- `Mass Flow Rate`

Enter a numerical quantity or CEL expression that specifies the value of the mass flow rate from side 1 to side 2 of the interface.

---

#### Note:

When imposing a mass flow rate at a domain interface, the CFX-Solver updates the pressure change to drive the mass flow rate toward the specified value. The update is based on an internally-estimated coefficient, which may not be optimal.

The **Pressure Update Multiplier** enables you to tune convergence behavior. Enter a numerical quantity or CEL expression that specifies the pressure update multiplier.

The default value is 0.25. If convergence is slow (as may occur for low Reynolds number flows), consider increasing the value. If convergence is unstable, consider decreasing the value. Note that values above 1 are permissible.

---

### 14.1.2.1.2. No Slip Wall

For a description of the options that influence flow on a wall boundary, see [Mass and Momentum in the CFX-Solver Modeling Guide](#).

#### 14.1.2.1.2.1. No Slip Wall: Wall Velocity

When set, this option enables you to specify the wall velocity. Various options are available for specifying the wall velocity.

### 14.1.2.1.3. Free Slip Wall

**Free Slip Wall** has no sub-options.

### 14.1.2.1.4. Side Dependent

Selecting this option allows a different boundary condition to be applied on each side of the interface. Each boundary condition is specified on the related interface boundary (which appears in the **Outline** tree view under the associated domain).

## 14.1.2.2. Heat Transfer

Allows the specification of additional models at the interface.

The **Heat Transfer** options are described in the following sections:

#### 14.1.2.2.1. Conservative Interface Flux

#### 14.1.2.2.2. Side Dependent

#### 14.1.2.2.1. Conservative Interface Flux

This option enables you to define the **Thermal Contact Resistance** or **Thin Material**, which are two ways of defining the same characteristics. That is, if you do not know the contact resistance, you can define the thin material and its thickness and have the solver derive the resistance.

The **Interface Model** options are:

- None

The default behavior (conservative interface flux) is applied to the interface. No additional models are applied.

- Thermal Contact Resistance

Enter a numerical quantity or CEL expression that specifies the value of the thermal contact resistance between side 1 and side 2 of the interface.

- Thin Material

Select a material and enter a numerical quantity or CEL expression that specifies the value of the thickness of the material spanning from side 1 to side 2 of the interface.

#### 14.1.2.2.2. Side Dependent

Selecting this option allows a different boundary condition to be applied on each side of the interface. Each boundary condition is specified on the related interface boundary (which appears in the **Outline** tree view under the associated domain).

#### 14.1.2.3. Electric Field

Allows the specification of additional models at the interface.

The **Electric Field** options are:

[14.1.2.3.1. Conservative Interface Flux](#)

[14.1.2.3.2. Side Dependent](#)

##### 14.1.2.3.1. Conservative Interface Flux

The **Interface Model** options are:

- None

The default behavior (conservative interface flux) is applied to the interface. No additional models are applied.

- Electrical Contact Resistance

Enter a numerical quantity or CEL expression that specifies the value of the electrical contact resistance between side 1 and side 2 of the interface.

##### 14.1.2.3.2. Side Dependent

Selecting this option allows a different boundary condition to be applied on each side of the interface. Each boundary condition is specified on the related interface boundary (which appears in the **Outline** tree view under the associated domain).

#### 14.1.2.4. Additional Variable

Allows the specification of additional models at the interface.

The **Additional Variable** options are described in the following sections:

[14.1.2.4.1. Conservative Interface Flux](#)

[14.1.2.4.2. Side Dependent](#)

##### 14.1.2.4.1. Conservative Interface Flux

The **Interface Model** options are:

- None

The default behavior (conservative interface flux) is applied to the interface. No additional models are applied.

- `Additional Variable Contact Resistance`

Enter a numerical quantity or CEL expression that specifies the value of the Additional Variable contact resistance between side 1 and side 2 of the interface.

#### 14.1.2.4.2. Side Dependent

Selecting this option allows a different boundary condition to be applied on each side of the interface. Each boundary condition is specified on the related interface boundary (which appears in the **Outline** tree view under the associated domain).

#### 14.1.2.5. Conditional Connection Control

**Conditional Connection Control** is an optional group of settings. These settings enable a CEL expression to control whether an interface is open (connected) or closed (not connected; a wall boundary is applied).

When **Conditional Connection Control** is selected, you have the following options:

- `Specified Open State`

Provide a CEL expression. The connection is open when the expression evaluates to true and closed when the expression evaluates to false.

- `Irreversible State Change`

Provide a CEL expression and an initial state (Open or Closed).

The state switches once the expression evaluates to true but then remains in that opposite state (that is, opposite to the initial condition) regardless of what happens to the expression after that point.

For details on conditional connection control, see [Conditional Connections in the CFX-Solver Modeling Guide](#).

### 14.1.3. Domain Interface: Solid Interface Models Tab

This tab is available when at least one side of the interface is on a porous domain that contains a solid.

The **Solid Interface Models** tab contains the following settings:

- **Heat Transfer**

These settings are basically the same as the corresponding settings described in [Heat Transfer \(p. 210\)](#).

Note that:

- Any additional heat transfer models are applied to the solid(s) in the porous domain(s).

- The `Conservative Interface Flux` option is available only if there are solids on both sides of the domain interface.

- **Additional Variable**

These settings are basically the same as the corresponding settings described in [Additional Variable \(p. 211\)](#).

## 14.1.4. Domain Interface: Mesh Connection Tab

The **Mesh Connection** tab contains the [Mesh Connection Method \(p. 213\)](#) settings.

### 14.1.4.1. Mesh Connection Method

You must specify a mesh connection method for all interface models.

#### 14.1.4.1.1. Mesh Connection: Option

The following options may be available, depending on other settings:

- **Automatic**
- **1:1** Direct (One-to-One)
- **GGI** (General Grid Interface)

For details on these options, see [Mesh Connection Options in the CFX-Solver Modeling Guide](#).

#### 14.1.4.1.2. Intersection Control

You can use the options described in this section to control the intersection of non-matching meshes for a particular interface.

---

**Note:**

You can also use Solver Controls to apply default controls for the intersection of all interfaces (settings that are overwritten by Intersection Control settings that you apply individually to domain interfaces using the settings below). See [Intersection Control \(p. 304\)](#) to learn how to apply default Intersection Control settings to all interfaces.

---

---

**Note:**

- If Direct (one-to-one) mesh connectivity is available, the solver will ignore the **Intersection Control** option and will instead use a 'topological intersection', that is, use the one-to-one information to generate the intersection data.
  - If you are restarting a run, the intersection step is skipped and the intersection data is read from the results file. This behavior can be overridden by setting the expert parameter `force_intersection` to `True`.
-

#### 14.1.4.1.2.1. Intersection Control: Option

The **Intersection Control** options for when the **Mesh Connection Option** is set to **GGI** or **Automatic** are as described below. The following options can be used to control the intersection of non-matching meshes. CFX provides the GGI (General Grid Interface) capability which determines the connectivity between the meshes on either side of the interface using an intersection algorithm. In general, two intersection methods are provided:

- **Bitmap Intersection:**

Two faces on either side of the interface which have to be intersected are both drawn into an equidistant 2D pixel map. The area fractions are determined by counting the number of pixels that reside inside both intersected faces (that is, within the union of the two faces). The area fraction for a face is then calculated by dividing the number of overlapping pixels by the total number of pixels in the face. This method is very robust.

- **Direct Intersection (Default):**

Two faces on either side of the interface are intersected using the Sutherland-Hodgeman clipping algorithm. This method computes the exact area fractions using polygon intersection, and is much faster and more accurate than the bitmap method.

The **Bitmap Resolution** controls the number of pixels used to fill the 2D pixel map (see description of the bitmap intersection method above). The higher this number, the more accurate the final calculation of the area fractions. In general, the default resolution of 100 should be sufficient but large differences in the mesh resolution on both sides of the interface as well as other mesh anomalies may require the bitmap resolution to be increased. Larger numbers will cause longer intersection times, for example, doubling the bitmap resolution will approximately quadruple the GGI intersection time.

The **Angle tolerance** option (in degrees) is used for `Direct` intersections only. In order for two faces to be able to intersect, the angular difference between their normals must be within the specified tolerance of 180 degrees.

Both **Intersection Control** options enable you to set the following:

##### **Permit No Intersection**

When the **Permit No Intersection** option is set, the solver will run when there is no overlap between the two sides of an interface. This parameter is mainly useful for transient cases where interface geometry is closing and opening during the run. For example, transient rotor-stator cases with rotating valves, or moving mesh cases where the GGI interface changes from overlap to non-overlap during the simulation both can exhibit this type of behavior. This parameter is not switched on by default.

##### **Discernible Fraction**

Controls the minimum area fraction below which partially intersected faces are discarded. The following default values used by the solver depend on the intersection method:

- **Bitmap Intersection:**  $1/(\text{Bitmap Resolution})^{1.5}$
- **Direct Intersection:** 1.0E-06

## Edge Scale Factor

Controls the value of the GGI edge scale factor. Control volume sector faces on GGI interfaces are detected as degenerate if two opposite edges are smaller than the GGI edge scale factor times the cube root of the corresponding sector volume. Those faces are not intersected.

## Periodic Axial Radial Tolerance

Used when determining if the surface represented by the interface is a constant axial or radial surface. For a rotational periodic GGI interface, the solver ensures that the ratio of the radial and axial extent compared to the overall extent of each interface side is bigger than the specified value and therefore, the interface vertices do not have the same radial or axial positions.

## Circumferential Normalized Coordinates Option

The **Circumferential Normalized Coordinates Option** is used to set the type of normalization applied to the axial or radial position coordinates ( $\eta$ ). Mesh coordinate positions on GGI interfaces using pitch change are transformed into a circumferential ( $\theta$ ) and axial or radial position ( $\eta$ ). The  $\eta$  coordinates span from hub to shroud and are normalized to values between 0 and 1. In cases where the hub and/or shroud curves do not match on side 1 and side 2, different approaches are available to calculate the normalized  $\eta$  coordinates based on side local or global minimum and maximum  $\eta$  values:

- Mixed (Default for Fluid Fluid interfaces): Normalization of  $\eta$  is based on local minimum and maximum  $\eta$  values as well as the  $\eta$  range of side 1. This method forces the hub curves on side 1 and 2 to align. Non-overlap regions adjacent to the shroud may be produced if the shroud curves are not the same.
- Global (Default for Fluid Solid Interfaces): Normalization of  $\eta$  is based on global minimum and maximum eta values. This method intersects side 1 and 2 unchanged from their relative positions in physical coordinates. If the hub and shroud curves do not match then non-overlap regions will be produced.
- Local: Normalization of  $\eta$  is done locally for each side of the interface. This method will always produce an intersection of side 1 and 2, but may cause undesirable scaling of the geometry in some cases.

## Face Search Tolerance Factor

A scaling factor applied to the element sized based separation distance, which is used to find candidates for intersection. For a given face on side 1 of the interface, candidate faces for intersection are identified on side 2 using an octree search algorithm. The octree search uses this tolerance to increase the sizes of the bounding boxes used to identify candidates. Making this parameter larger will increase the size of the bounding boxes, resulting in possible identification of more candidates.

## Face Intersection Depth Factor

A scaling factor applied to the element sized based separation distance used when performing the direct or bitmap intersection. The final intersection of faces is only applied to those faces that are closer to each other than a specified distance. This distance is calculated as the sum of the average depth of the elements on side 1 and side 2 of the interface. This



factor is applied as a scaling on the default distance. It might be necessary to adjust this factor if the normal element depth on the two interfaces sides varies a lot, or side 1 and 2 of the interface are separated by thin regions (for example, thin fin type geometries).

---

# Chapter 15: Boundary Conditions

---

Boundary conditions must be applied to all the bounding regions of your domains. Boundary conditions can be inlets, outlets, openings, walls, and symmetry planes.

Unspecified external regions are automatically assigned a no-slip, adiabatic wall boundary condition. Such regions assume the name `<Domain> Default`, where `<Domain>` corresponds to the name of the domain. Unspecified internal boundaries are ignored.

You can apply boundary conditions to any bounding surface of a 3D primitive that is included in a domain (including internal surfaces). If you choose to specify a boundary condition on an internal surface (for example, to create a thin surface), then boundary conditions must be applied to both sides of the surface.

This chapter describes:

- 15.1. Default Boundary Condition
- 15.2. Creating and Editing a Boundary Condition
- 15.3. Interface Boundary Conditions
- 15.4. Symmetry Boundary Conditions
- 15.5. Working with Boundary Conditions

Additional information on boundary conditions is available in:

- [The Purpose of Boundary Conditions in the CFX-Solver Modeling Guide](#)
- [Available Boundary Conditions in the CFX-Solver Modeling Guide](#)
- [Using Boundary Conditions in the CFX-Solver Modeling Guide](#)

## 15.1. Default Boundary Condition

---

You should be familiar with the concept of primitive and composite regions before reading this section. If you are not, see [Mesh Topology in CFX-Pre \(p. 152\)](#) for details.

When a domain is created, all of the bounding 2D regions that are not used elsewhere are assigned to a default boundary condition that is created automatically. These regions can be considered to be the boundary between the current domain and the rest of the "world". The boundary that is generated is given the name `<Domain name> Default`. When 2D primitives (or composites that reference them) are assigned to other boundary conditions and domain interfaces, they are removed from the `<Domain name> Default` boundary condition. The default boundary condition is a no-slip adiabatic wall, but this can be edited like any other boundary condition. Solid-world 2D primitives behave in a similar way.

## Removing Regions from the Default Domain

Fluid-solid regions are initially contained in the `<Domain Name> Default` boundary condition. When a CFX-Solver input file is written, or a user-defined domain interface is created, any fluid-solid regions referenced by this interface are removed from the default boundary.

If every region is assigned to another boundary condition, the `<Domain Name> Default` boundary object will cease to exist. In such a case, if a boundary condition is subsequently deleted, the `<Domain name> Default` wall boundary will be recreated for the unspecified region. Because the `<Domain name> Default` wall boundary condition is controlled automatically, you should never need to explicitly edit its **Location** list.


## Internal 2D Regions

Any 2D regions that lie within a domain are ignored unless a boundary condition is explicitly assigned (these are treated as thin surfaces). Each side of a fluid-fluid 2D primitive can have a different boundary condition, but most often both sides will be a wall. Thin surfaces are created by assigning a wall boundary condition to each side of a fluid-fluid 2D region. You can specify physics (such as thermal conduction) across thin surfaces in CFX-Pre by defining a domain interface. For details, see [Defining Domain Interfaces as Thin Surfaces in the CFX-Solver Modeling Guide](#).

## 15.2. Creating and Editing a Boundary Condition

---

To create a new boundary:

1. Select **Insert > Boundary** from the main menu, or click *Boundary*  on the main toolbar, or right-click a domain in the **Outline** tree view and select **Insert > Boundary**.

When creating a boundary from the main menu or toolbar, you may be required to select the appropriate domain for the boundary.

2. Enter a new name, if required, using the rules described in [Valid Syntax for Named Objects \(p. 95\)](#) and click **Apply**.

To edit an existing boundary:

1. Right-click the boundary's name in the **Outline** tree view.
2. Select **Edit**.

The details view for the boundary appears.

For more information on the edit command, see [Outline Tree View \(p. 37\)](#).

The details view describes the characteristics of a boundary condition on a series of tabs:

- [Boundary Basic Settings Tab \(p. 219\)](#)
- [Boundary Details Tab \(p. 221\)](#)
- [Boundary Fluid Values Tab \(p. 227\)](#)

- [Boundary Sources Tab \(p. 234\)](#)
- [Boundary Plot Options Tab \(p. 234\)](#)

## 15.2.1. Boundary Basic Settings Tab

This tab sets the type, location, coordinate frame, frame type (stationary or rotating), and other details, for each boundary condition as detailed in the following sections:

- 15.2.1.1. [Boundary Type](#)
- 15.2.1.2. [Location](#)
- 15.2.1.3. [Coordinate Frame](#)
- 15.2.1.4. [Frame Type](#)
- 15.2.1.5. [Profile Boundary Conditions](#)
- 15.2.1.6. [Use GT-SUITE Coupling](#)


### 15.2.1.1. Boundary Type

You can set **Boundary Type** to one of: Inlet, Outlet, Opening, Wall, Symmetry. *Interface* boundaries can be edited, but not created. For details, see [Available Boundary Conditions in the CFX-Solver Modeling Guide](#).

### 15.2.1.2. Location

You can choose the location of a boundary condition from a list containing all 2D composite and primitive regions. For details, refer to the following sections:

- [Mesh Topology in CFX-Pre \(p. 152\)](#)
- [Boundary Condition and Domain Interface Locations \(p. 154\)](#)

The drop-down list contains commonly used regions (all composite names and primitive names that are not referenced by any composites) and the extended list (displayed when clicking the *Ellipsis* icon ) contains all regions in a domain.

---

#### Tip:

- Hold the **Ctrl** key as you click to select multiple regions.
- With the **Location** drop-down list active, you can select regions by clicking them in the viewer with the mouse. This will display a small box containing the names of the regions that are available for selection.

---

### 15.2.1.3. Coordinate Frame

Coordinate frames are used to determine the principal reference directions of specified and solved vector quantities in your domain, and to specify reference directions when creating boundary conditions or setting initial values. By default, CFX-Pre uses `Coord 0` as the reference coordinate

frame for all specifications in the model, but this can be changed to any valid CFX-Pre coordinate frame. For details, see [Coordinate Frames \(p. 371\)](#) and [Coordinate Frames in the CFX-Solver Modeling Guide](#).

#### 15.2.1.4. Frame Type

When the boundary condition is in a rotating domain, the **Frame Type** setting affects whether the conditions you specify on the boundary (pressure, velocity, and so on) are interpreted relative to the (rotating) domain (**Frame Type** set to *Rotating*) or relative to the absolute (stationary) frame of reference (**Frame Type** set to *Stationary*), from the point of view of absolute versus relative quantities.

For cases where there is relative motion between the coordinate frame and the boundaries, the boundary conditions are evaluated at the appropriate coordinate frame, either absolute or relative. The mesh coordinates are either rotated or counter-rotated at the boundary to match the coordinate frame. However, this is not true for the global coordinate frame (Coord 0).

For details, refer to the following sections:

- [Cartesian Velocity Components in the CFX-Solver Modeling Guide](#)
- [Cylindrical Velocity Components](#)

#### 15.2.1.5. Profile Boundary Conditions

This option is available only if profile data is loaded. For instructions on loading a profile, see [Initialize Profile Data \(p. 106\)](#).

##### 15.2.1.5.1. Use Profile Data

Select **Use Profile Data** to define this boundary condition using the external data file previously initialized. Choose from the **Profile Name** list and click **Generate Values** to apply. CFX-Pre will automatically select options and values to best match the variables contained in the selected profile. Check the settings on the other tabs to ensure that the correct CEL expressions have been set. For details, see [Interpolation \(From File\) User Functions \(Profile Functions\) \(p. 423\)](#).

---

**Note:**

If the points supplied in the profile data file do not geometrically overlap with the intended boundary mesh then there may be inaccuracies in the interpolation of profile data to the mesh.

---

#### 15.2.1.6. Use GT-SUITE Coupling

Boundaries of type *Opening* can interact with GT-SUITE, obtaining values from GT-SUITE and sending values to GT-SUITE over the boundary. As a prerequisite, there must be at least one GT-SUITE Model object listed in the **Outline** tree view under **Simulation > Expressions, Functions and Variables > GT-SUITE Models**.

To have the boundary interact with GT-SUITE:

1. Select **GT-SUITE Boundary Conditions** > **Use GT-SUITE Coupling**.
2. Set **GT-SUITE Boundary Conditions** > **GT-SUITE Setup** > **Model** to the name of an existing GT-SUITE Model object.
3. On the **Boundary Details** tab, set various options to `GT-SUITE Coupling` as applicable.

---

**Note:**

For CFX, mass flow and normal speed directed *into* the domain are considered *positive*. GT-SUITE conventions can differ.

---

For details about using GT-SUITE with CFX, see [Coupling CFX to an External Solver: GT-SUITE Coupling Simulations in the CFX-Solver Modeling Guide](#).

### 15.2.2. Boundary Details Tab

Boundary value settings depend on characteristics of the flow. For instance, temperature is required at a boundary only if heat transfer is being modeled.

If you are changing the characteristics of the flow, ensure that boundary conditions are correctly updated. In most cases, CFX-Pre alerts you of the need to update settings in the form of physics validation errors. For details, see [Physics Message Window \(p. 45\)](#).

Example:

Suppose a domain is created, isothermal flow is specified, and an inlet boundary condition set. If flow characteristics are then altered to include heat transfer, the inlet specification must be changed to include the temperature of the fluid at that location.

More information on some of the settings is available:

- [Mass and Momentum in the CFX-Solver Modeling Guide](#)
- [Flow Direction in the CFX-Solver Modeling Guide](#)
- [Turbulence in the CFX-Solver Modeling Guide](#)
- [Heat Transfer in the CFX-Solver Modeling Guide](#)
- [Mesh Deformation in the CFX-Solver Modeling Guide](#)

Various settings are available on the **Boundary Details** tab, depending on the type of boundary condition:

- [Boundary Details: Inlet \(p. 222\)](#)
- [Boundary Details: Outlet \(p. 222\)](#)
- [Boundary Details: Opening \(p. 223\)](#)
- [Boundary Details: Wall \(p. 223\)](#)

- [Boundary Details: Symmetry](#) (p. 225)
- [Boundary Details: Interfaces](#) (p. 226)

### 15.2.2.1. Boundary Details: Inlet

#### 15.2.2.1.1. Flow Regime: Inlet

**Option** can be set to one of `Subsonic`, `Supersonic`, or `Mixed`. For details, refer to the following sections:

- [Inlet \(Subsonic\)](#) in the *CFX-Solver Modeling Guide*
- [Inlet \(Supersonic\)](#) in the *CFX-Solver Modeling Guide*
- [Inlet \(Mixed Subsonic-Supersonic\)](#) in the *CFX-Solver Modeling Guide*

#### 15.2.2.1.2. Mesh Motion: Inlet

The option for **Mesh Motion** is set to `Stationary` by default. For details, see [Mesh Deformation](#) in the *CFX-Solver Modeling Guide*.

### 15.2.2.2. Boundary Details: Outlet

#### 15.2.2.2.1. Flow Regime: Outlet

First, specify the flow regime option. For details, refer to the following sections:

- [Outlet \(Subsonic\)](#) in the *CFX-Solver Theory Guide*
- [Outlet \(Supersonic\)](#) in the *CFX-Solver Theory Guide*

#### 15.2.2.2.2. Mass and Momentum: Outlet

For details, see [Mass and Momentum](#) in the *CFX-Solver Modeling Guide*.

#### 15.2.2.2.3. Pressure Averaging: Outlet

This option appears when `Average Static Pressure` is selected under **Mass and Momentum**. For details, see [Average Static Pressure](#) in the *CFX-Solver Modeling Guide*.

#### 15.2.2.2.4. Thermal Radiation: Outlet

For details, see [Thermal Radiation](#) in the *CFX-Solver Modeling Guide*.

#### 15.2.2.2.5. Mesh Motion: Outlet

The option for **Mesh Motion** is set to `Stationary` by default. For details, see [Mesh Deformation](#) in the *CFX-Solver Modeling Guide*.

### 15.2.2.3. Boundary Details: Opening

#### 15.2.2.3.1. Mass and Momentum: Opening

For details, see [Mass and Momentum in the CFX-Solver Modeling Guide](#).

#### 15.2.2.3.2. Flow Direction: Opening

This option appears when a flow direction is required; that is, when one of `Opening Pres.` and `Dirn.` or `Static Pres.` and `Dirn.` is selected under **Mass and Momentum**. For details, see [Flow Direction in the CFX-Solver Modeling Guide](#).

#### 15.2.2.3.3. Loss Coefficient: Opening

For details, see [Loss Coefficient in the CFX-Solver Modeling Guide](#)

#### 15.2.2.3.4. Turbulence: Opening

For details, see [Turbulence in the CFX-Solver Modeling Guide](#).

#### 15.2.2.3.5. Heat Transfer: Opening

For details, see [Heat Transfer in the CFX-Solver Modeling Guide](#).

#### 15.2.2.3.6. Thermal Radiation: Opening

This is the same as specifying thermal radiation at an inlet. For details, see [Thermal Radiation in the CFX-Solver Modeling Guide](#).

#### 15.2.2.3.7. Component Details: Opening

The **Component Details** section appears when a variable composition/reacting mixture has been created for a single phase simulation, or a simulation with one continuous phase and particle tracking.

The mass fractions must sum to unity on all boundaries. With this in mind, highlight the materials you want to modify and enter the mass fraction. To enter an expression for the mass fraction,

click *Enter Expression*  and enter the name of your expression.

#### 15.2.2.3.8. Mesh Motion: Opening

The option for **Mesh Motion** is set to `Stationary` by default. For details, see [Mesh Deformation in the CFX-Solver Modeling Guide](#).

### 15.2.2.4. Boundary Details: Wall

#### 15.2.2.4.1. Wall Boiling Model: Wall

The **Wall Boiling Model** > **Option** setting is described in [Using a Wall Boiling Model in the CFX-Solver Modeling Guide](#).



For details on the wall boiling model, see [Wall Boiling Model in the CFX-Solver Modeling Guide](#).

#### 15.2.2.4.2. Mass And Momentum: Wall

**Option** can be set to one of No Slip Wall, Free Slip Wall, Finite Slip Wall, Specified Shear, Counter-rotating Wall, Rotating Wall or Fluid Dependent. For details, see [Mass and Momentum in the CFX-Solver Modeling Guide](#).

##### 15.2.2.4.2.1. Slip Model Settings

The **Slip Model** settings apply for finite slip walls.

The only available option is `Power Law`. You must provide the nominal slip speed ( $U_s$ ), the critical stress ( $\tau_c$ ), the slip power ( $m$ ), the pressure coefficient ( $B$ ), and the normalizing stress ( $\tau_n$ ).

For details about the finite slip wall model, see [Finite Slip Wall in the CFX-Solver Modeling Guide](#).

##### 15.2.2.4.2.2. Shear Stress Settings

The **Shear Stress** settings apply for walls with specified shear.

You specify the shear stress value directly, using a vector that points tangentially to the wall. The normal component of the vector that you specify is ignored.

##### 15.2.2.4.2.3. Wall Velocity Settings

The **Wall Velocity** settings apply for no slip walls, and walls with finite slip.

The **Wall Velocity Relative To** option can be used to set the wall velocity relative to either the `Boundary Frame` or `Mesh Motion` occurring on the boundary. You can select only the `Boundary Frame` option if the wall mesh motion is set to either `Unspecified`, `Parallel to Boundary` or `Surface of Revolution`. For details about no slip wall velocity, see [No Slip Wall in the CFX-Solver Modeling Guide](#).

If **Wall Velocity > Option** is set to `Cartesian Components`, you must specify the velocity in the X, Y, and Z axis directions. Similarly, if you choose `Cylindrical Components` then values are required for **Axial Component**, **Radial Component**, and **Theta Component**.

You can specify a counter rotating wall. For details, see [Counter-rotating Wall in the CFX-Solver Modeling Guide](#).

Specifying a `Rotating Wall` requires an angular velocity and, if the domain is stationary, an axis definition. For details, see [Rotating Wall in the CFX-Solver Modeling Guide](#).

##### 15.2.2.4.2.3.1. Axis Definition

If you select `Coordinate Axis`, a **Rotation Axis** is required. The `Two Points` method requires a pair of coordinate values specified as **Rotation Axis From** and **Rotation Axis To**.

#### 15.2.2.4.3. Wall Roughness

For details, see [Wall Roughness in the CFX-Solver Modeling Guide](#).

#### 15.2.2.4.4. Solid Motion: Wall

If the boundary is for a domain that involves solid motion, then the **Solid Motion > Boundary Advection** option may be available. If the velocity for the solid motion (specified in the **Boundary Details** tab for the domain) is directed into the domain everywhere on a boundary, and if you specify a fixed temperature or a fixed value of an Additional Variable on that boundary, then you should consider turning on the **Boundary Advection** option.

If you have specified a fixed temperature, then turning on the **Boundary Advection** option causes the advection of thermal energy into the solid domain at a rate that is consistent with the velocity normal to the boundary, the specified fixed temperature, and the material properties.

If you have specified a fixed value for an Additional Variable, then turning on the **Boundary Advection** option causes the advection of that Additional Variable into the solid domain at a rate that is in accordance with the velocity normal to the boundary, the specified fixed value of the Additional Variable, and, for mass-specific Additional Variables, the density of the solid material.

For a boundary where the solid is moving out of the domain, consider turning on the **Boundary Advection** option in order to allow thermal energy and Additional Variables to be advected out.

For details on setting up the solid motion model for a domain, see [Solid Motion \(p. 196\)](#). For details on Additional Variables, see [Additional Variables \(p. 405\)](#).

#### 15.2.2.4.5. Heat Transfer: Wall

For details, see [Heat Transfer in the CFX-Solver Modeling Guide](#).

#### 15.2.2.4.6. Wall Contact Model

For details, see [Wall Contact Model in the CFX-Solver Modeling Guide](#).

#### 15.2.2.4.7. Thermal Radiation: Wall

For details, see [Thermal Radiation in the CFX-Solver Modeling Guide](#).

#### 15.2.2.4.8. Mesh Motion: Wall

The option for **Mesh Motion** is set to `Stationary` by default. For details, see [Mesh Motion in the CFX-Solver Modeling Guide](#).

### 15.2.2.5. Boundary Details: Symmetry

Only **Mesh Motion** can be set in this tab for Symmetry boundary conditions. The option for **Mesh Motion** is set to `Unspecified` by default. For details, see [Mesh Deformation in the CFX-Solver Modeling Guide](#).

### 15.2.2.6. Boundary Details: Interfaces

The options for **Mass and Momentum**, **Turbulence**, **Heat transfer**, **Mesh Motion**, and **Additional Variables** are set to `Conservative Interface Flux` by default.

---

**Important:**

`Conservative Interface Flux` implies that the quantity in question will "flow" between the current boundary and the boundary on the other side of the interface. This means that `Conservative Interface Flux` must also be used on the boundary on the other side of the interface. Accordingly, the CFX-Solver will not be able to handle cases where `Conservative Interface Flux` is set on just one side of the interface, or where the quantity being transferred does not exist on the other side. CFX-Pre will issue a warning if either of these cases exist.

---

For details on **Nonoverlap Conditions**, refer to [Non-overlap Boundary Conditions](#).

### 15.2.2.7. Mesh Motion

When mesh deformation is selected for the domain that contains a boundary condition, mesh motion can be specified for the boundary on the **Boundary Details** tab.

The available options are:

- `Conservative Interface Flux`
- `Unspecified`
- `Stationary`
- `Specified Displacement`
- `Specified Location`
- `Periodic Displacement`
- `Parallel to Boundary`
- `Surface of Revolution`
- `Rigid Body Solution`

For details on these options, see [Mesh Motion Options in the CFX-Solver Modeling Guide](#).

For details on mesh deformation, see [Mesh Deformation in the CFX-Solver Modeling Guide](#).

See [Mesh Deformation \(p. 174\)](#) for information about activating mesh deformation for the domain.

### 15.2.2.8. Electric Field

When an electric field model is activated for a domain that contains a boundary condition, **Electric Field** can be specified for a boundary from the **Boundary Details** tab.

The available options are:

- Voltage
- Ground
- Flux In
- Zero Flux
- Electrical Current Transfer Coefficient
- Conservative Interface Flux
- Electric Field Contact Resistance

For details, see [Electric Field in the CFX-Solver Modeling Guide](#).

For information about activating an electric field model for a domain, see [Electromagnetic Model \(p. 184\)](#).

### 15.2.3. Nonoverlap Conditions Tab

The settings for nonoverlap conditions of an interface boundary are similar to the corresponding settings of wall boundaries, which are described under [Boundary Details: Wall \(p. 223\)](#).

For modeling details, see [Non-overlap Boundary Conditions in the CFX-Solver Modeling Guide](#).

### 15.2.4. Boundary Fluid Values Tab

The **Fluid Values** tab for a boundary condition object is used to set boundary conditions for each fluid in an Eulerian multiphase simulation and each particle material when particle tracking is modeled.

The **Boundary Conditions** list box contains the materials of the fluid passing through the boundary condition. Selecting a material from the list will create a frame with the name of the material and properties available to edit. These properties are detailed in the following sections.

#### 15.2.4.1. Fluid Values: Turbulence

**Turbulence > Option** can be set to any one of the following values. Unless otherwise specified, do not change any further turbulence settings.

- Low (Intensity = 1%)
- Medium (Intensity = 5%)
- High (Intensity = 10%)
- Intensity and Length Scale

For details, see [Intensity and Length Scale \(p. 228\)](#).

- Intensity and Eddy Viscosity Ratio

For details, see [Intensity and Eddy Viscosity Ratio \(p. 228\)](#).

- k and Epsilon

For details, see [k and Epsilon \(p. 228\)](#).

- k and Omega
- k and Eddy Viscosity Ratio
- k and Length Scale
- Reynolds Stresses and Epsilon
- Reynolds Stresses and Omega
- Reynolds Stresses and Eddy Viscosity Ratio
- Reynolds Stresses and Length Scale
- Default Intensity and Autocompute Length Scale
- Intensity and Autocompute Length

For details, see [Intensity and Auto Compute Length \(p. 228\)](#).

- Zero Gradient

#### **15.2.4.1.1. Intensity and Length Scale**

Enter a numeric value or an expression for **Value**, and specify a value for the eddy length scale.

#### **15.2.4.1.2. Intensity and Eddy Viscosity Ratio**

Enter a numeric value or an expression for **Value**, and specify a value for the eddy viscosity ratio.

#### **15.2.4.1.3. k and Epsilon**

Specify a turbulent kinetic energy value and a turbulent eddy dissipation value.

#### **15.2.4.1.4. Intensity and Auto Compute Length**

Enter a numeric value or an expression for **Value**.

### **15.2.4.2. Fluid Values: Volume Fraction**

**Volume Fraction** > **Option** can be set to:

- Value

If set to Value, you must enter a numeric value or an expression for the volume fraction for each fluid. Note that the total volume fractions of the fluids in the list box must be equal to 1.

- Zero Gradient

The volume fraction can also be set to `Zero Gradient`, which implies that the volume fraction gradient perpendicular to the boundary is zero. This setting can be useful for subcritical free surface flow when the free surface elevation is specified (via a pressure profile) at the outlet.

### 15.2.4.3. Fluid Values: Heat Transfer

If **Option** is set to `Static Temperature`, you must specify a value for the static temperature.

### 15.2.4.4. Fluid Values for Inlets and Openings

#### 15.2.4.4.1. Multiphase

1. Set the fluid velocity on the **Boundary Details** tab.
2. Select from the following:
  - `Normal Speed`
  - `Cartesian Velocity Components`
  - `Mass Flow Rate`

For details, see [Mass and Momentum in the CFX-Solver Modeling Guide](#).

3. Set the static pressure on the **Boundary Details** tab.
4. Select from the following:
  - `Normal to Boundary`
  - `Directional Components`

For details, see [Mass and Momentum in the CFX-Solver Modeling Guide](#).

5. Set turbulence quantities at the inlet boundary (if applicable).

For details, see [Turbulence in the CFX-Solver Modeling Guide](#).

6. Set the inlet temperature of each phase (if applicable).

For details, see [Heat Transfer in the CFX-Solver Modeling Guide](#).

7. Enter the volume fraction of the selected fluid at the inlet.

The total volume fraction summed over all the fluids must be equal to 1.

8. If one of the fluids is a variable composition mixture, specify the mass fractions of each of the components.

For details, see [Component Details: Opening \(p. 223\)](#).

#### 15.2.4.4.2. MUSIG settings

When the fluid selected in the list box at the top of the **Fluid Values** tab has a morphology of `Polydispersed Fluid`, size fractions must be specified for each of the size groups. The size fractions can be set to `Value` or `Automatic`. All size fractions set to `Automatic` are calculated to have the same value such that the overall sum of size fractions (including those that are specified by value) is unity. If all size fractions are set to `Value`, you must ensure that the specified size fractions sum to unity.

#### 15.2.4.4.3. Particle Tracking Settings for Inlets and Openings

##### 15.2.4.4.3.1. Phase List

Select the phase for which to set properties.

##### 15.2.4.4.3.2. Particle Behavior

Optionally, specify particle properties at the boundary.

When this check box is disabled, particles do not enter through this boundary.

##### 15.2.4.4.3.3. Mass and Momentum

For details, see [Mass and Momentum in the CFX-Solver Modeling Guide](#).

##### 15.2.4.4.3.4. Particle Position

For details, see [Particle Position in the CFX-Solver Modeling Guide](#).

##### 15.2.4.4.3.5. Particle Locations

For details, see [Particle Locations in the CFX-Solver Modeling Guide](#).

##### 15.2.4.4.3.6. Number of Positions

Select from `Direct Specification` or `Proportional to Mass Flow Rate`. For details, see [Number of Positions in the CFX-Solver Modeling Guide](#).

##### 15.2.4.4.3.7. Particle Mass Flow

For details, see [Particle Mass Flow Rate in the CFX-Solver Modeling Guide](#).

##### 15.2.4.4.3.8. Particle Diameter Distribution

For details, see [Particle Diameter Distribution in the CFX-Solver Modeling Guide](#).

##### 15.2.4.4.3.9. Heat Transfer

Available when heat transfer is selected. For details, see [Heat Transfer in the CFX-Solver Modeling Guide](#).

### 15.2.4.4.3.10. Component Details

Available when the particle phase has been set up as a variable composition mixture. For details, see [Component Details in the CFX-Solver Modeling Guide](#).

### 15.2.4.5. Fluid Values for Outlets

If you are using the inhomogeneous multiphase model and have selected the **Fluid Velocity** option on the **Boundary Details** tab, the fluid-specific velocity information is set on the tab shown below at an outlet boundary.

Specify the Mass and Momentum as:

- Normal Speed
- Cartesian Velocity Components
- Cylindrical Velocity Components
- Mass Flow Rate

For details, see [Mass and Momentum in the CFX-Solver Modeling Guide](#).

### 15.2.4.6. Fluid Values for Walls

#### 15.2.4.6.1. Particle Tracking Settings for Walls

This tab enables you to define particle behavior at walls. This is done by selecting a particle type from the list box and specifying its properties as outlined below:

- Select a particle-wall interaction option - for details, see [Settings for Particle-Wall Interaction \(p. 231\)](#).
- Specify an erosion model - for details, see [Erosion Model in the CFX-Solver Modeling Guide](#).
- Specify a particle-rough wall model - for details see [Particle-Rough Wall Model \(Virtual Wall Model\) in the CFX-Solver Modeling Guide](#).
- Specify the amount of mass absorbed at a wall - for details, see [Mass Flow Absorption in the CFX-Solver Modeling Guide](#).
- Define the particle behavior - Select this option to control the entry of particles and to specify particle properties at wall boundaries. The settings for this option are similar to those available for inlets and openings. For details, see [Particle Tracking Settings for Inlets and Openings \(p. 230\)](#).

#### 15.2.4.6.2. Settings for Particle-Wall Interaction

The particle-wall interaction can be controlled by selecting one of the following **Wall Interaction** options:

- **Equation Dependent** - This is the default option in Ansys CFX and requires the specification of the following **Velocity** settings:



- **Restitution Coefficient** - The droplet reflection at the wall can be controlled by specifying the values for **Perpendicular Coefficient** and **Parallel Coefficient**.

The impact of droplet collision and the resulting momentum change across the collision can be described by specifying the perpendicular and parallel coefficients of restitution. For details, see [Restitution Coefficients for Particles in the CFX-Solver Modeling Guide](#).

- **Minimum Impact Angle** - Select this check box if you want to specify the minimum impact angle. Below this impact angle, particles will be stopped with the fate `Sliding` along walls.
- **Wall Film** - When **Wall Interaction** is set to `Wall Film`, then the following **Wall Film Interaction** models can be selected:
  - **Stick to Wall** - This model enforces all particles that hit a wall to become part of the wall film. This option does not require any further settings.
  - **Elsaesser** - This model requires the specification of **Wall Material**.
  - **User Defined** - The settings for this option are similar to those described for [User Wall Interaction \(p. 232\)](#).

For details on various wall interaction options, see [Wall Interaction in the CFX-Solver Modeling Guide](#).

**User Wall Interaction** - This option is available when a Particle User Routine has been created. For details, refer to the following sections:

- [Particle User Routines \(p. 441\)](#)
- [Wall Interaction in the CFX-Solver Modeling Guide](#)

For additional modeling information on particle transport, see [Particle Transport Modeling in the CFX-Solver Modeling Guide](#).

## 15.2.4.7. Fluid Values for Interfaces

### 15.2.4.7.1. Fluid-Solid Interface, Fluid Side

The **Fluid Values** tab is available on the fluid side of a fluid-solid interface for an inhomogeneous multiphase setup.

When you are using the inhomogeneous multiphase model, you must use a no-slip wall or set a wall velocity. For details, see [Mass and Momentum in the CFX-Solver Modeling Guide](#).

When particle transport is selected, additional settings are available. These contain the same options as those that appear for wall boundaries. For details, refer to the following sections:

- [Restitution Coefficients for Particles in the CFX-Solver Modeling Guide](#)
- [Erosion Model in the CFX-Solver Modeling Guide](#)
- [Mass Flow Absorption in the CFX-Solver Modeling Guide](#).

Particles are introduced into the domain from this boundary. For details, see [Fluid Values for Inlets and Openings](#) (p. 229).

### 15.2.4.7.2. Fluid-Fluid and Periodic Interfaces

For periodic and fluid-fluid interfaces, `Conservative Interface Flux` is the only available option for all quantities and cannot be changed.

## 15.2.5. Boundary Solid Values Tab

### 15.2.5.1. Heat Transfer

#### 15.2.5.1.1. Adiabatic

The heat flux across the boundary is zero. The boundary is insulated.

#### 15.2.5.1.2. Fixed Temperature

The boundary is fixed at a specified temperature  $T_w$ .

#### 15.2.5.1.3. Heat Flux

A heat flux is specified across the boundary. A positive value indicates heat flux into the domain.

#### 15.2.5.1.4. Heat Transfer Coefficient

In this case, the heat flux at a boundary is implicitly specified using an external heat transfer coefficient,  $h_c$  and an outside or external boundary temperature,  $T_o$ . This boundary condition can be used to model thermal resistance outside the computational domain. The heat flux, when calculated using the **Heat Transfer Coefficient**, is:

$$q_w = h_c(T_o - T_w) \quad (15.1)$$

where  $T_o$  is the specified outside or external boundary temperature and  $T_w$  is the temperature at the boundary (edge of the domain).

#### 15.2.5.1.5. Conservative Interface Flux

`Conservative Interface Flux` implies that the heat flow in question will be between the current solid on this boundary and another solid on the other side of the interface. This means that `Conservative Interface Flux` must also be used on the boundary on the other side of the interface. Accordingly, the CFX-Solver will not be able to handle cases where `Conservative Interface Flux` is set on just one side of the interface, or where the quantity being transferred does not exist on the other side. CFX-Pre will issue a warning if either of these cases exist.

### 15.2.5.2. Additional Variables

See [Boundary Details and Fluid Values Tabs for Boundary Condition Objects](#) (p. 409) for details.

### 15.2.6. Fluid Pair Values Tab

This tab is used to override, for the boundary, the corresponding settings on the **Boundary Models** tab for the domain. For details, see [Boundary Models Tab \(p. 203\)](#).

---

**Note:**

If any of the settings are left unspecified, then the corresponding settings of the domain apply; if the domain settings are unspecified, then default values apply.

---

### 15.2.7. Boundary Sources Tab

Boundary sources, such as sources of mass (continuity), energy, radiation, Additional Variables, component mass fractions, and turbulence, can be specified at inlet, opening, outlet, interface, and wall boundaries. For details, see [Sources in the CFX-Solver Modeling Guide](#).

### 15.2.8. Boundary Plot Options Tab

The **Plot Options** tab enables you to create **Boundary Contour** and **Boundary Vector** graphics to display scalar and vector values at boundaries, respectively, as detailed in the following sections:

- [Boundary Contour \(p. 234\)](#)
- [Boundary Vector \(p. 234\)](#)

#### 15.2.8.1. Boundary Contour

Selecting this option and choosing a **Profile Variable** draws the boundary surface colored by the selected variable. The available variables depend on the settings on the **Boundary Details** and **Sources** tabs, as applicable. A legend appears by default showing the variable plotted on the boundary with a local range. You can turn off visibility for the legend and the plots by clearing the check box next to the boundary contour object associated with your boundary condition in the **Outline** tree view. You may have to click the + sign next to the boundary condition in order to view the contour object in the **Outline** tree view.

#### 15.2.8.2. Boundary Vector

Selecting this option draws vectors at the nodes of the boundary surface, pointing in the direction specified by the **Profile Vector Component** setting. The availability of vectors (and this option) depends on the settings on the **Boundary Details** and **Basic Settings** tabs. For example, vector plots are available if you specify **Basic Settings** > **Boundary Type** as **Inlet** and the **Boundary Details** > **Mass and Momentum** option as velocity components.

---

## 15.3. Interface Boundary Conditions

---

All domain interfaces automatically create boundaries of type **Interface** that contain the regions used in the domain interface. These boundaries are named `<Domain Interface Name> Side <Number>`. For example, for a domain interface named `myInterface`, the related boundary conditions would be called `myInterface Side 1` and `myInterface Side 2`. At least one of these

boundaries will be auto-created for each domain involved in the interface. You can edit these boundaries like any other boundary, but you cannot create new interface boundaries directly.

You will usually not need to edit an auto-generated Interface boundary, but options are available for fluid-solid interfaces (which can be considered a special case of wall boundaries). Settings and options available when editing interface boundaries can be configured. For details, refer to the following sections:

- [Boundary Details: Interfaces \(p. 226\)](#)
- [Fluid Values for Interfaces \(p. 232\)](#)

## 15.4. Symmetry Boundary Conditions

---

When specifying symmetry boundary conditions, select the locations from the drop-down list box and select the **Frame Type** if your domain is rotating. No further settings are required, and the same settings apply for fluid and solid domains.

For details, see [Symmetry Plane in the CFX-Solver Modeling Guide](#).

## 15.5. Working with Boundary Conditions

---

The topics in this section include:

- [Boundary Condition Visualization \(p. 235\)](#)
- [Profile Data and CEL Functions \(p. 236\)](#)

### 15.5.1. Boundary Condition Visualization

When you create a boundary condition in CFX-Pre, several things happen in the Viewer:

- Symbols for the boundary conditions are displayed in the viewer, based on type. The visibility of these symbols is determined by the **Label and Marker** control form. For details, see [Labels and Markers \(p. 60\)](#).
- Boundary condition symbols are shown at surface display line intersections.
- Regions contained in the boundary condition are highlighted according to settings specified under **Edit > Options**. For details, see [3D Viewer Toolbar \(p. 52\)](#).

If multiple boundary conditions are defined on a region of mesh, an error appears in the physics validation window below the viewer.

---

#### Note:

Inlets, outlets, and openings use arrow symbols that are locally normal to the boundary surface, irrespective of the actual direction specified for the boundary condition. It is possible to show arrows pointing in the specified direction by creating a Boundary Vector object. You can optionally turn off the default arrow symbols by clearing the check boxes on **Label**

**and Marker** control form (see above). Also see [Boundary Vector \(p. 234\)](#) and [Boundary Markers \(p. 60\)](#) for more details.

When using CFX-Pre within Ansys Workbench or with a pale viewer background color, the colors of these symbols are black in order to make them more visible.

---

## 15.5.2. Profile Data and CEL Functions

Profile data can be used to define a boundary condition based on a distribution of appropriate values.

### 15.5.2.1. Types of Discrete Profiles

- 1D profile uses one spatial coordinate to define the data position; for example,  $x$ ,  $y$ ,  $z$ , or a cylindrical value. This could be used to describe the axisymmetric flow down a cylindrical pipe (that is, the data values for a value of  $r$ ).
- 2D profile uses two spatial coordinates (Cartesian or polar); for example,  $(x, y)$ ,  $(x, z)$ ,  $(r, t)$ ,  $(a, t)$ , and so on. If you are importing the data from a 2D code on a planar boundary, you may want to use this as a boundary condition in a 3D case in CFX.
- 3D profile uses three spatial coordinates; for example,  $(x, y, z)$  or  $(r, t, a)$ . Among various uses of 3D Profile Data are boundary conditions, spatially varying fluid properties, Additional Variables, or equation sources.

### 15.5.2.2. Multiphase Boundary Condition Example

The syntax for using profile data in CEL expressions is described in [Using Interpolation \(From File\) Functions in CEL Expressions \(p. 428\)](#).

Consider a multiphase boundary condition set up using the following:

- The profile data file has a profile named `myProfile`
- One of the data fields is `Temperature [K]`

CFX-Pre enables a function such as `myProfile.water.Temperature(x, y, z)` to refer to a data field stored in the profile. This derived value can be assigned to a parameter, such as `Fixed Temperature`.

The expressions that are automatically generated in CFX-Pre for profile boundaries are simply the expressions in terms of interpolation functions. Modify them in the same way as a normal CEL expression.

For example, the expression `myProfile.Temperature(x, y, z)` could be modified to `2*myProfile.Temperature(2x, y, z)`. For details, see [Profile Boundary Conditions in the CFX-Solver Modeling Guide](#).

---

# Chapter 16: Initialization

---

Initialization is the process by which all unspecified solution field values are assigned at the beginning of a simulation. These values are commonly referred to as *initial values*. For steady-state simulations, they may be collectively referred to as an *initial guess*.

For steady-state simulations, initial values can be set automatically if a good initial guess is not known or is not required. Although accurate initial values may not always be available, a good approximation can reduce the time to solve a steady-state simulation and reduce the chance that the solution fails to converge due to diverging residuals. The more complicated the simulation and models used, the more important it becomes to start the solution process with sensible initial values. Advice about choosing sensible initial values is available in [Initialization Parameters in the CFX-Solver Modeling Guide](#).

For transient simulations, the initial values must be specified for all variables because the data describes the state at the simulation start time.

If available, the results from a previous simulation can be used to provide the initial values. In this case, any values chosen to be automatically set will be overridden by values from the initial values file(s). See [Reading the Initial Conditions from a File in the CFX-Solver Modeling Guide](#) for details.

Global and domain initialization settings may be specified. Global settings apply to only those domains that do not have their own initialization settings.

Information on modeling initial values is available in [Initial Condition Modeling in the CFX-Solver Modeling Guide](#).

## 16.1. Using the User Interface

---

The following topics will be discussed:

- [Domain: Initialization Tab \(p. 237\)](#)
- [Global Settings and Fluid Settings Tabs \(p. 238\)](#)

The **Global Settings** and **Fluid Settings** tabs for the global initialization object (listed as `Initialization` under `Simulation` in the tree view) contain settings that specify how initial values are to be determined, and, in some cases, the initial values themselves. They are accessible by clicking *Global*

*Initialization*  `t=0`, by selecting **Insert** > **Global Initialization**, or by editing the initialization object listed in the tree view under `Simulation`.

### 16.1.1. Domain: Initialization Tab

The Initialization tab for a domain contains settings that specify how initial values are to be determined, and, in some cases, the initial values themselves. It is accessible by editing a domain object.

### 16.1.1.1. Domain Initialization

This check box determines whether or not the domain is initialized based on its own settings or based on global initialization settings. When this check box is selected, an interface that is essentially the same as that for global initialization is displayed. Any initialization values defined on a per-domain basis will override values defined at the global level. For details, see [Global Settings and Fluid Settings Tabs](#) (p. 238). After specifying and applying domain initialization, an entry called `Initialization` is listed in the tree view under the applicable domain.

## 16.1.2. Global Settings and Fluid Settings Tabs

When a simulation involves only one fluid, the **Fluid Settings** tab is not available, but then all of its contents are added to the **Global Settings** tab.

### 16.1.2.1. Coordinate Frame Check Box

This check box determines whether or not a specified coordinate frame is used for interpreting initial conditions. If the check box is not selected, the default coordinate frame, `Coord 0`, is used.

#### 16.1.2.1.1. Coordinate Frame Check Box: Coordinate Frame

Select a coordinate frame to use for interpreting initial conditions. For details, see:

- [Coordinate Frames](#) (p. 371)
- [Coordinate Frames in the CFX-Solver Modeling Guide](#)
- [Coordinate Frame in the CFX-Solver Modeling Guide](#)

### 16.1.2.2. Frame Type Check Box

This check box determines whether or not a specified frame type is used for interpreting initial values of velocity. If the check box is not selected, the default frame of reference is used. The default frame of reference is stationary or rotating, depending on whether the domain is stationary or rotating, respectively.

#### 16.1.2.2.1. Frame Type

- `Stationary`

The frame of reference used to interpret initial values of velocity is the stationary frame of reference. For example, if the initial velocity throughout a domain is parallel to the rotation axis of the domain, the flow will initially have no swirl in the stationary frame of reference, even if the domain is rotating.

- `Rotating`

The frame of reference used to interpret initial values of velocity is that of the associated domain. For example, if the initial velocity throughout a domain is specified as being parallel to the rotation axis of the domain, and if the domain is rotating, the flow will have swirl in the stationary frame of reference.

For details, see [Frame Type](#) in the *CFX-Solver Modeling Guide*.

### 16.1.2.3. Initial Conditions: Velocity Type

- Cartesian
- Cylindrical

For details, see [Velocity Type](#) in the *CFX-Solver Modeling Guide*.

### 16.1.2.4. Initial Conditions: Cartesian Velocity Components

#### 16.1.2.4.1. Option

- Automatic

The initial velocity field is loaded from an initial values file, if one is available. If an initial values file is not available, the initial velocity field is computed from built-in algorithms. For details, see [Automatic](#) in the *CFX-Solver Modeling Guide*.

- Automatic with Value

The initial velocity field is loaded from an initial values file, if one is available. If an initial values file is not available, the initial velocity field is set to user-specified values. For details, see [Automatic with Value](#) in the *CFX-Solver Modeling Guide*.

#### 16.1.2.4.2. Velocity Scale Check Box

(applies only when **Option** is set to Automatic)

This check box determines whether or not a specified velocity scale is used. If the check box is not selected, a velocity scale will be calculated internally by the CFX-Solver, based on a weighted average value of velocity over all applicable Boundary Conditions (inlets, openings and outlets). Initial guess values that are calculated based on the internally calculated velocity scale may be unsuitable due to the shape of your domain, or, for example, due to a small, high-speed inlet which results in an over-prediction of the velocity magnitude.

##### 16.1.2.4.2.1. Velocity Scale Check Box: Value

Enter a numerical quantity or CEL expression for the velocity scale. This is not a normalized value; it is essentially the velocity magnitude that will be used for all applicable velocity vectors. For details, see [Velocity Scale](#) in the *CFX-Solver Modeling Guide*.

#### 16.1.2.4.3. U, V, W

(applies only when **Option** is set to Automatic with Value)

Enter a numerical quantity or CEL expression for each Cartesian velocity component. For details, see [Cartesian Velocity Components](#) in the *CFX-Solver Modeling Guide*.



## 16.1.2.5. Initial Conditions: Cylindrical Velocity Components

### 16.1.2.5.1. Option

For details, see [Option](#) (p. 239).

### 16.1.2.5.2. Velocity Scale Check Box

(applies only when **Option** is set to `Automatic`)

For details, see [Velocity Scale Check Box](#) (p. 239).

### 16.1.2.5.3. Axial Component, Radial Component, Theta Component

(applies only when **Option** is set to `Automatic with Value`)

Enter a numerical quantity or CEL expression for each cylindrical velocity component. For details, see [Cylindrical Velocity Components in the CFX-Solver Modeling Guide](#).

## 16.1.2.6. Initial Conditions: Static Pressure

### 16.1.2.6.1. Option

- `Automatic`

The initial static pressure field is loaded from an initial values file, if one is available. If an initial values file is not available, the initial static pressure field is computed from built-in algorithms.

- `Automatic with Value`

The initial static pressure field is loaded from an initial values file, if one is available. If an initial values file is not available, the initial static pressure field is set to user-specified values.

For details, see [Static Pressure in the CFX-Solver Modeling Guide](#).

### 16.1.2.6.2. Relative Pressure

(applies only when **Option** is set to `Automatic with Value`)

Enter a numerical quantity or CEL expression for the relative pressure.

For details, see [Static Pressure in the CFX-Solver Modeling Guide](#).

## 16.1.2.7. Initial Conditions: Turbulence

From **Option**, the various initial condition settings for turbulence are:

- `Low Intensity and Eddy Viscosity Ratio`: This sets intensity to 1% and viscosity ratio to 1.
- `Medium Intensity and Eddy Viscosity Ratio`: This sets intensity to 5% and viscosity ratio to 10.

- **High Intensity and Eddy Viscosity Ratio:** This sets intensity to 10% and viscosity ratio to 100.
- **Intensity and Eddy Viscosity Ratio:** Use this option to specify fractional intensity and eddy viscosity ratio.
- **Intensity and Length Scale:** Use this option to specify fractional intensity and length scale.
- **k and Epsilon:** Use this option to specify turbulence kinetic energy and turbulence eddy dissipation.
- **k and Omega:** Use this option to specify turbulence kinetic energy and turbulence eddy frequency.
- **k and Eddy Viscosity Ratio:** Use this option to specify turbulence kinetic energy and eddy viscosity ratio.
- **k and Length Scale:** Use this option to specify turbulence kinetic energy and length scale.
- **Reynolds Stresses and Epsilon:** Use this option to specify Reynolds Stresses and turbulence eddy dissipation.
- **Reynolds Stresses and Omega:** Use this option to specify Reynolds Stresses and turbulence eddy frequency.
- **Reynolds Stresses and Eddy Viscosity Ratio:** Use this option to specify Reynolds Stresses and eddy viscosity ratio.
- **Reynolds Stresses and Length Scale:** Use this option to specify Reynolds Stresses and length scale.

For additional details, see [K \(Turbulent Kinetic Energy\)](#), [Epsilon \(Turbulence Eddy Dissipation\)](#), and [Reynolds Stress Components in the CFX-Solver Modeling Guide](#).

#### 16.1.2.7.1. Fractional Intensity

- **Option:** Automatic

The fractional intensity field is loaded from an initial values file, if one is available. If an initial values file is not available, the fractional intensity field is computed automatically.

- **Option:** Automatic with Value

The fractional intensity field is loaded from an initial values file, if one is available. If an initial values file is not available, the fractional intensity field is set to user-specified values.

#### 16.1.2.7.2. Eddy Viscosity Ratio

- **Option:** Automatic

The eddy viscosity ratio field is loaded from an initial values file, if one is available. If an initial values file is not available, the eddy viscosity ratio field is computed automatically.

- **Option:** Automatic with Value

The eddy viscosity ratio field is loaded from an initial values file, if one is available. If an initial values file is not available, the eddy viscosity ratio field is set to user-specified values.

#### 16.1.2.7.3. Eddy Length Scale

- **Option:** Automatic

The eddy length scale field is loaded from an initial values file, if one is available. If an initial values file is not available, the eddy length scale field is computed automatically.

- **Option:** Automatic with Value

The eddy length scale field is loaded from an initial values file, if one is available. If an initial values file is not available, the eddy length scale field is set to user-specified values.

#### 16.1.2.7.4. Turbulence Kinetic Energy

- **Option:** Automatic

The turbulence kinetic energy field is loaded from an initial values file, if one is available. If an initial values file is not available, the turbulence kinetic energy field is computed automatically.

- **Option:** Automatic with Value

The turbulence kinetic energy field is loaded from an initial values file, if one is available. If an initial values file is not available, the turbulence kinetic energy field is set to user-specified values.

#### 16.1.2.7.5. Turbulence Eddy Dissipation

- **Option:** Automatic

The turbulence eddy dissipation field is loaded from an initial values file, if one is available. If an initial values file is not available, the turbulence eddy dissipation field is computed automatically.

- **Option:** Automatic with Value

The turbulence eddy dissipation field is loaded from an initial values file, if one is available. If an initial values file is not available, the turbulence eddy dissipation field is set to user-specified values.

#### 16.1.2.7.6. Turbulence Eddy Frequency

- **Option:** Automatic

The turbulence eddy frequency field is loaded from an initial values file, if one is available. If an initial values file is not available, the turbulence eddy frequency field is computed automatically.

- **Option:** Automatic with Value

The turbulence eddy frequency field is loaded from an initial values file, if one is available. If an initial values file is not available, the turbulence eddy frequency field is set to user-specified values.

#### 16.1.2.7.7. Reynolds Stress Components

- **Option:** `Automatic`

The Reynolds stress components fields are loaded from an initial values file, if one is available. If an initial values file is not available, the Reynolds stress components fields are computed automatically.

- **Option:** `Automatic with Value`

The Reynolds stress components fields are loaded from an initial values file, if one is available. If an initial values file is not available, the Reynolds stress components fields are set to user-specified values.

#### 16.1.2.8. Initial Conditions: Temperature

(applies only when heat transfer is active)

- `Automatic`

The initial temperature field is loaded from an initial values file, if one is available. If an initial values file is not available, the initial temperature field is computed from built-in algorithms.

- `Automatic with Value`

The initial temperature field is loaded from an initial values file, if one is available. If an initial values file is not available, the initial temperature field is set to user-specified values.

#### 16.1.2.9. Initial Conditions: Radiation Intensity

(applies only when using the `P1, Discrete Transfer`, or `Monte Carlo` model for **Thermal Radiation**)

##### 16.1.2.9.1. Option

- `Automatic`

The initial radiation intensity field and blackbody temperature field are loaded from an initial values file, if one is available. If an initial values file is not available, the initial radiation intensity field and blackbody temperature field are computed from built-in algorithms.

- `Automatic with Value`

The initial radiation intensity field and blackbody temperature field are loaded from an initial values file, if one is available. If an initial values file is not available, the initial radiation intensity field and blackbody temperature field are set to user-specified values.

For details, see [Radiation Intensity in the CFX-Solver Modeling Guide](#).

## 16.1.2.10. Initial Conditions: Mixture Fraction

### 16.1.2.10.1. Option

- Automatic

The initial mixture fraction field is loaded from an initial values file, if one is available. If an initial values file is not available, the initial mixture fraction field is computed from built-in algorithms.

- Automatic with Value

The initial mixture fraction field is loaded from an initial values file, if one is available. If an initial values file is not available, the initial mixture fraction field is set to user-specified values.

### 16.1.2.10.2. Mixture Fraction

Enter a numerical quantity or CEL expression that specifies the value of the mixture fraction throughout the domain.

## 16.1.2.11. Initial Conditions: Mixture Fraction Variance

### 16.1.2.11.1. Option

- Automatic

The initial mixture fraction variance field is loaded from an initial values file if, one is available. If an initial values file is not available, the initial mixture fraction variance field is computed from built-in algorithms.

- Automatic with Value

The initial mixture fraction variance field is loaded from an initial values file, if one is available. If an initial values file is not available, the initial mixture fraction variance field is set to user-specified values.

### 16.1.2.11.2. Mix. Fracn. Variance

Enter a numerical quantity or CEL expression that specifies the value of the mixture fraction variance throughout the domain.

## 16.1.2.12. Initial Conditions: Component Details

(applies only when the relevant fluid is a variable composition mixture)

### 16.1.2.12.1. List Box

This list box is used to select a component (of a fluid that is a variable composition mixture) in order to set its fluid-specific initialization options.

### 16.1.2.12.2. [component name]: Option

- Automatic

The initial mass fraction field is loaded from an initial values file, if one is available. If an initial values file is not available, the initial mass fraction field is computed from built-in algorithms.

- Automatic with Value

The initial mass fraction field is loaded from an initial values file, if one is available. If an initial values file is not available, the initial mass fraction field is set to user-specified values.

### 16.1.2.12.3. [component name]: Mass Fraction

Available when **Option** is set to `Automatic with Value`, you must enter a numerical quantity or CEL expression that specifies the value of the component mass fraction throughout the domain.

## 16.1.2.13. Initial Conditions: Additional Variable Details

This section is similar to **Component Details**, dealing with Additional Variables instead. For details, see [Initial Conditions: Component Details \(p. 244\)](#).

## 16.1.2.14. Fluid Specific Initialization

(applies only when multiple fluids are involved)

The fluid-specific initialization settings are grouped together, either in the **Fluid Specific Initialization** section or, in the case of global initialization, on the **Fluid Settings** tab.

## 16.1.2.15. Fluid Specific Initialization: List Box

This list box is used to select a fluid in order to set its fluid-specific initialization options.

## 16.1.2.16. Fluid Specific Initialization: [fluid name] Check Box

This check box determines whether or not the initialization options for the indicated fluid are specified explicitly or are left at default values.

## 16.1.2.17. Fluid Specific Initialization: [fluid name] Check Box: Initial Conditions

Most of the fluid-specific initial condition settings are described in this section as they appear in the non-fluid specific initial condition section in the case of a single-fluid simulation. Those that are not are described here.

### 16.1.2.17.1. Velocity Type

The velocity type can be either `Cartesian` or `Cylindrical`. For details, see [Velocity Type in the CFX-Solver Modeling Guide](#).

### 16.1.2.17.2. Volume Fraction: Option

- Automatic

The initial volume fraction field is loaded from an initial values file, if one is available. If an initial values file is not available, the initial volume fraction field is computed from built-in algorithms.

- Automatic with Value

The initial volume fraction field is loaded from an initial values file if one is available. If an initial values file is not available, the initial volume fraction field is set to user-specified values.

### 16.1.2.17.3. Volume Fraction: Volume Fraction

Enter a numerical quantity or CEL expression that specifies the value of the volume fraction throughout the domain. For details, see [Initial Conditions for a Multiphase Simulation in the CFX-Solver Modeling Guide](#).

### 16.1.2.18. Solid Specific Initialization

(applies only when there are solids in porous domains)

The solid-specific initialization settings are grouped together, either in the **Solid Specific Initialization** section or, in the case of global initialization, on the **Solid Settings** tab.

### 16.1.2.19. Solid Specific Initialization: List Box

This list box is used to select a solid in order to set its solid-specific initialization options.

### 16.1.2.20. Solid Specific Initialization: [solid name] Check Box

This check box determines whether or not the initialization options for the indicated solid are specified explicitly or are left at default values.

---

# Chapter 17: Subdomains

---

A subdomain is a 3D region within a predefined domain that can be used to specify values for volumetric sources. Fluid or porous subdomains (that is, when the parent domain is of type Fluid) allow sources of energy, mass, momentum, radiation, Additional Variables, components, and turbulence to be specified. Solid subdomains allow only sources of energy and radiation to be set.

This chapter describes:

- [Creating and Editing a Subdomain \(p. 247\)](#)
- [Subdomains Tab \(p. 248\)](#)
- [Basic Settings Tab \(p. 248\)](#)
- [Sources Tab \(p. 249\)](#)
- [Fluids Tab \(p. 252\)](#)
- [Mesh Motion Tab \(p. 252\)](#)

A domain must be created before a subdomain can be created. The location of a subdomain must be a 3D region that is part of a single parent domain. 3D primitives are implicitly included in a parent domain if 3D composites or assemblies are used in the domain location. A subdomain cannot span more than one domain, but you can create many subdomains in each domain. You should consider subdomain requirements when you generate a mesh, because subdomains must be created on existing 3D regions. Definitions for primitive and composite regions are available in [Mesh Topology in CFX-Pre \(p. 152\)](#).


Additional information on the physical interpretation of subdomain sources and modeling advice is available, as well as additional information on the mathematical implementation of sources, in [Sources in the CFX-Solver Modeling Guide](#).

The CFX Expression Language (CEL) can be used to define sources by creating functions of any CFX System Variables. For details, see [CEL Operators, Constants, and Expressions in the CFX Reference Guide](#).

## 17.1. Creating and Editing a Subdomain

---

To create a new subdomain:

1. Select **Insert > Subdomain** from the main menu, or click *Subdomain*  on the main toolbar, or right-click a domain in the **Outline** tree view and select **Insert > Subdomain**.

When creating a subdomain from the main menu or toolbar, you may be required to select the appropriate parent domain for the subdomain.



2. Enter a new name, if required, using the rules described in [Valid Syntax for Named Objects \(p. 95\)](#) and click **Apply**.

To edit an existing subdomain:

1. Right-click the subdomain's name in the **Outline** tree view.
2. Select **Edit**.

The details view for the subdomain appears.

For more information on the edit command, see [Outline Tree View \(p. 37\)](#).

## 17.2. Subdomains Tab

---

After entering a name for the subdomain, or selecting the subdomain to edit, the subdomain details view will be displayed. This contains two tabs, **Basic Settings** and **Sources**, that are accessed by selecting the tabs located across the top. You should complete each tab in turn, proceeding from left to right.

---

### Note:

Fluid sources are on their own tab.

---

- **Basic Settings:** Sets the location and the coordinate frame for the subdomain. For details, see [Basic Settings Tab \(p. 248\)](#).
- **Sources:** Defines volumetric source terms in the subdomain for single-phase simulations, or volumetric source terms that apply to all fluids in a multiphase simulation. For details, see [Sources Tab \(p. 249\)](#).
- **Fluid Sources:** Defines volumetric source terms that apply to individual fluid in a multiphase simulation.

## 17.3. Basic Settings Tab

---

The following settings are required on the **Basic Settings** tab.

- [Location \(p. 248\)](#)
- [Coordinate Frame \(p. 248\)](#)

### 17.3.1. Location

Select the region name that the subdomain will occupy. The location can be defined as multiple regions, assemblies and/or user 3D Regions. For details, see [Mesh Topology in CFX-Pre \(p. 152\)](#).

### 17.3.2. Coordinate Frame

By default, **Coordinate Frame** is set to `Coord 0`. You may use alternative coordinate frames. To create a new coordinate frame, select **Insert > Coordinate Frame** from the main menu. For details, see:

- [Coordinate Frames \(p. 371\)](#)
- [Coordinate Frames in the CFX-Solver Modeling Guide.](#)

## 17.4. Sources Tab

---

The volumetric sources that can be set in a subdomain depend on the physical models used in the simulation. Possible source types include mass (continuity), momentum, energy, radiation, Additional Variable, component mass fraction, and turbulence.

### 17.4.1. Single-Phase Fluid Sources

In a single-phase simulation, the volumetric or total source terms are set on the Sources tab. For details, see [Sources in the CFX-Solver Modeling Guide.](#)

1. Select the **Sources** check box to specify sources for the subdomain.
2. Select the **Momentum Source/Porous Loss** check box to specify a momentum source.

For details, see [Momentum Source/Porous Loss \(p. 249\)](#).

3. Select the equation source to specify.

Possible equation source types include: mass fraction, energy, continuity (mass), turbulence, and Additional Variable. For details, see:

- [Equation Sources \(p. 250\)](#)
- [Source Coefficient / Total Source Coefficient in the CFX-Solver Modeling Guide.](#)

#### 17.4.1.1. Momentum Source/Porous Loss

A source of momentum is introduced by setting X, Y and Z, or r, Theta and Axial components for the momentum source under **General Momentum Source**. All three components must be set if a general momentum source is defined. You can optionally specify a **Momentum Source Coefficient** to aid convergence. When employing a cylindrical coordinate frame, you must specify an axis using a rotation axis or two points.

In addition to specifying a general source of momentum, you can model porous loss in a flow using an isotropic or directional loss model. In each case, the loss is specified using either linear and quadratic coefficients, or permeability and loss coefficients. For the **Directional Loss** model, the loss in the transverse direction can be set using the `Loss Coefficient`, which multiplies the streamwise loss by the entered factor. When using the **Directional Loss** model, you must supply a streamwise direction. The direction can be specified with Cartesian or cylindrical coordinates. If you choose cylindrical coordinates, specify the axis using a rotation axis or two points.

For additional details on modeling momentum sources, see [Momentum Sources in the CFX-Solver Modeling Guide.](#)

### 17.4.1.2. Equation Sources

**Equation Sources** introduces source terms to a particular scalar equation.

#### 17.4.1.2.1. Component Mass Fractions

This will introduce a source of a particular component. A `Source` per unit volume or a `Total Source` can be used. The optional **Source Coefficient** or **Total Source Coefficient** provides improved convergence for nonlinear sources. For details, see [General Sources in the CFX-Solver Modeling Guide](#).

1. In the **Option** and **Source** fields, set a component source term for a mixture.

This can be an expression or value for the total source or the source per unit volume. For details, see [General Sources in the CFX-Solver Modeling Guide](#).

2. Set an optional **Total Source / Source Coefficient**.

For details, see [Source Coefficient / Total Source Coefficient in the CFX-Solver Modeling Guide](#).

#### 17.4.1.2.2. Additional Variables

A source for an Additional Variable can be set only if it is included in the parent domain and solved for using a transport equation. (Poisson and Diffusive transports can also have sources.) A `Source` per unit volume or a `Total Source` can be used. The optional **Source Coefficient** or **Total Source Coefficient** provides improved convergence for nonlinear sources. For details, see [General Sources in the CFX-Solver Modeling Guide](#).

1. In the **Option** and **Source** fields, set an Additional Variable Source term.

For details, see [General Sources in the CFX-Solver Modeling Guide](#).

2. Set an optional **Total Source / Source Coefficient**.

For details, see [Source Coefficient / Total Source Coefficient in the CFX-Solver Modeling Guide](#).

#### 17.4.1.2.3. Continuity

`Continuity` sources differ from other sources because you are introducing new fluid into the domain. Properties of the fluid entering the domain are required and appear in the **Variables** frame under the **Continuity** section. For details on the following settings, see [Mass \(Continuity\) Sources in the CFX-Solver Modeling Guide](#).

- **Continuity > Source:** Set a mass source value for the continuity equation.
- **Continuity > Option:** Set the `Fluid Mass Source` per unit volume or the `Total Fluid Mass Source`.
- **MCF/Energy Sink Option:** Select the appropriate sink option from `Local Mass Fractions and Temperature`, `Specified Mass Fractions and Local Temperature`, or `Specified Mass Fractions and Temperature`, as appropriate.
- Set a value for the **Mass Flux Pressure Coefficient**, **Total Mass Source Pressure Coefficient** or **Mass Source Pressure Coefficient**, as appropriate.

- Set a value for the **Mass Flux Volume Fraction Coefficient**, **Total Mass Source Volume Fraction Coefficient** or **Mass Source Volume Fraction Coefficient**, as appropriate.
- Set the variable values for the fluid that is introduced into the domain. The options available on this section depend on the physical models used in the simulation. If the continuity source is negative, then these parameters are not relevant except in the case when either *Specified Mass Fractions and Local Temperature*, or *Specified Mass Fractions and Temperature* have been selected for the **MCF/Energy Sink Option**.
  - **Additional Variables**: Set a value for each Additional Variables that is introduced with the mass source.
  - **Component Mass Fractions**: Set the mass fraction for each of the components in the mass source.
  - **Temperature**: Enter the temperature for the mass source.
  - Set the mass source turbulence quantities as required by the selected turbulence model such as **Turbulence Eddy Dissipation** and **Turbulence Kinetic Energy**.
  - **Velocity**: Set velocity components for the mass source.

#### 17.4.1.2.4. Turbulence Quantities

When the flow is turbulent, sources can be specified for the required turbulence quantities such as *Turbulence Eddy Dissipation* and *Turbulence Kinetic Energy*. A *Source per unit volume* or a *Total Source* can be used. The optional **Source Coefficient** or **Total Source Coefficient** provides improved convergence for nonlinear sources. For details, see [General Sources in the CFX-Solver Modeling Guide](#).

#### 17.4.1.2.5. Energy

An *Energy source* can be specified when the parent domain models heat transfer using the *Thermal Energy* or *Total Energy* model. A *Source per unit volume* or a *Total Source* can be used. The optional **Source Coefficient** or **Total Source Coefficient** provides improved convergence for nonlinear sources. For details, see [General Sources in the CFX-Solver Modeling Guide](#).

### 17.4.2. Bulk Sources for Multiphase Simulations

In a multiphase simulation, source terms that apply to all fluid in the simulation are set on the **Sources** tab. Bulk sources take account of the volume fraction of each phase when applying the source term. For details, see [Bulk Sources in the CFX-Solver Modeling Guide](#).

1. Select the **Bulk Sources** check box to specify bulk sources.
2. Sources are set in the same way as for single phase simulations. For details, see [Single-Phase Fluid Sources \(p. 249\)](#).

### 17.4.3. Multiplying Sources by Porosity

Selecting the **Multiply by Porosity** check box causes the equation source to be scaled by the porosity value. For example, if a porous domain has a volume porosity of 0.8, then if the **Multiply by**

**Porosity** check box is selected, 80% of the source is applied to the fluid; if the check box is not selected then 100% of the source is applied to the fluid.

## 17.5. Fluids Tab

---

Fluid sources are used in an Eulerian multiphase simulation to apply volumetric source terms to individual fluids, and in particle transport to model absorption of particles in the subdomain. For details, see [Particle Absorption \(p. 252\)](#).

1. Select a fluid from the **Fluid** list to set fluid-specific sources.
2. Toggle on the check box next to the fluid to expand the options.
3. Sources are set in the same way as for single phase simulations. For details, see [Single-Phase Fluid Sources \(p. 249\)](#).

It is important to note that these source terms are not automatically multiplied by the fluid volume fraction (that is, do not automatically reduce to zero as the volume fraction goes to zero). For details, see [Fluid-Specific Sources in the CFX-Solver Modeling Guide](#).

### 17.5.1. Particle Absorption

This setting is available when particle tracking is modeled. For details, see [Subdomains in the CFX-Solver Modeling Guide](#).

1. Select a particle type to activate particle absorption.
2. Set the absorption diameter to the desired value.

## 17.6. Mesh Motion Tab

---

When mesh deformation is selected for the domain that contains a subdomain, the **Mesh Motion** tab is available for the subdomain.

The available options are:

- Unspecified
- Stationary
- Specified Displacement
- Specified Location
- Rigid Body Solution

For details on these options, see [Mesh Motion Options in the CFX-Solver Modeling Guide](#).

For details on mesh deformation, see [Mesh Deformation in the CFX-Solver Modeling Guide](#).

See [Mesh Deformation \(p. 174\)](#) for information about activating mesh deformation for the domain.

---

# Chapter 18: Injection Regions

---

In many cases, especially early in a design process, it is not practical to resolve the geometry associated with a large number of small inflows or outflows. Using injection regions in CFX, you can instead create arrays of mass sources and sinks to represent these.

This chapter describes:


- 18.1. Creating and Editing an Injection Region
- 18.2. User Interface
- 18.3. User Functions for Injection Regions
- 18.4. Supported Capabilities and Limitations
- 18.5. Running Cases that involve Injection Regions
- 18.6. Postprocessing Injection Regions

## 18.1. Creating and Editing an Injection Region

---

You can add one or more injection regions to a domain.

To create an injection region:

1. Select **Insert > Injection Region** from the main menu, or click *Injection Region*  on the main toolbar, or right-click a domain in the **Outline** tree view and select **Insert > Injection Region**.

When creating an injection region from the main menu or toolbar, you may be required to select the appropriate domain for the injection region.

2. Enter a new name, if required, using the rules described in [Valid Syntax for Named Objects \(p. 95\)](#) and click **Apply**.

To edit an existing injection region:

1. Right-click the injection region's name in the **Outline** tree view.
2. Select **Edit**.

The details view for the injection region appears.

For more information on the edit command, see [Outline Tree View \(p. 37\)](#).

The details view describes the characteristics of an injection region on a series of tabs, which are described in [User Interface \(p. 254\)](#). To define an injection region, configure the settings in the details view, then click **OK** or **Apply**.

Injection regions are listed under the associated domain in the **Outline** tree.

Related information on sources is available in [Sources in the CFX-Solver Modeling Guide](#).

## 18.2. User Interface

---

The following topics are discussed:

- 18.2.1. Basic Settings Tab
- 18.2.2. Injection Details Tab
- 18.2.3. Injection Distribution Tab
- 18.2.4. Fluid Values Tab
- 18.2.5. Plot Options Tab

### 18.2.1. Basic Settings Tab

The **Basic Settings** tab defines the type, positions, and coordinate frame of the injection region.

#### 18.2.1.1. Region Type

The **Region Type** setting has the following options:

- Inlet

This option specifies that the flow is directed into the domain, creating mass sources at the injection positions.

- Outlet

This option specifies that the flow is directed out of the domain, creating mass sinks at the injection positions.

#### 18.2.1.2. Injection Location Type

The **Injection Location Type** settings specify which regions are considered to contain the injection region.

The available options are:

- Domain

This option specifies that the injection/removal takes place volumetrically at the parent domain mesh elements that are identified by the specified injection positions (described next).

- Selected 2D Regions

This option specifies that the injection/removal takes place on selected 2D regions of the parent domain, at mesh element faces that are identified by the specified injection positions (described next). The **Location** setting holds the selection of 2D regions.

### 18.2.1.3. Injection Positions

The **Injection Positions** settings specify a set of points. The points are used to locate places of injection.

The available options are:

- From Profile

Set **Profile Name** to the name of a profile user function whose coordinate fields provide the injection positions, then click **Generate Values**.

A user function specifies the coordinates for each injection position. For details, see [User Functions for Injection Regions \(p. 268\)](#).

- User Location

Set **User Location List** to the name of one or more locators that provide injection positions.

For example, you could specify a user surface; each node of the user surface would then be used as an injection position. For details, see [User Locations \(p. 377\)](#).

Injection positions on surfaces (but not injection positions within a volume) are located by projecting each point location along its respective local surface normal onto the surface (that is, the 2D region specified for the injection region location).

---

**Note:**

- In general, injection positions should be within the meshed domain. The exception is for domains that have rotationally periodic interfaces, where a periodic repetition of the domain sector would mean that the injection position would be located within the sector.
- The injection positions are mapped to the initial mesh at the beginning of a run. In moving mesh cases, the injection positions subsequently move with the mesh. Note that the geometry (for example, a hole) associated with an injection position can be distorted with mesh motion.

---

### 18.2.1.4. Coordinate Frame

Select a coordinate frame for the injection region.

Coordinate frames are used to determine the principal reference directions of specified and solved vector quantities in your domain, and to specify reference directions when creating boundary conditions or setting initial values. By default, CFX-Pre uses `Coord 0` as the reference coordinate frame for all specifications in the model, but this can be changed to any valid CFX-Pre coordinate frame. For details, see [Coordinate Frames \(p. 371\)](#) and [Coordinate Frames in the CFX-Solver Modeling Guide](#).



## 18.2.2. Injection Details Tab

The **Injection Details** tab defines geometric parameters and fluid conditions at the injection positions.

You can optionally use CEL expressions to specify details at the injection positions. For example, you can specify injection hole diameters using data from the same profile function that defines the injection positions (see [User Functions for Injection Regions \(p. 268\)](#)). To do this, you could set **Diameter** to `Injections.Diameter(x,y,z)` where `Injections` is the user function name in this example.

### 18.2.2.1. Injection Model

The available options are `Circular Hole`, `Cylindrical Hole`, and `Axisymmetric Slot`. The `Cylindrical Hole` and `Axisymmetric Slot` options are available only when, on the **Basic Settings** tab, **Injection Location Type > Option** is set to `Selected 2D Regions`.

With the `Circular Hole` option, each injection position has a circular hole with the specified **Diameter**.

With the `Cylindrical Hole` option, each injection position has a hole that is not necessarily circular; the shape is formed by the intersection of the selected 2D regions and the cylinder. The cylinder has the specified **Diameter**. The orientation of the cylinder is controlled by the **Hole Direction** settings (see [Hole Direction \(p. 256\)](#)).

With the `Axisymmetric Slot` option, each injection position has a rectangular hole that is aligned circumferentially to an axis of rotation. The size of the hole is defined by a **Slot Width** and a **Slot Pitch** (which is the angular extent of the slot in the circumferential direction). An additional **Axis Definition** might also be required if the parent domain is not already specified as rotating.

#### 18.2.2.1.1. Hole Direction

The **Hole Direction** settings define the direction of a cylinder that intersects with the selected 2D regions at an injection position to define the corresponding hole geometry. These settings are available only when **Injection Location Type** is set to `Selected 2D Regions` and **Injection Model > Option** is set to `Cylindrical Hole` (for the `Circular Hole` option, the hole is directed along the local normal of the selected 2D regions).

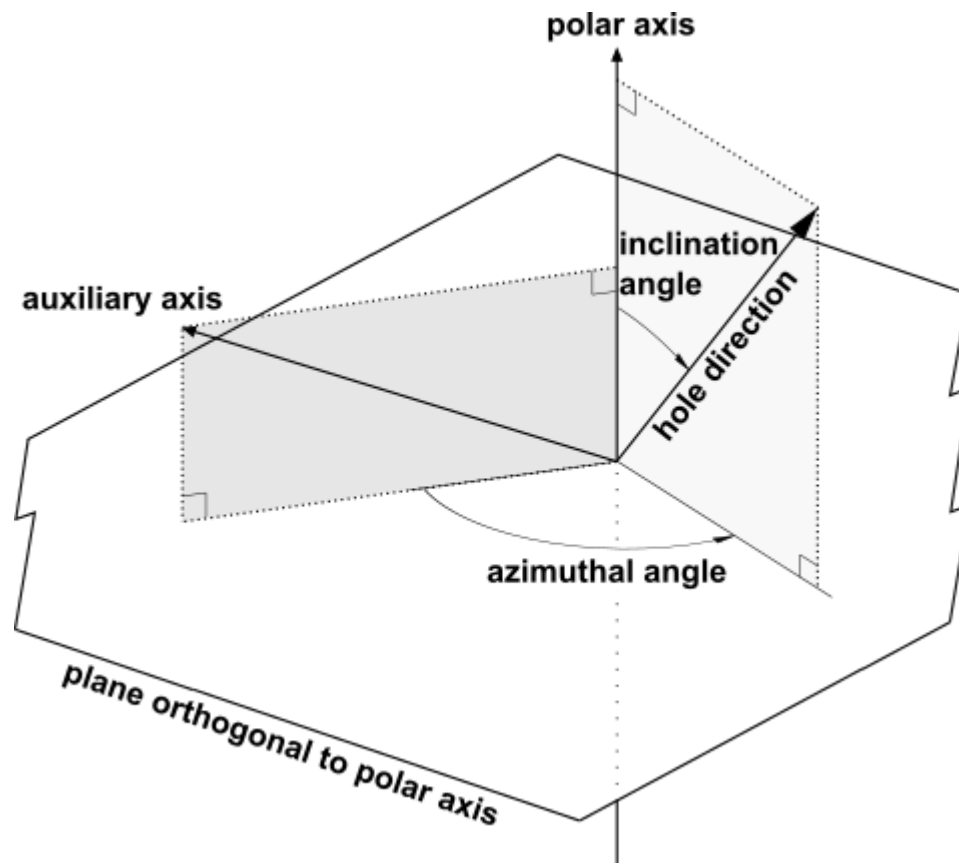
There are four options for the hole direction:

- `Normal to Boundary Condition`

The hole is directed along the local surface normal at the injection position.

- `Local Spherical Angles`

You must specify an **Inclination Angle** and an **Azimuthal Angle**, which are used within a local coordinate frame that is defined by **Polar Axis** and **Auxiliary Axis** direction vectors (which are not parallel). The hole direction is determined by rotating away from the local **Polar Axis**, towards the **Auxiliary Axis**, by the **Inclination Angle**, then further rotating around the local **Polar Axis**, in a direction consistent with the right-hand rule, by the **Azimuthal Angle**.



- Cylindrical Components

The hole is directed along the specified direction vector.

- Cartesian Components

The hole is directed along the specified direction vector.

### 18.2.2.2. Mass and Momentum

The only available option is Injection Mass Flow Rate.

Specify a positive value or expression for **Mass Flow Rate**; negative values are clipped to zero. If you need to change a source to a sink or vice versa, change the **Region Type** setting on the **Basic Settings** tab.

Indicate the applicability of the specified mass flow rate by selecting an option for **Mass Flow Rate Area**:

- Total for All Sectors

The specified mass flow rate is the total mass flow rate for the complete 360° geometry, including all rotationally periodic sectors.

- As Specified

The specified mass flow rate is the mass flow rate for the specified region, whether or not the latter includes all sectors and irrespective of the number of injection positions.

- Per Injection Position

The specified mass flow rate is the mass flow rate for each individual injection position.

If profile data is used to specify the mass flow rate then using the Per Injection Position option ensures that the correct data value is applied to each injection position.

### 18.2.2.3. Flow Direction

This setting defines the direction of the flow from the hole.

The options are:

- Injection Model (default)

The fluid is directed along the specified **Hole Direction**.

- Normal to Boundary Condition

The fluid is directed along the local surface normal at the injection position.

- Cartesian Components

The fluid is directed along the specified direction vector.

- Cylindrical Components

The fluid is directed along the specified direction vector.

### 18.2.2.4. Turbulence

Select an appropriate turbulence option for the fluid entering the domain at the injection positions.

**Turbulence > Option** can be set to any one of the following values

- Low (Intensity = 1%)
- Medium (Intensity = 5%)
- High (Intensity = 10%)
- Intensity and Eddy Viscosity Ratio
- k and Eddy Viscosity Ratio

### 18.2.2.5. Heat Transfer

Specify a static or total temperature for the fluid entering the domain at the injection positions.

If there are multiple frames of reference and the parent domain is rotating, indicate the frame of reference for a specified total temperature.

### 18.2.2.6. Additional Variables

Specify the value of each Additional Variable for the fluid at the injection positions.

### 18.2.3. Injection Distribution Tab

The **Injection Distribution** tab appears when, on the **Injection Details** tab, **Injection Model** > **Option** is set to `Cylindrical Hole`.

The **Injection Distribution** setting controls whether a distribution function is applied to the variables near the injection positions.

The following topics are discussed:

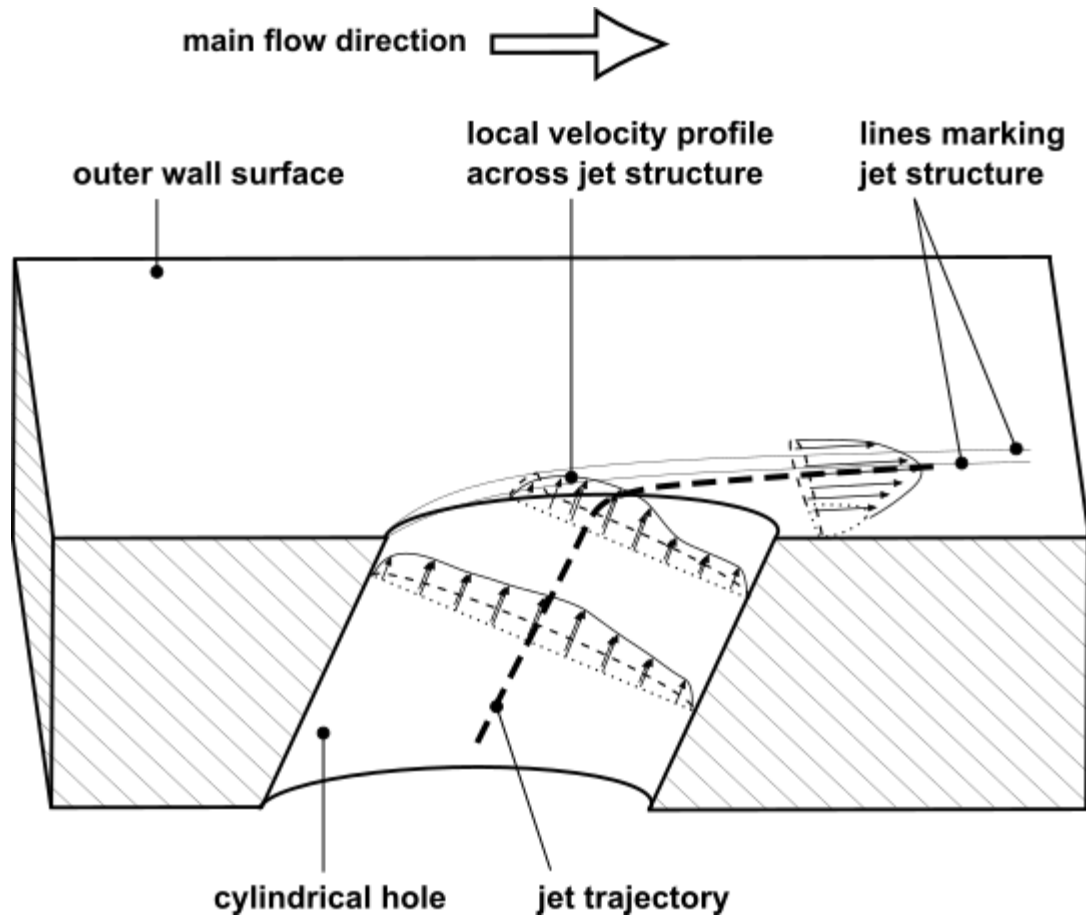
[18.2.3.1. Injection Distribution Basics](#)

[18.2.3.2. Injection Distribution Settings for the Jet in Crossflow Option](#)

#### 18.2.3.1. Injection Distribution Basics

For a cylindrical hole with a defined direction, you might want to control details about how mass is injected into the domain. In reality, the magnitude and direction of the fluid velocity might vary locally within the immediate vicinity of a hole exit, as would other quantities such as the local fluid temperature and turbulence intensity. These variations are a result of external conditions such as the relative velocity of any fluid already within the domain that is approaching the hole exit to form a cross-flow. These variations are also influenced by local geometry, such as the shape and direction of the hole. The settings on the **Injection Distribution** tab enable you to control details about how mass is injected into the domain and details about the local inflow conditions. As the mesh is refined near the injection positions, these local secondary flow patterns and condition variations can be resolved.

There is only one option for the type of injection distribution: `Jet in Crossflow`. The `Jet In Crossflow` option assumes that the fluid from each hole is injected from a boundary (that includes the hole exit) into a crossflow. However, the flow equation source terms for mass, momentum, and so on, are applied throughout a small source volume near each hole exit. This source volume has a shape that approximates the jet structure. A typical jet structure bends with the main flow direction after leaving a wall from an injection hole, and has varying velocity profiles along its trajectory, as shown in [Figure 18.1: Jet from a Wall into a Crossflow \(p. 260\)](#).

**Figure 18.1: Jet from a Wall into a Crossflow**

### 18.2.3.2. Injection Distribution Settings for the Jet in Crossflow Option

The `Jet In Crossflow` option has settings to specify:

- A detail about the crossflow: the **Inflow Boundary** from which the crossflow originates
- Details about how mass is injected: injection trajectory, injection shape, injection variables.

The exact shape of each hole source volume is described in terms of the injection trajectory and cross-sectional shape. Injection variables carry other properties of the jets.

The following topics are discussed:

[18.2.3.2.1. The Inflow Boundary Setting](#)

[18.2.3.2.2. Injection Trajectory](#)

[18.2.3.2.3. Injection Shape](#)

[18.2.3.2.4. Injection Variables](#)

#### 18.2.3.2.1. The Inflow Boundary Setting

The inflow boundary is the reference inlet for the domain. During a run, CFX-Solver uses this boundary to calculate the strength of the crossflow (local blowing ratio).

### 18.2.3.2.2. Injection Trajectory

The injection trajectory is a curvilinear flow path that the fluid follows, the upstream end being within the hole and the downstream end being past the hole exit, out in the crossflow. The trajectory extends approximately one hole diameter upstream and downstream of the hole exit. [Figure 18.1: Jet from a Wall into a Crossflow \(p. 260\)](#) shows an example of an injection trajectory.

**Injection Trajectory > Option** can only be set to `Standard`. This trajectory model, which is described by auf dem Kampe, Völker, and Zehe (2010) [231], takes into account:

- The hole direction, which can be thought of as the direction of the drilling that created the hole (**Injection Details** tab > **Injection Model** > **Hole Direction**),
- The crossflow conditions near the hole exit such as the local blowing ratio and the density ratio, and
- The trajectory model parameters.

The trajectory is calculated in terms of local hole coordinates. These coordinates are shown in [Figure 18.2: Local Hole Coordinates \(p. 261\)](#). The local x coordinate is parallel to the **Hole Direction** projected onto the boundary. The local z coordinate is normal to the boundary. The local y coordinate is transverse. The origin of the local hole coordinates is at the specified hole center position.

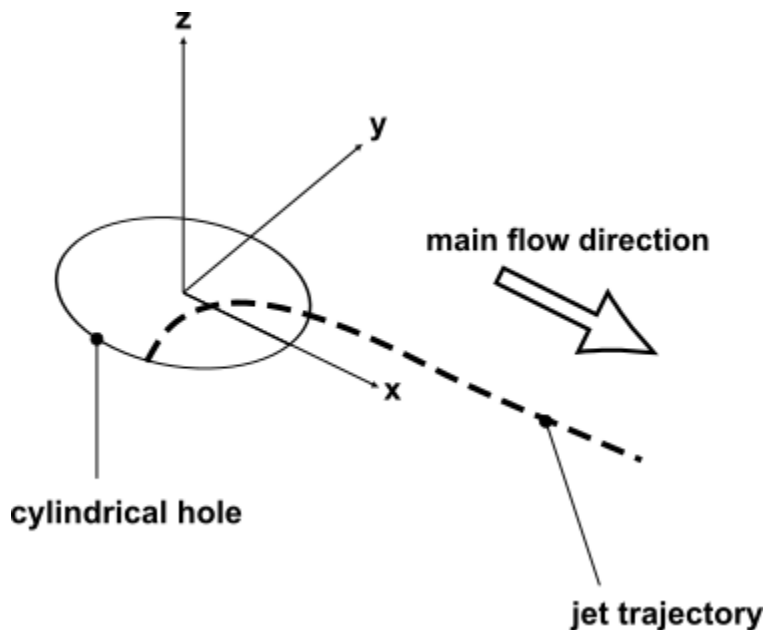
---

#### Note:

Drilling paths that are precisely normal to the boundary should be avoided because they have an arbitrary local coordinate system. A small angle, for example 1 degree, should be used instead.

---

**Figure 18.2: Local Hole Coordinates**



Inside the hole passage, the trajectory follows the drilling direction. In the region of the hole exit and beyond the hole exit, Equation 18.1 (p. 262) finds the y and z coordinates of points along the trajectory as a function of the local x coordinate.

$$f_{\phi}\left(\frac{x}{d}\right) = \frac{\phi_0}{d} + \sqrt{A_{\phi} \frac{IR}{\sin\psi_{\text{real}}}} \cdot \ln \left[ \frac{\sqrt{\left(\frac{x-x_0}{d}\right)^2 + 2B_{\phi} \frac{x-x_0}{d} + A_{\phi} \frac{IR \cot^2 \psi_{\text{real}}}{\sin\psi_{\text{real}}} + \frac{x-x_0}{d} + B_{\phi}}}{\sqrt{A_{\phi} \frac{IR \cot^2 \psi_{\text{real}}}{\sin\psi_{\text{real}}} + B_{\phi}}} \right] \quad (18.1)$$

where:

- $x$  is the x-coordinate in local hole coordinates.
- $\phi$  stands for y when the equation is being solved for y-coordinates of the trajectory, or z when the equation is being solved for z-coordinates of the trajectory.
- $x_0/d$  is **Relative Offset Y** when the equation is being solved for y-coordinates of the trajectory, or 0 (zero) when the equation is being solved for z-coordinates of the trajectory.
- $\phi_0/d$  is **Relative Reference Y** when the equation is being solved for y-coordinates of the trajectory, or **Relative Reference Z** when the equation is being solved for z-coordinates of the trajectory.
- $A_{\phi}$  is **A Coefficient Y** when the equation is being solved for y-coordinates of the trajectory, or **A Coefficient Z** when the equation is being solved for z-coordinates of the trajectory.
- $B_{\phi}$  is **B Coefficient Y** when the equation is being solved for y-coordinates of the trajectory, or **B Coefficient Z** when the equation is being solved for z-coordinates of the trajectory.
- $\psi_{\text{real}}$  is the effective azimuthal angle (**Eff. Azimuthal Angle**), which is usually zero, when the equation is being solved for y-coordinates of the trajectory, or the effective elevation angle (**Eff. Elevation Angle**) when the equation is being solved for z-coordinates of the trajectory.
- $d$  is the local hole diameter.
- $IR$  is the momentum ratio, which is automatically derived from the local blowing ratio and the local density ratio.

**Relative Offset Y**, **Relative Reference Y**, and **Relative Reference Z** dictate where the trajectory intersects the boundary at the hole exit. The A and B coefficients describe how the local y and local z coordinates change.

Under **Spherical Angles**, the effective angles describe the direction of the trajectory downstream of the hole exit:

- The Effective Azimuthal Angle (**Eff. Azimuthal Angle**) measures the local direction of the jet trajectory, as an angle in the local xy plane, measured from the x-axis to the projection of the trajectory's tangent line on the local xy plane.
- The Effective Elevation Angle (**Eff. Elevation Angle**) measures the local direction of the jet trajectory, as an angle in the local xz plane, measured from the x-axis to the projection of the trajectory's tangent line on the local xz plane.

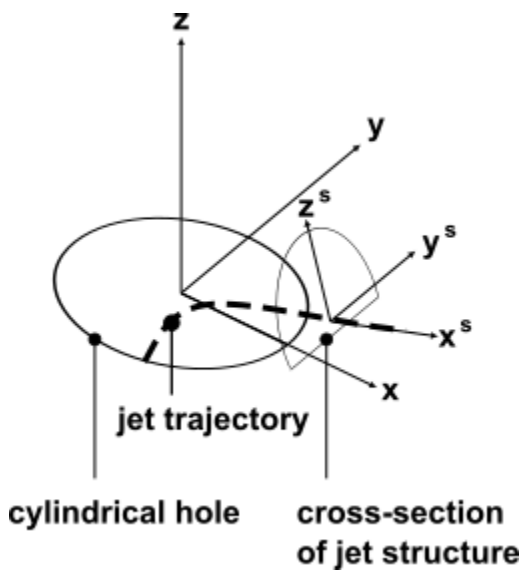
- You can optionally set a minimum value (**Min. Elevation Angle**) for the elevation angle. The trajectory is clipped if and where the effective elevation angle would become less than the set minimum, thereby preventing the trajectory from becoming parallel to the boundary. Too low an effective elevation angle could create numerical problems, especially if the trajectory were to lie flat on the boundary.

### 18.2.3.2.3. Injection Shape

The jet structure, which forms the source volume, is constructed from a series of cross-sections that are perpendicular to the trajectory. The outline shape of these cross-sections are calculated according to the **Injection Shape** settings.

Figure 18.3: Local Hole Coordinates and a Jet Cross-section showing Cross-sectional Coordinates (p. 263) shows an example of a jet that has a simple circular cross-section. The cross section has its own coordinates, with  $x^s$  being along the trajectory. Note that the cross-section shown in this example has a flat edge as a result of being truncated at the boundary. The **Injection Shape** settings control the shape of jet cross-section before any truncation by boundaries.

**Figure 18.3: Local Hole Coordinates and a Jet Cross-section showing Cross-sectional Coordinates**



**Injection Shape > Option** can only be set to `Radial Fourier Series`, where the local cross-section radius around the trajectory is a function, in the form of a truncated Fourier series, of the polar angle around the trajectory. The cross-sections are mathematically modeled using Equation 18.2 (p. 263):

$$r_{csb}^s = \frac{a_0}{2} + \sum_{k=1}^2 a_k \cdot \cos(k\varphi^s) + b_k \cdot \sin(k\varphi^s) \quad (18.2)$$

where:

- $a_0/d$  is **Fourier A0 Coefficient**,
- $a_1/d$  is **Fourier A1 Coefficient**,



- $b_1/d$  is **Fourier B1 Coefficient**,
- $a_2/d$  is **Fourier A2 Coefficient**,
- $b_2/d$  is **Fourier B2 Coefficient**,
- $d$  is the hole diameter,
- $\varphi$  is the polar angle in the local  $y^s z^s$  plane, measured from the local  $y^s$ -axis towards the local  $z^s$ -axis,
- $r_{csb}^s$  is the cross-sectional radius in that direction,
- $s$  denotes the local trajectory frame.

The coefficients can be adjusted to fine tune the injection shape if required, and need not be constant.

Note that the coefficients of the Fourier series are normalized by the hole diameter. If, for example, the A0 coefficient is set to 1, and the other coefficients are set to 0, then the cross-sections are circular and centered on the trajectory, and the source volume for each hole is a truncated cylinder, possibly curved depending on the strength of the crossflow.

---

**Note:**

The shape model is described by auf dem Kampe, Völker, and Zehe (2010) [231].

---

#### 18.2.3.2.4. Injection Variables

For each primitive solution variable relevant to the current simulation (for example, velocity, temperature, and so on) you can select an option to describe how that variable varies within the source volume at each hole.

There are several available options for specifying variations, as described in the following sections:

18.2.3.2.4.1. Uniform

18.2.3.2.4.2. Quadratic Radial Polynomial

18.2.3.2.4.3. Value

18.2.3.2.4.4. Standard Jet Velocity

18.2.3.2.4.5. Cartesian Vector Components

18.2.3.2.4.6. Cylindrical Vector Components

Each option either:

- Defines a distribution function, where variations in scalars or vector magnitudes are expressed in terms of normalized values, or
- Directly defines values for scalar quantities, vector magnitudes, or vector components.

For each injection variable controlled by a distribution function, the magnitude given by the distribution function is automatically scaled so that the net quantity applied over the source volume at each hole (for example momentum, enthalpy) is approximately constant irrespective of the form of the distribution function. Also, the velocity magnitude is scaled to be consistent with the mass flow. This scaling is done in a similar way to that shown by auf dem Kampe and Völker (2010) [232].

#### 18.2.3.2.4.1. Uniform

With the `Uniform` option, the variable has the same value everywhere within the source volume.

Note that for a vector quantity such as velocity, the `Uniform` option means that the *magnitude* is uniform; the direction is always parallel to the trajectory.

#### 18.2.3.2.4.2. Quadratic Radial Polynomial

The `Quadratic Radial Polynomial` option models variation of a variable in the radial direction from the trajectory, using a quadratic function operating on the normalized radial coordinate. In the case of a vector variable, this distribution function dictates the magnitude of the vector. The direction is always parallel to the tangent at the nearest point on the trajectory.

The equation for the model is shown in [Equation 18.3 \(p. 265\)](#).

$$\psi = A + B \left( \frac{r^s}{r_{csb}^s} \right) + C \left( \frac{r^s}{r_{csb}^s} \right)^2 \quad (18.3)$$

Here,  $r^s$  is the radial position from the trajectory in the local trajectory frame, and  $r_{csb}^s$  is the cross-sectional radius in that frame ([Equation 18.2 \(p. 263\)](#)). The ratio of the two therefore ranges from 0 to 1.

The model requires three dimensionless coefficients:

- `A Coefficient` for the constant,
- `B Coefficient` for the first order term, and
- `C Coefficient` for the second order term.

Note that setting these coefficients to 1,0,0, respectively, is effectively the same as choosing the `Uniform` option (see [Uniform \(p. 265\)](#)).

#### 18.2.3.2.4.3. Value

You can specify the value(s) of a scalar variable using the `Value` option, which allows a CEL expression. For example, you could use a CEL expression to specify a position-dependent scalar value.

#### 18.2.3.2.4.4. Standard Jet Velocity

The `Standard Jet Velocity` option is available only for vector variables (for example, `Velocity`) and is an extension of the `Quadratic Radial Polynomial` option.

This option facilitates the setup of a counter-rotating vortex secondary flow pattern within the jet. The model for this flow pattern is described by auf dem Kampe, Völker, and Zehe (2010) [231].

The governing equations for the velocity components,  $u^s$ ,  $v^s$ , and  $w^s$ , in cross-sectional coordinate directions  $x^s$ ,  $y^s$ , and  $z^s$ , respectively, are written as functions of transformed (in fact, rotated)  $y^s$  and  $z^s$  coordinates, called  $y_r$  and  $z_r$ , respectively, which are calculated as follows:

$$\begin{pmatrix} y_r \\ z_r \end{pmatrix} = \begin{pmatrix} \cos B_{v,w} & \sin B_{v,w} \\ -\sin B_{v,w} & \cos B_{v,w} \end{pmatrix} \begin{pmatrix} \frac{y^s}{r_{csb}^s} \\ \frac{z^s}{r_{csb}^s} \end{pmatrix} \quad (18.4)$$

Here,  $B_v$  and  $B_w$  are angles used to rotate the local  $s$  coordinate frame for the calculation of velocity components  $v^s$  and  $w^s$  respectively.

Using the transformed coordinates  $y_r$  and  $z_r$ , the velocity components are calculated using the following equations:

$$\begin{aligned} u^s &= A_u + B_u \left( \frac{r^s}{r_{csb}^s} \right) + C_u \left( \frac{r^s}{r_{csb}^s} \right)^2 \\ v^s &= A_v \left[ \frac{\cos(C_v y_r)}{1 + (D_v z_r)^2} - \frac{\cos(D_v z_r)}{1 + (C_v y_r)^2} \right] \\ w^s &= A_w \left[ \frac{\cos(C_w y_r)}{1 + (D_w z_r)^2} \right] \end{aligned} \quad (18.5)$$

where:

- Coefficients  $A_u$ ,  $B_u$ , and  $C_u$ , are non-dimensional.
- $C_v$ ,  $C_w$ ,  $D_v$ , and  $D_w$  are parameters that affect the distributions of velocity components  $v^s$  and  $w^s$ .

#### 18.2.3.2.4.5. Cartesian Vector Components

The `Cartesian Vector Components` option is available only for vector variables. Specify the vector components directly, optionally using CEL expressions. The vector components, **U**, **V**, and **W**, are aligned with the trajectory's  $x$ ,  $y$ , and  $z$  axes, respectively.

#### 18.2.3.2.4.6. Cylindrical Vector Components

The `Cylindrical Vector Components` option is available only for vector variables. Specify the vector components directly, optionally using CEL expressions. The **Axial Component** is aligned with the trajectory's  $x$ -axis.

### 18.2.4. Fluid Values Tab

If one of the fluids is a variable composition mixture, specify the mass fractions of each of the components.

For details, see [Component Details: Opening \(p. 223\)](#).

## 18.2.5. Plot Options Tab

The **Plot Options** tab enables you to create graphics to display scalar and vector values at injection regions, as detailed in the following sections:

### 18.2.5.1. Color by Parameter Value

### 18.2.5.2. Injection Region Vector

#### 18.2.5.1. Color by Parameter Value

Selecting this option and choosing a variable draws injection region symbols colored by the selected variable. The available variables depend on the settings on the **Injection Details** and **Fluid Values** tabs, as applicable. A legend appears by default showing the variable used for coloring the injection region symbols. You can turn off visibility for the legend and the plots by clearing the check box next to the object associated with your injection region in the **Outline** tree view.

#### 18.2.5.2. Injection Region Vector

The **Injection Region Vector** option is available:

- If you have a setting (for example, **Flow Direction**) that is set using `Cartesian Components` or `Cylindrical Components`, or
- If you have set up a hole direction using the `Cylindrical Hole` option.

Selecting this option draws vectors at the injection regions, pointing in the direction specified by the **Vector Components** setting. The available vectors are:

- Direct vector specifications from Injection Details and other settings.

Vectors are formed from grouped parameter settings that form vectors, for example a **Flow Direction** specification using the `Cartesian Components` or `Cylindrical Components` option.

The specified components of these vectors are always applied (without conversion) in the local coordinate frame.

- `Cylindrical Hole Direction`

Vectors are properly aligned with the cylindrical holes, taking into account all of the **Hole Direction** settings (see [Hole Direction \(p. 256\)](#)).

The direction of such vectors shown in CFX-Pre may differ slightly from the direction calculated by CFX-Solver if the Hole Direction specification makes use of the `Normal to Boundary Condition` option, especially where surface curvature is high.

A scaling parameter is available to adjust the length of the vectors. For details, see [Injection Region Vectors \(p. 86\)](#).

## 18.3. User Functions for Injection Regions

To specify injection positions for an injection region, you can define a user function defined by data in a `.csv` file. The data format is the same as for a profile file, which is described in [Profile Data Format \(p. 424\)](#).

An example `.csv` file is shown below:

```
[Name]
Injections

[Parameters]
Minimum Injection Temperature = 328.0 [K]
Coefficient ABC = 42.9
Parameter123 = -600.0

[Spatial Fields]
x, y, z

[Data]
x [ m ], y [ m ], z [ m ], Diameter [ m ]
3.18766475e-01, -4.35341895e-01, 7.27896512e-01, 0.00210000e+00
2.91383922e-01, 3.52843165e-01, 2.80668914e-01, 0.00213000e+00
3.61989558e-01, -3.94276977e-01, 1.41301310e+00, 0.00213000e+00
```

The `[Parameters]` section can be used to define values that are constant across all injection positions.

Each row in the `[Data]` section describes a different injection position. Some columns are required for coordinate data; other columns are optional, and can hold property data.

You can generate a template for an injection region data file using CFD-Post. For details, see [Using CFD-Post to Design an Injection Region Data File \(p. 268\)](#).

You must load the injection region data file into CFX-Pre and then initialize the data (**Tools > Initialize Profile Data**) in order to prepare the data for use in defining the injection positions and properties.

For details on user functions defined by `.csv` files, see [Interpolation \(From File\) User Functions \(Profile Functions\) \(p. 423\)](#).

---

### Note:

You must define at least three injection positions (rows) in any `.csv` file definition. However, not all of the positions need to be within the mesh.

---

### 18.3.1. Using CFD-Post to Design an Injection Region Data File

You can generate a template for an injection region data file (`.csv`) using CFD-Post as follows:

1. Generate the location.
2. Export a profile file (**File > Export**), setting **Type** to `BC-Profile`, **Location** to the previously generated location, **Profile Type** to `Custom`, and selecting variables for additional columns as needed.

## 18.4. Supported Capabilities and Limitations

The following functions are available in the CFX-Solver for evaluations of quantities over injection positions:

- `area`, `areaAve`, `areaInt`
- `ave`
- `count`
- `inside()`
- `massFlow`, `massFlowAve`, `massFlowAveAbs`, `massFlowInt`
- `maxVal`, `minVal`
- `rmsAve`
- `sum`

For example, for an injection region that lies on a surface, the `area` function returns the total area of all injection holes of the injection region.

Like other CEL functions, the functions listed above return conservative values by default. That is, the returned values are derived from the conservative values of the vertices that are directly affected by the injection positions. Alternatively, by using the `hybrid` suffix (see [Quantitative CEL Functions in Ansys CFX in the CFX Reference Guide](#)), these functions can use the values you specify at the injection positions.

If you are using shortcut menus to form CEL expressions, note that injection regions appear under **Physics Locators > Other**, not under **2D** or **3D**.

Injection regions are not compatible with:

- Multi-phase flow

### Other Limitations:

- When the solver control advanced option of **Surface Overlap Type** is set to `Mapped Faces` (see [Injection Control \(p. 306\)](#)), injection positions must be geometrically separated enough to avoid having multiple injection positions directly influence the same mesh elements or vertices.
- For transient blade row cases, hybrid solution values do not correctly show the influence of injection regions on boundary vertices that are directly influenced by any injection region. However, in all cases, conservative solution values are expected to be properly influenced by injection regions.
- When using the `Selected 2D Regions` option, the mapping of the holes on the mesh assumes that, in the region of each hole, the mesh surface is approximately planar. Any significant variation of the mesh normal direction across a given hole cross-section can lead to inaccurate mapping.
- Injection regions cannot be defined with a selected 2D region that has 'thin' mesh topology. In particular, the 2D region cannot contain a thin surface and cannot generally have two or more geometrically coincident faces.

## 18.5. Running Cases that involve Injection Regions

---

- In CFX-Solver Manager, monitors are created for all injection regions, and are listed under the `FLOW/INJECTION REGION` plot line variables (see [Plot Lines Tab in the CFX-Solver Manager User's Guide](#)).
- In the CFX-Solver Output (`.out`) file, injection regions are listed in the section `Boundary Flow and Total Source Term Summary`. The effects of injection regions are therefore included in any domain imbalance calculation.

## 18.6. Postprocessing Injection Regions

---

For injection regions defined via a `.csv` file, you can visualize the injection positions in CFD-Post by defining a point cloud as follows:

1. Select **Insert** > **Location** > **Point Cloud**, and specify a name for the point cloud.
2. Set **Method** to `From File`.
3. Set **File** to the name of the injection region data file (`.csv`).
4. Click **Apply**.

If a `.csv` file contains multiple data sets, you can create multiple point clouds as follows:

1. Select **File** > **Import** > **Import Surface, Line or Point Data**.

The **Import Surface, Line or Point Data** dialog box appears.

2. Set **File** to the name of the `csv` file.
3. Set **Import As** to `Point Cloud`.
4. Click **OK**.

Point clouds are listed in the **Outline** tree view, under `User Locations and Plots`.

---

### Note:

For transient blade row cases, hybrid solution values do not correctly show the influence of injection regions on boundary vertices that are directly influenced by any injection region. However, in all cases, conservative solution values are expected to be properly influenced by injection regions.

---

---

# Chapter 19: Source Points

---

Source points are sources that act on a single mesh element. The location of the point is entered in Cartesian coordinates, and the source is created for the element whose center is closest to the specified point. Source points appear as red spheres in the Viewer. For a transient run or run with a moving mesh, the closest element is identified once at the start and is used for the remainder of the run.

This chapter describes:

- 19.1. [Basic Settings Tab](#)
- 19.2. [Sources Tab](#)
- 19.3. [Fluid Sources Tab](#)
- 19.4. [Sources in Solid Domains](#)
- 19.5. [Source Points and Mesh Deformation](#)

Sources are specified in a way similar to subdomain sources with the exceptions that momentum and radiation sources cannot be specified and only "Total Source" values can be entered. For details, see [Subdomains](#) (p. 247).

The visibility of source points can be turned on and off using the check box in the tree view. For details, see [Object Visibility](#) (p. 51) and [Outline Tree View](#) (p. 37).

Additional information on sources is available in [Sources in the CFX-Solver Modeling Guide](#).

---

## Note:

You can define injection regions to conveniently create fluid mass sources or sinks at a set of positions on selected surfaces or within the volume of a selected domain. For details, see [Injection Regions](#) (p. 253).

---

## 19.1. Basic Settings Tab

---

The **Basic Settings** tab defines the coordinate frame and the point coordinates for the source point.

1. Enter **Cartesian Coordinates** for the source point.  
Points entered are relative to the selected coordinate frame.
2. Use the default coordinate frame or a user-specified coordinate frame. For details, see:
  - [Coordinate Frames in the CFX-Solver Modeling Guide](#)
  - [Coordinate Frame Basic Settings Tab](#) (p. 371).



The default coordinate frame, `Coord 0`, will be used as the basis for the entered Cartesian coordinates, unless you have created your own coordinate frame and have selected it from the drop-down list.

## 19.2. Sources Tab

---

The point sources that can be set depend on the physical models used in the simulation. Possible source types include: mass (continuity), energy, radiation, Additional Variable, component mass fraction, turbulence.

### 19.2.1. Single-Phase Fluid Sources

1. Select the **Sources** check box to specify sources for the point.
2. Select the equation source to specify.

Possible equation source types include: mass fraction, energy, continuity (mass), turbulence, Additional Variable. For details, see [Source Coefficient / Total Source Coefficient in the CFX-Solver Modeling Guide](#).

3. Select the **Continuity** check box to set sources for the continuity equation.

For details, see [Mass \(Continuity\) Sources in the CFX-Solver Modeling Guide](#).

Additional information is available in:

- [Sources in the CFX-Solver Modeling Guide](#)
- [General Sources in the CFX-Solver Modeling Guide](#).

#### 19.2.1.1. Component Mass Fractions

This is only available when mixtures are included in the fluids list in the domain. You can specify a **Total Source** and an optional **Total Source Coefficient** for improved convergence for strongly varying sources. For details, see [General Sources in the CFX-Solver Modeling Guide](#).

#### 19.2.1.2. Additional Variables

Set the **Total Source** for the Additional Variable and an optional **Total Source Coefficient**. A source for an Additional Variable can be set only if it is solved for.

#### 19.2.1.3. Continuity

Continuity sources differ from other sources because you are introducing new fluid into the domain. Properties that are required of the fluid, which is entering the domain, appear in the **Variables** section of the form. These values are not used if the source is negative, because no new fluid is introduced into the subdomain. For details, see [Mass \(Continuity\) Sources in the CFX-Solver Modeling Guide](#).

### 19.2.1.3.1. Continuity Option

The value of the mass source is set using the **Total Fluid Mass Source** option. For details, see [Mass \(Continuity\) Sources in the CFX-Solver Modeling Guide](#).

### 19.2.1.3.2. Additional Variables

Set a value for any Additional Variables that are introduced with the mass source. For details, see [Mass \(Continuity\) Sources in the CFX-Solver Modeling Guide](#).

### 19.2.1.3.3. Component Mass Fractions

Enter mass fractions of each of the components in the mass source. For details, see [Mass \(Continuity\) Sources in the CFX-Solver Modeling Guide](#).

### 19.2.1.3.4. Temperature

Enter the temperature for the mass source.

### 19.2.1.3.5. Velocity

Set velocity components for the mass source.

## 19.2.1.4. Energy

A total source for the energy equation can be set. The optional **Total Source Coefficient** provides improved convergence for strongly varying sources. An energy source can be specified when the parent domain models heat transfer using the thermal energy or total energy models.

## 19.2.1.5. Turbulence Eddy Dissipation or Turbulence Kinetic Energy

When the flow is turbulent, a total source for the Turbulence Eddy Dissipation or Turbulence Kinetic Energy can be specified. The optional **Total Source Coefficient** provides improved convergence for strongly varying sources. For details, see [General Sources in the CFX-Solver Modeling Guide](#).

## 19.2.2. Multiphase Bulk Sources

In a multiphase simulation, source terms that apply to all fluids in the simulation are set on the **Sources** tab. Bulk sources take account of the volume fraction of each phase when applying the source term. For details, see [Bulk Sources in the CFX-Solver Modeling Guide](#).

- Select **Bulk Sources** to specify bulk sources. Bulk sources apply to all fluids in a multiphase simulation.

## 19.2.3. Multiplying Sources by Porosity

Selecting the **Multiply by Porosity** check box causes the equation source to be scaled by the porosity value. For example, if a porous domain has a volume porosity of 0.8, then if the **Multiply by Porosity** check box is selected, 80% of the source is applied to the fluid; if the check box is not selected then 100% of the source is applied to the fluid.

## 19.3. Fluid Sources Tab

---

Fluid sources can be set when more than one fluid is selected. The options depend on the type of simulation you are running, and whether bulk sources are used.

1. Select the fluid from the **Fluid Specific Source Point Sources** list
2. Select the check box next to the selected variable to enter a source, then select the **Continuity** check box.
3. Enter a value for **Total Source**.
4. Optionally, enter a total mass source coefficient for either pressure or volume fraction.

For details, see [Source Coefficient / Total Source Coefficient in the CFX-Solver Modeling Guide](#).

5. Set the values for the continuity equation.

For details, see [Mass \(Continuity\) Sources in the CFX-Solver Modeling Guide](#).

## 19.4. Sources in Solid Domains

---

Source points can exist in solid domains to provide sources of thermal energy and radiation.

- Select **Energy** to enable an energy source.

For details, see [General Sources in the CFX-Solver Modeling Guide](#).

## 19.5. Source Points and Mesh Deformation

---

When mesh deformation is selected, the volume element whose centroid is closest to the source point at the beginning of the simulation will move with the mesh (if that part of the mesh is deforming). The location of the point source will, therefore, move as the mesh deforms. For details, see [Mesh Deformation \(p. 174\)](#).

---

# Chapter 20: Rigid Bodies

---

This chapter describes:

## 20.1. Rigid Body User Interface

A rigid body is a solid object that moves through a fluid without deforming. Its motion is dictated by the fluid forces and torques acting upon it, plus any external forces (such as gravity) and external torques. Within Ansys CFX, a rigid body can be modeled in two ways:

- A rigid body can be defined by a collection of 2D regions that form its faces. When a rigid body is modeled in this way, the rigid body itself does not need to be meshed. Mesh motion is used to move the mesh on the rigid body faces in accordance with the solution of the rigid body equations of motion.
- Alternatively, an immersed solid can be defined as a rigid body. In this case the motion of the immersed solid is dictated by the solution of the rigid body equations of motion.

Implementation information for Rigid Bodies is available in [Rigid Body Theory in the CFX-Solver Theory Guide](#).

## 20.1. Rigid Body User Interface

---

The user interface for setting up a rigid body defined as a collection of 2D regions is presented in the following sections:

### 20.1.1. Insert Rigid Body Dialog Box

### 20.1.2. Basic Settings Tab


### 20.1.3. Dynamics Tab

### 20.1.4. Initial Conditions Tab

The user interface for setting up a rigid body implemented as an immersed solid consists of rigid body settings found on the **Basic Settings** tab for the immersed solid domain. These settings are essentially the same as those on the **Basic Settings** and **Dynamics** tabs in the details view for the rigid body object.

### 20.1.1. Insert Rigid Body Dialog Box

Use this method to simulate a rigid body by using a collection of 2D regions.

The **Insert Rigid Body** dialog box is used to initiate the creation of a new rigid body object. It is accessible by clicking *Rigid Body* , or by selecting **Insert > Rigid Body**.

---

**Note:**

Another way to simulate a rigid body is by using an immersed solid that uses a **Domain Motion** option of *Rigid Body Solution*. In this case, the rigid body settings appear in the details view for the immersed solid domain. All of the rigid body settings for an immersed solid appear on the **Basic Settings** tab for the immersed solid domain. The analogous settings for the rigid body object are distributed between the **Basic Settings** and **Dynamics** tabs of the rigid body object. Although there are initialization settings on the **Initialization** tab of the rigid body object, there are no corresponding initialization settings for a rigid body defined as an immersed solid.

---

Immersed solids are described in [Immersed Solids in the CFX-Solver Modeling Guide](#).

## 20.1.2. Basic Settings Tab

The **Basic Settings** tab for the rigid body object has the following settings:

### 20.1.2.1. Mass

Specify the mass of the rigid body. The mass is used in the calculation of translational acceleration due to applied forces. It is also used in combination with the specified gravity vector (the gravity vector specified with the rigid body, not with any buoyancy model) to calculate the force due to gravity. For details, see [Gravity](#) (p. 280).

The force due to gravity acts through the specified center of mass. For details, see [Center of Mass](#) (p. 280).

### 20.1.2.2. Location

Specify all 2D regions (belonging to fluid domains) that physically contact the faces of the rigid body. The forces and torques exerted by the fluid on all of these faces will contribute to the motion of the rigid body. Note that if the rigid body is defined as an immersed solid, then the location is automatically just the location of the immersed solid, and no further location setting is required.

To have these faces move automatically in accordance with the rigid body solution, you must also specify that these faces have mesh motion provided by the rigid body solution. To do this, use the **Boundary Details** tab in the details view for the boundary containing the faces. Additional faces or mesh elements, which do not form part of the rigid body, can also have their mesh motion partially or wholly specified by the rigid body solution. To do this, use the **Boundary Details** tab for boundaries and the **Mesh Motion** tab for subdomains. Applying the rigid body solution to additional faces or mesh elements could be used, for example, to help control the way the mesh distorts as the rigid body moves.

### 20.1.2.3. Coordinate Frame

You should specify a stationary coordinate frame that has its origin at the center of mass of the (physical) rigid body when the body is in its position at the start of the simulation. The coordinate frame you specify must have the same orientation as the axes used to define the mass moment of inertia of the body in its initial position (see [Mass Moment of Inertia \(p. 277\)](#)). In this documentation, this coordinate frame is referred to as the *rigid body coordinate frame*. The position and orientation of the rigid body are always calculated relative to this coordinate frame, and all other settings for the rigid body object (for example, gravity and degrees of freedom) are with respect to this coordinate frame, unless otherwise specified in this documentation.

If you choose a coordinate frame that does not have the position of the rigid body at the start of the simulation, then you must specify an initial position for the rigid body (on the **Initial Conditions** tab) that corresponds to the translation required to move the coordinate frame into the rigid body's initial position. See [Initial Conditions Tab \(p. 280\)](#) for more information. Note that initial conditions settings are not currently available for rigid bodies defined as immersed solids.

### 20.1.2.4. Mass Moment of Inertia

Specify absolute values of the components of the mass moment of inertia for the rigid body with respect to a coordinate frame that:

- Has the same initial orientation as the rigid body coordinate frame
- Has its origin at the rigid body center of mass
- Moves rigidly with the rigid body.

The mass moment of inertia components are used to define the mass moment of inertia matrix described in [Rotational Equations of Motion in the CFX-Solver Theory Guide](#).

For example, **YY Component** is defined as:

$$\int ((x-x_G)^2 + (z-z_G)^2) dm$$

and **XY Component** is defined as:

$$\int (x-x_G)(y-y_G) dm$$

where the center of mass is given by  $(x_G, y_G, z_G)$ , and  $dm$  is a differential element of mass.

### 20.1.3. Dynamics Tab

The **Dynamics** tab for the rigid body object contains settings for the following items that influence the motion of the rigid body:



- External forces
- External torques

- Degrees of freedom of movement
- Gravity vector.

With the exception of **Rotational Degrees of Freedom** all dynamics settings are in comparison to the rigid body coordinate frame.

The **Dynamics** tab has the following settings:

### 20.1.3.1. External Force Definitions

External forces other than gravity are specified in the **External Force Definitions** frame. Use the *Add new item*  and *Remove selected item*  icons to add or remove external force definitions from the list.

Each external force definition has one of the following options:

- None

This setting deactivates the external force.

- Spring



These settings enable you to set up a spring force that is applied to the rigid body. Specify the neutral position with respect to the rigid body coordinate frame via the **Linear Spring Origin** settings. Whenever the center of mass of the rigid body is at the **Linear Spring Origin**, the applied spring force is zero. The spring force develops as the center of mass of the rigid body moves away from the neutral position, with a force-to-displacement ratio specified by the **Linear Spring Constant**. For details on how the center of mass is located, see [Center of Mass \(p. 280\)](#).

- Value

These settings enable you to specify an external force using Cartesian force components (along the principal axes of the rigid body coordinate frame).

The external forces are applied through the center of mass of the rigid body so that they cause translational accelerations only (and not any rotational accelerations).

### 20.1.3.2. External Torque Definitions

External torques are specified in the **External Torque Definitions** frame (in the details view). Use the *Add new item*  and *Remove selected item*  icons to add or remove external torque definitions from the list.

Each external torque definition has one of the following options:

- None

This setting deactivates the external torque.

- Spring

These settings enable you to set up a spring torque that is applied to the rigid body. Specify the neutral orientation (with respect to the rigid body coordinate frame) via the **Equilibrium Orientation** settings. You define the orientation using three angular displacements (specifically, Euler angles), which are applied by the software using the ZYX convention:

- The **Euler Angle Z** setting modifies the orientation by a rotation about the Z axis (using the right-hand rule to determine the direction).
- The **Euler Angle Y** setting then further modifies the orientation by a rotation about the (modified) Y axis (using the right-hand rule to determine the direction).
- The **Euler Angle X** setting then further modifies the orientation by a rotation about the (twice modified) X axis (using the right-hand rule to determine the direction).

These Euler angles are described in more detail in [Rigid Body Motion in the CFX-Solver Modeling Guide](#).

Whenever the orientation of the rigid body matches that specified by the **Equilibrium Orientation** settings, the applied spring torque is zero. The spring torque develops as the rigid body rotates away from the neutral orientation, with a torque-to-angular-displacement ratio specified by the **Torsional Spring Constant** settings.

The torque develops due to changes in Euler angles, but is applied on the corresponding axes of the rigid body coordinate frame. The torque spring implementation described here is recommended for use only when both the equilibrium orientation and the rigid body orientation can be described with small Euler angles (a few degrees).

- Value

These settings enable you to specify an external torque using Cartesian torque components around the principal axes of the rigid body coordinate frame.

### 20.1.3.3. Degrees of Freedom

#### 20.1.3.3.1. Translational Degrees of Freedom

This setting determines the combination of axes along which the rigid body may translate. In this context, the axes are those of the rigid body coordinate frame.

#### 20.1.3.3.2. Rotational Degrees of Freedom

This setting determines the axes about which the rigid body may rotate. In this context, the axes are of a coordinate frame that:

- Has the same initial orientation as the rigid body coordinate frame
- Has its origin at the rigid body center of mass
- Moves rigidly with the rigid body.



### 20.1.3.4. Gravity

Specify a gravity vector in the rigid body coordinate frame that defines the downward direction and the magnitude of free-fall acceleration due to gravity.

### 20.1.4. Initial Conditions Tab

The **Initial Conditions** tab contains settings for initializing the rigid body solver. These settings are described in the following subsections.

---

**Note:**

These settings are not available for rigid bodies defined as immersed solids.

---

**Note:**

For the settings on the **Initial Conditions** tab, the **Automatic** option provides initial values of zero, and the **Automatic with Value** option provides initial values as specified. In both cases, the values used at the beginning of a *restarted* run are taken from the results of the previous run. When you restart a run, select **Continue History From** and choose initial values from the previous results. To access this option, you should be in the **Define Run** dialog box of CFX-Solver Manager and both **Initial Values Specification** and **Continue History From** must be selected.

---

#### 20.1.4.1. Center of Mass

These settings define the location (in the rigid body coordinate frame) of the center of mass of the rigid body at the start of the simulation. These settings should be set with non-zero values only if the origin of the rigid body coordinate frame is not at the center of mass of the rigid body at the start of the simulation. If non-zero values are required, then they represent the translation from the origin of the rigid body coordinate frame to the position of the rigid body center of mass at the start of the simulation.

---

**Note:**

Setting non-zero values for the **Center of Mass** settings does not impose a mesh motion at the start of the simulation; rather, the initial position of the mesh is assumed to represent the physical location of the rigid body at the start of the simulation.

---

**Note:**

It is recommended that you define a rigid body coordinate frame that has its origin located at the initial center of mass of the rigid body (that is, the center of mass of the rigid body as positioned in the initial mesh). You could then set all the **Center of Mass** settings on the **Initial Conditions** tab to zero (or select the **Automatic** option to set the initial values to zero automatically).

---

### **20.1.4.2. Linear Velocity**

These settings initialize the translational velocity of the rigid body in the rigid body coordinate frame. The velocity components are taken as being along the principal axes of the rigid body coordinate frame.

### **20.1.4.3. Angular Velocity**

These settings initialize the angular velocity of the rigid body in the rigid body coordinate frame. The angular velocity components are taken as being about the principal axes of the rigid body coordinate frame, using the right-hand rule to establish direction.

### **20.1.4.4. Linear Acceleration**

These settings initialize the translational acceleration of the rigid body in the rigid body coordinate frame. The acceleration components are taken as being along the principal axes of the rigid body coordinate frame.

### **20.1.4.5. Angular Acceleration**

These settings initialize the angular acceleration of the rigid body in the rigid body coordinate frame. The angular acceleration components are taken as being about the principal axes of the rigid body coordinate frame, using the right-hand rule to establish direction.



---

# Chapter 21: Units and Dimensions

---

This chapter describes:

- 21.1. Units Syntax
- 21.2. Using Units in CFX-Pre
- 21.3. Setting the Solution Units

## 21.1. Units Syntax

---

Dimensional quantities are defined in units or a combination of units. For example, mass can have units of [kg], [g] or [lb] (and many others); pressure can have units of [atm], [N m<sup>-2</sup>] and [Pa] (and many others).

The general units syntax in CFX is defined as [multiplier|unit|^power] where multiplier is a multiplying quantity (such as mega, pico, centi, and so on), unit is the unit string (kg, m, J, and so on), and power is the power to which the unit is raised. When typing units in expression, they must be enclosed by square brackets, [...]. You will usually not see the brackets when selecting units from a list of commonly used units. In general, units declarations must obey the following rules:

- A units string consists of one or more units quantities, each with an optional multiplier and optional power. Each separate units quantity is separated by one or more spaces.
- Short forms of the multiplier are usually used. n stands for nano, μ stands for micro, c for centi, k for kilo, m for milli, M for mega and G for giga.
- Powers are denoted by the ^ (*caret*) symbol. A power of 1 is assumed if no power is given.

---

### Note:

The / operator is not supported, so a negative power is used for unit division (for example, [kg m<sup>-3</sup>] corresponds to kilograms per cubic meter).

---

- If you enter units that are inconsistent with the physical quantity being described, then a dialog box will appear informing you of the error, and the units will revert to the previous units.
- Units do not have to be given in terms of the fundamental units (mass, length, time, temperature, angle and solid angle). For instance, Pa (Pascals) and J (Joules) are both acceptable as parts of unit strings.
- Units strings are case sensitive; for example, Kg and KG are both invalid parts of units strings.

To give the units of dynamic viscosity, which has the dimensions of Mass Length<sup>-1</sup> Time<sup>-1</sup>, the unit string  $[kg\ m^{-1}\ s^{-1}]$  (or  $[lb\ ft^{-1}\ hr^{-1}]$ ) is valid.

---

**Note:**

The following unit strings are *invalid*:

$[kg/(metre\ sec)]$	$[kg/(ms)]$
$[kg/m/s]$	$[kg/(m.s)]$
$[kg/(m\ s)]$	$[kg/(m*s)]$
$[kg/(m\ sec)]$	$[lb/(ft\ hr)]$

---

## 21.2. Using Units in CFX-Pre

---

There are a number of details views in CFX-Pre that require the entry of physical quantities. For example, when you set the physical properties for a fluid, or enter values for boundary conditions, the units in which you input the data must be selected.

A list of possible units for the quantity of interest is provided, but you may want to use an expression for the quantity, in which case you must specify the units. You can use any units that are consistent with the quantity you are describing. The default units in CFX-Pre are SI.

The units selector is automatically filled in using the default units for the quantity. You can select other commonly used units for that quantity from the drop-down list in the units selector.

### 21.2.1. Units Commonly Used in CFX

CFX-Pre provides you with a choice of several commonly used units to ease the task of specifying quantities and converting results.

The full list of quantities where commonly used units are available is given in the following table:

Quantity	Commonly used units
Velocity	$[m\ s^{-1}]$ $[km\ hr^{-1}]$ $[mile\ hr^{-1}]$ $[ft\ s^{-1},\ knot]$
Volumetric Flow	$[m^3\ s^{-1}]$ $[litre\ s^{-1}]$ $[gallon\ hr^{-1}]$ $[gallonUSliquid\ hr^{-1}]$


Quantity	Commonly used units
Mass Flow	[kg s <sup>-1</sup> ] [tonne s <sup>-1</sup> ] [lb s <sup>-1</sup> ]
k (turbulence kinetic energy)	[m <sup>2</sup> s <sup>-2</sup> ] [J kg <sup>-1</sup> ]
Epsilon (turbulence dissipation rate)	[m <sup>2</sup> s <sup>-3</sup> ] [J kg <sup>-1</sup> s <sup>-1</sup> ]
Pressure	[Pa] [N m <sup>-2</sup> ] [bar] [torr] [mm Hg] [psi] [psf]
Concentration	[m <sup>-3</sup> ] [litre <sup>-1</sup> ] [foot <sup>-3</sup> ]
Dynamic Viscosity	[kg m <sup>-1</sup> s <sup>-1</sup> ] [centipoise] [g cm <sup>-1</sup> s <sup>-1</sup> ] [N s m <sup>-2</sup> ] [Pa s] [dyne s cm <sup>-2</sup> ] [lb ft <sup>-1</sup> hr <sup>-1</sup> ] [lbf s ft <sup>-2</sup> ]
Thermal Conductivity	[W m <sup>-1</sup> K <sup>-1</sup> ] [cal cm <sup>-1</sup> s <sup>-1</sup> K <sup>-1</sup> ] [BTU (ft <sup>2</sup> s (F/ft)) <sup>-1</sup> ] [BTU (ft <sup>2</sup> hr (F/ft)) <sup>-1</sup> ]

Quantity	Commonly used units
Specific Heat Capacity	[J kg <sup>-1</sup> K <sup>-1</sup> ] [cal g <sup>-1</sup> K <sup>-1</sup> ] [J g <sup>-1</sup> K <sup>-1</sup> ] [BTU lb <sup>-1</sup> F <sup>-1</sup> ]
Thermal Expansivity	[K <sup>-1</sup> ]
Kinematic Diffusivity	[m <sup>2</sup> s <sup>-1</sup> ] [cm <sup>2</sup> s <sup>-1</sup> ]
Acceleration	[m s <sup>-2</sup> ] [ft s <sup>-2</sup> ]
Temperature	[K] [C] [R] [F]
Density	[kg m <sup>-3</sup> ] [g cm <sup>-3</sup> ] [lb ft <sup>-3</sup> ]
Mass Concentration	[kg m <sup>-3</sup> ] [g l <sup>-1</sup> ]
Mass Fraction	[kg kg <sup>-1</sup> ] [g kg <sup>-1</sup> ]
Length	[m] [mm] [foot] [in]
Mass Flow in	[kg s <sup>-1</sup> ] [tonne s <sup>-1</sup> ] [lb s <sup>-1</sup> ]
Volumetric Flow in	[m <sup>3</sup> s <sup>-1</sup> ] [litre s <sup>-1</sup> ] [gallon hr <sup>-1</sup> ]

Quantity	Commonly used units
	[gallonUSliquid hr <sup>-1</sup> ]
Heat Transfer Coefficient	[W m <sup>-2</sup> K <sup>-1</sup> ]
Heat Flux in	[W m <sup>-2</sup> ]
Time	[s] [min] [hr]
Shear Strain rate	[s <sup>-1</sup> ]
Energy Source	[W m <sup>-3</sup> ] [kg m <sup>-1</sup> s <sup>-3</sup> ]
Energy Source Coefficient	[W m <sup>-3</sup> K <sup>-1</sup> ] [kg m <sup>-1</sup> s <sup>-3</sup> K <sup>-1</sup> ]
Momentum Source	[kg m <sup>-2</sup> s <sup>-2</sup> ]
Momentum Source Lin. Coeff.	[kg m <sup>-3</sup> s]
Momentum Source Quad. Coeff.	[kg m <sup>-4</sup> ]
Per Time	[s <sup>-1</sup> ]
Angle	[radian] [degree]
Angular Velocity	[radian s <sup>-1</sup> ] [rev min <sup>-1</sup> ]
Specific Enthalpy	[J kg <sup>-1</sup> ] [m <sup>2</sup> s <sup>-2</sup> ]
Energy	[J]

### 21.2.2. Defining Your Own Units

The commonly used units array is only a subset of the possible units you can use in CFX-Pre. Each unit is a combination of one or more base dimensions. To specify your own units for a quantity, click

the *Enter Expression* icon  for the associated variable and type the value and units into the data area using the syntax. For details, see [Units Syntax \(p. 283\)](#).

There are many base units to choose from; most units in common use are valid as parts of unit strings. You can specify any quantity in any valid units as long as you adhere to the units definition syntax.



## 21.3. Setting the Solution Units

---

There are two sets of units in CFX-Pre: the units visible when selecting **Edit > Options** from the main menu, which are also used for mesh import and transformation, and the solution units set in the **Solution Units** details view (available from the main menu under **Insert > Solver**). The solution units are the units that the CFX-Solver writes in the results file. For details, see [Units \(p. 89\)](#).

Setting the solution units does not alter the units you can use to define quantities in CFX-Pre. These are the units in which the results file is written. Additionally, these are the units assumed in the summary at the end of the CFX-Solver Output file, when data such as variable range and forces on walls is presented.

When postprocessing a results file in CFD-Post, the units used are not necessarily those used in the results file. CFD-Post will convert to your preferred units.

Most common units can be used for the solution units; however, some important restrictions apply:

- The temperature solution units must be an absolute scale; for example, Kelvin [K] or Rankine [R]. Celsius and Fahrenheit cannot be used. Temperature quantities elsewhere in CFX-Pre can be set in Celsius and Fahrenheit.
- The solution units must not be changed when restarting a run. The units in the initial guess file will assume the units used in the current CFX-Solver definition (.def) file.
- You must not change the length units outside of CFX-Pre, for example, by editing the CCL in a CFX-Solver input file. The mesh is written to the CFX-Solver input file using the length units; therefore, once the CFX-Solver input file has been written, these units should not change.

---

# Chapter 22: Solver Control

---

**Solver Control** is used to set parameters that control the CFX-Solver during the solution stage.

This chapter describes:

- 22.1. Basic Settings Tab
- 22.2. Equation Class Settings Tab
- 22.3. Particle Control Tab
- 22.4. Rigid Body Control Tab
- 22.5. Advanced Options Tab

You can find further help on setting solver parameters in [Advice on Flow Modeling in the CFX-Solver Modeling Guide](#).

## 22.1. Basic Settings Tab

---

The **Basic Settings** tab controls following common and simulation specific parameters:

- [Basic Settings: Common](#) (p. 289)
- [Basic Settings for Steady-State Simulations and for Transient Blade Row Cases that use the Harmonic Balance Transient Method](#) (p. 292)
- [Basic Settings for Transient Simulations](#) (p. 293)
- [Immersed Solid Control](#) (p. 293)

### 22.1.1. Basic Settings: Common

#### 22.1.1.1. Advection Scheme

For details, see [Advection Scheme Selection in the CFX-Solver Modeling Guide](#).

#### 22.1.1.2. Transient Scheme

The **Transient Scheme** settings are available for steady-state simulations and for transient blade row cases that use the Harmonic Balance transient method. For details, see [Transient Scheme in the CFX-Solver Modeling Guide](#).

#### 22.1.1.3. Turbulence Numerics

The **Turbulence Numerics** options are `First Order` and `High Resolution`. The `First Order` option uses `Upwind` advection and the `First Order Backward Euler` transient

scheme. The `High Resolution` option uses `High Resolution` advection and the `High Resolution` transient scheme.

For details, see [Advection Scheme Selection in the CFX-Solver Modeling Guide](#) and [Transient Scheme in the CFX-Solver Modeling Guide](#).

---

**Note:**

The settings on the [Equation Class Settings Tab \(p. 295\)](#) will override the **Turbulence Numerics** settings.

---

#### 22.1.1.4. Convergence Criteria

For details, see [Monitoring and Obtaining Convergence in the CFX-Solver Modeling Guide](#).

- **Residual Type:** select either `RMS` or `MAX`.
- **Residual Target:** specify a value for the convergence.

For details, see [Residual Type and Target Levels in the CFX-Solver Modeling Guide](#).

- **Conservation Target:** optionally specify the fractional imbalance value. The default value is `0.01`.

For details, see [Conservation Target in the CFX-Solver Modeling Guide](#).

#### 22.1.1.5. Elapsed Wall Clock Time Control

Select the `Maximum Run Time` option if you want to stop your run after a maximum elapsed time (wall clock time).

If you select this option the flow solver will automatically attempt to estimate the time to complete the next timestep or outer loop iteration. The estimated time is the average time that it takes to solve a previous iteration (includes the time to assemble and solve the linear equations, radiation and particle tracking) plus the average time it is taking to write any `Standard` backup or transient files. The time estimate currently does not include the time used by processes external to the flow solver. This includes mesh refinement, interpolation and FSI with Mechanical.

#### 22.1.1.6. Interrupt Control

Interrupt control conditions are used to specify criteria for stopping a solver run independently of any specified **Elapsed Wall Clock Time Control** settings.

You can set **Option** to:

- `Any Interrupt`

This option causes a solver run to terminate if the specified convergence conditions (specified under **Convergence Conditions**) are met or if any of the user-defined interrupts (specified under **User Interrupt Conditions**) are triggered.

- `All Interrupts`

This option causes a solver run to terminate only if the specified convergence conditions are met and all user-defined interrupts are triggered.

The **Convergence Conditions** settings indicate which of the convergence criteria constitute an interrupt condition:

- `Default Conditions`

With this option, the convergence criteria selected are those normally used for the type (steady-state, transient, harmonic balance) of run.

- `Selected Conditions`

With this option, you can manually select the convergence criteria from a list.

User-defined interrupt control conditions are specified using logical expressions. These expressions are evaluated by CFX-Solver and are reported in the CFX-Solver Output file. After executing each iteration (of a steady-state case) or time step (of a transient case), the solver evaluates all internal termination conditions and all user-defined interrupt control conditions. If any or all (depending on the interrupt control option) of these conditions are *true*, then solver execution stops and the outcome is written to the CFX-Solver Output file.

Typically, interrupt control conditions are defined by single-valued logical expressions. However, single-valued mathematical expressions can also be used. In this case, a single-valued mathematical expression is considered to be *true* if and only if the result of the expression is greater than or equal to 0.5. Otherwise it is deemed to have a value of *false*. For a discussion on logical expressions, see [CFX Expression Language Statements in the CFX Reference Guide](#).

You should consider setting interrupt control conditions if you expect your case to exhibit transient convergence behavior. For details and an example, see [Using Interrupt Control in Cases with Transient Convergence Behavior in the CFX-Solver Modeling Guide](#).

---

**Note:**

To prevent a steady-state run from being interrupted before reaching the intended minimum number of iterations, each interrupt condition should include a requirement that the minimum number of iterations has been completed. For example, an interrupt condition could begin with `"aitern>5 && "` in order to require that a minimum number of 5 iterations has been reached before the interrupt control condition can evaluate to *true*.

---

### 22.1.1.6.1. List Box

The list box is used to select interrupt control conditions for editing or deletion. Interrupt control conditions can be created or deleted with icons that appear beside the list box.

#### 22.1.1.6.1.1. [Interrupt Condition Name]

Enter a logical expression to specify an interrupt control condition.

### 22.1.1.7. Junction Box Routine

If you have created any Junction Box Routine objects, select those to include in this Solver run.

For details, see [User Junction Box Routines in the CFX-Solver Modeling Guide](#).

## 22.1.2. Basic Settings for Steady-State Simulations and for Transient Blade Row Cases that use the Harmonic Balance Transient Method

### 22.1.2.1. Convergence Control: Min. Iterations

The minimum number of iterations the CFX-Solver will run.

### 22.1.2.2. Convergence Control: Max. Iterations

The maximum number of iterations the CFX-Solver will run.

For details, see [Max. Iterations in the CFX-Solver Modeling Guide](#).

### 22.1.2.3. Convergence Control: Fluid Timescale Control

Sets the method of time scale control for a simulation. For details, see [Time Scale Control in the CFX-Solver Modeling Guide](#).

For steady-state simulations, and for transient blade row simulations that use the Harmonic Balance transient method, the following options are available:

- **Auto Timescale:** For details, see [Auto Timescale in the CFX-Solver Modeling Guide](#) and [Automatic Time Scale Calculation in the CFX-Solver Theory Guide](#).

- **Length Scale Option**

Three options are available: Conservative, Aggressive or Specified Length Scale.

- **Local Timescale Factor:** For details, see [Local Time Scale Factor in the CFX-Solver Modeling Guide](#).
- **Physical Timescale:** For details, see [Physical Time Scale in the CFX-Solver Modeling Guide](#).

### 22.1.2.4. Convergence Control: Solid Timescale Control

For steady-state simulations, and for transient blade row simulations that use the Harmonic Balance transient method, two options are available if the simulation involves a solid domain, and/or a solid within a porous domain:

- **Auto Timescale**
- **Physical Timescale**

When **Solid Timescale** is set to Auto Timescale, the option Solid Timescale Factor is available. For details, see [Solid Time Scale Control in the CFX-Solver Modeling Guide](#).

## 22.1.3. Basic Settings for Transient Simulations

### 22.1.3.1. Convergence Control

You will have already specified the number of timesteps under **Analysis Type**. For details, see [Analysis Type](#) (p. 161).

#### 22.1.3.1.1. Min. Coeff. Loops

This option determines the minimum number of iterations per timestep, and has a default value of 1.

---

**Important:**

If a compressible transient flow is undertaken with only one iteration per time step, then the solution can be incorrect if the **Heat Transfer** option is not set to **Total Energy**, or if heat transfer is not included in the simulation. This is due to the CFX-Solver not extrapolating the pressure at the start of the time step in these circumstances. This means that density is not extrapolated, and so the solver cannot calculate an accurate value for the time derivative of density on the first iteration. The workaround for this problem is to either run with at least two iterations per time step, or to use the **Total Energy** option for **Heat Transfer**.

---

#### 22.1.3.1.2. Max. Coeff. Loops

This option determines the maximum number of iterations per timestep, and has a default value of 10. For details, see [Max. Iter. Per Step in the CFX-Solver Modeling Guide](#).

#### 22.1.3.1.3. Fluid Timescale Control

**Coefficient Loops** is the only available option.

#### 22.1.3.1.4. Solid Timescale Control

To control the solid timescale for transient runs (including transient blade row runs), use the controls provided in the details view for the solid domain, on the **Solver Control** tab. For details, see [Timescale Control for Solid Domains \(Transient Cases; Transient Blade Row Cases with Harmonic Balance\)](#) (p. 204).

## 22.1.4. Immersed Solid Control

The immersed solid is represented as a source term in the fluid equations that drives the fluid velocity to match the solid velocity. The size of the source term is controlled by the **Momentum Source Scaling Factor** setting, which can be set globally (in the global **Solver Control** settings) or for individual immersed solids (on the immersed solid domain **Solver Control** tab). The default value of 10 is acceptable most of the time. If robustness problems are encountered, the scaling factor may be decreased (for example by a factor of 2), but at the expense of accuracy; the difference between the fluid velocity and the specified solid velocity will generally increase, even if only by a small amount.

**Boundary Model** has two options:

- None

This is the default option.

- Modified Forcing

This option enables modifications to the immersed solid source term to take better account of the boundaries of the immersed solid and their effect on the flow.

If you set **Boundary Model** to `Modified Forcing`, then the **Boundary Tracking** setting is available.

The **Boundary Tracking** setting has two options:

- Boundary Face Extrusion

Once **Boundary Model** is selected, this is the default option.

The `Boundary Face Extrusion` option requires you to specify two settings that must be set appropriately so that each near-wall node can be reliably projected onto the best location on the immersed boundary face. These settings are:

– **Extrusion Distance**

This is the distance by which each immersed boundary face is extruded outward from, and normal to, the immersed boundary to form an imaginary volume. A bounding box, that consists of faces of constant x, y, and z coordinate, is then constructed around each imaginary volume. If a near-wall node falls within the bounding box, then CFX-Solver will project that near-wall node normal to and onto the plane of the boundary face. If the projected location lies outside the face, other mechanisms take effect to find the nearest suitable projected location.

The value of **Extrusion Distance** should be of the order of magnitude of the length of a fluid mesh element in the vicinity of the immersed boundary.

– **Extrusion Tolerance**

This is a factor by which the solver will multiply the edge length of the bounding box for each imaginary volume. The bounding box is mentioned in the description for the **Extrusion Distance** setting. An overly large value will cause near-wall nodes to be mapped to locations on boundary faces that are not necessarily the closest face.

The default value for **Extrusion Tolerance** is 0.01.

- Search Through Elements

When using this option, CFX-Solver will search through elements near the immersed boundary and project the near-immersed-boundary fluid nodes onto the face edge or vertices of the immersed solid element. This option requires no user input.

This method is less accurate than the `Boundary Face Extrusion` option, but can be used for cases where no suitable **Extrusion Distance** can be determined.

For details, see [Immersed Boundary Tracking in the CFX-Solver Modeling Guide](#).

## 22.2. Equation Class Settings Tab

---

Equation class settings enable you to control some aspects of the CFX-Solver on an equation class basis. For example, you can set:

- A different time scale
- A different advection scheme
- A different transient scheme
- Convergence control parameters for each class of equation
- Criteria parameters for each class of equation

You might want to make such changes if you suspect that a convergence problem is caused by a particular equation class.

- Select the equation class for which to specify settings.
- The settings in this tab are a subset of those found on the **Basic Settings** tab (see [Basic Settings: Common](#) (p. 289)).

For details, see [Timestep Selection in the CFX-Solver Modeling Guide](#) and [Advection Scheme Selection in the CFX-Solver Modeling Guide](#).

The settings you specify on this form will override those on the **Basic Settings** tab. Any equation classes that are unspecified will use the parameters set on the **Basic Settings** tab. The number and type of equation classes depends on the specific physics of the problem.

For details, see [Controlling the Timescale for Each Equation in the CFX-Solver Modeling Guide](#).

---

### Note:

For a free surface simulation, you cannot set the advection scheme for the volume fraction (vf) equation class because the CFX-Solver uses a special compressive advection scheme.

---

## 22.3. Particle Control Tab

---

This section is visible when particle tracking has been selected. For details, see [Particle Solver Control in the CFX-Solver Modeling Guide](#).

## 22.4. Rigid Body Control Tab

---

The following options handle control of the rigid body solver.

The **Update Frequency** setting controls when the rigid body solver updates the position of the rigid body. The options are:

- `Every Iteration` (steady-state simulations only)



This option causes the rigid body solver to update the position of the rigid body at the beginning of each iteration.

- `Every Time Step` (transient simulations only)

This option causes the rigid body solver to update the position of the rigid body at the beginning of each time step.

This option is recommended only for weakly-coupled systems (for example, where fluid flow changes within a single time step have little effect on the rigid body motion) or when the time step is very small.

- `Every Coefficient Loop` (transient simulations only)

This option:

- Causes the rigid body solver to update the position of the rigid body at the beginning of each coefficient loop within each time step.
- Is recommended when there is tight coupling between the fluid flow and the motion of the rigid body.
- Is more computationally expensive than the `Every Time Step` option
- Can cause issues if the flow solver requires several coefficient loop iterations to stabilize the applied forces/torques between calls to the rigid body solver.

- `General Coupling Control`

This option causes the rigid body solver to update the position of the rigid body during each "stagger" (or coupling) iteration.

During every stagger iteration, the following sequence occurs:

1. The rigid body solver reads the latest calculated forces and torques.
2. The rigid body solver solves to determine the new rigid body state (position, orientation, velocity, and so on).
3. The CFX flow solver reads the new rigid body state.
4. The CFX flow solver solves all the field equations, including mesh motion where appropriate, by performing outer loop iterations (steady-state cases) or coefficient loop iterations (transient cases) until either the maximum number of iterations/coefficient loops is reached, or until all the field equations have converged. The mesh motion solution uses the latest rigid body state in order to calculate the mesh motion on all boundaries or subdomains set to use the rigid body solution. If the rigid body is an immersed solid, then the latest rigid body state is used to calculate the motion of the immersed solid domain.

Stagger iterations are repeated until the maximum number of stagger iterations is reached or until the time step (transients runs) or simulation (steady-state runs) is deemed to have converged, when the following criteria are met:

- The rigid body solver has converged (angular equations only),

- The force, torque and mesh motion data transferred between the CFX flow solver and the rigid body solver satisfy the appropriate convergence criteria, AND
- All the CFX field equations (mesh motion, mass and momentum, energy, and so on) have converged to the appropriate convergence criteria.

When the time step (transient runs) is converged, then this guarantees an implicit solution of all solution fields for each time step, providing the most stability when there is a strong coupling between the fluid flow and the motion of the rigid body.

The **Internal Coupling Step Control** settings set limits on the minimum and maximum number of stagger iterations.

When the `Every Iteration`, `Every Coefficient Loop`, or `General Coupling Control` option is selected, the **Internal Coupling Data Transfer Control** settings are available. These settings include:

- **Mesh Motion Data Transfer Control**
- **Force Data Transfer Control**
- **Torque Data Transfer Control**

For each of these **Internal Coupling Data Transfer Control** settings, you may:

- Optionally specify an **Under Relaxation Factor** for aiding convergence. The default value is 0.75 for all three transferred quantities. If a case has difficulty converging due to overshoots in the force, torque and/or mesh motion quantities passed between the two solvers, then the under-relaxation factor for one or more quantities can be reduced to damp down the overshoots. Values that are too small will tend to increase the number of iterations needed for convergence. On the other hand, for a case where the force, torque and mesh motion are steady and do not change much from time step to time step, it may be possible to raise the under-relaxation factors to 1 to speed up convergence.

---

#### Note:

The rigid body solver does not apply under-relaxation to **Mesh Motion Data Transfer Control** in the following situations:

- For all time steps when **Update Frequency** is set to `Every Time Step`
  - For the initial coefficient loop when **Update Frequency** is set to `Every Coefficient Loop`
  - For the initial stagger iteration when **Update Frequency** is set to `General Coupling Control`
- 

#### Note:

The rigid body solver also does not apply under-relaxation to **Force Data Transfer Control** and **Torque Data Transfer Control**, in the following situation:

- For all time steps when **Update Frequency** is set to `Every Time Step`
-

- Optionally specify a **Convergence Target**.

The convergence target is the target value of the convergence measures described below.

- For **Mesh Motion Data Transfer Control**, the convergence measure is the maximum of:

- The distance from the previous to the current center of mass, divided by the greater of the previous and current distances from the center of mass to the rigid body coordinate frame origin. Here, "previous" refers to the previous stagger iteration, coefficient loop, and so on — whenever the rigid body solver was last called.

- $\sqrt{(\delta q_1)^2 + (\delta q_2)^2 + (\delta q_3)^2 + (\delta q_0 - 1)^2}$ , where  $\delta q = q^n [q^{n-1}]^{-1}$ ,  $n$  refers to the current iteration (stagger iteration, coefficient loop, and so on — whenever the rigid body solver was last called),  $n-1$  refers to the previous iteration, and  $q = [q_0, q_1, q_2, q_3]$  is the orientation quaternion, with  $q_0$  being the scalar component and  $q_1, q_2, q_3$  being the vector components. Here, all quantities are with respect to the rigid body coordinate frame.

- For **Force Data Transfer Control**, the convergence measure is:

The magnitude of the change (from the previous value to the current value) in net force divided by the greater of the previous magnitude of the net force and the current magnitude of the net force, where "previous" refers to the previous stagger iteration, coefficient loop, and so on — whenever the rigid body solver was last called.

- For **Torque Data Transfer Control**, the convergence measure is:

The magnitude of the change (from the previous value to the current value) in net torque divided by the greater of the previous magnitude of the net torque and the current magnitude of the net torque, where "previous" refers to the previous stagger iteration, coefficient loop, and so on — whenever the rigid body solver was last called.

The convergence measures are plotted on the **Rigid Body Convergence** tab in CFX-Solver Manager when running a simulation that involves the rigid body solver. For details on the plots available for such runs, see [Monitor Plots related to Rigid Bodies in the CFX-Solver Modeling Guide](#).

The **Angular Momentum Equation Control** settings are:

- **Integration Method**

An iterative process is used to calculate the solution to the angular momentum equation.

Choose one of the following options to control the integration method for this process:

- First Order Backward Euler

This option provides first-order accuracy, and is vulnerable to gimbal lock problems. For details, see [First Order Backward Euler in the CFX-Solver Theory Guide](#).

- Simo Wong

This option provides second-order accuracy. For details, see [Simo Wong Algorithm in the CFX-Solver Theory Guide](#).

The **Iteration Convergence Criterion > Iteration Convergence** setting controls the degree to which an orientation must remain the same between successive iterations in order for the solution to be considered converged. The value is a normalized quantity; the default should be sufficient for most purposes.

**The Maximum Number of Iterations > Max. Iterations** setting (which controls the maximum number of iterations used by the Simo Wong integration method) can be increased if you see any warnings in the solver output that indicate a lack of convergence for the integration method.

## 22.5. Advanced Options Tab

---

The parameters on the **Advanced Options** tab should not need to be changed for most simulations.

### Optimization Level

For good solver performance in terms of convergence rate and robustness, it is sometimes necessary to modify the solver control **Optimization Level** setting. This setting selects different sets of pre-set parameter values that affect performance. Higher values generally result in higher convergence rate with lower convergence robustness. However, the effect on performance is case dependent.

Optimization level descriptions follow:

- 1

Standard behavior

- 2

Artificial wall removal is modified. This optimization level is recommended if, especially in a multiphase run, artificial walls are created as the solver iterates but those walls are not removed or take a long time to be removed. Note that if a solution cannot be found in the absence of these artificial walls, the walls will remain present, for example when a re-circulation region occurs at an outlet.

- 3

Second order interpolation is used whenever possible. Consequently, the final solution will be affected depending on the mesh resolution. In addition, the computation of advection coefficients is modified to reduce mesh dependence. This optimization level is recommended for use when high aspect ratio mesh cells are present, for example close to walls. This optimization level is also recommended for non-Newtonian fluid flows that do not show convergence at a reasonable rate. Note that these changes might compromise convergence stability so this level of optimization should be used with caution.

---

#### Note:

- Non-default optimization levels should be used only if there is a lack of solver performance after trying standard solver control options.

- Optimization Level controls are available in several places in the Solver Control advanced options, providing global control as well as control over linear solver numerics and multiphase-related numerics.
- 

## Dynamic Model Control

When **Global Dynamic Model Control** is selected, it enables some special modes to be implemented in the solver for the first few iterations or timesteps. For details, see [Advanced Options: Dynamic Model Control in the CFX-Solver Modeling Guide](#).

For details on **Turbulence Control**, see [Turbulence Control in the CFX-Solver Modeling Guide](#).

For details on **Combustion Control**, see [Advanced Combustion Controls in the CFX-Solver Modeling Guide](#).

For details on **Hydro Control**, see [Hydro Control in the CFX-Solver Modeling Guide](#).

## Pressure Level Information

Sets an X/Y/Z location for reference pressure and a pressure level.

For details, see [Pressure Level Information in the CFX-Solver Modeling Guide](#).

## Thermal Radiation Control

For details, see [Thermal Radiation Control in the CFX-Solver Modeling Guide](#).

## Body Forces

Under this option, `Volume-Weighted` should be generally used except for free surface cases.

For details, see [Body Forces in the CFX-Solver Modeling Guide](#).

## Interpolation Scheme

For details, see [Interpolation Scheme in the CFX-Solver Modeling Guide](#).

## Multicomponent Energy Diffusion

This option is available when a multicomponent flow is used with a heat transfer equation (that is, thermal or total energy). For details, see [Multicomponent Energy Diffusion in the CFX-Solver Modeling Guide](#). The possible options are:

- `Automatic`: uses unity Lewis number when no component diffusivities specified and no algebraic slip model; uses generic assembly when necessary
- `Generic Assembly`: sets default component diffusivities to unity Schmidt number  $Sc = 1$ ; generic treatment of energy diffusion term with support for user defined component diffusivities and algebraic slip model

- **Unity Lewis Number:** sets  $Le = 1$ ; single diffusion term, rather than separate term for contribution of every component, resulting in faster solver runs; the default molecular diffusion coefficient for components is derived from thermal conductivity

---

**Note:**

Forcing unity Lewis number mode when not physically valid may lead to inconsistent energy transport. Therefore this setting is not recommended.

---

## Temperature Damping

For details, see [Temperature Damping in the CFX-Solver Modeling Guide](#).

## Velocity Pressure Coupling

The **Rhie Chow Option** controls the details of the Rhie Chow pressure dissipation algorithm. **Fourth Order** ensures that the dissipation term vanishes rapidly under mesh refinement. However, it can sometimes induce wiggles in the pressure and velocity fields; for example, near shocks. The **Second Order** option damps out these wiggles more rapidly, but is also less accurate. The **High Resolution** option uses **Fourth Order** as much as possible, but blends to **Second Order** near pressure extrema. It is a good choice for high speed flow. The default is **Fourth Order** for most simulations, but **High Resolution** is automatically chosen if **High Speed Numerics** is activated under **Compressibility Control** on the **Solver Control** tab. The **High Resolution** option may occasionally be useful in other situations as well. For example, if you observe the simulation diverging and continuity residuals are significantly higher than the momentum residuals prior to divergence.

## Compressibility Control

The following options control parameters that affect solver convergence for compressible flows.

The **Total Pressure Option** controls the algorithm used for static-to-total conversions (and vice versa). There are three possible settings:

### Incompressible

The incompressible assumption is used in all situations.

### Automatic

The total pressure is calculated depending on the equation of state.

### Unset

This is equivalent to **Automatic** when the total energy model is used, and **Incompressible** otherwise.

For further details, see [Total Pressure in the CFX-Solver Theory Guide](#).

The **Automatic** option may experience robustness problems for slightly compressible fluids (such as compressible liquids). In such cases, you should consider using the **Incompressible** option instead.

When the **High Speed Numerics** option is selected, special numerics are activated to improve solver behavior for high-speed flow, such as flow with shocks. This setting causes the following behavioral changes:

- It activates a special type of dissipation at shocks to avoid a transverse shock instability called the carbuncle effect (which may occur if the mesh is finer in the transverse direction than in the flow direction).
- It activates the High Resolution Rhie Chow option to reduce pressure wiggles adjacent to shocks.
- It sets the nodal pressure gradients to zero at all pressure boundaries as well as openings.
- For steady-state flows, it modifies the default relaxation factors for the advection blend factor and gradients.

The **Clip Pressure for Properties** option enables the solver to accept negative absolute pressures in the converged solution. For simulations involving compressible flow, the absolute pressure should not be negative. However, the pressure field required to satisfy the governing equations on a finite mesh may not necessarily satisfy this condition. By default, the solver is robust to a pressure field that may want to temporarily lead to negative pressures, but not if negative pressures are present in the converged solution. The solver can be made robust to negative absolute pressures in the converged solution by activating this parameter, which clips the absolute pressure to a finite value when evaluating pressure-dependent properties such as density.

## Linear Solver Control

With **Option** set to `Automatic`, you can select an optimization level.

For good solver performance in terms of convergence rate and robustness, it is sometimes necessary to modify the linear solver control **Optimization Level** setting. This setting selects different sets of pre-set parameter values that affect performance. Higher values generally result in higher convergence rate with lower convergence robustness. However, the effect on performance is case dependent.

Optimization level descriptions follow:

- 1

Standard behavior

- 2

Convergence testing is stricter. If the linear solver diverges significantly, the solver is stopped without updating the principal variable fields of the failed equation system. Special controls are applied at start-up to avoid solution failure, for example when initial velocities are directed into walls.

- 3

The smoothing applied by the linear solver as it iterates is modified to improve solution behavior at short mesh length scales. This can be helpful for parallel processing runs and for multiphase runs with very strong volume fraction gradients. This optimization level is recommended only for parallel runs or multiphase runs where there are free surfaces or strong volume fraction gradients because,

in other cases, this optimization level might merely increase the CPU time used without providing any benefit.

---

**Note:**

- Non-default optimization levels should be used only if there is a lack of solver performance after trying standard solver control options.
  - Optimization Level controls are available in several places in the Solver Control advanced options, providing global control as well as control over linear solver numerics and multiphase-related numerics.
- 

## Multiphase Control

The following options handle control of solver details specific to multiphase flows.

When the **Volume Fraction Coupling** option is set to *Segregated*, the solver solves equations for velocity and pressure in a coupled manner, followed by solution of the phasic continuity equations for the volume fractions. With the *Coupled* option, the solver implicitly couples the equations for velocity, pressure, and volume fraction in the same matrix. The coupled volume fraction algorithm is particularly beneficial for buoyancy-dominated flows, such as buoyant free surface problems. The coupled volume fraction algorithm is also extremely beneficial when used in conjunction with models that include forces that are proportional to volume fraction gradients, such as the turbulence dispersion force in Eulerian multiphase flows and the solids pressure force in Eulerian multiphase models of granular flows (see [Interphase Momentum Transfer Models in the CFX-Solver Theory Guide](#)). For models of this nature, convergence of the segregated volume fraction algorithm tends to require much smaller physical time scales than are required by the coupled volume fraction algorithm.

---

**Note:**

If you encounter convergence difficulties when using a turbulence dispersion model in conjunction with *Segregated* volume fraction coupling, then you should consider reducing the physical time scale, or switching to *Coupled* volume fraction coupling.

---

---

**Note:**

If the residuals fail to drop reasonably well while solving a case that uses the *Coupled* option, it may be beneficial to reduce the mass flow under-relaxation parameter, `relax mass`, below its default value. For details on this parameter, see [Convergence Control Parameters in the CFX-Solver Modeling Guide](#).

---

The **Initial Volume Fraction Smoothing** option can be set to *None* or *Volume-Weighted*. If the initial conditions for volume fraction have a discontinuity, startup robustness problems may occur. Choosing *Volume-Weighted* smoothing of these volume fractions might improve startup robustness.

Under **Optimization Level**, you can select an optimization level.

For good solver performance in terms of convergence rate and robustness, it is sometimes necessary to modify the multiphase **Optimization Level** setting. This setting selects different sets of pre-set



parameter values that affect performance. Higher values generally result in higher convergence rate with lower convergence robustness. However, the effect on performance is case dependent.

Optimization level descriptions follow:

- 1

Standard behavior

- 2

This optimization level is strongly recommended for any cases with phase change or for which non-drag forces are present. Improved linearization of non-drag forces is applied and interphase mass transfer is modified to ensure physical behavior in the limit that one of the phases becomes very dilute.

- 3

Further improvements to the linearization of non-drag forces are applied together with improved damping of the forces in the limit that the continuous phase becomes dilute. In addition, if wall boiling is being modeled, the energy equation has under-relaxation applied to improve stability in the wall boundary layer.

- 4

This optimization level attempts to improve convergence rate, but might reduce convergence stability.

---

**Note:**

- Non-default optimization levels should be used only if there is a lack of solver performance after trying standard solver control options.
- Optimization Level controls are available in several places in the Solver Control advanced options, providing global control as well as control over linear solver numerics and multiphase-related numerics.

---

## Intersection Control

You can use the options described in this section to control the intersection of non-matching meshes. The parameters that you set here are applied to all interfaces where Intersection Control settings have not been applied individually in domain interface definitions. (See [Intersection Control \(p. 213\)](#) to learn how to apply Intersection Control settings to individual interfaces.)

CFX provides the GGI (General Grid Interface) capability, which determines the connectivity between the meshes on either side of the interface using an intersection algorithm. In general, two intersection methods are provided:

- Bitmap Intersection:

Two faces on either side of the interface which have to be intersected are both drawn into an equidistant 2D pixel map. The area fractions are determined by counting the number of pixels that reside inside both intersected faces (that is, within the union of the two faces). The area fraction for

a face is then calculated by dividing the number of overlapping pixels by the total number of pixels in the face. This method is very robust.

- Direct Intersection (Default):

Two faces on either side of the interface are intersected using the Sutherland-Hodgeman clipping algorithm. This method computes the exact area fractions using polygon intersection, and is much faster and more accurate than the bitmap method.

---

**Note:**

- If Direct (one-to-one) mesh connectivity is available, the solver will ignore the **Intersection Control** option and will instead use a 'topological intersection', that is, use the one-to-one information to generate the intersection data.
  - If you are restarting a run, the intersection step is skipped and the intersection data is read from the results file. This behavior can be overridden by setting the expert parameter `force_intersection` to True.
- 

The **Bitmap Resolution** controls the number of pixels used to fill the 2D pixel map (see description of the bitmap intersection method above). The higher this number, the more accurate the final calculation of the area fractions. In general, the default resolution of 100 should be sufficient but large differences in the mesh resolution on both sides of the interface as well as other mesh anomalies may require the bitmap resolution to be increased. Larger numbers will cause longer intersection times, for example, doubling the bitmap resolution will approximately quadruple the GGI intersection time.

When the **Permit No Intersection** option is set, the solver will run when there is no overlap between the two sides of an interface. This parameter is mainly useful for transient cases where interface geometry is closing and opening during the run. For example, transient rotor-stator cases with rotating valves, or moving mesh cases where the GGI interface changes from overlap to non-overlap during the simulation both can exhibit this type of behavior. This parameter is not switched on by default.

The **Discernible Fraction** option controls the minimum area fraction below which partially intersected faces are discarded. The following default values used by the solver depend on the intersection method:

- Bitmap Intersection:  $1/(\text{Bitmap Resolution})^{1.5}$
- Direct Intersection: 1.0E-06

The idea is that intersection inaccuracies should not lead to tiny area fractions that have no impact on the solution.

The **Edge Scale Factor** option is used to control the detection of degenerate faces. Degenerate faces are detected by comparing the face edge lengths with a characteristic length of the volume touching the face. Degenerated faces will not be intersected and therefore, intersected faces of zero size are discarded so that problems with the 2D projection of those faces are avoided.

The **Periodic Axial Radial Tolerance** option is used when determining if the surface represented by the interface is a constant axial or radial surface. For a rotational periodic GGI interface, the solver ensures that the ratio of the radial and axial extent compared to the overall extent of each interface side is bigger than the specified value and therefore, the interface vertices do not have the same radial or axial positions.

The **Circumferential Normalized Coordinates Option** is used to set the type of normalization applied to the axial or radial position coordinates ( $\eta$ ). Mesh coordinate positions on GGI interfaces using pitch change are transformed into a circumferential ( $\theta$ ) and axial or radial position ( $\eta$ ). The  $\eta$  coordinates span from hub to shroud and are normalized to values between 0 and 1. In cases where the hub and/or shroud curves do not match on side 1 and side 2, different approaches are available to calculate the normalized  $\eta$  coordinates based on side local or global minimum and maximum  $\eta$  values:

- **Mixed** (Default for Fluid Fluid interfaces): Normalization of  $\eta$  is based on local minimum and maximum  $\eta$  values as well as the  $\eta$  range of side 1. This method forces the hub curves on side 1 and 2 to align. Non-overlap regions adjacent to the shroud may be produced if the shroud curves are not the same.
- **Global** (Default for Fluid Solid Interfaces): Normalization of  $\eta$  is based on global minimum and maximum  $\eta$  values. This method intersects side 1 and 2 unchanged from their relative positions in physical coordinates. If the hub and shroud curves do not match then non-overlap regions will be produced.
- **Local**: Normalization of  $\eta$  is done locally for each side of the interface. This method will always produce an intersection of side 1 and 2, but may cause undesirable scaling of the geometry in some cases.

The **Face Search Tolerance Factor** is a scaling factor applied to the element sized based separation distance used to find candidates for intersection. For a given face on side 1 of the interface, candidate faces for intersection are identified on side 2 using an octree search algorithm. The octree search uses this tolerance to increase the sizes of the bounding boxes used to identify candidates. Making this parameter larger will increase the size of the bounding boxes, resulting in possible identification of more candidates.

The **Face Intersection Depth Factor** is a scaling factor applied to the element sized based separation distance used when performing the direct or bitmap intersection. The final intersection of faces is only applied to those faces that are closer to each other than a specified distance. This distance is calculated as the sum of the average depth of the elements on side 1 and side 2 of the interface. This factor is applied as a scaling on the default distance. It might be necessary to adjust this factor if the normal element depth on the two interfaces sides varies a lot, or side 1 and 2 of the interface are separated by thin regions (for example, thin fin type geometries).

## Injection Control

The **Injection Control** settings are available if any domain has an injection region defined with **Injection Location Type** set to `Selected 2D Regions` (see [Injection Location Type \(p. 254\)](#)).

The **Injection Control > Surface Injection Control** settings are described next.

The **Surface Overlap Type** options are:

- `Mapped Faces` (default)

If the mesh faces at an injection position are smaller than the hole size, the injection at that position influences multiple mesh faces.

Note that, with the `Mapped Faces` option, injection positions must be geometrically separated enough to avoid having multiple injection positions directly influence the same mesh elements or vertices.

**Intersection Control** settings are made available. These are identical to the **Intersection Control** settings of domain interfaces, described in [Intersection Control: Option \(p. 214\)](#).

- `Nearest Face`

At each injection position, the injection influences only the mesh element face that contains the hole center, irrespective of the comparative sizes of the hole and mesh element face.

---

**Note:**

If the hole size at a given injection position is less than the size of the local mesh element faces, the `Nearest Face` option is preferred.

---

In order for CFX-Solver to locate an injection position on a surface, the specified position (for example, a point from a `.csv` file) must lie within a certain distance of the surface. That distance is the product of:

- The specified value of **Extrusion Factor**, and
- The largest radius associated with any injection position of the injection region.

If you do not specify a value for **Extrusion Factor** then the default value of 5 applies. The CFX-Solver Output file notes any injection positions that could not be located on a surface (even those that are intentionally not located on any surface).

The **Intersection Control** settings are available if **Surface Overlap Type** is set to `Mapped Faces`. These are identical to the **Intersection Control** settings of domain interfaces, described in [Intersection Control: Option \(p. 214\)](#).



---

# Chapter 23: Output Control

---

The **Output Control** panel is used to manage the way files are written by the solver. If a transient simulation is running, you can control which variables will be written to transient results files, and how frequently files will be created. Results can be written at particular stages of the solution by writing backup files after a specified number of iterations. These backup files can be loaded into CFD-Post so that the development of the results can be examined before the solution is fully converged. Monitor data can also be written to track the solution progress. Particle tracking data can be written for postprocessing in CFD-Post. Surface data can be exported.


This chapter describes:

[23.1. User Interface](#)

[23.2. Working with Output Control](#)

## 23.1. User Interface

---

The **Output Control** dialog box is accessible by clicking *Output Control* , by selecting **Insert > Solver > Output Control**, or by editing the `Output Control` object listed in the tree view under `Simulation > Solver`. You can also edit the CCL directly to change the object definition; for details, see [Using the Command Editor \(p. 487\)](#).

The topics in this section include:

[23.1.1. Results Tab](#)

[23.1.2. Backup Tab](#)

[23.1.3. Transient Results Tab](#)

[23.1.4. Transient Statistics Tab](#)

[23.1.5. Monitor Tab](#)

[23.1.6. Particles Tab](#)

[23.1.7. Export Tab](#)

[23.1.8. Common Settings](#)

### 23.1.1. Results Tab

The **Results** tab for the `Output Control` object contains settings that control the content of the results file that is written at the end of a solver run. In the case of a transient run, the results file contains information from the last timestep.

#### 23.1.1.1. Option

See [Option \(p. 340\)](#).

### 23.1.1.2. File Compression

See [File Compression](#) (p. 341).

### 23.1.1.3. Output Variables List

See [Output Variables List](#) (p. 341).

### 23.1.1.4. Include Mesh Check Box

See [Include Mesh Check Box](#) (p. 341).

### 23.1.1.5. File Format Check Box

See [File Format Check Box](#) (p. 341).

### 23.1.1.6. Output Equation Residuals Check Box

See [Output Equation Residuals Check Box](#) (p. 342).

### 23.1.1.7. Output Boundary Flows Check Box

See [Output Boundary Flows Check Box](#) (p. 343).

### 23.1.1.8. Output Variable Operators Check Box

See [Output Variable Operators Check Box](#) (p. 343).

### 23.1.1.9. Extra Output Variables List

When the **Extra Output Variables List** is selected, you can specify any variable that is not included in the results file by default. For more details, see [Variables in ANSYS CFX in the CFX Reference Guide](#).

### 23.1.1.10. Output Particle Boundary Vertex Fields Check Box

See [Output Particle Boundary Vertex Fields Check Box](#) (p. 343).

## 23.1.2. Backup Tab

The **Backup** tab contains settings that specify the content of backup files, and the timesteps at which the files are written. The purpose of the backup file is to ensure that a solver run can be restarted. Backup files can be used to restart the simulation from the point where the error occurred, saving time and computational resources.

### 23.1.2.1. List Box

This list box is used to select `Backup Results` objects for editing or deletion. `Backup Results` objects can be created or deleted with the icons that appear beside the list box.

The union of all requested backup file content, across all `Backup Results` objects applicable for a given iteration, is written as a single backup file for that iteration. If no backup file content is specified for a given iteration in any `Backup Results` object, then no backup file is written for that iteration.

## 23.1.2.2. [Backup Results Name]

### 23.1.2.2.1. Option

See [Option](#) (p. 340).

### 23.1.2.2.2. File Compression

See [File Compression](#) (p. 341).

### 23.1.2.2.3. Output Variables List

See [Output Variables List](#) (p. 341).

### 23.1.2.2.4. Include Mesh Check Box

See [Include Mesh Check Box](#) (p. 341).

### 23.1.2.2.5. Output Equation Residuals Check Box

See [Output Equation Residuals Check Box](#) (p. 342).

### 23.1.2.2.6. Output Boundary Flows Check Box

See [Output Boundary Flows Check Box](#) (p. 343).

### 23.1.2.2.7. Output Variable Operators Check Box

See [Output Variable Operators Check Box](#) (p. 343).

### 23.1.2.2.8. Extra Output Variables List

When the **Extra Output Variables List** is selected, you can specify any variable that is not included in the results file by default. For more details, see [Variables in Ansys CFX in the CFX Reference Guide](#).

### 23.1.2.2.9. Output Particle Boundary Vertex Fields Check Box

See [Output Particle Boundary Vertex Fields Check Box](#) (p. 343).

### 23.1.2.2.10. Include Tracks of One-way Coupled Particles Check Box

Select or clear the check box to include or exclude tracks of one-way coupled particles to be written to backup files. This option is available only for cases that involve one-way coupled particles. For details, see [Particle Fluid Pair Coupling Options in the CFX-Solver Modeling Guide](#).



### 23.1.2.2.11. Output Frequency: Option

See [Output Frequency Options \(p. 344\)](#).

## 23.1.2.3. Backup Data Retention

### 23.1.2.3.1. Option

- Delete Old Files

Deletes the previous backup file when a new backup file is created. This only corresponds to backups created with a specified output frequency. It does not apply to backup files created manually during a run. It also does not apply to backup files requested by a System Coupling system.

- Keep All Files

Previous backup files are not deleted when a new backup file is created.

## 23.1.3. Transient Results Tab

The **Trn Results** tab for the `Output Control` object contains settings that specify the content of transient results files, and the timesteps at which the files are written. Each transient results file contains results for a particular timestep. The transient results files are written in addition to the full results file that will be written at the end of the transient simulation.

The settings on the **Transient Results** tab are analogous to those on the **Backup** tab; for details, see [Backup Tab \(p. 310\)](#).

### 23.1.3.1. List Box

This list box is used to select `Transient Results` objects for editing or deletion. `Transient Results` objects can be created or deleted with the icons that appear beside the list box.

Only one transient results file is written at a given time regardless of how many transient results file objects exist. Each `Transient Results` object will add information to the transient results file for that timestep. Thus, the resulting transient results file is a union of the data requested by all `Transient Results` objects for that timestep.

### 23.1.3.2. [Transient Results Name]

#### 23.1.3.2.1. Option

See [Option \(p. 340\)](#).

#### 23.1.3.2.2. File Compression

See [File Compression \(p. 341\)](#).

### 23.1.3.2.3. Output Variables List

See [Output Variables List](#) (p. 341).

### 23.1.3.2.4. Include Mesh Check Box

See [Include Mesh Check Box](#) (p. 341).

### 23.1.3.2.5. File Format Check Box

See [File Format Check Box](#) (p. 341).

### 23.1.3.2.6. Output Equation Residuals Check Box

See [Output Equation Residuals Check Box](#) (p. 342).

### 23.1.3.2.7. Output Boundary Flows Check Box

See [Output Boundary Flows Check Box](#) (p. 343).

### 23.1.3.2.8. Output Variable Operators Check Box

See [Output Variable Operators Check Box](#) (p. 343).

### 23.1.3.2.9. Extra Output Variables List

When the **Extra Output Variables List** is selected, you can specify any variable that is not included in the results file by default. For more details, see [Variables in Ansys CFX in the CFX Reference Guide](#).

### 23.1.3.2.10. Output Particle Boundary Vertex Fields Check Box

See [Output Particle Boundary Vertex Fields Check Box](#) (p. 343).

### 23.1.3.2.11. Output Frequency

See [Output Frequency Options](#) (p. 344).

## 23.1.3.3. Transient Blade Row Results

**Transient Blade Row Results** settings are available when there is at least one disturbance defined in the details view of the `Transient Blade Row Models` object.

You can control the output of variables related to the Transient Blade Row model via the following settings:

- **Option**

Choose the level of output: `Essential`, `Selected Variables`, or `None`.

- **File Compression**

Choose the level of file compression as a trade-off between file size and speed.

- **Output Variables List**

This setting is required for the `Selected Variables` option.

You can multi-select variables from a list (using the **Ctrl** key) to have them added to the output.

- **Extra Output Variables List**

This setting is available for the `Essential` option.

You can multi-select variables from a list (using the **Ctrl** key) to have them added to the output.

Data compression is based on a Fourier series. The **Data Compression** settings control the number of Fourier coefficients to store in the output and the portion of the simulation over which these coefficients are accumulated.

You can specify the number of Fourier coefficients to be stored in the output.

The **Accumulate at Period** setting controls when the Fourier Coefficients should start being accumulated. The available options are:

- `Moving Averages`

The solver continuously accumulates and updates Fourier Coefficients during the run.

---

**Note:**

Once you have specified the output for transient blade row results, there is no need to create intermediate transient results files.

---

For instructions on setting up and using Transient Blade Row models, see [Transient Blade Row Modeling in the CFX-Solver Modeling Guide](#).

## 23.1.4. Transient Statistics Tab

The **Trn Stats** tab for the `Output Control` object contains settings that specify the transient statistics data to be included in the results files; for details, see [Working with Transient Statistics \(p. 345\)](#).

### 23.1.4.1. List Box

This list box is used to select `Transient Statistics` objects for editing or deletion. `Transient Statistics` objects can be created or deleted with the icons that appear beside the list box.

### 23.1.4.2. [Transient Statistics Name]

#### 23.1.4.2.1. Option

The following statistics are evaluated at each node over the history of the simulation:

- `Arithmetic Average`
- `Minimum`

- Maximum
- Standard Deviation
- Root Mean Square
- Full

### 23.1.4.2.2. Output Variables List

See [Output Variables List](#) (p. 341).

If a selected output variable is a vector or a tensor, the selected operation **Option** is applied to each component of the variable separately. The result for each component is included in the results file. For example, if `Velocity` is selected as a variable, using the `Maximum` option, the results file will include the maximum value of each of the three velocity components. Note that the maximum value of each of the components will likely have occurred at different times during the simulation. Also note that the magnitude of the resulting velocity components will not be the same as the maximum of the velocity magnitude (the latter can be determined by using the `Maximum` option for an additional variable defined to be the velocity magnitude).

### 23.1.4.2.3. Start Iteration List Check Box

This check box determines whether or not a **Start Iteration List** is used. If this check box is cleared, statistics will start (or continue, for restart runs) accumulation during the first timestep of the current run.

#### 23.1.4.2.3.1. Start Iteration List Check Box: Start Iteration List

Enter a comma-separated list of iteration numbers corresponding to the variables selected in the **Output Variables List**. If the start iteration list contains fewer entries than the **Output Variables List**, then the final start iteration in the list is applied for all remaining output variables.

The start iteration for a given transient statistic specifies the timestep index at which statistic accumulation begins. Prior to that timestep, statistics are initialized, as outlined in [Working with Transient Statistics](#) (p. 345).

---

**Note:**

In the case of restarted transient runs, iteration numbers are interpreted as the total accumulated timestep index rather than the index for the current run.

---

### 23.1.4.2.4. Stop Iteration List Check Box

This check box determines whether or not a **Stop Iteration List** is used. If this check box is cleared, the statistics will continue accumulation until the end of the run.

#### 23.1.4.2.4.1. Stop Iteration List Check Box: Stop Iteration List

Enter a comma-separated list of iteration numbers corresponding to the variables selected in the **Output Variables List**. If the stop iteration list contains fewer entries than the **Output Variables List**, then the final stop iteration in the list is applied for all remaining output variables.

The stop iteration for a given transient statistic specifies the timestep index at which statistic accumulation ceases. After that timestep, statistics are simply not modified.

---

**Note:**

In the case of restarted transient runs, start and stop iterations are interpreted as the total accumulated timestep index rather than the index for the current run.

---

### 23.1.5. Monitor Tab

The **Monitor** tab for the `Output Control` object contains settings that specify monitor output. The following types of information can be monitored as a solution proceeds:

- Primitive or derived solution variables
- Fluid Properties
- Expressions.

When monitoring expressions, the expression must evaluate to a single number; for details, see [Working with Monitors \(p. 347\)](#).

#### 23.1.5.1. Monitor Objects Check Box

This check box determines whether or not monitor data is generated as a solution proceeds. If it is selected, the following settings are available:

23.1.5.1.1. Monitor Coeff. Loop Convergence

23.1.5.1.2. Monitor Balances: Option

23.1.5.1.3. Monitor Forces: Option

23.1.5.1.4. Monitor Residuals: Option

23.1.5.1.5. Monitor Totals: Option

23.1.5.1.6. Monitor Particles: Option

23.1.5.1.7. Efficiency Output Check Box

23.1.5.1.8. Monitor Points And Expressions

23.1.5.1.9. Aerodynamic Damping

23.1.5.1.10. Monitor GT-SUITE

23.1.5.1.11. Radiometer

23.1.5.1.12. Monitor Surfaces

##### 23.1.5.1.1. Monitor Coeff. Loop Convergence

(applies only for transient cases)

This check box determines whether or not monitor data is output within coefficient (inner) loops. Regardless of the setting, data will be output for each timestep.

### 23.1.5.1.2. Monitor Balances: Option

- Full

Mass, Momentum, and other balances are written to the solver monitor file.

- None

### 23.1.5.1.3. Monitor Forces: Option

- Full

Forces and moments on wall boundaries are written to the solver monitor file.

It is important to note that these forces and moments do not include reference pressure effects. You can include reference pressure effects in the force calculation by setting the expert parameter `include pref in forces = t`

It is also important to note that for rotating domains in a transient run, forces and moments on wall boundaries are evaluated in the reference frame fixed to the initial domain orientation. These quantities are not influenced by any rotation that might occur during a transient run or when a rotational offset is specified. However, results for rotating domains in a transient run may be in the rotated position (depending on the setting of **Options** in CFD-Post) when they are subsequently loaded into CFD-Post for postprocessing.

- None

### 23.1.5.1.4. Monitor Residuals: Option

- Full

RMS/max residuals are written to the solver monitor file.

- None

### 23.1.5.1.5. Monitor Totals: Option

- Full

Flow and source totals (integrals over boundaries) are written to the solver monitor file.

- None

### 23.1.5.1.6. Monitor Particles: Option

- Full

If Lagrangian Particle Tracking information is included in the simulation, force, momentum, energy, mass flow rates, and source data for particles are written to the solver monitor file.

- None

### 23.1.5.1.7. Efficiency Output Check Box

This check box determines whether or not the device efficiency can be monitored in CFX-Solver Manager. When selected it also activates field efficiency output to CFD-Post. If activated, the following information must be specified:

#### 23.1.5.1.7.1. Inflow Boundary

A single boundary condition region of type INLET.

#### 23.1.5.1.7.2. Outflow Boundary

A single boundary condition region of type OUTLET.

#### 23.1.5.1.7.3. Efficiency Type

Choose between **Compression**, **Expansion**, and **Both Compression and Expansion**.

For more information, see [Activating Efficiency Output](#)

#### 23.1.5.1.7.4. Efficiency Calculation Method

For each of the efficiency types, two efficiency calculation options are possible: **Total to Total** and **Total to Static**.

For more information, see [Isentropic Efficiency and Total Enthalpy](#)

### 23.1.5.1.8. Monitor Points And Expressions

The following settings are discussed:

[23.1.5.1.8.1. Monitor Points And Expressions: List Box](#)

[23.1.5.1.8.2. \[Monitor Name\]: Option](#)

[23.1.5.1.8.3. \[Monitor Name\]: Output Variables List](#)

[23.1.5.1.8.4. \[Monitor Name\]: Cartesian Coordinates](#)

[23.1.5.1.8.5. \[Monitor Name\]: Cylindrical Coordinates](#)

[23.1.5.1.8.6. \[Monitor Name\]: Expression Value](#)

[23.1.5.1.8.7. \[Monitor Name\]: Coordinate Frame](#)

[23.1.5.1.8.8. \[Monitor Name\]: Monitor Location Control](#)

[23.1.5.1.8.9. \[Monitor Name\]: Monitor Statistics](#)

#### 23.1.5.1.8.1. Monitor Points And Expressions: List Box

This list box is used to select monitor point and expression objects for editing or deletion. These objects can be created or deleted with the icons that appear beside the list box.

#### 23.1.5.1.8.2. [Monitor Name]: Option

- Cartesian Coordinates

Monitor point data includes variable values at the node closest to the specified point. A crosshair will be displayed in the viewer to indicate the monitored node.

- `Cylindrical Coordinates`

Specify the monitor point location in terms of **Position Axial Comp.**, **Position Radial Comp.**, and **Position Theta Comp.** values.

Monitor point data includes variable values at the node closest to the specified point. A crosshair will be displayed in the viewer to indicate the monitored node.

---

**Note:**

This option disables the ability to specify the points by picking in the Viewer.

---

- `Expression`

An expression is monitored.

### 23.1.5.1.8.3. [Monitor Name]: Output Variables List

(applies only when **Option** is set to `Cartesian Coordinates` or `Cylindrical Coordinates`)

Select the variables to monitor.

---

**Tip:**

Hold the **Ctrl** key when clicking to select multiple variables.

---

### 23.1.5.1.8.4. [Monitor Name]: Cartesian Coordinates

(applies only when **Option** is set to `Cartesian Coordinates`)

Enter coordinates for the point location to monitor.

---

**Tip:**

After you click a coordinate entry area, all of the coordinate entry areas turn yellow to show that you are in Picking mode. You can then select locations from the viewer using the mouse. To manipulate the object in the viewer while in Picking mode, use the viewer icons (rotate, pan, zoom) in the toolbar. You can end Picking mode by changing the keyboard focus (by clicking in another field, for example).

---

### 23.1.5.1.8.5. [Monitor Name]: Cylindrical Coordinates

(applies only when **Option** is set to `Cylindrical Coordinates`)



Enter coordinates for the point location to monitor in terms of **Position Axial Comp.**, **Position Radial Comp.**, and **Position Theta Comp.** values.

---

**Note:**

This option disables the ability to specify the points by picking in the Viewer.

Cylindrical coordinates are defined with respect to the global coordinate frame, unless a local coordinate frame is selected.

---

**23.1.5.1.8.6. [Monitor Name]: Expression Value**

(applies only when **Option** is set to `Expression`)

Enter a CEL expression that evaluates to a single number that is to be monitored.

**23.1.5.1.8.7. [Monitor Name]: Coordinate Frame**

Set the coordinate frame used for the monitor point or expression; for details, see [Coordinate Frames in the CFX-Solver Modeling Guide](#).

For a monitor point that uses Cartesian or cylindrical coordinates, the coordinate frame is used to interpret the specified coordinates. For a monitor point that uses an expression, the coordinate frame affects all CEL functions that are used in the expression and that return the component of a vector (for example, `force_x()`).

**23.1.5.1.8.8. [Monitor Name]: Monitor Location Control**

(applies only when **Option** is set to `Cartesian Coordinates` or `Cylindrical Coordinates`)

**Monitor Location Control** settings determine where the monitor is placed with respect to the mesh, and how often the monitor position is updated. The following **Monitor Location Control** options are available:

**Interpolation Type**

- `Nearest Node`

This option causes monitor point data to be taken from the mesh node nearest to the specified coordinates. The nearest mesh node is determined initially and again every time the monitor position is updated (based on the **Position Update Frequency** setting).

- `Trilinear`

This option causes monitor point data to be interpolated to the specified coordinates using a weighted average of the data from the vertices of the containing mesh element. The containing mesh element, and the values of the weighting factors, are determined

initially and again every time the position is updated (based on the **Position Update Frequency** setting).

---

**Note:**

If **Interpolation Type** is set to `Trilinear` and the specified coordinates are not contained by the current mesh, CFX-Solver reverts to `Nearest Node` behavior. As a result, you may receive unexpected monitor output. However, if the monitor position updates, normal `Trilinear` behavior resumes whenever the coordinates are contained by the mesh. This might occur if the simulation includes a moving mesh or a stationary monitor point in a rotating domain.

---

**Domain Name**

- If the check box is selected, you must set the domain name to which the specified coordinates will be restricted.
- If the check box is cleared, CFX-Solver searches over all domains for a location match. In this case, the matching domain for a stationary monitor point can potentially change each time the monitor point is updated. This might occur if, for example, the simulation includes a moving mesh or has multiple frames of reference.

**Frame Type**

- If the check box is selected, choose whether the specified coordinates are relative to the `Local` frame (which could be rotating) or the `Stationary` frame. If the simulation involves transient rotating domains, and the frame type is set to `Stationary`, then the monitor location will acknowledge any rotational periodic interfaces that you have set up, and periodic expansion will be assumed.
- If the check box is cleared, the specified coordinates are, by default, relative to the local frame.

---

**Note:**

If you position a stationary monitor point too close to a frame-change interface, such as a Transient Rotor-Stator or sliding mesh interface, then the monitor point may not consistently find the nearest vertices in the same domain as the mesh moves. This can lead to unpredictable output values from the monitor. You should restrict such monitors to one of the domains on either side of the interface using **Domain Name**.

---

**Position Update Frequency**

Determine how frequently the monitor position is updated, if at all. This option is relevant only in simulations that include a moving mesh or transient rotating domains (steady-state rotating domains are frozen and, therefore, do not actually rotate). Otherwise, the monitor point interpolation does not change and the position remains constant.

- `Initial Mesh Only`

The monitor point interpolation is based on the initial mesh.

- `Every Timestep` or `Every Iteration`

The monitor point interpolation is updated as the simulation proceeds.

### 23.1.5.1.8.9. [Monitor Name]: Monitor Statistics

(applies only when **Option** is set to `Expression`)

You can select one or more statistical quantities (for example, `Arithmetic Average` and `Standard Deviation`) to be evaluated for the expression value over a moving interval. This interval represents the recent history of the solution. When you run the simulation, CFX-Solver computes the statistical quantities in addition to the expression value.

---

**Note:**

- Monitor statistics are not supported for cases that use variable time steps.
  - You can refer to monitor statistics in CEL expressions. For details, see [Using Monitors in CEL](#) (p. 349).
- 

The following **Monitor Statistics** options are available:

#### Interval Option

Control how frequently the values of the selected statistics are updated. Each update, the values are calculated using the most recent iterations that fit into the interval.

- `Moving Interval`

Updates statistical values every iteration.

- `Previous Complete Interval`

Updates statistical values at the completion of the previous interval. The values remain fixed until each subsequent interval is complete.

---

**Note:**

Statistical quantities are only plotted for complete intervals, and will not be displayed before the first interval is complete.

---

#### Statistics List

Select statistics for CFX-Solver to evaluate. You can select multiple statistics using the **Ctrl** key.

- `Arithmetic Sum`

Sums all values of the monitor within the current interval.

- **Arithmetic Average**  
Calculates the arithmetic mean of the current interval.
- **Difference from the Mean**  
Subtracts the arithmetic average of the current interval from the monitor value at the end of the interval.
- **Maximum**  
Displays the maximum monitor value that occurs within the current interval.
- **Minimum**  
Displays the minimum monitor value that occurs within the current interval.
- **Root Mean Square**  
Calculates the square root of the arithmetic mean of the squares of the monitor values within the current interval.
- **Standard Deviation**  
Calculates the sample standard deviation of the current interval.
- **Time Integral**  
Approximates the time integral of the monitor over the interval, where time is in solution units. This option is available only for transient simulations.

### Interval Definition

Control the size of the evaluation interval. You can select different options for steady-state and transient cases.

The following options are available for steady-state cases:

- **Iteration**  
Specifies the interval as a number of solver iterations.

The following settings are available for transient cases:

- **Time Period**  
Specifies the interval as a number of operational periods. This setting is only available to transient blade row cases. The period length is the value set in the **Transient Blade Row Models** object, under **Transient Method > Time Period**.
- **Time**  
Specifies the interval as an amount of simulation time.
- **Timestep**

Specifies the interval as a number of time steps.

---

**Note:**

When a case with a monitor statistic is restarted with a different **Interval Definition** setting, the statistic may be discontinuous at the start of the simulation.

---

### 23.1.5.1.9. Aerodynamic Damping

Aerodynamic damping monitors are available for transient blade row cases that involve wall boundaries that have the `Periodic Displacement` mesh motion boundary condition.

Aerodynamic damping per vibration cycle is a measure of the net energy transferred mechanically from the blade to the fluid over one vibration cycle of the blade. The damping value is evaluated by computing the total work that the blade surface imparts on the fluid over the time period corresponding to a given mode of vibration. For details on the calculation, see [Equation 6.2 in the CFX-Solver Modeling Guide](#).

A positive value of aerodynamic damping indicates that the vibration is damped (for the frequency being studied). Conversely, a negative value of aerodynamic damping indicates that the vibration is undamped.

The following settings are discussed:

[23.1.5.1.9.1. Aerodynamic Damping: List Box](#)

[23.1.5.1.9.2. \[Aerodynamic Damping Name\]: Option](#)

[23.1.5.1.9.3. \[Aerodynamic Damping Name\]: Normalization](#)

[23.1.5.1.9.4. \[Aerodynamic Damping Name\]: Location Type](#)

[23.1.5.1.9.5. \[Aerodynamic Damping Name\]: Integration Multiplier](#)

[23.1.5.1.9.6. \[Aerodynamic Damping Name\]: Integration Interval](#)

#### 23.1.5.1.9.1. Aerodynamic Damping: List Box

This list box is used to select aerodynamic damping monitor objects for editing or deletion. These objects can be created or deleted with the icons that appear beside the list box.

#### 23.1.5.1.9.2. [Aerodynamic Damping Name]: Option

- Full Period Integration

With this option set, the damping value calculation involves an integration over the last *complete* integration interval. In this context, the first integration interval starts at the beginning of the run (or, in the case of restarts, the initial run). The size of an integration interval is displayed as a read-only value, as described below. The damping value is calculated only upon the completion of each integration interval.

- Moving Integration Interval

With this option set, the damping value calculation involves an integration over a time interval that has a duration of one integration interval and that ends at the current time step. The damping value is calculated for each time step.

#### 23.1.5.1.9.3. [Aerodynamic Damping Name]: Normalization

Enter a value and units for normalization of the aerodynamic damping value. The aerodynamic damping value, which has units of energy and represents the work done per period, is normalized by the specified normalization value, which also has units of energy, resulting in a dimensionless normalized value of aerodynamic damping.

#### 23.1.5.1.9.4. [Aerodynamic Damping Name]: Location Type

Specify a mesh region or a boundary that represents the surface over which integration is carried out as part of the aerodynamic damping calculation. Only boundaries that have a `Periodic Displacement` mesh motion option (as specified on the **Boundary Details** tab), or mesh regions that are used in such boundaries, may be selected.

#### 23.1.5.1.9.5. [Aerodynamic Damping Name]: Integration Multiplier

Enter an integer that multiplies with the period of mesh motion to yield the **Integration Interval**.

#### 23.1.5.1.9.6. [Aerodynamic Damping Name]: Integration Interval

This is a read-only value indicating the size of the time interval over which integration is carried out as part of the aerodynamic damping calculation. It is equal to the period of mesh motion multiplied by the specified **Integration Multiplier**. The period of mesh motion is the inverse of the **Frequency** value specified on the **Boundary Details** tab (see [Periodic Displacement in the CFX-Solver Modeling Guide](#)) for the boundary over which integration is carried out as part of the aerodynamic damping calculation.

### 23.1.5.1.10. Monitor GT-SUITE

Using the Monitor GT-SUITE settings, you can add or delete solution monitors for GT-SUITE interfaces and GT-SUITE Turboshaft functions. These monitors display the variable values supplied to CFX by GT-SUITE.

---

#### Note:

You can alternatively or also add solution monitors for GT-SUITE interfaces and GT-SUITE Turboshaft functions as part of GT-SUITE initialization. For details, see [Monitoring Tab \(p. 113\)](#).

---

Use the icons beside the list to add or delete monitors. There are **Location** settings to indicate which boundaries to monitor. There are **Variable Selection** settings to indicate which variables to monitor on those boundaries.

### 23.1.5.1.11. Radiometer

(applies only when using the Discrete Transfer or Monte Carlo thermal radiation model)

A radiometer is a user defined point in space that monitors the irradiation heat flux (not incident radiation) arriving at the required location. The user specification involves much more than just the location of the sensor, as it also requires the viewing direction, its temperature and some numerical controls for each particular sensor.

By default, radiometers are ideal and the efficiency factor is 1.

A cyan arrow with a cross-hair is used to denote the location of each sensor in the viewer.

For details, see [Radiometers in the CFX-Solver Modeling Guide](#).

The following settings are discussed:

23.1.5.1.11.1. Radiometer: List Box

23.1.5.1.11.2. [Radiometer Name]: Option

23.1.5.1.11.3. [Radiometer Name]: Cartesian Coordinates

23.1.5.1.11.4. [Radiometer Name]: Temperature

23.1.5.1.11.5. [Radiometer Name]: Quadrature Points

23.1.5.1.11.6. [Radiometer Name]: Coordinate Frame Check Box

23.1.5.1.11.7. [Radiometer Name]: Coordinate Frame Check Box: Coordinate Frame

23.1.5.1.11.8. [Radiometer Name]: Diagnostic Output Level Check Box

23.1.5.1.11.9. [Radiometer Name]: Diagnostic Output Level Check Box: Diagnostic Output Level

23.1.5.1.11.10. [Radiometer Name]: Direction: Option

23.1.5.1.11.11. [Radiometer Name]: Direction: X Component, Y Component, Z Component

#### **23.1.5.1.11.1. Radiometer: List Box**

This list box is used to select radiometer objects for editing or deletion. These objects can be created or deleted with the icons that appear beside the list box.

#### **23.1.5.1.11.2. [Radiometer Name]: Option**

- **Cartesian Coordinates**

#### **23.1.5.1.11.3. [Radiometer Name]: Cartesian Coordinates**

Enter Cartesian coordinates that describe the location of the radiometer. These coordinates are interpreted in the coordinate frame associated with the radiometer. For details, see [Coordinate Frames \(p. 371\)](#) and also [Coordinate Frames in the CFX-Solver Modeling Guide](#).

#### **23.1.5.1.11.4. [Radiometer Name]: Temperature**

Enter the temperature of the radiometer.

#### **23.1.5.1.11.5. [Radiometer Name]: Quadrature Points**

Enter the number of rays used for ray tracing from the radiometer.

### 23.1.5.1.11.6. [Radiometer Name]: Coordinate Frame Check Box

This check box determines whether the coordinate frame used to interpret the location and direction specifications of the radiometer will be specified or left at the default of `Coord 0`.

### 23.1.5.1.11.7. [Radiometer Name]: Coordinate Frame Check Box: Coordinate Frame

Select a coordinate frame to interpret location and direction specifications of the radiometer. For details, see [Coordinate Frames \(p. 371\)](#) and also [Coordinate Frames in the CFX-Solver Modeling Guide](#).

### 23.1.5.1.11.8. [Radiometer Name]: Diagnostic Output Level Check Box

This check box determines whether the diagnostic output level will be specified or left at the default of 0.

### 23.1.5.1.11.9. [Radiometer Name]: Diagnostic Output Level Check Box: Diagnostic Output Level

Enter a number greater than zero. The CFX-Solver will write the ray traces to a series of polylines in a `.csv` file that can be visualized in CFD-Post. This can be used to determine if the number of quadrature points is optimal. For details, see [Radiometers in the CFX-Solver Modeling Guide](#).

### 23.1.5.1.11.10. [Radiometer Name]: Direction: Option

- **Cartesian Components**

### 23.1.5.1.11.11. [Radiometer Name]: Direction: X Component, Y Component, Z Component

(applies only when **Monitor Objects check box: Radiometer: [Radiometer Name]: Direction: Option** is set to `Cartesian Components`)

Enter a numerical quantity or CEL expression for each Cartesian component of a direction vector that represents the orientation of the radiometer.

### 23.1.5.1.12. Monitor Surfaces

Monitor Surfaces are defined on 2D User Locations (see [User Locations \(p. 377\)](#)) and can be used for monitoring and post-processing selected variables at the User Surface location in Ansys CFD-Post.

Monitor surfaces can be used to observe the solution at a Monitor Surface location while the solver is in progress (see [Monitor Run in Progress in the CFD-Post User's Guide](#)).

---

#### Note:

- Monitor surfaces (and by extension live monitoring of solutions in CFD-Post) are not supported for transient blade row cases or Ansys Workbench.
- Monitor Surfaces are always defined in the stationary frame of reference and cannot move relative to any frame of reference, or with any rotating domain.



- For steady-state cases, the selected solution variables associated with Monitor Surfaces are calculated by the solver and are stored in Solution Monitoring (.smn) files that can be found in the run directory. The purpose of these files is to enable you to monitor, in CFD-Post, the development of the solution using the Monitor Run in Progress functionality.

Solution Monitoring files can also be reviewed in CFD-Post after the run has ended. However, their use is limited because they represent only unconverged solution data.

---

The following settings are discussed:

### 23.1.5.1.12.1. Monitor Surfaces: List Box

### 23.1.5.1.12.2. [Monitor Surface Name]: Option

### 23.1.5.1.12.3. [Monitor Surface Name]: Coordinate Frame

### 23.1.5.1.12.4. [Monitor Surface Name]: Output Variables List

### 23.1.5.1.12.5. [Monitor Surface Name]: Output Location List

### 23.1.5.1.12.6. [Monitor Surface Name]: File Compression Level

### 23.1.5.1.12.7. [Monitor Surface Name]: Output Frequency

### 23.1.5.1.12.8. Monitor Location Data Retention

#### **23.1.5.1.12.1. Monitor Surfaces: List Box**

This list box is used to select Monitor Surface objects for editing or deletion. Monitor Surface objects can be created or deleted with the icons that appear beside the list box.

#### **23.1.5.1.12.2. [Monitor Surface Name]: Option**

Only User Locations can be used to define a Monitor Surface.

#### **23.1.5.1.12.3. [Monitor Surface Name]: Coordinate Frame**

Select a coordinate frame to define the Monitor Surface in. To create a new coordinate frame, see [Creating a New Coordinate Frame \(p. 371\)](#). By specifying a coordinate frame that rotates, the monitor surface will effectively remain stationary relative to that frame, and rotate relative to the global frame. This enables simpler live monitoring and post-processing for cases with rotating domains.

#### **23.1.5.1.12.4. [Monitor Surface Name]: Output Variables List**

See [Output Variables List \(p. 341\)](#).

Select the output variables to write to the results file at the Monitor Surface location. Conservative values for these variables will be interpolated from the volume data onto the Monitor Surface.

#### **23.1.5.1.12.5. [Monitor Surface Name]: Output Location List**

Choose the User Location to write solution data from.

### 23.1.5.1.12.6. [Monitor Surface Name]: File Compression Level

See [File Compression](#) (p. 341).

### 23.1.5.1.12.7. [Monitor Surface Name]: Output Frequency

See [Output Frequency Options](#) (p. 344).

### 23.1.5.1.12.8. Monitor Location Data Retention

This behaves in the same way as the [Backup Data Retention](#) (p. 312) option on the **Backup** tab but only applies to monitor location data files.

## 23.1.6. Particles Tab

The **Particles** tab for the `Output Control` object contains settings that specify whether the particle data is recorded, and details of how the data is collected and recorded.

This tab is available only when the morphology option is set to `Particle Transport Fluid` or `Particle Transport Solid` in CFX-Pre; for details, see [Basic Settings Tab](#) (p. 168).

The particle data is initially written to particle track files, which contain a specified level of detail about particles involved in your simulation. The files are written to the directory with the same name as your current run. An option on the **Particles** tab controls whether or not the track files are retained after their data is copied into the final results file (and any backup results files).

### 23.1.6.1. Particle Track File Check Box

This check box determines whether or not to customize the type and amount of particle track data recorded in the results file.

#### 23.1.6.1.1. Option

- **All Track Positions (default)**

Point data is collected for all track positions, as determined by the Track Positions setting.

- **Specified Position Interval**

Point data is collected for a subset of all track positions. The entire set of track positions is determined by the Track Positions setting. The subset is controlled by the Interval setting. For example, if Track Position Interval is left at its default value of 1, then the result is the same as setting Option to All Track Positions. Setting Interval to 2 will cause point data to be collected for every second track position; setting Interval to 3 will cause point data to be collected for every third track position, and so on.

- **Specified Distance Spacing**

Point data is collected for evenly-spaced points along each track. The spacing is controlled by this parameter, and represents a physical distance.

- **Specified Time Spacing**

Point data is collected for points along each track with the points spaced by time according to this parameter. The physical distance between data collection points is therefore a function of the particle velocity along each track.

- **None**

This option can be used to avoid writing any track information. This might be useful if you are not interested in particle tracks or want to avoid the additional disk space required to store the tracks. If this option is set, no tracks will be available in CFD-Post. In contrast to the track file information, sources are required for a clean re-start of a particle case and must be written to the results file.

---

**Note:**

For a transient run, final particle positions are always added to the track information, and therefore can be seen at the end of a run.

---

### 23.1.6.1.2. Track Positions Check Box

(applies only when **Particle Track File Check Box: Option** is set to `All Track Positions` or `Specified Position Interval`)

This check box determines whether the **Track Positions** setting will be specified or left at the default value: `Element Faces`.

### 23.1.6.1.3. Track Positions Check Box: Track Positions

- **Control Volume Faces**

Points are written each time a sub-control volume boundary is crossed. This produces the more precise and larger track files than the other option.

- **Element Faces**

Points are written to the track file each time a particle crosses the boundary of an element.

### 23.1.6.1.4. Interval

(applies only when **Particle Track File Check Box: Option** is set to `Specified Position Interval`)

Enter an integer that specifies the spacing (in terms of points) between points along the tracks.

### 23.1.6.1.5. Track Distance Spacing

(applies only when **Particle Track File Check Box: Option** is set to `Specified Distance Spacing`)

Enter a numerical quantity that specifies the physical distance interval between successive points on the track. Data will be collected only for those points.

### 23.1.6.1.6. Track Time Spacing

(applies only when **Particle Track File Check Box: Option** is set to `Specified Time Spacing`)

Enter a numerical quantity that specifies the physical time interval between successive points on the track. Data will be collected only for those points.

### 23.1.6.1.7. Track Printing Interval Check Box

This check box determines whether the Track Printing Interval setting will be specified or left at the default value: 1.

### 23.1.6.1.8. Track Printing Interval Check Box: Interval

Output data is collected for every  $n$ th particle track, where  $n$  is the specified number.

### 23.1.6.1.9. Keep Track File Check Box

This check box determines whether or not the track files are kept. When the track files are kept, they can be found below the working directory in a directory that has the same name as the run. For example, for the first solution of `dryer.def`, the track files are kept in a directory called `dryer_001`.

The data will be copied into the results file regardless of whether or not the track files are kept. CFD-Post can extract the track file data from the results file for postprocessing.

### 23.1.6.1.10. Track File Format Check Box

This check box determines whether the track file format will be specified, or left at the solver default value: `Unformatted`. The track file will remain in the working directory after finishing a run only if you select the **Keep Track File** option to force the solver to *not* delete it.

### 23.1.6.1.11. Track File Format Check Box: Track File Format

- `Formatted`

Formatted track files are in human-readable ASCII format but take up much more disk space than unformatted track files.

The general structure of formatted ASCII track files will print the Number of Particle Positions in a Block at the top of the file preceding repetitions of the following:

```
Particle Track Number
X Position
Y Position
Z Position
Traveling Time
Traveling Distance
Particle Diameter
Particle Number Rate
Particle Mass Fraction Component 1
Particle Mass Fraction Component 2
....
Particle Mass Fraction Component n
Particle U Velocity
Particle V Velocity
Particle W Velocity
```

Particle Temperature  
Particle Mass

---

**Note:**

Particle Mass Fraction Component 1- n only appear for multi-component particle materials and Particle Temperature only appears when heat transfer is activated.

---

- Unformatted

Unformatted track files are written in a non-readable, binary, format.

### 23.1.6.2. Particle Histogram

This option allows you to define particle histogram data of track variables on user-specified boundary patches and/or particle injection regions.

In steady-state, particle histograms are written/overwritten each time particles are tracked. In transient simulations, particle histograms are over written within a time step loop (i.e. coefficient loops). The histograms of the last coefficient loop of a time step is stored in the run directory.

#### 23.1.6.2.1. Histogram Type

Each of the histogram types are obtained by splitting the range of data into bins of equal size. The main difference between each type is the method in which the data is counted.

- **Raw**

For each bin, you count the number of points from the data set that fall into each bin.

- **Normalized**

For each bin in the Normalized histogram, you normalize the count in terms of the total number of observations.

- **Accumulated**

In the Accumulated histogram, the vertical axis count is a cumulative count of the current bin plus all bins for smaller values of the response variable.

- **Accumulated and Normalized**

For the Accumulated and Normalized histogram option, the vertical axis count is a cumulative normalized count of the current bin plus all bins for smaller values of the response variable.

#### 23.1.6.2.2. Number of Bins

Number of discrete intervals or classes that accumulate data is divided into based on the frequency of the data.

### 23.1.6.2.3. Particle Definition

The Particle Definition allows you to select the particle type for which the histogram output is created. **x** and **y** variable lists only show particle track variables filtered according to the particle type selected in **Particle Definition**.

### 23.1.6.2.4. x Variable

The response variable. Any valid particle track variable is supported. Response variables outside of the user specified min/max limits are stored in bins '1' or 'n'.

### 23.1.6.2.5. y Variable

The count variable. For the **Raw** histogram type, this variable can hold any particle track variable. For all other histogram types, this variable is restricted to the particle number rate option only.

---

**Note:**

Particle number rates are evaluated without distinguishing incoming particles from outgoing particles.

---

### 23.1.6.2.6. x Var. Lower Bound

Lower bound for histogram response variable (X axis). Histogram lower bound variables use context-sensitive dimensions that depend on the dimension of y-variable specified.

### 23.1.6.2.7. x Var. Upper Bound

Upper bound for histogram response variable (X axis). Histogram upper bound variables use context-sensitive dimensions that depend on the dimension of y-variable specified.

### 23.1.6.2.8. Sample Location

Specify a boundary patch or particle injection region against which the histogram will be plotted.

---

**Note:**

You can only select individual boundary patches or particle injection regions in CFX-Pre.

---

### 23.1.6.2.9. Export Format

All particle histogram data is written to file in a **CFX CSV Histogram** format. You can specify the output file name in CCL using the **Filename Prefix** option. If no user-defined filename can be found, the following file naming convention applies:

- Steady-state run:

<Particle Type Alias Name>\_<Histogram Alias Name>.csv

For example: MyParticle\_Histo1.csv

- Transient run:

<Particle Type Alias Name>\_<Histogram Alias Name>\_<Timestep Number>.csv

For example: MyParticle\_Histo2\_3.csv



### 23.1.6.2.10. Export Frequency

Only supported export frequency option is: **Every Timestep**.

## 23.1.6.3. Transient Particle Diagnostics

This section is available for transient simulations using particle tracking and enables you to output various particle data; for details, see [Transient Particle Diagnostics in the CFX-Solver Modeling Guide](#).

### 23.1.6.4. Transient Particle Diagnostics: List Box


Shows the current Transient Particle Diagnostics outputs. You can click  to create a new diagnostics output file or click  to delete an existing one.

### 23.1.6.5. Transient Particle Diagnostics: [Transient Particle Diagnostics Name]

#### 23.1.6.5.1. Option

- Particle Penetration
- Total Particle Mass
- User Defined - This option can be used to specify a user-defined **Diagnostic Routine** to evaluate the diagnostics information based on particle variables specified in **Particle Variables List**. Optionally, you can also select the **Monitored Values List** check box and specify a comma-separated list of names for monitored values. For details, see [User Diagnostics Routine in the CFX-Solver Modeling Guide](#).

#### 23.1.6.5.2. Particles List

Select particles to be used for output from the drop-down list, or click  and select from the Particles List dialog box.

#### 23.1.6.5.3. Spray Mass Frac.

The fraction of the total spray mass contained within an imaginary cone, the half-angle of which is the spray angle. The cone tip is at the point of injection and the cone axis is parallel to the direction of injection.

#### 23.1.6.5.4. Penetration Origin and Direction: Option

- Specified Origin and Direction

### 23.1.6.5.5. Penetration Origin and Direction: Injection Center

Enter the Cartesian coordinates of the center of injection.

### 23.1.6.5.6. Penetration Origin and Direction: Injection Direction

#### 23.1.6.5.6.1. Option

- Cartesian Components

Specify the Cartesian components (Direction X Comp., Direction Y Comp., and Direction Z Comp.) of the injection direction.

### 23.1.6.5.7. Axial Penetration: Option

- Axial Penetration

See [Spray Penetration in the CFX-Solver Modeling Guide](#) in [Transient Particle Diagnostics in the CFX-Solver Modeling Guide](#) for details.

- None

### 23.1.6.5.8. Radial Penetration: Option

- Radial Penetration

See [Spray Penetration in the CFX-Solver Modeling Guide](#) in [Transient Particle Diagnostics in the CFX-Solver Modeling Guide](#) for details.

- None

### 23.1.6.5.9. Normal Penetration: Option

- Normal Penetration

See [Spray Penetration in the CFX-Solver Modeling Guide](#) in [Transient Particle Diagnostics in the CFX-Solver Modeling Guide](#) for details.

- None

### 23.1.6.5.10. Spray Angle: Option

- Spray Angle

See [Spray Penetration in the CFX-Solver Modeling Guide](#) in [Transient Particle Diagnostics in the CFX-Solver Modeling Guide](#) for details.

- None

#### 23.1.6.5.10.1. Spray Angle: Spray Radius at Penetration Origin Check Box

Enable to specify a spray radius for the penetration origin.



### 23.1.6.5.10.2. Spray Angle: Spray Radius at Penetration Origin Check Box: Spray Radius

Enter a penetration origin spray radius.

## 23.1.7. Export Tab

The **Export** tab for the `Output Control` object is used to specify export files to be written by the CFX-Solver; for details, see [Working with Export Results \(p. 350\)](#).

---

### Note:

For cases involving mesh motion, the export of wall boundary data is limited to stationary walls.

---

### 23.1.7.1. List Box

This list box is used to select `Export Results` objects for editing or deletion. `Export Results` objects can be created or deleted with the icons that appear beside the list box.

### 23.1.7.2. [Export Name]: Option

This option allows you to export a specified list of **Particle Track Data** on specified boundary patches or particle injection regions. The particle track data is written to a file and can be imported and manipulated after the run. This option requires you to define a list of particle track variables to be exported, as well as a location where the data is sampled from.

### 23.1.7.3. [Export Name]: Export Format

The **Export Format** check box determines whether the export format will be specified or left at its default of `CGNS`.

The only available export format option for **Particle Track Data** is `CFX CSV`.

#### 23.1.7.3.1. Filename Prefix Check Box

This check box determines whether or not a user-specified prefix is used in the filenames of exported files. By default, CFX-Solver will use the object name as the filename prefix. For details, see [File Naming Conventions \(p. 350\)](#).

#### 23.1.7.3.2. Filename Prefix Check Box: Filename Prefix

Specify a prefix to use in the filenames of exported files.

#### 23.1.7.3.3. Include Topology Check Box

(applies only when the export format option is set to `CFX CSV BC Profile`)

This check box determines whether or not the CFX-Solver will include the mesh connectivity data in the exported profile file. With the mesh connectivity data included, the exported profile file can be imported into CFD-Post for visual examination. For example, you might want to visualize

the pressure harmonics from a blade flutter case or a blade row interaction case before using the profile in Ansys Mechanical.

#### 23.1.7.3.4. Include File Info Header Check Box

(applies only when the export format option is set to CFX CSV BC Profile)

This check box determines whether or not the CFX-Solver will include the file information header in the exported profile file. The file information header provides the version number of CFX-Solver, the date, the time, and the name of the case.

#### 23.1.7.3.5. Precision Check Box

(applies only when the export format option is set to CFX CSV BC Profile)

This check box determines whether or not a user-specified precision is used for real data.

#### 23.1.7.3.6. Precision Check Box: Precision

(applies only when the export format option is set to CFX CSV BC Profile)

Specify the number of significant digits in the fractional part of real number data in the exported CFX CSV BC Profile file. Valid values range from 0 to 20. By default, the CFX-Solver writes 8 significant digits in the fractional part. A value of 0 causes the CFX-Solver to write 8 significant digits. If an insufficient precision is specified for a case involving a very fine mesh, the coordinates of two close mesh nodes can become identical due to truncation of their digits.

### 23.1.7.4. [Export Name]: Export Frequency

#### 23.1.7.4.1. Option

- **Time List**

See [Time List](#) (p. 344).

- **Time Interval**

See [Time Interval](#) (p. 344).

- **Iteration List**

See [Iteration List](#) (p. 344).

- **Iteration Interval**

See [Iteration Interval](#) (p. 344).

- **Every Iteration**

- **End of Run**

This is only available option for **Particle Track Data**.

### 23.1.7.5. [Export Name]: Export Surface

#### 23.1.7.5.1. List Box

This list box is used to select `Export Surface` objects for editing or deletion. `Export Surface` objects can be created or deleted with the icons that appear beside the list box. Solution fields are required either on individual 2D boundary regions or on composite boundary regions.

#### 23.1.7.5.2. [Export Surface Name]: Option

- Selected Variables
- Acoustic Dipole/Acoustic Rotating Dipole
- Harmonic Forced Response

The `Harmonic Forced Response` option is only available for **Transient Blade Row** analysis.

#### 23.1.7.5.3. [Export Surface Name]: Particle Definition List

**Particle Definition** allows you to select the particle type for which the **Particle Track Data** is exported to file. The Output Variables list will only show particle track variables filtered according to the particle type selected in **Particle Definition**.

#### 23.1.7.5.4. [Export Surface Name]: Output Boundary List

(applies only when **Option** is set to `Selected Variables` or `Acoustic Dipole/Acoustic Rotating Dipole`)

See [Output Boundary List and Output Region Naming \(p. 351\)](#).

For the **Particle Track Data** option, particle track variables can only be sampled from boundary patches and particle injection regions.

#### 23.1.7.5.5. [Export Surface Name]: Location Type

(applies only when **Option** is set to `Harmonic Forced Response`)

For a `Harmonic Forced Response` analysis, you can use the **Location Type** setting to choose how location(s) for exported data are selected. The options are:

- **Boundary**: Enables you to select a boundary condition patch for data export.
- **Mesh Regions**: Enables you to select one or more mesh regions for data export. For example, you can choose to export data from an entire blade surface, or only the pressure side, suction side, or blade tip, in order to help map CFD results to a structural mesh.

---

**Note:**

This feature is only supported for:

- Time Domain Cases: blade flutter, inlet disturbance, Transient Rotor Stator (TRS)
- Harmonic Analysis Cases: blade flutter

---

### 23.1.7.5.6. [Export Surface Name]: Output Variables List

(applies only when **Option** is set to `Selected Variables`). For details, see [Output Variables List](#) (p. 341).

For the **Particle Track Data** option, only particle track variables can be specified.

### 23.1.7.5.7. [Export Surface Name]: Output Nodal Area Vectors Check Box

(Applies only when **Option** is set to `Selected Variables`)

To export acoustic data for LMS noise analysis, select this check box, and the **Value** check box contained within **Output Nodal Area Vectors**.

### 23.1.7.5.8. [Export Surface Name]: Excitation Frequency

(Applies only when **Option** is set to `Harmonic Forced Response`)

- **Engine Order:** Enables you to specify the engine order excitation for a blade row interaction analysis, in order to export the first harmonics of the surface pressure for the specific engine order excitation. For details, see [Case 4: Harmonic Forced Response in the CFX-Solver Modeling Guide](#). For the time integration method, you may not be able capture the specific mode if the time integration (number of timesteps per period) detail is not compatible with the engine order selected. Note that specifying an engine order of zero results in the export of time averaged pressure values.
- **Blade Boundary:** Enables you to export the first harmonics of pressure fields corresponding to blade vibration frequency from the blade flutter analysis. This output will be used to obtain the aerodynamic damping in the Harmonic Forced Response analysis.

### 23.1.7.5.9. [Export Surface Name]: Accumulation Period

(Applies only when **Option** is set to `Harmonic Forced Response`)

For a `Harmonic Forced Response` analysis, you can use the **Accumulation Period** option to specify the time over which the exported data is gathered to export the most accurate results. You can choose between:

- **Last Period:** You should use this option if the signal is fully periodic to the corresponding fundamental frequency.

For the `Engine Order` option under **Excitation Frequency**, the specified engine order must correspond to the fundamental frequency or its harmonics, where the fundamental frequency is the disturbance frequency being modeled.

For the `Blade Boundary` option under **Excitation Frequency**, the fundamental frequency corresponds to the vibration frequency of the specified blade boundary.

- **Full Revolution:** You should use this option if the signal is fully periodic over a time period corresponding to the blade passing through a full 360 degrees.

For the `Engine Order` option under **Excitation Frequency**, you can specify any integer engine order that may not correspond to any harmonics of the fundamental disturbance frequency.

For the `Blade Boundary` option under **Excitation Frequency**, the blade vibration frequency must correspond to an integer multiple of the rotor rotation frequency.

## 23.1.8. Common Settings

### 23.1.8.1. Option

- `Selected Variables`

Selected vertex fields are written to the results file. The fields are chosen from the **Output Variables List**. No clean restart is possible from these files.

- `Boundary Only` (available only on the **Trn Results** tab)

The selected transient results are written only to the vertices on the boundaries, reducing the size of each `.trn` file but also restricting the variable data available for postprocessing to that on the boundaries. Interior data will be undefined and, as such, this option is not recommended for postprocessing moving mesh simulations. Where available, hybrid data from each requested variable will be written, otherwise conservative values will be used. Data is not exported on symmetry boundaries or 1:1 periodic interfaces.

- `Smallest` (not available on the **Results** tab)

Mesh data and all solution vertex fields are written. A restart is possible from these files, but the restart will not be clean (you can expect a temporary increase in residual values).

- `Essential`

The smallest file that preserves a clean restart is written. This includes data written in the `Smallest` category and the following:

- GGI control surface fields
- Boundary face solution arrays
- Mass flow data fields

- `Standard`

This contains data written in the `Essential` category and the following:

- Hybrid fields
- Postprocessing fields

- `Solver Monitor`

Only the data needed to monitor a run, such as residuals and monitor data, are written to the results file. No mesh data or solution vertex fields are written. This option can be used, for example, to build an operating map without having to retain numerous results files.

- None

Used when no output of results is required (for example, during solver performance benchmarking).

### 23.1.8.2. File Compression

- None

This offers no compression.


- Default

This is a compromise between disk space and processing load.

- Best Speed Least Compression
- Low Speed Most Compression

You may want to increase the compression level for large backup files, or if you do not have much disk space.

### 23.1.8.3. Output Variables List

Allows you to select the output variables to write to the results file. Select the desired variables from the list or click the  icon to select from a list of all variables. **Output Variable List** is available only when the option is set to `Selected Variables` or `Boundary Only`.

### 23.1.8.4. Include Mesh Check Box

Adds the mesh to the output.

When **Include Mesh** is selected, the mesh data will be written to the results file. Using the **Include Mesh** option will enable postprocessing of the results file and will make restarting possible.

### 23.1.8.5. File Format Check Box

This check box determines whether the **File Format** setting will be specified, or left at the solver default value: `CFX Native`.

The **File Format** options are:

- `CFX Native`

The CFX-Solver writes a `.res` file, or, for multiple-configuration or operating point cases, an `.mres` file plus multiple `.res` files.

- `Common Fluids Format`

The CFX-Solver writes Common Fluids Format (CFF) files and, for monitoring operations in the CFX-Solver Manager, a minimal `.res` file of type `Solver Monitor`.

Note that neither the CFF nor `.res` file written using this option contains sufficient information for a restart.

- CFX Native and Common Fluids Format

The CFX-Solver writes a `.res` file and CFF files.

CFF files are written as a set of case and data files in the run directory, plus a fluids project file (`.flprj`) that references the set. The CFF case and data files have extensions `.cas.cff` and `.dat.cff`, respectively.

CFF files are for post-processing outside of CFD-Post. CFX-Solver has limitations for writing CFF files. Consequently, if you are planning to load results from CFX into CFD-Post, it is generally better to load `.res` or `.mres` files than to load CFF files.

Note that the CFX-Solver cannot write CFF files for cases that involve:

- Multiple-configuration
- Operating points
- Particles
- Radiation
- Transient Blade Row

Note that there are limitations on the data that CFX-Solver can write to CFF files. The following data is not written:

- Hybrid values, mass flow values, and any solution variables that are not located at mesh vertices
- Profile data that is held in an external file
- Solution residual and monitor data
- Boundary-only variables
- The domain-specific reference pressure and the value of expert parameter `include pref in forces` which could have been used to include reference pressure effects in force calculations

### 23.1.8.6. Output Equation Residuals Check Box

When **Output Equation Residuals** is set to `All`, the equation residuals for all equations are written to the results file for steady-state solutions. The residuals can then be viewed in CFD-Post. They appear as ordinary variables available from the full list of variables. This parameter replaces the expert parameter `output eq residuals`.

### 23.1.8.7. Output Boundary Flows Check Box

When **Output Boundary Flows** is set to `All`, equation flows, including mass flows, are written to the file you have set up. These flows enable accurate evaluations of forces, heat flows, and mass flow related calculations in CFD-Post. These flows are always written for `Standard` backup/results and transient files. They are not written for `Selected Variables` and `Smallest` transient files, unless the **Output Boundary Flows** parameter is set to `All`.

### 23.1.8.8. Output Variable Operators Check Box

This check box is available for results, backup results, and transient results, when using one of the following options: `Selected Variables`, `Smallest`, and `Essential`. Selecting this check box exposes the **Output Var. Operators** setting, which can have a value of `All` or `None`. Setting the value to `All` causes the applicable output file to include all available operators that the solver has calculated (for example, gradients, High Resolution betas) for the variables listed under **Output Variables List** or **Extra Output Variables List**. For the `Standard` option, the **Output Variable Operators** check box is not shown and all available operators are written.

### 23.1.8.9. Output Particle Boundary Vertex Fields Check Box

For cases with particles, when **Output Particle Boundary Vertex Fields** is selected, the following boundary vertex fields are written:

- Inlets, outlets, openings and interfaces:
  - Mass flow density
  - Momentum flow density
  - Energy flow density
- Walls:
  - Mass flow density
  - Stress
  - Erosion rate density.

For transient cases, the following additional boundary vertex fields are written:

- Inlets, outlets, openings and interfaces:
  - Time integrated mass flow density
  - Time integrated momentum flow density
  - Time integrated energy flow density
- Walls:
  - Time integrated mass flow density



- Time integrated erosion rate density.

For additional details, see [Particle Boundary Vertex Variables in the CFX Reference Guide](#)

### **23.1.8.10. Output Frequency Options**

This determines how often backup and transient results files are written during a run.

#### **23.1.8.10.1. Timestep Interval**

Enter a number that specifies the number of timesteps between the writing of each results file.

#### **23.1.8.10.2. Timestep List**

Enter a comma-separated list of timestep numbers that specifies the timesteps at which files are written.

#### **23.1.8.10.3. Time Interval**

Enter a number that specifies the simulation time interval between the writing of each file. The simulation time interval is added to a running total that starts at the simulation start time. An iteration within half a timestep of the current total has a file written.

#### **23.1.8.10.4. Time List**

Enter a comma-separated list of simulation times that specifies the iterations at which files are written.

#### **23.1.8.10.5. Every Timestep**

No further input is needed. A backup or transient results file is written for every timestep in a transient simulation.

#### **23.1.8.10.6. Every Iteration**

No further input is needed. A backup or transient results file is written for every iteration.

#### **23.1.8.10.7. Iteration Interval**

Enter a number that specifies the number of iterations between the writing of each file.

#### **23.1.8.10.8. Iteration List**

Enter a comma-separated list of iteration numbers that specifies the iterations at which files are written.

#### **23.1.8.10.9. Wall Clock Time Interval**

(Only available for backup files)

Enter a number that specifies the wall clock time interval between the writing of each backup file.

This is used to create backup files every so often in real time. For example, on an overnight simulation you might choose to have a backup file created every two hours, regardless of how many iterations or timesteps had been performed.

#### 23.1.8.10.10. None

No results files will be written. You might choose this option to temporarily disable writing backup or transient files while keeping the definition of what to include in those files so that you can easily re-enable writing those files.

## 23.2. Working with Output Control

---

The following topics will be discussed:

- [Working with Transient Statistics \(p. 345\)](#)
- [Working with Monitors \(p. 347\)](#)
- [Working with Export Results \(p. 350\)](#)

### 23.2.1. Working with Transient Statistics

This section describes the generation and output of running statistics for solution variables. The available statistics are arithmetic average, minimum, maximum, root-mean-square (RMS) and standard deviation.

This follows from the same statistical theory that is used to determine statistical Reynolds Stresses, for example,  $\rho \overline{u'v'}$  in turbulence modeling.

---

#### Note:

Transient statistics of boundary-only variables are not reliable on interfaces that are re-intersected, for example due to:

- Movement resulting from mesh deformation, or
  - The use of a `Transient Rotor Stator` frame change/mixing model.
- 

#### 23.2.1.1. Statistic Initialization and Accumulation

Arithmetic averages, RMS values, and the minimum and maximum values are initialized using the solution values and are updated with each time step.

Standard deviations are initialized with a value of zero. The type of standard deviation that is computed is the *sample* standard deviation.

Whenever the standard deviation is computed, the arithmetic average and RMS statistics are also computed.

### 23.2.1.2. Statistics as Variable Operators

Transient statistics are operators that act on variables (both conservative and hybrid values) identified in the **Output Variables List**. Like other variable operators, the data written to results files have names like `<variable>.<statistic>` where `<variable>` is the name of the specified variable and `<statistic>` is one of the following:

- `Trnmin` (Minimum)
- `Trnmax` (Maximum)
- `Trnavg` (Arithmetic average)
- `Trnrms` (Root mean square)
- `Trnsdv` (Standard deviation)

---

**Tip:**

To output transient statistics for intermediate results, be sure to select the **Output Variable Operators** check box on the **Transient Results** tab.

Choose the **Full** option if all variable operators are desired.

---

A significant consequence of treating transient statistics as operators is that only one instance of a `<variable>.<statistic>` exists during the entire simulation. For example, even if multiple transient statistics objects containing the arithmetic average of velocity are requested, only one statistic will ever exist. The potential for specifying different start (stop) iterations for these transient statistics objects is addressed by using the earliest (latest) value specified; that is, statistics are accumulated over the largest range of timesteps possible as defined by the start and stop iterations for all transient statistics objects.

---

**Note:**

If you want to re-initialize a given statistic (that is, remove the history from the statistic), you must shut down and restart the simulation with a new start (stop) iteration. This step is required to ensure that the new statistic accumulation interval is not included when searching for the earliest and latest start and stop iteration values, respectively.

---

### 23.2.1.3. Using Statistics with Transient Rotor-Stator Cases


You can use transient statistics to examine the convergence of a transient/rotor stator case. This is done by obtaining averaged variable data over the time taken for a blade to move through one pitch. By comparing consecutive data sets, you can examine if a pseudo steady-state situation has been reached. Variable data averaged from integer pitch changes should be the same if convergence has been achieved.

Each of the variables that are created by the CFX-Solver can be used in CFD-Post to create plots or perform quantitative calculations.

## 23.2.2. Working with Monitors


### 23.2.2.1. Setting up Monitors

To add basic monitors:


1. Click *Output Control*  or select Insert > Solver > Output Control from the main menu.  
The **Output Control** dialog box will appear.
2. Click the **Monitor** tab.
3. Select **Monitor Objects**.
4. Select which variables to monitor (Balances, Forces, Residuals, Totals, Particles).

By default, all of the listed quantities are monitored.

To add monitors for points and/or expressions:

1. In the **Monitor Points and Expressions** frame, click *Add new item* .  
The **Insert Monitor Point** dialog box appears, asking for the name of a new monitor.
2. Enter a name, or accept the default name, and then click **OK**.  
The [Monitor Name] frame expands to show a set of input fields.
3. Specify the settings for the monitor object. For details, see [Monitor Points And Expressions \(p. 318\)](#).
4. Add more monitor objects as desired.

To add monitors for aerodynamic damping (for transient blade row runs that involve boundaries that move periodically):

1. In the **Aerodynamic Damping** frame, click *Add new item* .  
The **Insert Aerodynamic Damping** dialog box appears, asking for the name of a new monitor.
2. Enter a name, or accept the default name, and then click **OK**.  
The [Aerodynamic Damping] frame expands to show a set of input fields.
3. Specify the settings for the monitor object. For details, see [Aerodynamic Damping \(p. 324\)](#).
4. Add more monitor objects as desired.

Click **OK** or **Apply** to set the definitions of all of the monitor objects. All monitor points will be displayed in the viewer.

### 23.2.2.2. Transient/ Mesh Deformation Runs

For a transient run or run with a moving mesh, the monitor point location can either update with the mesh or it can remain based off of the initial mesh. For details, see [\[Monitor Name\]: Monitor Location Control](#) (p. 320).

### 23.2.2.3. Output Information

Information on the variables to be monitored is given near the start of the CFX-Solver Output file. In the following example, the variables **Velocity** and **Pressure** were requested for the **Output Variables List** in the .ccl file.

```

+-----+
|                               User Defined Monitor Points                               |
+-----+
Monitor Point: my monitor point
Domain: rotor
Monitor vertex location (x,y,z):      4.101E-01,  5.703E-02,  5.263E-02
User specified location (x,y,z):      4.100E-01,  5.618E-02,  4.951E-02
Distance to user specified location:                                     3.231E-03
Valid variables from output variable list:
    Air Ideal Gas.Velocity u
    Air Ideal Gas.Velocity v
    Air Ideal Gas.Velocity w
    Pressure
+-----+

```

The "Monitor vertex location" shows the actual location that is being monitored (the closest vertex to the "User specified location"). The "Distance to user specified location" shows the difference between the specified and actual monitoring location.

The "Output variable list" shows the full name of all variables that will be monitored.

### 23.2.2.4. Expression

When using the `Expression` option for a monitor, the results of the evaluated expression are output at each iteration. Enter an expression that evaluates to a single value at each timestep. The following are examples of expressions that could be monitored:

- `force()@MainWall / (0.5*areaAve(density)@Inlet * areaAve(vel)@Inlet * area()@MainWall)`
- `volumeAve(CO2.mf)@Domain 1`

The variable names should be preceded with the fluid name when applicable. You can view a list of variable names in the **Expression** details view by right-clicking in the **Definition** window when editing an expression.

### 23.2.2.5. Viewing Monitor Values during a Run

You can view a plot of monitor point values during a solver run. For details, see [Monitors Tab in the CFX-Solver Manager User's Guide](#).

### 23.2.2.6. Viewing Monitor Point Values after a Run

You can view monitor point values easily in the CFX-Solver Manager after a run. For details, see [Exporting Plot Variables and Coordinates in the CFX-Solver Manager User's Guide](#).

You can also extract monitor point values from a `.res` file using the `cfx5dfile` command. The following syntax is used:

```
cfx5dfile <file> -read-monitor
```

where `<file>` is a CFX-Solver input or results file containing monitor point information. The output is sent to standard output (you may want to add a redirect to write the output to a text file, for example:

```
cfx5dfile <file> -read-monitor > out.txt
```

The output is produced as a list of variable names, followed by a comma-delimited list of values that correspond to the order of variable names. One line of these values is produced for every iteration that has been carried out.

You can enter:

```
cfx5dfile -help
```

to obtain more information.

### 23.2.2.7. Using Monitors in CEL

Within CEL expressions, you can use the `probe` function to refer to monitors and monitor statistics. For example:

- If `Monitor Point 1` is an expression-based monitor (a monitor point with **Option** set to `Expression`), with `Standard Deviation` in the specified statistics list:

- You can obtain the value of (the expression being monitored by) `Monitor Point 1` at the current time step with:

```
probe(Expression Value)@Monitor Point 1
```

(Here, "Expression Value" is not an example; it is part of the syntax.)

- You can obtain the standard deviation of the value of `Monitor Point 1` over the current interval (the size of which is specified in the definition of `Monitor Point 1`) with:

```
probe(Expression Value.Standard Deviation)@Monitor Point 1
```

- If `Monitor Point 2` is a location-based monitor (a monitor with **Option** set to `Cartesian Coordinates` or `Cylindrical Coordinates`):

- You can obtain the value of variable `Pressure` at `Monitor Point 2` at the current time step with:

```
probe(Pressure)@Monitor Point 2
```

---

**Note:**

You cannot directly obtain statistics from a location-based monitor. As a workaround, you can obtain statistics from an expression-based monitor, with the expression including a probe of a variable at the location-based monitor.

---

**Note:**

The `probe` function does not calculate a value for a statistic until the defined interval is complete. As a result, a "probe" function that retrieves a statistic from an expression-based monitor:

- Should not be used in a CEL expression for a quantity that is required at all times, for example, a boundary condition.
  - Causes any interrupt control condition in which it is used to be invalid (and ignored) until the required interval is complete.
- 

For details on setting up monitor statistics, see [\[Monitor Name\]: Monitor Statistics \(p. 322\)](#).

For an example of using a monitor statistic as part of interrupt control, see [Using Interrupt Control in Cases with Transient Convergence Behavior in the CFX-Solver Modeling Guide](#).

### 23.2.3. Working with Export Results

The topics in this section include:

- [File Naming Conventions \(p. 350\)](#)
- [Propagating Older Time Step Values \(p. 351\)](#)
- [Output Boundary List and Output Region Naming \(p. 351\)](#)
- [Output Variables List \(p. 351\)](#)

#### 23.2.3.1. File Naming Conventions

Each instance of an `Export Results` object will correspond to a particular group of files in a transient run.

Transient data is written into a series of files, named using the form:

```
<prefix>_<timestep>.<extension>
```

- `<prefix>` defaults to the `Export Results` object name unless you override this with the parameter "Filename Prefix"
- `<timestep>` is a string containing the timestep number

- `cgns` is the only available `<extension>`

The mesh information (mesh coordinates and nodal area vectors, if applicable) is written into a separate file to save disk space, because the mesh information does not change with time. The mesh filename is of the form:

```
<prefix>_mesh.cgns
```

A link is created for each solution file (`<prefix>_<timestep>.cgns`) to map the mesh coordinates to the mesh file (`<prefix>_mesh.cgns`). If you write your own reader, you need not open the mesh file separately to read in the mesh coordinates for each solution file.

### 23.2.3.2. Propagating Older Time Step Values

When exporting results to 3rd party applications, it is possible that values from an earlier time step are written at time steps where true data does not exist - for example at time steps where minimal results files were requested in the solver control setup. This is done because some 3rd party software can only successfully read exported files when a consistent number of variables exists in each file. It is up to you to recognize that these "placeholder variables" are not accurate at the particular time step.

### 23.2.3.3. Output Boundary List and Output Region Naming

The default behavior of this parameter is to attempt to create one composite boundary region per domain. The boundary condition patches do not have to be in the same domain so that rotating dipole sources and regular dipole sources will be contained in the same export file if this is desired.

Each export surface object name is unique. If the export surface lies within one domain, the name of each exported surface will simply be the `Export Surface` object name. If the export surface lies across multiple domains, a region will be exported for each domain spanned by the export surface. Such regions are named using the form "`<domain name>.<export surface name>`".

### 23.2.3.4. Output Variables List

When either of the acoustic options are selected, the output variables list is implied. For both acoustic options, there is output for pressure on vertices and the surface mesh (x, y, z and topology). For the rotating dipole option there is also output for nodal area vectors.





---

# Chapter 24: Transient Blade Row Models

---

This chapter describes the `Transient Blade Row Models` object that can appear in the **Outline** tree.

For instructions on setting up and using Transient Blade Row models, see [Transient Blade Row Modeling in the CFX-Solver Modeling Guide](#).

This chapter describes:

- 24.1. Inserting a New Transient Blade Row Models Object
- 24.2. Transient Blade Row Models Details View

## 24.1. Inserting a New Transient Blade Row Models Object

---

[Setting up a Transient Blade Row Model in the CFX-Solver Modeling Guide](#) describes a procedure for inserting a Transient Blade Row Model object.

To insert a `Transient Blade Row Models` object:

1. Do one of the following:
  - Right-click `Flow Analysis 1` (or whichever analysis is applicable) in the **Outline** tree, then select **Insert > Transient Blade Models** from the shortcut menu.
  - Select `Flow Analysis 1` (or whichever analysis is applicable) in the **Outline** tree, then select **Insert > Transient Blade Row Models** from the main menu.
2. Click **OK**.

## 24.2. Transient Blade Row Models Details View

---

The **Transient Blade Row Models** details view is available after you insert the object of the same name. For details, see [Inserting a New Transient Blade Row Models Object](#) (p. 353).

The following sections describe the settings in the **Transient Blade Row Models** details view:

- 24.2.1. Transient Blade Row Model Settings
- 24.2.2. Transient Method

### 24.2.1. Transient Blade Row Model Settings

The **Transient Blade Row Model > Option** setting can have one of the following values:

- None

This option disables Transient Blade Row modeling. It is useful when running transient simulations without any transformations.

- Profile Transformation

The **Transient Method** settings (**Time Period**, **Time Steps**, and **Time Duration**) must be configured. These settings are described in [Transient Method \(p. 359\)](#).

- Time Transformation

To help the solver apply the Time Transformation method to the applicable domains, you must accurately characterize a disturbance. For details, see [Time Transformation Disturbance Settings \(p. 354\)](#).

The **Transient Method** settings (**Time Period**, **Time Steps**, and **Time Duration**) must be configured. These settings are described in [Transient Method \(p. 359\)](#).

- Fourier Transformation

To help the solver apply the Fourier Transformation method to the applicable domains, you must accurately characterize a disturbance. For details, see [Fourier Transformation Disturbance Settings \(p. 356\)](#).

The **Transient Method** settings (**Time Period**, **Time Steps**, and **Time Duration**) must be configured. These settings are described in [Transient Method \(p. 359\)](#).


### 24.2.1.1. Time Transformation Disturbance Settings

The `Time Transformation` method improves solution accuracy over that of the Profile Transformation method when there is unequal pitch between the blade row subsets on each side of a domain interface that uses the Transient Rotor Stator frame change/mixing model.

A given domain can receive a disturbance from:

- An upstream or downstream domain interface that uses the Transient Rotor Stator frame change/mixing model. For example, the wake from a blade can pass through a domain interface and disturb the downstream domain(s).
- Inlet or outlet (or opening) boundaries. For example, a boundary condition could use a CEL function that depends on space and time in order to mimic the wake of an upstream blade.

The `Time Transformation` method can be applied to a domain that has an inlet or outlet disturbance.

To characterize a disturbance, create a new item by clicking *Add new item* , enter a unique name for the disturbance, then specify information about the disturbance in one of the following ways:

- Rotor Stator option

This option can be applied only when the disturbance originates from a domain interface that uses the Transient Rotor Stator frame change/mixing model.

After selecting this option:

1. Select a **Domain Interface** from the existing domain interfaces that use the Transient Rotor Stator frame change/mixing model.
2. For each side of the interface (**Side 1** and **Side 2**, which correspond to the first and second sides of the selected domain interface, respectively) specify the **Option** setting.

The **Option** setting for each side is used to determine the domain(s) to which the Time Transformation method is applied. The options for each side are:

– Automatic

This option causes the solver to identify the domain(s) directly touching the applicable side of the interface. This option is suitable when all the domains that are affected by the disturbance are in contact with the applicable side of the interface.

– Domain List

This option enables you to manually select the domain(s) that are on the applicable side of the interface and that are affected by the disturbance. You might need to use this option if, for example, a blade row is modeled with two domains with one being downstream of the other, in which case you would select both domains as being on one side of the interface, even though one of those domains does not touch the interface.

– None

This option prevents the present disturbance (but not any other disturbances) from applying the Time Transformation method to any domain that is on the applicable side of the interface.

For details, see [Case 1: Transient Rotor Stator Single Stage in the CFX-Solver Modeling Guide](#).

- Rotational Flow Boundary Disturbance option

Use this option to characterize a disturbance that originates from a boundary condition (for example, an inlet or outlet boundary condition that is specified using one or more CEL expressions that depend on space and time).

After selecting this option:

1. Set **Domain Name** to the name of the domain(s) that are affected by the disturbance.
2. Set the **Signal Motion** settings:
  - Setting **Option** to *Stationary* causes the signal to be stationary in the absolute (stationary) frame of reference.
  - Setting **Option** to *Rotating* enables you to select a coordinate frame as a way of specifying the signal motion. Any boundary using this coordinate frame will be made transient periodic with the period calculated from the pitch and rotating speed of the signal. For details on moving coordinate frames, see [Frame Motion Settings \(p. 375\)](#).
3. Specify information about the external blade row that creates the disturbance in the **External Passage Definition** settings:

- For a case with rotational periodicity, specify **Passages in 360**: the number of passages in 360° of the external blade row that creates the disturbance. Also specify **Pass. in Component**: the number of passages in the external blade row that creates the disturbance.
- 4. For a case with rotational periodicity, the **Passages in 360** and **Pass. in Component** settings on the domain **Basic Settings** tab apply.

For details, see [Case 2: Flow Boundary Disturbance in the CFX-Solver Modeling Guide](#).

## 24.2.1.2. Fourier Transformation Disturbance Settings


### 24.2.1.2.1. Basic Settings Tab

The `Fourier Transformation` method improves solution accuracy over that of the `Profile Transformation` method when there is unequal pitch between the blade row subsets on each side of a domain interface that uses the `Transient Rotor Stator frame change/mixing` model.

The `Fourier Transformation` method works for compressible as well as incompressible flows and for all ranges of disturbance pitch ratios from small to very large. For example, the `Fourier Transformation` method can handle a once-per-revolution disturbance on a blade passage, which is a situation that cannot be handled with the `Time Transformation` method.

A given domain can receive a disturbance from:

- Inlet or outlet (or opening) boundaries. For example, a boundary condition could use a `CEL` function that depends on space and time in order to mimic the wake of an upstream blade.
- An upstream or downstream domain interface that uses the `Transient Rotor Stator frame change/mixing` model. For example, the wake from a blade can pass through a domain interface and disturb the downstream domain(s).

To characterize a disturbance, create a new item by clicking *Add new item* , enter a unique name for the disturbance, then specify information about the disturbance in one of the following ways:

- `Rotor Stator` option

This option can be applied only when the disturbance originates from a domain interface that uses the `Transient Rotor Stator frame change/mixing` model.

After selecting this option:

1. Select a **Domain Interface** from the existing domain interfaces that use the `Transient Rotor Stator frame change/mixing` model.
2. For each side of the interface (**Side 1** and **Side 2**, which correspond to the first and second sides of the selected domain interface, respectively) select the `Phase Corrected` option and then specify the **Phase Corrected Intf.** and **Sampling Domain Intf.** settings.

The **Phase Corrected Intf.** setting specifies the periodic GGI-only interface (having rotational periodicity) to which to apply the phase shift with respect to the sampling domain interface signal.

The **Sampling Domain Intf.** setting specifies the non-periodic non-frame-change GGI-only interface on which the Fourier coefficients will be accumulated. This is the interface between a pair of adjacent blades in a given component.

For details, see [Case 1: Transient Rotor Stator Single Stage in the CFX-Solver Modeling Guide](#).

- Rotational Flow Boundary Disturbance option

Use this option to characterize a disturbance that originates from a boundary condition (for example, an inlet or outlet boundary condition that is specified using one or more CEL expressions that depend on space and time).

After selecting this option:

1. Specify the **Phase Corrected Intf.** and **Sampling Domain Intf.** settings.

The **Phase Corrected Intf.** setting specifies the periodic GGI-only interface (having rotational periodicity) to which to apply the phase shift with respect to the sampling domain interface signal.

The **Sampling Domain Intf.** setting specifies the non-periodic non-frame-change GGI-only interface on which the Fourier coefficients will be accumulated. This is the interface between a pair of adjacent blades in a given component.

2. Set the **Signal Motion** settings:

- Setting **Option** to *Stationary* causes the signal to be stationary in the absolute (stationary) frame of reference.

- Setting **Option** to *Rotating* enables you to select a rotating coordinate frame as a way of specifying the signal motion. Any boundary using this coordinate frame will be made transient periodic with the period calculated from the pitch and rotating speed of the signal. For details on moving coordinate frames, see [Frame Motion Settings \(p. 375\)](#).

3. Specify information about the external blade row that creates the disturbance in the **External Passage Definition** settings:

- For a case with rotational periodicity, specify **Passages in 360**: the number of passages in 360° of the external blade row that creates the disturbance. Also specify **Pass. in Component**: the number of passages in the external blade row that creates the disturbance.

4. For a case with rotational periodicity, the **Passages in 360** and **Pass. in Component** settings on the domain **Basic Settings** tab apply.

For details, see [Case 2: Flow Boundary Disturbance in the CFX-Solver Modeling Guide](#).

- Blade Flutter option

Use this option to characterize a disturbance that originates from the periodic motion of a wall boundary.

After selecting this option:

1. Specify the **Phase Corrected Intf.** and **Sampling Domain Intf.** settings as applicable.

The **Phase Corrected Intf.** setting specifies the periodic GGI-only bitmap-intersection-method interface (having rotational periodicity) to which to apply the phase shift with respect to the sampling domain interface signal.

The **Sampling Domain Intf.** setting specifies the non-periodic non-frame-change GGI-only interface on which the Fourier coefficients will be collected. This is the interface between a pair of adjacent blades in a given component.

2. Specify the **Blade Boundary.**

The **Blade Boundary** setting specifies the wall boundary with a `Periodic Displacement` mesh motion boundary condition that creates the disturbance. This boundary contains the information required (frequency, and phase angle) to compute the phase shift to be applied on the rotationally periodic boundaries.

For details, see [Case 3: Blade Flutter in the CFX-Solver Modeling Guide](#).

---

**Note:**

You must ensure that the disturbance is periodic in time when using the `Rotational Flow Boundary Disturbance` option.

---

### 24.2.1.2.2. Advanced Options Tab

It is possible to specify some further parameters to control the Fourier Transformation model. These parameters are specified on the **Advanced Options** tab of the **Transient Blade Row Models** details view. This tab becomes visible once you set **Transient Blade Row Model > Option** to `Fourier Transformation`.

On the **Advanced Options** tab, you can select **Fourier Transformation Control > Frequency Filtering**.

#### 24.2.1.2.2.1. Frequency Filtering

Frequency filtering is available on the **Advanced Options** tab under **Fourier Transformation Control**. Frequency filtering enables filtering of undesired frequencies that might be present in the solution and that could trigger instabilities. The `Scaled Domain Damping` option requires that you set a value for **Damping Scale Factor**. This value is a dimensionless positive number that has a lower bound of `0.0`, which specifies that there is no filtering. The recommended value for **Damping Scale Factor** is `1.0`.

## 24.2.2. Transient Method

You can select the methodology for obtaining a transient solution: `Time Integration`, or `Harmonic Balance`. For additional information, see [Transient versus Harmonic Solution Method in the CFX Reference Guide](#).

---

### Note:

If you restart a case from existing Fourier Transformation results, you do not have to specify the minimum number of periods for the specified method.

---

The following topics are discussed:

[24.2.2.1. Time Integration](#)

[24.2.2.2. Harmonic Balance](#)

### 24.2.2.1. Time Integration

The **Time Period** settings are used to establish a value for the time period in which each disturbance of interest cycles an integer number of times. The options for obtaining this number are:

- `Value`

This option requires that you specify a value for the time period directly.

- `Passing Period`

When this option is selected, the time period is calculated automatically as:

- For inlet disturbance cases:

The signal pitch divided by the relative velocity between the signal and the domain.

- For transient rotor stator cases:

The result of dividing the blade pitch by the absolute value of the angular velocity of the rotor(s), for the specified domain.

Note that the blade pitch is indirectly specified via the **Passages in 360** setting on the **Basic Settings** tab for the domain.

This resulting time period is displayed as the **Passing Period**.

- `Automatic`

When this option is selected, the time period is calculated automatically. The calculation uses the information specified for each disturbance, and the blade pitch of the domain. Note that the blade pitch of a domain is indirectly specified via the **Passages in 360** setting on the **Basic Settings** tab for the domain.

The time period is given a value such that, during one period, each disturbance undergoes an integer number of cycles. This time period is displayed as the **Passing Period**.



The **Min. Timesteps / Per.** is computed and displayed; it is the minimum number of time steps required to resolve each disturbance cycle period into an integer number of time steps.

In a typical transient simulation, you can specify the time step size directly. However, when using a Transient Blade Row model, the time step size is set indirectly.

The **Time Steps** settings control the size of a time step indirectly in one of two ways, depending on the option you choose:

- `Number of Timesteps per Period`

The time step size is computed by dividing the **Time Period** setting by a number, **Timesteps/Period**, that you specify.

- `Timestep Multiplier` (available only when the **Time Period** option is set to **Automatic**)

The time step size is computed by dividing the **Time Period** setting by the number of time steps per period, where the latter is the product of **Min. Timesteps / Per.** and **Timestep Multiplier**.

You must ensure that the number of time steps per period is sufficient to resolve each disturbance.

The **Time Duration** settings control the length of the simulation directly by controlling the number of periods. This setting also indirectly controls the total number of time steps in the simulation.

CFX-Pre automatically determines, lists, and uses the domains with the longest passing period. For the Fourier Transformation results to be sensible, CFX-Solver requires a minimum of 15 of the longest periods for the transient rotor stator cases, and a minimum of five of the longest periods for the inlet disturbance cases. Alternatively, you can use the extended list to select any other domain. If you pick a domain from the extended list, which does not have the longest passing period, then CFX-Pre will warn you. In this case, you will have to ensure that the **Number of Periods per Run** is increased to match what is needed for the domain with the longest passing period.

#### 24.2.2.2. Harmonic Balance

Specify the number of modes to retain in the approximation.

---

**Note:**

You cannot set up a case with Harmonic Balance and a Stage (Mixing-Plane) interface.

---

For guidelines on Harmonic Analysis, see [Harmonic Analysis in the CFX-Solver Theory Guide](#).

---

# Chapter 25: Mesh Adaption

---

Mesh Adaption in CFX is the process by which the mesh is selectively refined in areas that are affected by the adaption criteria specified. This means that as the solution is calculated, the mesh can automatically be made finer or coarser in locations where solution variables change rapidly, in order to resolve the features of the flow in these regions.

Each mesh element is given an Adaption Level. Each time the element is split into smaller elements, the new elements are assigned an Adaption Level that is one greater than the element it was generated from. The maximum number of Adaption Levels is controlled to prevent over-refinement.

In CFX, mesh adaption is only available for single domain, steady-state problems with limited physics. A full list of limitations can be found in [Mesh Adaption Limitations in the CFX-Solver Theory Guide](#). The Mesh Adaption process is performed by CFX-Solver. However, the parameters that control the Adaption process are defined in CFX-Pre on the **Mesh Adaption** form.

This chapter describes:

- [Overview \(p. 361\)](#)
- [Setting Up Mesh Adaption \(p. 363\)](#)
- [The Details View for Mesh Adaption \(p. 364\)](#)
- [Advanced Topic: Adaption with 2D Meshes \(p. 367\)](#)

## 25.1. Overview

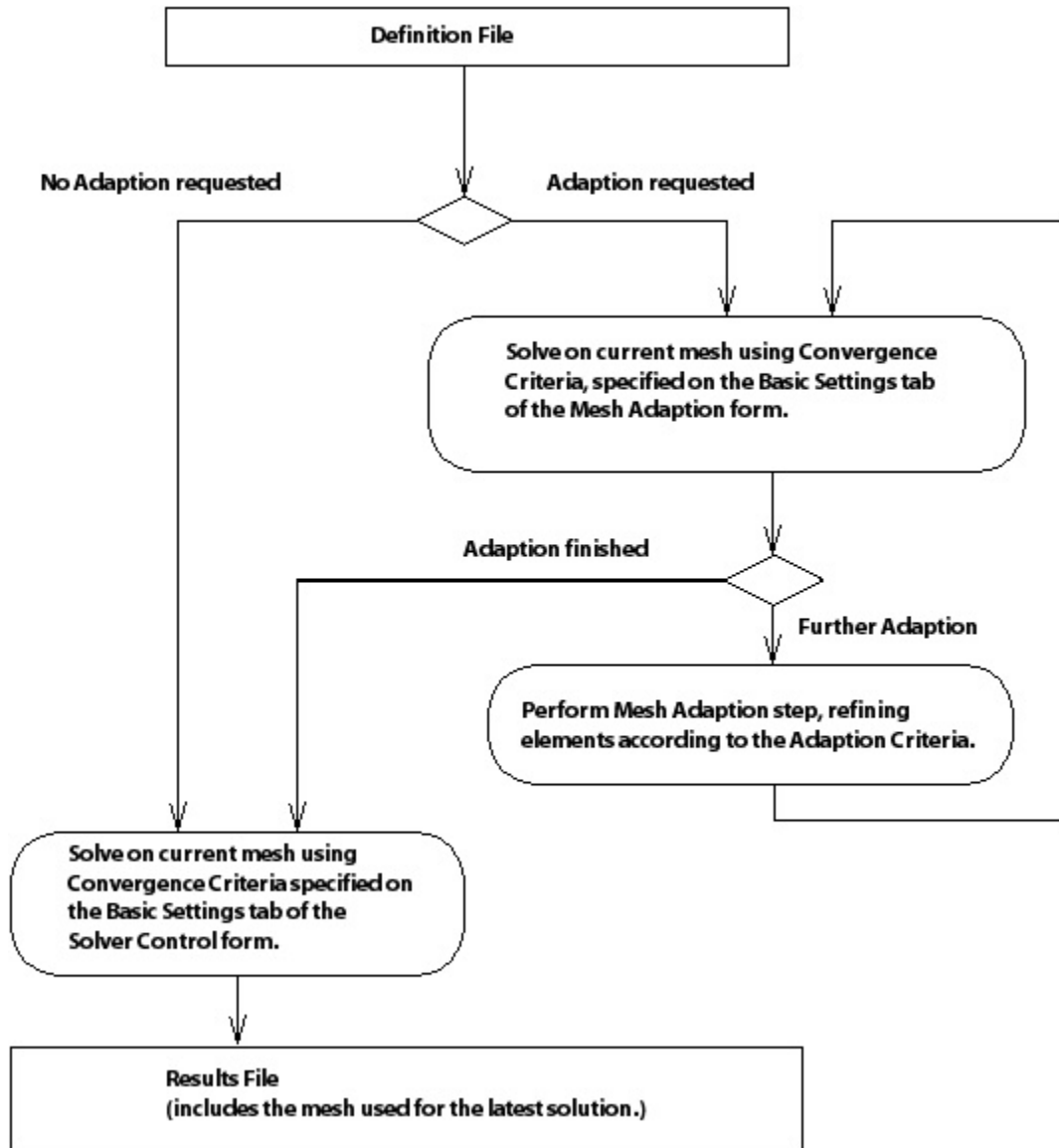
---

The following will take place when CFX-Solver is run (on steady-state problems). The process is shown in the diagrammatic form below ([Figure 25.1: Mesh Adaption Process \(p. 362\)](#)).

1. The CFX-Solver solves for solution variables using the mesh that is contained in the CFX-Solver input file, or specified using an initial values file. The CFX-Solver uses Convergence Criteria that have been specified on the **Basic Settings** tab of the **Mesh Adaption** form; the Convergence Criteria specified on the **Solver Control** form is not used at this stage.
2. A Mesh Adaption Step (one loop of the adapt-solve cycle) takes place. Using the solution calculated in this first step, together with the Adaption Criteria specified on the **Mesh Adaption Basic Settings** form, the mesh is refined in selected areas. For details, see [Mesh Adaption in the CFX-Solver Theory Guide](#).
3. The CFX-Solver solves for solution variables using the mesh created by the Mesh Adaption step. The CFX-Solver uses the Convergence Criteria specified on the **Basic Settings** tab of the **Mesh Adaption** form; the Convergence Criteria specified on the **Solver Control** form is not used at this stage.

4. Steps 2 and 3 are repeated until the **Max. Num. Steps** (specified on the **Basic Settings** of the **Mesh Adaption** form) is reached.
5. Finally, CFX-Solver solves for solution variables using the mesh that was created by the final Mesh Adaption step. The Convergence Criteria used by the CFX-Solver at this stage are those specified on the **Solver Control** form.

**Figure 25.1: Mesh Adaption Process**



The Mesh Adaption step itself consists of the following:

1. The Adaption Criteria are applied to each edge of each element in the mesh.
2. Nodes are added to the existing mesh according to the Adaption Criteria. The number of nodes added is dependent on the total number of nodes to be added and the node allocation parameter.
3. The solution already calculated on the older mesh is linearly interpolated onto the new mesh.

If the CFX-Solver is being run in parallel, then each "Solve" step is preceded by a mesh partitioning step.

Additional information on how elements are selected for adaption, how elements are divided, and the limitations of mesh adaption in CFX is available in [Mesh Adaption in the CFX-Solver Theory Guide](#).

## 25.2. Setting Up Mesh Adaption

---

To set up mesh adaption:

1. Select **Insert > Solver > Mesh Adaption**.

The **Mesh Adaption** dialog box appears.

2. Select or clear **Activate Adaption**.

If selected, Mesh Adaption performs when the solver is run.

3. Under **Region List**, select the regions to adapt.

For details, see [Region List \(p. 364\)](#).

4. Select or clear **Save Intermediate Files**.

For details, see [Save Intermediate Files \(p. 364\)](#).

5. Specify the required **Adaption Criteria**.

For details, see [Adaption Criteria \(p. 364\)](#).

6. Specify the required **Adaption Method**.

For details, see [Adaption Method \(p. 365\)](#).

7. Specify **Adaption Convergence Criteria**.

For details, see [Adaption Convergence Criteria \(p. 366\)](#).

8. Select Adaption Variables from the Variables List.

9. Enter the Maximum Number of Adaption Steps (Max. Num. Steps) allowed.

The default is 3. For details, see [Max. Num. Steps \(p. 365\)](#).

10. Select how many nodes should be present in the adapted mesh. Options are:

- **Multiple of Initial Mesh**
- **Final Number of Nodes**

For details, see [Option \(p. 365\)](#).

11. Select an option for **Adaption Method > Option**.

For details, see [Adaption Method \(p. 365\)](#).

12. Specify the **Adaption Convergence Criteria** to be used for each adaption step.

For details, see [Adaption Convergence Criteria](#) (p. 366).

This is different from the Convergence Criteria used to terminate the run, which is set in Solver Controller. For details, see [Basic Settings Tab](#) (p. 289).

13. Switch to the **Advanced Options** tab or click **OK** to accept the mesh adaption settings. For details, see [Advanced Options Tab](#) (p. 366).

## 25.3. The Details View for Mesh Adaption

---

The following tabs are presented on the details view of **Mesh Adaption**:

- [Basic Settings Tab](#) (p. 364)
- [Advanced Options Tab](#) (p. 366)

### 25.3.1. Basic Settings Tab

The following sections describe the options available on the **Basic Settings** tab for **Mesh Adaption**.

#### 25.3.1.1. Region List

**Region List** contains the names of all 3D Regions and Assemblies in the problem. Select any or all of the 3D regions to be used for mesh adaption.

---

**Note:**

Mesh adaption cannot be used in multidomain simulations or in cases with external solver coupling. Mesh adaption also cannot be used for transient, mesh-motion, radiative-tracking, or particle-transport cases.

---

#### 25.3.1.2. Save Intermediate Files

If **Save Intermediate Files** is selected, an intermediate results file is saved immediately before each mesh adaption step begins. At the end of the run, these intermediate files are stored in a subdirectory with the same name as the run, in the directory that contains the CFX-Solver results file.

#### 25.3.1.3. Adaption Criteria

For each adaption step, and for each mesh element, the adaption criteria are applied, and mesh elements meeting the adaption criteria are refined. There are two methods of specifying how the adaption criteria are specified. For details, see [Adaption Method](#) (p. 365).

##### 25.3.1.3.1. Variables List

The Variables List is used to select the variables that make up part of the Adaption Criteria.

During the adaption process, if only one variable is selected, the value of the variable is observed for each element defining the selected regions specified by the Region List. The maximum variation in value of the variable along any edge of an element is used to decide whether the element

is to be modified. If multiple variables are selected, the maximum of variation of all the variables for a given element is used to decide whether or not an element should be modified.

To save unnecessary processing, it is important to ensure that variables selected will vary during the calculation. For instance, do not select Density as a variable for an incompressible flow calculation.

### 25.3.1.3.2. Max. Num. Steps

The number of steps that the adaption process performs is specified by **Max. Num. Steps**. It is recommended that you select a number between 1 and 5.

---

#### Note:

If CFX-Solver runs on the CFX-Solver input file and finishes normally, this number of Adaption Steps will take place. If CFX-Solver is stopped and then restarted from the results file produced, only the remaining number of Adaption Steps will take place in the restarted run.

---

### 25.3.1.3.3. Option

The number of nodes in the final mesh generated by the adaption process is controlled by the value selected in **Option**.

Select **Final Number of Nodes**, to specify the number of nodes in the final mesh, or **Multiple of Initial Mesh**, which enables specification of the number of nodes in the final mesh as a multiple of the initial mesh.

If **Multiple of Initial Mesh** is selected, it is also necessary to specify a **Node Factor** multiplier greater than 1.2. If **Final Number of Nodes** is selected, then specify the number of Nodes in Adaption Mesh that is no more than a factor of five greater than the number of nodes in the initial mesh.

---

#### Note:

The final mesh will not contain exactly the number of nodes specified in either case. For details, see [Mesh Adaption in the CFX-Solver Theory Guide](#).

---

## 25.3.1.4. Adaption Method

The **Adaption Method** used by the adaption process to apply the **Adaption Criteria** is controlled by the options specified in the **Adaption Method** section of the form.

### 25.3.1.4.1. Option

The **Adaption Method** is specified as either **Solution Variation**, or **Solution Variation \* Edge Length**.

If **Solution Variation \* Edge Length** is selected, the **Adaption Criteria** takes account of both the variation of the solution variable over the edge of the element and the length of the edge. The result of having applied the **Adaption Criteria** to each edge of an element is then multiplied

by the length of the edge. The maximum value of all the edges of the element is used to decide whether an element is to be refined. This means that in areas of the flow where the solution variation is similar, adaption will take place preferentially in regions where the mesh length scale is largest.

### 25.3.1.4.2. Minimum Edge Length

When **Solution Variation** is specified by **Option**, the Adaption Criteria is applied using only the maximum variation of a solution variable across any edge associated with an element. This option does not use the length of the edge in the calculation. In this case, consider specifying a **Minimum Edge Length** to avoid refining the mesh too far at a discontinuity in the solution. For details, see [Mesh Adaption in the CFX-Solver Theory Guide](#).

### 25.3.1.5. Adaption Convergence Criteria

The convergence criteria used to specify when the CFX-Solver stops running on the original and intermediate meshes is specified in the **Adaption Convergence Criteria** section of the form. The available parameters are the same as those used to determine the final convergence of the solution. For details, see [Basic Settings Tab \(p. 289\)](#).

## 25.3.2. Advanced Options Tab

It is possible to specify some further parameters to control the adaption process, these are specified on the **Advanced Options** tab of the **Mesh Adaption** form.

### 25.3.2.1. Node Alloc. Parameter

Setting the node allocation parameter, **Node Alloc. Parameter**, controls how much the mesh is refined at each adaption step. For some problems, it may be that it is better to refine a lot in the early steps, and for others it may be more appropriate to refine more in the later steps.

**Node Alloc. Parameter** is used as follows. On the  $n$ th adaption step, approximately  $S_n$  nodes are added to the mesh, where

$$S_n = \frac{M n^{-c}}{\sum_n n^{-c}}$$

$M$  is the maximum number of nodes that can be added to the original mesh calculated from having applied the **Adaption Criteria** to the selected regions and  $c$  is the value of **Node Alloc. Parameter**. For details, see [Adaption Criteria \(p. 364\)](#).

When **Node Alloc. Parameter** is set to 0, then the same number of nodes is added for each adaption step. When **Node Alloc. Parameter** is negative, more nodes are added in the later adaption steps. When it is positive, more nodes are added in the earlier adaption steps. The table below shows the percentage of nodes that will be added at each adaption step when **Max. Num. Steps** is set to a value of 3 and different values of **Node Alloc. Parameter** are specified.

<b>Node Alloc. Parameter</b>	<b>Step 1</b>	<b>Step 2</b>	<b>Step 3</b>
-2.0	7.14	28.57	64.28

<b>Node Alloc. Parameter</b>	<b>Step 1</b>	<b>Step 2</b>	<b>Step 3</b>
-1.0	16.66	33.33	50.00
0.0	33.33	33.33	33.33
0.5	43.77	30.95	25.27
1.0	54.54	27.27	18.18
2.0	73.47	18.36	8.16

It is recommended that you set a value for **Node Alloc. Parameter** in the range -2 to 2.

### 25.3.2.2. Number of Levels

The value of this parameter specifies the maximum number of times any element of the original mesh can be subdivided. It must not be greater than **Max. Num. Steps**.

## 25.4. Advanced Topic: Adaption with 2D Meshes

2D Meshes are not supported in CFX. However, CFX can use 3D meshes that are one element thick. This section outlines a workaround for such meshes when using mesh adaption.

CFX 2D meshes contain hexahedral, prismatic and potentially pyramidal and tetrahedral elements. In the support of prismatic inflation in 3D meshes, special restrictions are applicable to the adaption of prismatic elements. If adaption is applied to a 3D mesh that is one element thick, an error will result. In order to work around this, set the environment variable `CFX5_REFINER_NO_TRICOLUMNS` to 1 before starting CFX-Pre. Note that any refinement that takes place will be 3D refinement that will introduce additional elements in the third dimension. This environment variable can also be used when the input mesh has prismatic elements that have opposite triangular faces on the boundary. This can arise if the mesh has been imported from a mesh generation tool other than CFX-Mesh.





---

# Chapter 26: Expert Control Parameters

---

This chapter describes:

## 26.1. Modifying Expert Control Parameters

All geometry, domain, boundary condition, mesh, initial value and solver parameter information is written to the CFX-Solver Input (.def) File.

The CFX-Solver input file contains the relevant information required by the CFX-Solver to conduct the CFD simulation. This information mainly consists of numerical values that set up the simulation, as well as controls for the CFX-Solver.

Many of these parameters are set in CFX-Pre. For example, on the **Solver Control** panel, you can set the accuracy of the numerical discretization. *Expert control parameters* are settings that do not require modification for the majority of CFD simulations, and cannot immediately be set through the CFX-Pre interface. For details, see [CFX-Solver Expert Control Parameters in the CFX-Solver Modeling Guide](#).

Note that expert control parameters are only intended for use by customers who are experienced in using CFX, or who have been instructed to use expert control parameters by Ansys customer support. Use of these parameters is not fully supported, and may cause unexpected or unintended consequences for both the performance of the CFX-Solver and the quality of the results. For details, see [When to Use Expert Control Parameters in the CFX-Solver Modeling Guide](#).

## 26.1. Modifying Expert Control Parameters

---

1. Select **Insert > Solver > Expert Parameter**.

The **Expert Parameters** details view appears.

2. Make changes to the appropriate sections on any of the following tabs: **Discretization, Linear Solver, I/O Control, Convergence Control, Physical Models, Particle Tracking, or Model Overrides**.

Making changes requires selecting options and setting specific values. For parameters with a small number of choices (such as logical parameters), select an option from the drop-down list. Other parameters require data entry (such as real numbers).

For details on each of the parameters listed on these tabs, see [CFX-Solver Expert Control Parameters in the CFX-Solver Modeling Guide](#).

3. Repeat the previous step as desired.
4. Click **OK**.



---

# Chapter 27: Coordinate Frames

---

By default, all quantities used in CFX-Pre are defined with reference to the global Cartesian coordinate frame. In some cases, it is useful to use a different coordinate frame for specifying boundary conditions, initial conditions, source components or spatially varying material properties. It is possible to specify one or more local coordinate frame objects to which you can then refer. This chapter describes the user interface for creating local coordinate frame objects. For details, see [Coordinate Frames in the CFX-Solver Modeling Guide](#).

Local `Coordinate Frame` objects may be defined in terms of Cartesian coordinates. Here, the numbers 1, 2, 3 are used to denote the Cartesian X, Y, Z axes.

This chapter describes:

- 27.1. [Creating a New Coordinate Frame](#)
- 27.2. [Coordinate Frame Basic Settings Tab](#)

## 27.1. Creating a New Coordinate Frame

---

1. Select **Insert > Coordinate Frame**.

The **Insert Coordinate Frame** dialog box appears

2. Set **Name** to a unique name for the coordinate frame.

`Coord 0` cannot be used as a name as it is the default coordinate frame. For details, see [Valid Syntax for Named Objects \(p. 95\)](#).

3. Click **OK**.

The coordinate frame details view appears, with the **Basic Settings** tab visible.

4. Specify the basic settings.

For details, see [Coordinate Frame Basic Settings Tab \(p. 371\)](#).

5. Click **OK**.

An object named after the coordinate frame is created and listed in the tree view under `Simulation`.

## 27.2. Coordinate Frame Basic Settings Tab

---

The **Basic Settings** tab for a coordinate frame object contains settings that specify the coordinate frame. It is accessible by creating a new coordinate frame or by editing a coordinate frame listed in the tree view.

- 27.2.1. Coordinate Frame: Option
- 27.2.2. Coordinate Frame: Centroid
- 27.2.3. Coordinate Frame: Direction
- 27.2.4. Coord Frame Type
- 27.2.5. Reference Coord Frame
- 27.2.6. Origin
- 27.2.7. Z-Axis Point
- 27.2.8. X-Z Plane Point
- 27.2.9. Frame Motion Settings
- 27.2.10. Visibility Check Box

## 27.2.1. Coordinate Frame: Option

### 27.2.1.1. Point and Normal

This method can be used to make Cartesian coordinate frames. The origin of the coordinate frame is set to the centroid of a specified 2D region (which is available only when a mesh exists). Axis 3 of the coordinate frame is then computed as locally normal to the 2D region; its direction can be reversed if not satisfactory (after examining the coordinate frame representation in the viewer). Optionally, Axis 1 may be given a prescribed orientation about Axis 3 by specifying a point that is interpreted as lying on the 1-3 plane with a positive Axis 1 coordinate. If such a point is not specified explicitly, the global origin is used. Axis 2 is found by the right-hand rule.

### 27.2.1.2. Axis Points

This method can be used to make Cartesian coordinate frames. The coordinate frame is created by specifying three points. It is important to understand how these three points are used to create a coordinate frame. For details, see [Coordinate Frame Details in the \*CFD-Post User's Guide\*](#).

## 27.2.2. Coordinate Frame: Centroid

### 27.2.2.1. Location

(applies only when **Option** is set to `Point and Normal`)

Select a 2D region or combination of 2D regions, the centroid of which will be used as the origin of the coordinate frame.

---


**Tip:**

Hold the **Ctrl** key when clicking to select multiple regions.

---

---

**Tip:**

With *Single Select*  selected, you may click locations in the viewer to make them available for selection.

---

### 27.2.2.2. Centroid Type

(applies only when **Option** is set to `Point` and `Normal`)

- **Absolute**

The true centroid position is used. If the specified region is not planar, the absolute centroid may not lie on the surface.

- **Nearest Point on Mesh**

The mesh node nearest to the true centroid is used.

### 27.2.3. Coordinate Frame: Direction

#### 27.2.3.1. Invert Normal Axis Direction Check Box

(applies only when **Option** is set to `Point` and `Normal`)

This check box determines whether or not the direction of the Z axis is reversed from that initially determined.

#### 27.2.3.2. Point on Reference Plane 1-3 Check Box

(applies only when **Option** is set to `Point` and `Normal`)


This check box determines whether or not the  $\theta=0$  direction is defined explicitly by a point in the 1-3 plane. If selected, you must specify that point.

### 27.2.3.3. Point on Reference Plane 1-3 Check Box: Coordinate

Enter global Cartesian coordinates that define a point on the 1-3 plane of the coordinate frame. The direction of this point from the nearest point on Axis 3 is the direction of Axis 1. Axis 1 corresponds with the radial direction for  $\theta=0$ .

---

**Tip:**

With *Single Select*  selected, you may click 2D locations in the viewer to select their corresponding points.

---

### 27.2.4. Coord Frame Type

(applies only when **Option** is set to `Axis Points`)

Coordinates interpreted with reference to the coordinate frame will be interpreted as **Cartesian**. For details, see [Cartesian Coordinate Frames in the CFX-Solver Modeling Guide](#).

### 27.2.5. Reference Coord Frame

(applies only when **Option** is set to `Axis Points`)

Select an existing coordinate frame to use as a basis for interpreting the point data that defines the present coordinate frame.

### 27.2.6. Origin

(applies only when **Option** is set to `Axis Points`)

In the reference **Coordinate Frame**, enter coordinates that define the origin of the present coordinate frame.

---

**Tip:**

With *Single Select*  selected, you may click 2D locations in the viewer to select their corresponding points.

---


### 27.2.7. Z-Axis Point

(applies only when **Option** is set to `Axis Points`)

In the reference **Coordinate Frame**, enter coordinates that define a point on the positive side of the Z axis of the present **Coordinate Frame**.

---

**Tip:**

With *Single Select*  selected, you may click locations in the viewer.

---

### 27.2.8. X-Z Plane Point


(applies only when **Option** is set to `Axis Points`)

In the reference **Coordinate Frame**, enter coordinates that define a point on the X-Z plane of the present **Coordinate Frame**.

The direction of this point from the nearest point on the Z axis is the direction of the X axis.

---

**Tip:**

With *Single Select*  selected, you may click locations in the viewer.

---

### 27.2.9. Frame Motion Settings

The options are:

- Stationary
- Rotating

You can specify a constant angular velocity for the coordinate frame by setting **Option** to `Rotating` and specifying a (constant) value (which could be in the form of a CEL expression) for the **Angular Velocity**. You will also have to specify a *stationary* axis about which the rotation occurs, using the **Axis Definition** settings.

A boundary condition can use a rotating coordinate frame as its local frame of reference in order to cause an applied profile (for example, a pressure profile) to move.

### 27.2.10. Visibility Check Box

This check box determines whether or not the graphics representation of the coordinate axis is displayed in the viewer.





---

# Chapter 28: User Locations

---

User locations enable you to specify an arbitrary set of points and connectivities that are not directly associated with the volume mesh. User locations have uses such as:

- Providing a location on which the solver can write solution data.

You can observe the solution on some user locations while the solver is in progress.

Postprocessing on user locations in CFD-Post provides a more efficient way to view data in regions of known interest and can reduce the size of transient results files.

For details, see [Monitor Surfaces \(p. 327\)](#).

- Specifying injection region locations.

For details, see [Injection Positions \(p. 255\)](#) and [User Functions for Injection Regions \(p. 268\)](#).

User locations that are generated in CFX-Pre (that is, user lines that use the `Two Points` option and user points that are specified directly by their coordinates) are stored internally in the CFX-Solver input file. By default, the user location data is updated (recalculated) if necessary at the start of the CFX-Solver run. Controls are available to override this behavior: see [Solver Tab \(p. 455\)](#).

This chapter describes:

[28.1. User Surface](#)

[28.2. User Line](#)

[28.3. User Line Cloud](#)

[28.4. User Point](#)

[28.5. User Point Cloud](#)

## 28.1. User Surface

---

You can define a user surface from a `.CSV` file, for example one that was exported from CFD-Post. For details on the export of surface data from CFD-Post, see [USER SURFACE Data Format in the CFD-Post User's Guide](#).

### Creating a User Surface

1. Select **Insert > User Locations > User Surface**.

The **Insert User Surface** dialog box appears.

2. Set **Name** to a unique name for the user surface and click **OK**.

The details view for the user surface appears.

3. Select a **Data Storage** option:

- Use External File

Data is defined by the file you specify under **File Name**. The CFX-Solver will use the data stored in that file.

---

**Note:**

When submitting a run to a cluster or running on a different machine, the path to the `.csv` file must be valid on that cluster/machine.

---

- Internal

Data is read from the file that you select after clicking **Copy Data Into Setup**. This data is copied into the case definition when you click **Apply** or **OK**. The CFX-Solver will use the data stored with the case definition.

Note that, after you click **Copy Data Into Setup**, the **Copy Data Into Setup** dialog box appears, where you specify the data file to read. If the specified file has an incorrect format, an error message will appear; after making corrections to such a file, click **Reload** to reassess the format of the file.

4. In the details view, optionally select **Visibility** to view the surface in the graphics window.

---

**Note:**

The displayed surface will not be automatically updated if the user surface input file is subsequently modified, even when using the `Use External File` data storage option.

---

5. Click **OK**.

An object named after the user surface is created and listed in the tree view under `Simulation > User Locations`.

## 28.2. User Line

---

You can define a user line, which may be a polyline, by specifying points using either Cartesian (x, y, z) or cylindrical (Axial, Radial, Theta) coordinates. The coordinates must be specified relative to the global coordinate frame, `Coord 0`.

### Creating a User Line

1. Select **Insert > User Locations > User Line**.

The **Insert User Line** dialog box appears.

2. Set **Name** to a unique name for the user line and click **OK**.

The details view for the user line appears.

3. Select an option for defining the line:

- From File

Using the `From File` option, you can define a user line from a `.csv` file, for example one that was exported from CFD-Post. For details on the export of polyline data from CFD-Post, see [POLYLINE Data Format in the CFD-Post User's Guide](#).

Select a **Data Storage** option:

- Use External File

Data is defined by the file you specify under **File Name**. The CFX-Solver will use the data stored in that file.

---

**Note:**

When submitting a run to a cluster or running on a different machine, the path to the `.csv` file must be valid on that cluster/machine.

---

- Internal

Data is read from the file that you select after clicking **Copy Data Into Setup**. This data is copied into the case definition when you click **Apply** or **OK**. The CFX-Solver will use the data stored with the case definition.

Note that, after you click **Copy Data Into Setup**, the **Copy Data Into Setup** dialog box appears, where you specify the data file to read. If the specified file has an incorrect format, an error message will appear; after making corrections to such a file, click **Reload** to reassess the format of the file.

Line connectivity can be defined but is not required; if connectivity data is missing then the connectivity is dictated by the order of the points in the file.

- Two Points

Specify the total number of points (at least 2) in the line and the two end points of the line.

You can specify the coordinates of the end points using your choice of coordinate system:

- A Cartesian (x, y, z) coordinate system.

The coordinates must be specified relative to the global coordinate frame, `Coord 0`.

- A Cylindrical (Axial, Radial, Theta) coordinate system.

The coordinates must be specified relative to the global coordinate frame, `Coord 0`.

- A coordinate system defined by a profile function.

The end points of the line are defined by two points that are specified in custom coordinates (determined by the reference profile function; see [Coordinate System Definition \(p. 426\)](#)). The intermediate points on the line are created equally spaced in the custom coordinate system. This may result in a curved line when all of the points are mapped to Cartesian coordinates. If some of the intermediate points cannot be mapped to Cartesian coordinates, and **Ignore Unmapped Intermediate Points** is selected, then these points are skipped, and the line will have fewer points than specified and will be broken into segments. If either of the end points cannot be mapped, or if intermediate points cannot be mapped and **Ignore Unmapped Intermediate Points** is not selected, then the line will not be generated and an error message will be given.

---

**Note:**

The intermediate points are calculated as equally-spaced points in the specified coordinate system; for end points specified in cylindrical coordinates, the intermediate points are equally spaced in cylindrical coordinates and the line will not necessarily be a straight line.

---

4. In the details view, optionally select **Visibility** to view the line in the graphics window.
5. Click **OK**.

An object named after the user line is created and listed in the tree view under `Simulation > User Locations`.

## 28.3. User Line Cloud

---

You can define a user line cloud, which consists of a field of lines and points along those lines. The lines start at a specified wall boundary and run normal to the wall boundary (except at wall boundary edges, where the boundary normal is not well defined; see below).

A user line cloud can be used to apply field data, such as temperature, a short distance away from where it is sampled. For example, when modeling wall boiling, the temperature near a wall can be applied at the wall in order to compute the boundary layer liquid temperature. For each line in the line cloud, the temperature at each point along the line is sampled and then used to calculate a representative temperature value for the line, which is then applied in the vicinity of the line, within a radius of influence, via an object (for example, the `lineCloudAve` function) that references the line cloud.

### Creating a User Line Cloud

1. Select **Insert > User Locations > User Line Cloud**.

The **Insert User Line Cloud** dialog box appears.

2. Set **Name** to a unique name for the user line cloud and click **OK**.

The details view for the user line cloud appears.

3. Configure the **Boundary Line Location** and **Boundary Line Type** settings.

In the **Boundary Line Location** settings, **Option** can be set to:

- All Walls

The line cloud is generated over all walls in all domains.

- Boundary List

The line cloud is generated over the boundaries specified by **Boundary List**.

In the **Boundary Line Location** settings, optionally select **Line Origin Mesh Location** and set **Location** to one of the following:

- Face Center
- Face Integration Points
- Vertex

In the **Boundary Line Type** settings, **Option** can be set to:

- Fixed Length

Specify the value for **Line Length** to be used by every line in the line cloud.

In the **Boundary Line Type** settings, optionally select **Local to Domain** to cause lines to terminate on domain boundaries; line points outside domain boundaries are effectively deleted. If **Local To Domain** is not selected then lines can span domains.

In the **Boundary Line Type** settings, optionally select **Line Influence Radius to Active Length Ratio** and specify a value for **Ratio**. If **Line Influence Radius to Active Length Ratio** is not selected then the default ratio of 0.1 takes effect. A larger ratio increases the radius of influence of a given line, affecting more of the surrounding field values. Note that the radius of influence of a given line is directed normal to (not parallel to) the line.


In the **Boundary Line Type** settings, **Line Point Distribution** > **Option** can be set to:

- Uniform

The points along each line are evenly spaced.

Specify a value for **Line Point Distribution** > **Number of Points** to specify the number of points per line in the line cloud.

4. Optionally configure the **Line Averaging** settings. These settings control how variables are line averaged, offering such control on a per variable basis.

Start by creating a new **Line Averaging** list item, using the *Add new item*  icon to the right of the list.

If you select **Line Averaging Distance Fraction**, then you can specify, as a value between 0.0 and 1.0, the fraction of the line length over which data is averaged. For example, if you specify a value of 0.25, then only the first quarter of each line, starting at the wall end, is used to compute the variable averages.

If you do not select **Line Averaging Distance Fraction**, then a value of 1.0 takes effect.

If you select **Line Variables List** then you can make the averaging settings (currently only **Line Averaging Distance Fraction**) variable-specific.

- Optionally configure the **Line Interpolation** settings. These settings control how the calculated line cloud values (line averages) are interpolated to the mesh vertices, offering some control on a per variable basis. The mesh vertices just referenced are those that are found within the space occupied by the line cloud.

Optionally select **Number of Nearest Interpolation Points** and set **Value** to the number of line cloud values (not necessarily unique values) to use in the interpolation. For each mesh vertex to which values are to be interpolated, a number of nearby line cloud points are used to identify candidate lines. Of those lines, the specified number (**Number of Nearest Interpolation Points** > **Value**) are selected in order of nearest line origin. The same line can be selected more than once if more than one line cloud point that is near the mesh vertex belongs to the same line.

Optionally select **Cloud Point Distance Tolerance**, which is the distance (1E-6 m by default) within which a line point is considered to be coincident with a mesh vertex. If a line point is considered to be coincident with a mesh vertex, the line point value is directly assigned to the mesh vertex (bypassing the rest of the interpolation process for that mesh vertex).

Optionally select **Variable Interpolation Distance Weighting** to control the weighting for line cloud values in the interpolation process. The weighting for a particular line can be based on the distance from the mesh vertex to the nearest point on the line (**Point Distance**) or based on the distance from that nearest point on the line to the line origin (**Origin Distance**). The weighting is inversely proportional to this distance.

Optionally configure the **Line Interpolation Variable Options** settings. These settings offer per variable control over the weighting used in the interpolation process. Unspecified variables are interpolated as per the **Variable Interpolation Distance Weighting** specification (mentioned above).

Start by creating a new **Line Interpolation Variable Options** list item, using the *Add new item*  icon to the right of the list.

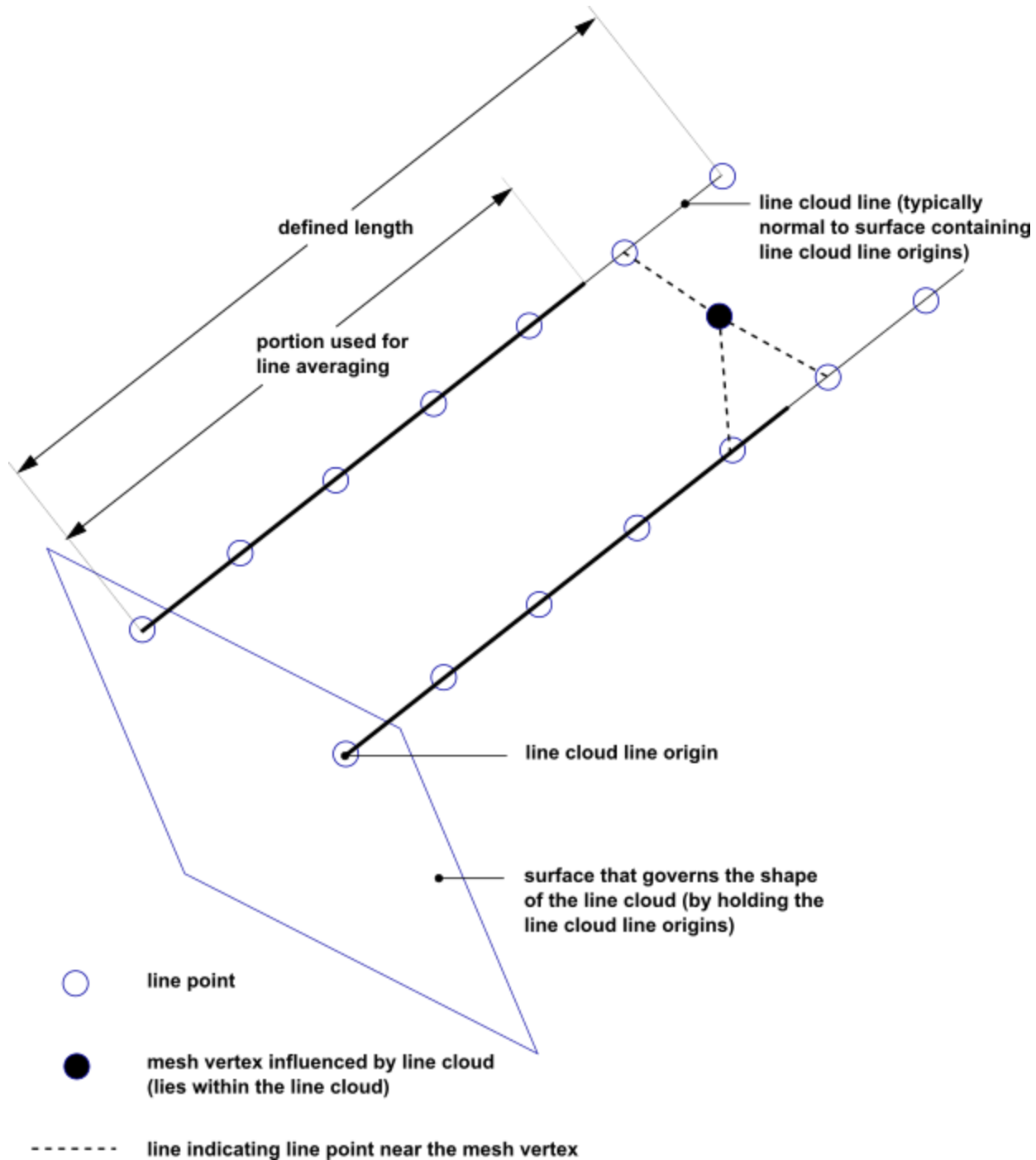
- Click **OK**.

An object named after the user line cloud is created and listed in the tree view under **Simulation** > **User Locations**.

The lines of a line cloud are generally perpendicular to the nearby boundary. Due to discretization error, some lines near boundary edges might not point along the local normal direction. If a boundary includes an edge, for example if two adjacent walls are in the same boundary, then the lines of the line cloud project into the domain using approximately the average of the face normals on either side of the edge.

[Figure 28.1: Line Cloud Illustration \(p. 383\)](#) shows two lines of a line cloud and a mesh vertex that is influenced by the three nearest line points.

Figure 28.1: Line Cloud Illustration



The process by which a user line cloud generates line averaged values can be summarized as follows:

1. Field values are interpolated to the line points.
2. For a given line from a user line cloud:
  - A piece-wise representation of the line data is constructed using the values at the line points.



- The line average is computed by integrating the applicable variable over a length, then dividing by that length, which is the applicable length as determined by **Line Averaging Distance Fraction**.
- The line average value is assigned to the line.

3. For a given mesh vertex:

Nearby line points are used to identify lines in the line cloud. A selection of these lines have their (line average) values interpolated to the mesh vertex.

The resulting mesh vertex values, which represent line averaged values, are accessible in objects that reference the user line cloud.

To visualize a user line cloud, create a location-based monitor that uses the user line cloud, then, after running the simulation, postprocess the resulting `.CSV` file using CFD-Post.

---

**Note:**

- User line clouds are not compatible with cases involving:
  - Mesh deformation
  - Mesh adaption
  - Harmonic Analysis
  - Time Transformation
- User line clouds are created at the start of a solver run and are not updated to take account of any changes in the model topology or mesh during the simulation. For example, if a domain interface uses a conditional connection ([Conditional Connections in the CFX-Solver Modeling Guide](#)) then the lines are unaffected by the closing of the GGI interface.
- User line cloud lines terminate on mesh boundaries; line points outside the mesh are effectively deleted.

---

## 28.4. User Point

---

You can define a user point by specifying either Cartesian ( $x, y, z$ ) or cylindrical (Axial, Radial, Theta) coordinates. The coordinates must be specified relative to the global coordinate frame, `Coord 0`.

### Creating a User Point

1. Select **Insert > User Locations > User Point**.

The **Insert User Point** dialog box appears.

2. Set **Name** to a unique name for the user point and click **OK**.

The details view for the user point appears.

3. You can specify the coordinates of the point using your choice of coordinate system:
  - A Cartesian (x, y, z) coordinate system.  
The coordinates must be specified relative to the global coordinate frame, `Coord 0`.
  - A Cylindrical (Axial, Radial, Theta) coordinate system.  
The coordinates must be specified relative to the global coordinate frame, `Coord 0`.
  - A coordinate system defined by a profile function.  
The point is specified in custom coordinates (determined by the reference profile function; see [Coordinate System Definition \(p. 426\)](#)).
4. In the details view, optionally select **Visibility** to view the point in the graphics window.
5. Click **OK**.  
An object named after the user point is created and listed in the tree view under `Simulation > User Locations`.

## 28.5. User Point Cloud

---

You can define a user point cloud from a `.csv` file, for example one that was exported from CFD-Post. The user point cloud format is the same as the user line format (that is, the polyline data format) except that line connectivity data should not be present; connectivity data will be ignored if it is present. For details on the export of polyline data from CFD-Post, see [POLYLINE Data Format in the CFD-Post User's Guide](#).

The point cloud must contain at least one point.

### Creating a User Point Cloud

1. Select **Insert > User Locations > User Point Cloud**.  
The **Insert User Point Cloud** dialog box appears.
2. Set **Name** to a unique name for the user point cloud and click **OK**.  
The details view for the user point cloud appears.
3. Select a **Data Storage** option:
  - Use External File

Data is defined by the file you specify under **File Name**. The CFX-Solver will use the data stored in that file.

---

**Note:**

When submitting a run to a cluster or running on a different machine, the path to the `.csv` file must be valid on that cluster/machine.

---

- Internal

Data is read from the file that you select after clicking **Copy Data Into Setup**. This data is copied into the case definition when you click **Apply** or **OK**. The CFX-Solver will use the data stored with the case definition.

Note that, after you click **Copy Data Into Setup**, the **Copy Data Into Setup** dialog box appears, where you specify the data file to read. If the specified file has an incorrect format, an error message will appear; after making corrections to such a file, click **Reload** to reassess the format of the file.

4. In the details view, optionally select **Visibility** to view the point cloud in the graphics window.

---

**Note:**

The displayed point cloud will not be automatically updated if the user point cloud input file is subsequently modified, even when using the `Use External File` data storage option.

---

5. Click **OK**.

An object named after the user point cloud is created and listed in the tree view under `Simulation > User Locations`.

---

# Chapter 29: Materials and Reactions

---

The tree view contains a `Materials` object and a `Reactions` object, which contain all the currently available materials and reactions.

A **Material** details view is available for editing the properties of new or existing fluids, solids, and mixtures. The new or modified materials can then be selected for use in your simulation or reaction definitions. For details, see [Materials \(p. 387\)](#).

A **Reaction** details view is available for editing the properties of new or existing reactions. For details, see [Reactions \(p. 401\)](#).

---

## Note:

You can set only those material properties that will be used in the CFD model. For example, you can set the buoyancy properties only if your model involves buoyant flow.

You can use CEL to define fluid property variation through an expression if it is required. For example, you could define `Dynamic Viscosity` to be a function of `Temperature`.

---

## 29.1. Materials

---

The **Material** details view, accessible by editing a material from the tree view or by creating a new material, is used to prepare materials for availability in a simulation.


The following topics will be discussed:

- [29.1.1. Library Materials](#)
- [29.1.2. Material Details View: Common Settings](#)
- [29.1.3. Material Details View: Pure Substance](#)
- [29.1.4. Material Details View: Fixed Composition Mixture](#)
- [29.1.5. Material Details View: Variable Composition Mixture](#)
- [29.1.6. Material Details View: Homogeneous Binary Mixture](#)
- [29.1.7. Material Details View: Reacting Mixture](#)
- [29.1.8. Material Details View: Hydrocarbon Fuel](#)

### 29.1.1. Library Materials

CFX-Pre provides an extensive list of library materials. Properties for these have already been defined and are known to CFX-Pre. If you modify a library material during a simulation using the **Material** details view, the modified definition is stored in the simulation file and is therefore local to your simulation.

On the **Outline** tab, right-click `Materials` and select **Import Library Data** to open the **Select Library Data to Import** dialog box. From here, you can select a material to load.

If necessary, you can open the **File to Import** dialog box by clicking *Browse* . The dialog box will open with the default location: `<CFXROOT>/etc/materials-extra/`. This directory contains CCL files that can be used to load additional materials into CFX-Pre (for example, Redlich Kwong, IAPWS, or Interphase Mass Transfer materials).

After importing material(s) from a library, you should inspect the material property data to ensure that it is adequate for the specific application.

If you want to use a material defined in one simulation in another simulation, the recommended method is to use the Export and Import CCL features to load the material definition from a local file. This is done by exporting CCL objects of type `LIBRARY:LIBRARY/MATERIAL`. For details, see [Import CCL Command \(p. 70\)](#) and [Export CCL Command \(p. 71\)](#).

## 29.1.2. Material Details View: Common Settings

### 29.1.2.1. Option

Any material can consist of one or more materials. If a material contains only a single pure species, then it is known as a pure substance. If it contains more than one species, then it is known as a mixture. The materials are assumed to be mixed at the molecular level in the mixture.

The type of material is set using the following options:

- The `Pure Substance` option should be used to create a fluid whose properties, such as viscosity, density, or molar mass, are known. All existing and newly created pure substances appear in the materials list and you can then create mixtures from them. For details, see [Material Details View: Pure Substance \(p. 391\)](#).
- The `Fixed Composition Mixture` option should be used to create a mixture with fixed mass fractions of each material. The mass fraction of each material is specified and is not allowed to change during the course of the simulation in space or time. For details, see [Material Details View: Fixed Composition Mixture \(p. 395\)](#).
- The `Variable Composition Mixture` option should be used to create a mixture whose mass fractions are allowed to change during the course of a simulation in space and time. The mass fraction of each material is not specified when defining the fluid. You can use a fixed composition mixture as a material in a variable composition mixture.

For details, see [Material Details View: Variable Composition Mixture \(p. 396\)](#).

- The `Homogeneous Binary Mixture` option applies to equilibrium phase change calculations. For details, see [Material Details View: Homogeneous Binary Mixture \(p. 398\)](#).
- The `Reacting Mixture` option is used for a chemical reaction, such as combustion.

For details, see [Material Details View: Reacting Mixture \(p. 399\)](#).

- The `Hydrocarbon Fuel` option. For details, see [Material Details View: Hydrocarbon Fuel \(p. 400\)](#).

## 29.1.2.2. Material Group

The **Material Group** filter is used to group materials by type, as well as restrict what materials can be mixed when the physical models include reactions or phase change. A material can be a member of more than one material group if it has a consistent set of properties. **Material Group** will always be set to at least one of the following:

### 29.1.2.2.1. User

Any user-defined materials, not assigned to one of the other groups, are shown in or can be added to this group. For example, materials loaded from a previous CFX-Pre simulation are shown in this group.

### 29.1.2.2.2. Air Data

This group contains Ideal Gas and constant property air. Constant properties are for dry air at both 0 [C], 1 [atm] (STP) and 25 [C], 1 [atm].

### 29.1.2.2.3. CHT Solids

Contains solid substances that can be used for solid domains when performing conjugate heat transfer modeling.

### 29.1.2.2.4. Calorically Perfect Ideal Gases

Contains gases that obey the Ideal Gas Law.

### 29.1.2.2.5. Constant Property Gases / Liquids

These groups contain gas and liquid substances with constant properties.

The gas properties are calculated at STP (0 [C] and 1 [atm]). Gas materials in this group can be combined with NASA SP-273 materials for use in combustion modeling simulations.

### 29.1.2.2.6. Dry/Wet Redlich Kwong

No materials appear in this group by default, they must be loaded from a pre-supplied materials file. All materials in this group use the built-in Redlich-Kwong equation of state and are suitable for performing equilibrium, homogeneous, phase change modeling.

For any given pure substance, there are three different materials. There is a material with a **RK** tag, used for dry vapor calculations, and three materials with **RKv**, **RKl** and **RKlv** suffixes, which are used for equilibrium phase change (wet vapor) calculations.

### 29.1.2.2.7. Dry/Wet Redlich Kwong RGP

No materials appear in this group by default, they must also be loaded from a pre-supplied materials file. All materials in this group use the Redlich-Kwong equation of state with properties specified in a TASCflow RGP file. These materials are suitable for performing equilibrium, homogeneous, phase change modeling.

Like the built-in Redlich Kwong group, for any given pure substance there are three different materials. There is a material with a **RK** tag, used for dry vapor calculations, and three materials

with  $RK_v$ ,  $RK_l$  and  $RK_{lv}$  suffixes, which are used for equilibrium phase change (wet vapor) calculations.

#### **29.1.2.2.8. Dry/Wet Peng Robinson RGP**

No materials appear in this group by default, they must also be loaded from a pre-supplied materials file. All materials in this group use the Redlich-Kwong equation of state with properties specified in a TASCflow RGP file. These materials are suitable for performing equilibrium, homogeneous, phase change modeling.

Like the built-in Redlich Kwong group, for any given pure substance there are three different materials. There is a material with a  $RK$  tag, used for dry vapor calculations, and three materials with  $RK_v$ ,  $RK_l$  and  $RK_{lv}$  suffixes, which are used for equilibrium phase change (wet vapor) calculations.

#### **29.1.2.2.9. Dry/Wet Soave Redlich Kwong RGP**

No materials appear in this group by default, they must also be loaded from a pre-supplied materials file. All materials in this group use the Redlich-Kwong equation of state with properties specified in a TASCflow RGP file. These materials are suitable for performing equilibrium, homogeneous, phase change modeling.

Like the built-in Redlich Kwong group, for any given pure substance there are three different materials. There is a material with a  $RK$  tag, used for dry vapor calculations, and three materials with  $RK_v$ ,  $RK_l$  and  $RK_{lv}$  suffixes, which are used for equilibrium phase change (wet vapor) calculations.

#### **29.1.2.2.10. Dry/Wet Steam**

No materials appear in this group by default, they must also be loaded from a pre-supplied materials file. Materials in this group use the IAPWS equation of state. Again, the materials are suitable for either dry or wet steam calculations.

#### **29.1.2.2.11. Gas Phase Combustion**

Contains materials that can be used for gas phase combustion. All materials in this group use the Ideal Gas equation of state. The specific heat capacity, enthalpy and entropy for each of the materials are specified using the NASA SP-273 format. For details, see [NASA Format in the CFX-Solver Modeling Guide](#).

#### **29.1.2.2.12. Interphase Mass Transfer**

This group contains materials that can be used for Eulerian or Lagrangian interphase mass transfer. This group currently contains a number of materials that have liquid or gas reference states, which are consistent for performing phase change calculations. The gas phases use the ideal gas equation of state and temperature-dependent specific heat capacity. The associated liquid phases use constant properties.

#### **29.1.2.2.13. Particle Solids**

Contains a list of solids that can be used in Particle Tracking calculations.


#### 29.1.2.2.14. Soot

This group contains solid substances that can be used when performing soot calculations.

#### 29.1.2.2.15. Water Data

This group contains liquid and vapor water materials with constant properties. The materials in this group can be combined with NASA SP-273 materials for use in combustion modeling simulations.

### 29.1.2.3. Material Description

This parameter can be toggled on to view a detailed description of the substance. Click *Edit the Material Description*  to edit the description (to a maximum of 120 alphanumeric characters).

### 29.1.2.4. Thermodynamic State

This parameter sets the state of a substance to solid, liquid or gas. There are certain limitations imposed by selecting a particular state. For example, a solid must always have at least density, specific heat capacity and thermal conductivity specified.

### 29.1.2.5. Coordinate Frame

For material properties that are set using expressions containing X, Y, or Z, you may want to supply a custom coordinate frame as the basis for evaluation of such properties. For details, see [Coordinate Frame \(p. 391\)](#), [Coordinate Frames \(p. 371\)](#), and [Coordinate Frames in the CFX-Solver Modeling Guide](#).

## 29.1.3. Material Details View: Pure Substance

The following topics will be discussed:

- [Basic Settings Tab \(p. 391\)](#)
- [Material Properties Tab \(p. 392\)](#)

### 29.1.3.1. Basic Settings Tab

The **Basic Settings** tab is used to set the type of material, its state, an optional description and an optional coordinate frame.

1. Set the **Material Group**.  
For details, see [Material Group \(p. 389\)](#).
2. The **Material Description** field is optional.  
For details, see [Material Description \(p. 391\)](#).
3. Select the **Thermodynamic State**.  
For details, see [Thermodynamic State \(p. 391\)](#).



4. Optionally set a custom coordinate frame for any material properties that depend on expressions in X, Y, or Z.

For details, see [Coordinate Frame \(p. 391\)](#), [Coordinate Frames \(p. 371\)](#), and [Coordinate Frames in the CFX-Solver Modeling Guide](#).

### 29.1.3.2. Material Properties Tab

There are two main categories specifying properties of a pure substance: `General Material` and `Table`. A `General Material` can have its thermodynamic, transport and radiation properties defined in the most general manner using any of the built-in flow solver models, constants, or CEL expressions. A table material uses a TASCflow RGP file to look up the required values. For details, see [Table \(p. 394\)](#).

#### 29.1.3.2.1. General Material

**General Materials** can have their **Equation of State** set to the following options:

- [Equation of State - Value \(p. 392\)](#).
- [Equation of State - Ideal Gas \(p. 393\)](#).
- [Equation of State - Real Gas \(p. 393\)](#).

For details on equations of state, see [Equation of State in the CFX-Solver Modeling Guide](#).

##### 29.1.3.2.1.1. Equation of State - Value

The following tab appears when **Equation of State** is set to `Value`. `Value` uses whichever model for density that is supplied by the user. For example, the equation of state model could be a constant or a CEL expression.

- **Equation of State** > **Option**. For details, see [Option in the CFX-Solver Modeling Guide](#).
- Specify the **Density** and **Molar Mass**. An expression can be used for **Density** that depends on temperature and/or pressure. In this case, the CFX-Solver may build property tables in order to calculate enthalpy and entropy. If you use this option, check the table generation settings.

Additional information on Material Properties is available in:

- [Specific Heat Capacity in the CFX-Solver Modeling Guide](#)
- [Transport Properties in the CFX-Solver Modeling Guide](#)
- [Radiation Properties in the CFX-Solver Modeling Guide](#)
- [Buoyancy Properties in the CFX-Solver Modeling Guide](#)
- [Electromagnetic Properties in the CFX-Solver Modeling Guide](#).

### 29.1.3.2.1.2. Equation of State - Ideal Gas

- If you set the specific heat capacity using a CEL expression, the solver will build tables for enthalpy and entropy. If you use this option, check the table generation settings.
- For an ideal gas, specify the **Molar Mass**. For details, see [Molar Mass in the CFX-Solver Modeling Guide](#).

Additional information on ideal gas is available in:


- [Ideal Gas in the CFX-Solver Modeling Guide](#)
- [Reference State Properties in the CFX-Solver Modeling Guide](#)
- [Transport Properties in the CFX-Solver Modeling Guide](#).

### 29.1.3.2.1.3. Equation of State - Real Gas

The `Real Gas` option can be specified to model non-ideal gases and some liquid phase properties. Set **Model** to one of the following:

- Aungier Redlich Kwong (the default model)
- Peng Robinson
- Soave Redlich Kwong
- Standard Redlich Kwong.

To load the Real Gas materials into CFX-Pre:

1. On the **Outline** tab, right-click `Materials` and select **Import Library Data**.
2. In the **Select Library Data to Import** dialog box, click *Browse*  and open the `MATERIALS-redkw.ccl`, `MATERIALS-sredkw.ccl`, or `MATERIALS-pengrob.ccl` file, which contain pre-defined real gas materials model.
3. Select the group of materials to load and click **OK**.
4. Complete all of the data fields on the **Materials Properties** tab to use a Real Gas equation of state, then click **OK**.


Additional information on Real Gas models is available in:

- [Real Gas in the CFX-Solver Modeling Guide](#)
- [Specific Heat Capacity in the CFX-Solver Modeling Guide](#)
- [Transport Properties in the CFX-Solver Modeling Guide](#).
- [Real Gas Properties in the CFX-Solver Theory Guide](#)

### 29.1.3.2.2. Table

**Table** uses a CFX-TASCflow real gas property (RGP) file to load real fluid property data (see [Real Fluid Properties in the CFX-Solver Modeling Guide](#)). You can load all of the RGP files that are supplied with CFX quickly by following the instructions given in [Loading an .rgp file in the CFX-Solver Modeling Guide](#). When defining materials that use data in tables not supplied with CFX, the definition is carried out separately by specifying the filename and component name for each material in turn. When **Table** is selected, the following form appears:

TASCflow RGP file **Table Format** is the only type supported for CFX.

1. Click *Browse*  beside **Table Name** to browse to the file containing the Real Gas Property Table data.
2. Enter the **Component Name** (as an RGP file can contain many components).

The component name corresponds to the name of a component in an RGP file. You may need to open the RGP file in a text editor to discover the exact name of the component you want to select. For details, see [Detailed .rgp File Format in the CFX-Solver Modeling Guide](#).

### 29.1.3.2.3. Table Generation

For some equation of state and specific heat capacity settings (such as Redlich Kwong, IAPWS, and general materials having variable density and specific heat set with CEL expressions), the CFX-Solver builds internal property tables for efficient property evaluation. The most commonly required table is enthalpy as a function of temperature and possibly pressure. This table is built if the specific heat capacity is a function of temperature, and, possibly pressure. Entropy tables are also used to convert static and total pressure (or vice versa). For example, at a boundary condition you may specify the total pressure and the flow solver will use entropy tables to calculate the static pressure. When using CEL expressions for density and specific heat capacity the solver uses an adaptive algorithm to control the generation of the tables. In some cases, it may be necessary to alter some table generation details, as described by the following parameters:

#### 29.1.3.2.3.1. Minimum and Maximum Temperature

These correspond to the lower and upper temperature bounds of the table. The selected values should exceed the expected temperature range somewhat, but to keep the size of the table from becoming too big, it should not exceed the expected range by a factor much greater than 2.

#### 29.1.3.2.3.2. Minimum and Maximum Absolute Pressure

These correspond to the lower and upper absolute pressure bounds of the table. As with the temperature bounds, the selected values should exceed the expected absolute pressure range, but not by too much.

#### 29.1.3.2.3.3. Error Tolerance

The table generation algorithm used by the solver is adaptive, and may cluster values where needed to resolve nonlinearities in the property definitions. The table generation is required to satisfy an error tolerance, defined as the relative error between the interpolation error and the exact value. The default tolerance (0.01 for enthalpy and 0.03 for entropy) is usually adequate.

#### 29.1.3.2.3.4. Maximum Points

This parameter specifies the maximum number of points (values) for each table dimension. Fewer points may be required if the error criterion is met sooner. The default value of 100 is usually adequate.

---

**Note:**

If the error tolerance cannot be met with the specified maximum number of points, the CFX-Solver will revert to a uniform table with a resolution set to the maximum number of points.

---

#### 29.1.3.2.3.5. Pressure/Temperature Extrapolation

This controls the solver behavior when evaluating properties at temperatures or pressures beyond the table range. If extrapolation is activated, the property will be extrapolated based on its slope at the table boundary; otherwise, the value at the table boundary will be used. In either case, a message is written to the CFX-Solver Output file that an out-of-bounds has occurred. If this happens, you should consider increasing the table range accordingly.

### 29.1.4. Material Details View: Fixed Composition Mixture

This panel describes the **Material** details view when creating a fixed composition mixture. Fixed composition mixtures can consist of pure substances only and not other fixed composition mixtures. You can include any combination of materials in a fixed composition mixture. To combine materials from different material groups, however, you must first select the **Material Groups** that contain those materials. For example, to select `Air at 25 C` and `Water at 25 C`, you would first need to select the groups `Air Data` and `Water Data`.

For details, see [Multicomponent Flow in the CFX-Solver Modeling Guide](#).

#### 29.1.4.1. Basic Settings Tab

1. Select the **Material Group**(s) that contain the required materials.
2. Use **Materials List** to add new materials to the mixture.
3. The **Material Description** field is optional.

For details, see [Material Description \(p. 391\)](#).

4. Select the **Thermodynamic State**.

For details, see [Thermodynamic State \(p. 391\)](#).

5. Optionally set a custom coordinate frame for any material properties that depend on expressions in X, Y or Z.

For details, see [Coordinate Frame \(p. 391\)](#), [Coordinate Frames \(p. 371\)](#), and [Coordinate Frames in the CFX-Solver Modeling Guide](#).

6. Select a material, from the **Child Materials** list box.

7. Enter the fixed **Mass Fraction** of the material within the mixture.

The sum of the mass fractions for all the materials in a mixture **must be 1**.

### 29.1.4.2. Mixture Properties Tab

**Mixture Properties** can be used to explicitly set values when the **Ideal Mixture** model produces unsatisfactory results. When you create a fixed composition mixture, the mixture properties are determined by mass averaging the properties of the component materials. In some cases, the ideal mixture rule used by the CFX-Solver may not be representative of the mixture properties. You can customize the individual thermodynamic and transport properties by selecting the appropriate check boxes and directly specifying the mixture properties.

For details, see [Mixture Properties \(Fixed, Variable, Reacting\)](#) in the *CFX-Solver Modeling Guide*.

For additional information on options for specific heat capacity, see [Specific Heat Capacity](#) in the *CFX-Solver Modeling Guide*.

### 29.1.5. Material Details View: Variable Composition Mixture

This panel describes the **Material** details view when creating a variable composition mixture. Components of a variable composition mixture can be pure substances and fixed composition mixtures. For details, see [Multicomponent Flow](#) in the *CFX-Solver Modeling Guide*.

#### 29.1.5.1. Basic Settings Tab

1. Select the **Material Group(s)** that contain the required materials.
2. Use **Materials List** to add new materials to the mixture.
3. The **Material Description** field is optional.

For details, see [Material Description](#) (p. 391).

4. Select the **Thermodynamic State**.

For details, see [Thermodynamic State](#) (p. 391).

5. Optionally set a custom coordinate frame for any material properties that depend on expressions in X, Y or Z.

For details, see [Coordinate Frame](#) (p. 391), [Coordinate Frames](#) (p. 371), and [Coordinate Frames in the CFX-Solver Modeling Guide](#).

#### 29.1.5.2. Mixture Properties Tab

In some cases, the ideal mixture rule used by the CFX-Solver may not be representative of the mixture properties.

When you create a variable composition mixture (and also when you create a reacting mixture), the thermodynamic mixture properties can be determined by:

- Mass averaging the properties of the component materials (by choosing the corresponding `Ideal Mixture` option), or
- Applying custom functions based on the properties and mass fractions of the component materials (by choosing options other than `Ideal Mixture`).

The **Thermodynamic Properties** settings include:

- **Equation of State**, with these options:

- `Ideal Mixture` - This option uses mass averaging of the properties of the component materials.

- `Mixture Density` - You specify values and/or CEL expressions for:

- Density

- The partial derivative of density with respect to pressure (at constant temperature), and

- The partial derivative of density with respect to temperature (at constant pressure).

These quantities can be functions of component mass fractions and component properties (such as density).

- `Mixture Pressure` - You specify values and/or CEL expressions for:

- Absolute pressure

- The partial derivative of absolute pressure with respect to specific volume (at constant temperature), and

- The partial derivative of absolute pressure with respect to temperature (at constant specific volume).

These quantities can be functions of component mass fractions and component properties.

- **Specific Heat Capacity**, with these options:

- `Ideal Mixture`

- `Mixture Specific Heat Capacity` - You specify the type of heat capacity (as being based on constant volume or constant pressure). You also specify values and/or CEL expressions for:

- Specific heat capacity

- Specific enthalpy

- Specific entropy

These quantities can be functions of component mass fractions and component properties (such as specific heat capacity, static enthalpy, and static entropy, respectively).

When entering expressions for the **Thermodynamic Properties** settings, terms that involve component properties should adhere to the general variable syntax described in [Quantitative CEL Functions in Ansys CFX](#).

You can override the individual transport properties by selecting the appropriate options and directly specifying the mixture properties.

For details, see [Mixture Properties \(Fixed, Variable, Reacting\)](#) in the *CFX-Solver Modeling Guide*.

For additional information on options for specific heat capacity, see [Specific Heat Capacity in the CFX-Solver Modeling Guide](#).

## 29.1.6. Material Details View: Homogeneous Binary Mixture

Homogeneous binary mixtures are used to define the phase boundary between two chemically equivalent materials in different thermodynamic states. For example, you could define the vapor pressure curve between water and steam. The vapor pressure curve is used by the flow solver to determine the saturation properties of the two materials. A homogeneous binary mixture is required for running the Equilibrium phase change model. Additionally, it can be used with the Eulerian multiphase thermal phase change model or the Lagrangian particle tracking evaporation model.

The **Basic Settings** tab is used to specify the two materials that form the mixture. On the **Saturation Properties** tab, the saturation properties can be specified.

### 29.1.6.1. Basic Settings Tab

1. Select the **Material Group(s)**.
2. Select the two constituent materials for the binary mixture.
3. The **Material Description** field is optional.

For details, see [Material Description](#) (p. 391).

### 29.1.6.2. Saturation Properties Tab

#### 29.1.6.2.1. General

The `General` option can be used to specify the saturation temperature or Antoine coefficients for materials that do not use a table or Redlich Kwong equation of state. If you set **Pressure > Option** to `Antoine Equation` option, then the flow solver automatically calculates saturation temperature. For details, see [Antoine Equation in the CFX-Solver Modeling Guide](#). If you set **Pressure > Option** to `Value`, you must specify the saturation pressure and the corresponding saturation temperature. For details, see [Using a General Set-up in the CFX-Solver Modeling Guide](#).

#### 29.1.6.2.2. Table

Files of type (\* . rgp) are filtered from the list of files in the current directory.

### 29.1.6.2.3. Real Gas

When **Real Gas** is chosen, the saturation properties are calculated using the material properties specified for the constituent components, and there is no need to set any values. As a consequence, the material properties for components in the mixture must all use the same Real Gas equation of state. For details, see [Using a Real Gas Equation of State in the CFX-Solver Modeling Guide](#).

### 29.1.6.2.4. Table Generation

For details, see [Table Generation \(p. 394\)](#).

## 29.1.7. Material Details View: Reacting Mixture

The following topics will be discussed:

- [Basic Settings Tab \(p. 399\)](#)
- [Mixture Properties Tab \(p. 399\)](#)

### 29.1.7.1. Basic Settings Tab

A reacting mixture contains at least one reaction. For details, see [Reactions \(p. 401\)](#). The details for each of the components are set under **Component Details** on the **Fluid Models** tab when defining your domain. For details, see [Fluid Models Tab \(p. 175\)](#).

1. Select the **Material Group** types.
2. Select the reactions from the **Reactions List**.
3. The **Material Description** field is optional.

For details, see [Material Description \(p. 391\)](#).

4. Select the **Thermodynamic State**.

For details, see [Thermodynamic State \(p. 391\)](#).

5. Optionally set a custom coordinate frame for any material properties that depend on expressions in X, Y, or Z.

For details, see [Coordinate Frame \(p. 391\)](#), [Coordinate Frames \(p. 371\)](#), and [Coordinate Frames in the CFX-Solver Modeling Guide](#).

6. From **Additional Materials List**, select any additional inert materials (which do not take part in any reaction).

### 29.1.7.2. Mixture Properties Tab

Mixture properties for reacting mixtures are the same as for variable composition mixtures. For details, see [Mixture Properties Tab \(p. 396\)](#).

For details on modeling reacting mixtures, see [Using Combustion Models in the CFX-Solver Modeling Guide](#).



## 29.1.8. Material Details View: Hydrocarbon Fuel

The following topics will be discussed:

- [Basic Settings Tab \(p. 400\)](#)
- [Proximate/ Ultimate Analysis Tab \(p. 400\)](#)
- [Mixture Materials Tab \(p. 400\)](#)

More information about hydrocarbon fuel models is available in [Hydrocarbon Fuel Model Setup in the CFX-Solver Modeling Guide](#) and in [Hydrocarbon Fuel Analysis Model in the CFX-Solver Theory Guide](#).

### 29.1.8.1. Basic Settings Tab

This tab is identical to the Basic Settings tab for pure substances. For details, see [Basic Settings Tab \(p. 391\)](#).

### 29.1.8.2. Proximate/ Ultimate Analysis Tab

For details, see [Hydrocarbon Fuel Analysis Model in the CFX-Solver Theory Guide](#).

### 29.1.8.3. Mixture Materials Tab

The material components in the model need to be mapped to solver alias names. For example, carbon dioxide could be represented in the solver by CO2, Carbon Dioxide CO2, CO2 modified, and so on.

**Particle Mixture** defines the components of the hydrocarbon fuel particles. The names for the ash, char and raw combustible component materials must be given.

**Gas Mixture** is for identifying the components of the gas-phase reacting mixture. Two options are available:

- `Mixture` asks for the name of the associated gas-phase material (reacting mixture) and provides parameters to identify the components of the gas phase, which are relevant for the hydrocarbon fuel model.
- `Mixture with HCN NO` additionally enables entering the names for the gas components involved in the fuel-nitrogen model.

Note that here the components of the gas phase are identified only for the hydrocarbon fuel model. The reacting mixture material still must be created with all its components in the same way as for gaseous combustion. It may have additional components in addition to those identified here.

**Binary Mixture** is for defining the homogeneous binary mixture material, which describes the heat transfer between the particle and the fluid mixture. For the two materials in the binary mixture you should specify the raw combustible material for the particle and the volatiles fuel for the gas phase.

## 29.2. Reactions

The **Reaction** details view, accessible by editing a reaction from the tree view or by creating a new reaction, is used to prepare reactions for availability in a simulation.

Once a reaction is created, it is available for inclusion in a fluid that is a reacting mixture or a variable composition mixture. For details, see:

- [Material Details View: Reacting Mixture \(p. 399\)](#)
- [Material Details View: Variable Composition Mixture \(p. 396\)](#)

### 29.2.1. Basic Settings Tab

Four types of reactions can be created on the **Basic Settings** tab:

- [Single Step \(p. 401\)](#)
- [Multiple Step \(p. 402\)](#)
- [Flamelet Library \(p. 403\)](#)
- [Multiphase \(p. 403\)](#)

### 29.2.2. Single Step

This option displays three other tabs in addition to **Basic Settings**. One of them is a **Reactants** tab, displaying a list of reactants and specifying the ratio with which they react together and the order of the reaction. A list of products is also set on the **Products** tab and includes the ratio with which they are produced. **Reaction Rates** has optional forward and backward reaction rates and third body terms can be applied.

#### 29.2.2.1. Single Step: Basic Settings

1. Optionally, select **Reaction Description** to enter a description for the reaction (to a maximum of 120 alphanumeric characters).
2. Optionally, specify any additional materials for this reaction using the **Additional Materials List**.
3. Optionally, select **Reaction or Combustion** to set a reaction or combustion model.

Any settings specified here will override the choice of models selected on the **Fluid Models** tab of the domains form (unless the choice of models on the **Fluid Models** form is set to None).

This is implemented to enable reaction-step specific combustion modeling for multi-step reactions. For details, see [Reaction-Step Specific Combustion Model Control in the CFX-Solver Modeling Guide](#).

#### 29.2.2.2. Single Step: Reactants

For details, see [Multiphase: Reactants \(p. 403\)](#).

1. **Option** assumes the value `Child Materials` when creating a reaction involving one phase.
2. Choose the reactant(s) to add from the **Materials List** drop-down.
3. Enter the **Stoichiometric Coefficient** for the each of the selected components.
4. Optionally, specify a reaction order.

If the reaction order is not entered, it will default to the same value as the stoichiometric coefficient.

### 29.2.2.3. Single Step: Products

The **Products** tab is identical to the **Reactants** tab, with the only difference being that the settings here apply to the products instead of the reactants.

### 29.2.2.4. Single Step: Reaction Rates

1. For each of **Forward Reaction Rate** and **Backward Reaction Rate**, **Option** defines the reaction rate dependency. Select from:
  - [Arrhenius in the CFX-Solver Modeling Guide](#)
  - [Arrhenius with Temperature PDF in the CFX-Solver Modeling Guide](#)
  - [Expression in the CFX-Solver Modeling Guide](#)
2. The **Pre Exponential Factor** and **Temperature Exponent** are required elements for the `Arrhenius` reaction type.
3. The temperature limit list (**Lower Temperature** and **Upper Temperature**) is required for the `Arrhenius with Temperature PDF` reaction type.
4. **Reaction Activation** enables `Activation Temperature` or `Activation Energy` to be set.
5. Some reactions require a **Third Body Term** to proceed.  
For details, see [Third Body Terms in the CFX-Solver Theory Guide](#).
6. You can define a reaction in terms of a dependency, equilibrium or an expression.  
For details, see [Reaction Rate Types in the CFX-Solver Modeling Guide](#).

### 29.2.3. Multiple Step

A list of **Single Step** reactions is required to define a **Multi Step** reaction.

1. Set **Option** to `Multi Step`.


Hold the **Ctrl** key to select multiple reactions from the list.

Alternatively, click *Select from a second list*  to open the **Materials List** list box.

2. Optionally, select **Reaction Description** to enter a description for the reaction.

### 29.2.4. Flamelet Library

A flamelet library is imported with optional customization of the `Laminar Burning Velocity`.

1. Set **Option** to `Flamelet Library`.
2. Click *Browse*  to browse to the flamelet library file. The file that contains your flamelet library should be selected.
3. Optionally, select **Reaction Description** to enter a description for the reaction.
4. Select **Laminar Burning Velocity** to specify an expression for the laminar flame speed definition.

When using a flamelet library, the definition for the library is specified in the **Reaction** details view. The name, library file and, optionally, laminar flame speed definition is specified. The reaction can then be used in a fluid that is a variable composition mixture. For details, see [Material Details View: Variable Composition Mixture](#) (p. 396). and [Laminar Flamelet with PDF Model in the CFX-Solver Modeling Guide](#).

### 29.2.5. Multiphase

This option is used to create reactions between more than one phase. For details, see [Reaction Models in the CFX-Solver Modeling Guide](#).

#### 29.2.5.1. Multiphase: Basic Settings

The setup of multiphase reactions is carried out by selecting the reaction **Option** to `Multiphase`. For details, see [Multiphase Reactions and Combustion in the CFX-Solver Modeling Guide](#).

1. For multiphase reactions the only option available for the **Material Amount Option** is `Mass Coefficient`.
2. Optionally, select **Reaction Description** to enter a description for the reaction.

#### 29.2.5.2. Multiphase: Reactants

Multiphase reactions are specified in terms of `Parent Materials` (the phase containing a reacting material), and `Child Materials` (the reacting materials themselves).

The **Parent Materials List** contains the phases from which reacting materials are selected.

1. For the currently selected parent material, (such as `Coal`), select the reactant materials from that phase from the materials list (for example, `Coal > Materials List`).

If a participating phase is a pure substance, it should be selected as both a parent and child material.

2. For each child material, enter a mass coefficient.
3. **Reaction Order** is only required for reactions of type `Mass Arrhenius`.  
If unset, it defaults to 1.

### 29.2.5.3. Multiphase: Products

The setup on the **Products** tab is identical to the **Reactants** tab. For details, see [Multiphase: Reactants](#) (p. 403).

### 29.2.5.4. Multiphase: Multiphase Reactions

The **Multiphase Reaction Rate > Option** can be one of:

- [Mass Arrhenius in the CFX-Solver Modeling Guide](#)
- [Gibb Char Oxidation Model in the CFX-Solver Modeling Guide](#)
- [Field Char Oxidation Model in the CFX-Solver Modeling Guide](#).

---

# Chapter 30: Additional Variables

---

Additional Variables are non-reacting scalar components that can be transported through the flow.

Modeling information for Additional Variables is available:

- [Additional Variables in the CFX-Solver Modeling Guide](#)
- [Additional Variables in Multiphase Flow in the CFX-Solver Modeling Guide](#)

Implementation information for Additional Variables in multiphase flow is available:

- [Additional Variables in the CFX-Solver Theory Guide](#)
- [Additional Variables in Multiphase Flow in the CFX-Solver Theory Guide](#)

This chapter describes the procedure for creating an Additional Variable and the user interfaces used to define and apply Additional Variables:

[30.1. User Interface](#)

[30.2. Creating an Additional Variable](#)

## 30.1. User Interface


---

The following topics are discussed:

- [Insert Additional Variable Dialog Box \(p. 405\)](#)
- [Basic Settings Tab for Additional Variable Objects \(p. 405\)](#)
- [Fluid Models and Fluid Specific Models Tabs for Domain Objects \(p. 406\)](#)
- [Boundary Details and Fluid Values Tabs for Boundary Condition Objects \(p. 409\)](#)

### 30.1.1. Insert Additional Variable Dialog Box

The **Insert Additional Variable** dialog box is used to initiate the creation of a new Additional Variable.

It is accessible by clicking the *Additional Variable* icon , or by selecting **Insert > Expressions, Functions and Variables > Additional Variable**.

### 30.1.2. Basic Settings Tab for Additional Variable Objects

The **Basic Settings** tab is used to define the fundamental properties of an Additional Variable. It is accessible by creating a new Additional Variable or by editing an Additional Variable listed in the tree view.

### 30.1.2.1. Variable Type

- `Specific`

The Additional Variable is solved on a per-unit-mass basis. For details, see [Volumetric and Specific Additional Variable in the CFX-Solver Modeling Guide](#).

- `Volumetric`

The Additional Variable is solved on a per-unit-volume basis. For details, see [Volumetric and Specific Additional Variable in the CFX-Solver Modeling Guide](#).

- `Unspecified`

The Additional Variable is defined in terms of an algebraic expression using CEL. For details, see [Unspecified Additional Variables in the CFX-Solver Modeling Guide](#).

### 30.1.2.2. Units

Specify the units that describe the Additional Variable. For details, see [Additional Variables in the CFX-Solver Modeling Guide](#).

### 30.1.2.3. Tensor Type

The Additional Variable's **Tensor Type** can be set to `Scalar` or `Vector`. If an Additional Variable is defined as type `Vector`, the components of a vector algebraic equation can be defined at the domain level.

Vector Additional Variables cannot be directly referenced in CEL expressions. The syntax for referencing a component of a vector Additional Variable is as follows:

```
<Component Name>.<Additional Variable Name>_x
```

## 30.1.3. Fluid Models and Fluid Specific Models Tabs for Domain Objects

The settings for Additional Variables on the **Fluid Models** tab are used to enable Additional Variables in the domain. For multiphase simulations, settings for unspecified and volumetric Additional Variables are available only on the **Fluid Specific Models** tab. For specific Additional Variables homogeneous transport equations can be set on the **Fluid Models** tab or on a per-fluid basis on the **Fluid Specific Models** tab if the Additional Variable has been set to `Fluid Dependent` on the **Fluid Models** tab.

### 30.1.3.1. Additional Variable Details: List Box

This list box is used to select an Additional Variable in order to set the details of its application to the domain.

### 30.1.3.2. Additional Variable Details: [Additional Variable Name] Check Box

This check box determines whether or not the Additional Variable is to be modeled in the domain.

### 30.1.3.2.1. Option

#### 30.1.3.2.1.1. Transport Equation

The transport of the Additional Variable of type `Volumetric` is modeled by a transport equation. For details, see [Additional Variables in the CFX-Solver Theory Guide](#).

#### 30.1.3.2.1.2. Diffusive Transport Equation

The transport of the Additional Variable is modeled by a transport equation. The advection term is dropped from the transport equation. For details, see [Additional Variables in the CFX-Solver Theory Guide](#).

#### 30.1.3.2.1.3. Homogeneous Transport Equation

The transport of the Additional Variable is modeled by a transport equation. This option is available only on the **Fluid Models** tab and only for multiphase flows (that is, only for homogeneous applications). For details, see:

- [Homogeneous Additional Variables in Multiphase Flow in the CFX-Solver Modeling Guide](#)
- [Homogeneous Additional Variables in Multiphase Flow in the CFX-Solver Theory Guide](#).

#### 30.1.3.2.1.4. Homogeneous Diffusive Transport Equation

The transport of the Additional Variable is modeled by a transport equation. The advection term is dropped from the transport equation. This option is available only on the **Fluid Models** tab and only for multiphase flows (that is, only for homogeneous applications). For details, see:

- [Homogeneous Additional Variables in Multiphase Flow in the CFX-Solver Modeling Guide](#)
- [Homogeneous Additional Variables in Multiphase Flow in the CFX-Solver Theory Guide](#).

#### 30.1.3.2.1.5. Poisson Equation

The transport of the Additional Variable is modeled by a transport equation. The transient and advection terms are dropped from the transport equation. For details, see [Additional Variables in the CFX-Solver Theory Guide](#).

#### 30.1.3.2.1.6. Homogeneous Poisson Equation

The transport of the Additional Variable is modeled by a transport equation. The transient and advection terms are dropped from the transport equation. This option is available only on the **Fluid Models** tab and only for multiphase flows (that is, only for homogeneous applications). For details, see:

- [Homogeneous Additional Variables in Multiphase Flow in the CFX-Solver Modeling Guide](#)
- [Homogeneous Additional Variables in Multiphase Flow in the CFX-Solver Theory Guide](#).

#### 30.1.3.2.1.7. Fluid Dependent

When the `Fluid Dependent` option is selected, the Additional Variable model details can be set for each fluid on the **Fluid Specific Models** tab.



### 30.1.3.2.1.8. Algebraic Equation

A given quantity or CEL expression specifies the value of the Additional Variable throughout the domain. Application of this option is, in the context of the fluids to which the Additional Variable is applied, effectively the same as setting the Additional Variable type to Unspecified.

### 30.1.3.2.1.9. Vector Algebraic Equation

A total of three given quantities, CEL expressions, or both, specifies the vector value of the Additional Variable throughout the domain. Application of this option is, in the context of the fluids to which the Additional Variable is applied, effectively the same as setting the Additional Variable type to Unspecified.

### 30.1.3.2.2. Value

(applies only when **Additional Variable Details: [Additional Variable name] Check Box: Option** is set to **Algebraic Equation**)

Enter a numerical quantity or CEL expression that specifies the value of the Additional Variable throughout the domain.

### 30.1.3.2.3. Kinematic Diffusivity Check Box

(applies only when **Additional Variable Details: [Additional Variable name] Check Box: Option** is set to **Transport Equation, Diffusive Transport Equation, or Poisson Equation**)

When running a **Transport Equation** Additional Variable, this check box determines whether the molecular diffusion term is added to the transport equation for the Additional Variable. For turbulent flow, the turbulent diffusion term (which is a consequence of averaging the advection term) is automatically included. Setting the kinematic diffusivity to zero includes the turbulent diffusion term only.

You must select this check box when using the Poisson equation or diffusive transport equation. If you do not, a blue warning message will appear to remind you.

### 30.1.3.2.4. Kinematic Diffusivity Check Box: Kinematic Diffusivity

(applies only when **Additional Variable Details: [Additional Variable name] check box: Option** is set to **Transport Equation, Poisson Equation, Diffusive Transport Equation**)

Enter a numerical quantity or CEL expression that specifies the value of the kinematic diffusivity throughout the domain.

### 30.1.3.2.5. AV Properties for Fluid: Frame Overview

(applies only for homogeneous Additional Variables)

The settings contained in this frame are used to optionally specify the kinematic diffusivity of the selected Additional Variable. The kinematic diffusivity may differ for each fluid in the domain. The solver calculates a single effective kinematic diffusivity based on the kinematic diffusivity of the Additional Variable in each fluid. For details, see [Homogeneous Additional Variables in Multiphase Flow in the CFX-Solver Theory Guide](#).

### 30.1.3.2.6. AV Properties for Fluid: List Box

This list box is used to select a fluid in the domain in order to optionally specify the kinematic diffusivity of the selected Additional Variable in that fluid.

### 30.1.3.2.7. AV Properties for Fluid: [Fluid Name] Check Box

This check box determines whether or not the kinematic diffusivity of the selected Additional Variable in the selected fluid is specified. Not specifying the kinematic diffusivity implies that the Additional Variable is non-diffusive.

### 30.1.3.2.8. AV Properties for Fluid: [Fluid Name] Check Box: Kinematic Diffusivity

(applies only when **Additional Variable Details: [Additional Variable name] Check Box: Option** is set to **Homogeneous Transport Equation, Homogeneous Diffusive Transport Equation, or Homogeneous Poisson Equation**)

Enter a numerical quantity or CEL expression that specifies the value of the kinematic diffusivity, throughout the domain, of the selected Additional Variable in the selected fluid.

### 30.1.3.2.9. Vector xValue, Vector yValue, and Vector zValue

(applies only when **Additional Variable Details: [Additional Variable Name] Check Box: Option** is set to **Vector Algebraic Equation**)

Enter a numerical quantity or CEL expression for each vector algebraic equation component.

## 30.1.4. Boundary Details and Fluid Values Tabs for Boundary Condition Objects

The **Boundary Details** and **Fluid Values** tabs for a boundary condition object contain settings that specify the values, fluxes, and transfer coefficients of Additional Variables at the boundary condition location. These tabs are accessible, when applicable, by editing a boundary condition object.

The Additional Variables that require specification are those that have been applied to the domain (to which the boundary condition belongs) in a form other than an algebraic equation.

For single phase flow, the Additional Variable settings are on the **Boundary Details** tab. For multiphase flow, the Additional Variable settings for homogeneous Additional Variables are on the **Boundary Details** tab and those for fluid-specific Additional Variables are either on the **Boundary Details** tab or the **Fluid Values** tab.

The types of boundary conditions that may allow the specification of Additional Variables are:

- Inlet
- Opening
- Wall
- Outlet

### 30.1.4.1. Additional Variables: List Box

This list box is used to select an Additional Variable in order to set the details of its boundary condition specification.

### 30.1.4.2. Additional Variables: [Name]

#### 30.1.4.2.1. Option

- Zero Flux
- Value
- Flux in

This option is applicable for Wall boundary conditions and, for Poisson and Diffusive transport models, Inlet boundary conditions.

- Transfer Coefficient
- Wall Flux In

This option is applicable for multiphase flow only.

- Wall Transfer Coefficient

This option is applicable for multiphase flow only.

#### 30.1.4.2.2. Value

(applies when **Additional Variables: [Additional Variable Name]: Option** is set to **Value** or **Transfer Coefficient**)

#### 30.1.4.2.3. Flux

(applies when **Additional Variables: [Additional Variable Name]: Option** is set to **Flux in**)

#### 30.1.4.2.4. Transfer Coefficient

(applies when **Additional Variables: [Additional Variable Name]: Option** is set to **Transfer Coefficient**)

## 30.2. Creating an Additional Variable

---

1. Click the *Additional Variable* icon  or select **Insert** > **Expressions, Functions and Variables** > **Additional Variable**.

The **Insert Additional Variable** dialog box appears.

2. Set **Name** to a unique name for the new Additional Variable. For details, see [Valid Syntax for Named Objects \(p. 95\)](#).

3. Click **OK**.

The **Additional Variable** details view opens on the **Basic Settings** tab.

4. Specify the basic settings. For details, see [Basic Settings Tab for Additional Variable Objects](#) (p. 405).
5. Click **OK**.

An object named after the Additional Variable is created and listed in the tree view under **Expressions, Functions and Variables > Additional Variables**.



---

# Chapter 31: Expressions

---

The **Expressions** workspace is used to generate and edit expressions using the CFX Expression Language (CEL), which you can then use in CFX-Pre in place of almost any numeric value (as long as the correct units are returned by the expression).

---

## Note:

In an expression, a term that has no units can be added to a term that has angular units, in which case the software internally applies radians to the term that has no units.

---

---

## Note:

CFX-Pre evaluates CEL expressions with single (not double) precision.

---

---

## Important:

There is some CEL that works in CFX-Pre and CFX-Solver, but not in CFD-Post. Any expression created in CFX-Pre and used as a Design Exploration output parameter could potentially cause fatal errors during the Design Exploration run, so you should create all expressions for Design Exploration output parameters in CFD-Post.

---

This chapter describes:

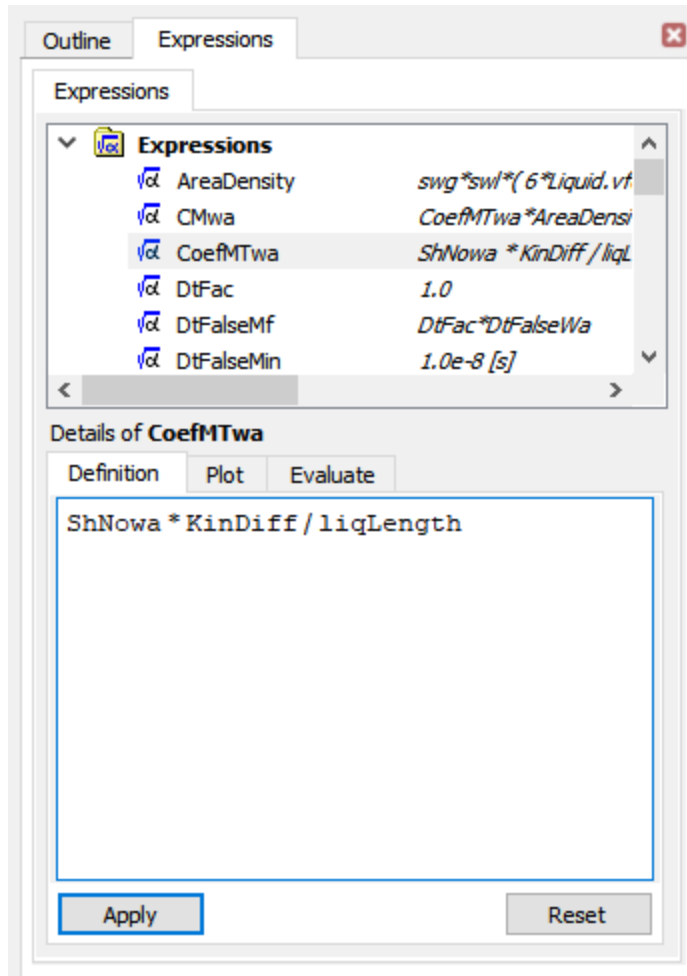
- 31.1. Expressions Workspace
- 31.2. Creating an Expression
- 31.3. Modifying an Expression
- 31.4. Importing or Exporting an Expression

## 31.1. Expressions Workspace

---

By double-clicking **Expressions** in the **Outline** workspace, or by inserting or editing an existing expression, the **Expressions** workspace opens in a new tab (see [Figure 31.1: Sample Expressions Workspace \(p. 414\)](#)). This workspace consists of a tree view and a details view. The following tabs are available in the details view:

- **Definition**, used to edit the definition of an expression selected in the **Expressions** tree view. For details, see [Definition \(p. 414\)](#).
- **Plot**, used to plot an expression versus a variable. For details, see [Plot \(p. 415\)](#).
- **Evaluate**, used to evaluate an expression when all quantities on which it depends are given. This is useful for verifying that an expression is correctly specified. For details, see [Evaluate \(p. 415\)](#).

**Figure 31.1: Sample Expressions Workspace**

### 31.1.1. Definition

CEL expressions can be defined using any combination of constants, variables, mathematical functions and other CEL expressions. For details, see [CFX Expression Language \(CEL\) in the CFX Reference Guide](#).

#### Tip:

Right-clicking in the **Definition** window provides access to a list of all available variables, expressions, locators, functions and constants. Although valid values can be chosen from each of the various lists, the validity of the expression itself is not checked until you click **Apply**.

Additional Variables can be used in expressions as soon as they have been completely specified. After they have been created, they appear in the list of available variables when right-clicking in the **Definition** window. For details, see:

- [CFX Expression Language \(CEL\) in the CFX Reference Guide](#)
- [CEL Operators, Constants, and Expressions in the CFX Reference Guide](#).

Click **Reset** to undo changes made after opening the CEL expression for editing.

### 31.1.2. Plot

The **Plot** tab is used to plot the selected expression against one variable. CFX-Pre automatically finds the variables associated with an expression, even if the expression depends on another expression.

For example, when previewing the expression `halfRadius`, defined as `0.5*radius`, where `radius` is an expression that depends on the variables `x` and `y`, CFX-Pre presents `x` and `y` as the variables upon which `halfRadius` depends.

1. Set up an expression in the **Definition** tab, or open an existing expression. Click the **Plot** tab.
2. Under **Number of Points**, set the number of sample data points for the plot.

Sample points are connected by line segments to approximate the functional relationship.

3. Under **Expression Variables**, select the independent variable.
4. Set the range for the independent variable.
5. Set **Fixed Value** for all of the remaining independent variables.
6. Click **Plot Expression** to view the resulting chart.

The **Plot Expression** button changes to **Define Plot**. This can be clicked after viewing the plot in order to make adjustments to the plot specification.

### 31.1.3. Evaluate


The **Evaluate** tab is used to evaluate an expression when all variables upon which the equation depends are specified. CFX-Pre automatically finds the variables associated with an expression, even if the expression depends on another expression.

For example, when previewing the expression `halfRadius`, defined as `0.5*radius`, where `radius` is an expression that depends on the variables `x` and `y`, CFX-Pre presents `x` and `y` as the variables upon which `halfRadius` depends.

1. Under **Expression Variables**, enter values for all listed variables.
2. Click **Evaluate Expression**.

The resulting expression is evaluated using the given variable values.

## 31.2. Creating an Expression

1. You can create an expression using any of the following methods:
  - On the **Outline** tab, right-click `Expressions` and select **Insert > Expression**.
  - Click *Expression*  in the main toolbar.



- Select **Insert > Expressions, Functions and Variables > Expression** from the menu bar.

Whichever method you choose, the **Insert Expression** dialog box appears.

2. Under **Name**, type a name for the new expression.
3. Click **OK**.
4. In the **Expressions** details view, under **Definition**, enter an expression. For details on using the **Definition** area, see [Definition \(p. 414\)](#).
5. Click **Apply**.
6. Optionally, select **Plot** or **Evaluate** to examine the expression.

---

**Note:**

In the **Expressions** details view, you can right-click and select **Insert > Expression** to add more expressions, but only if no existing expression is highlighted.

---

### 31.3. Modifying an Expression

---

1. In the **Expressions** tree view, double-click any expression, or right-click an expression and select **Edit**.

The **Expression** details view displays the definition of the expression.

2. Under **Definition**, modify the expression. For details on using the **Definition** area, see [Definition \(p. 414\)](#).
3. Make any desired changes and click **Apply**.
4. Optionally, select **Plot** or **Evaluate** to examine the expression.

### 31.4. Importing or Exporting an Expression

---

Expressions can be imported and exported in simulations. For details, see:

- [Import CCL Command \(p. 70\)](#)
- [Export CCL Command \(p. 71\)](#)

Any number of CCL objects can be exported; this section describes exporting only expressions to a file.

#### 31.4.1. Importing CCL Expressions

You can import expressions using the **Import CCL** function. For details, see [Import CCL Command \(p. 70\)](#).

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

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

Append imports expressions and overwrites any that currently exist in memory. Expressions that do not match ones being imported are not changed. Replace deletes all expressions in memory before importing.

3. Select a location from which to import.
4. Select a file to import.
5. Click **Open**.

---

**Important:**

Take care when importing CCL files because data can be overwritten.

---

### 31.4.2. Exporting CCL Expressions

You can export expressions using the **Export CCL** function. For details, see [Export CCL Command \(p. 71\)](#).

1. Select **File > Export > CCL** from the main menu bar.
2. Clear **Save All Objects**.
3. Under **Save All Objects**, select `LIBRARY > CEL > EXPRESSIONS`.
4. Select a location to which to export.
5. Enter a name for the exported file.
6. Click **Save**.



---

# Chapter 32: User Functions

---

User functions you create are listed under `Expressions, Functions and Variables > User Functions` in the **Outline** tree view.

There are several types of user functions:

- `Interpolation (Data Input)` — An interpolation function that you specify directly.

You can define a 1D set of points or a 3D cloud of points. This type of function supports the definition of one variable, with a value at each point.

- `Interpolation (From File)` — An interpolation function defined by data in a profile file.

Such functions are also known as *profile functions*. This type of function supports the definition of multiple variables, each having a value at each point.

- `Table` — An index-based function defined by data in a table file.

This type of function supports the definition of multiple variables, each having a value for each table row.

- `Functional Mockup Unit Instance` — A function that can return FMU input variables, FMU input parameters, or FMU output variables.
- `User Function` — A user CEL function driven by an existing user CEL routine object.
- `GT-SUITE Patch` — A function that supports boundary conditions that are interactive between CFX and GT-SUITE.
- `GT-SUITE Turboshaft` — A function that provides CEL access to the torque transferred to, and the shaft speed received from, the corresponding GT-SUITE turboshaft.

This chapter describes:

- 32.1. [Interpolation \(Data Input\) User Functions](#)
- 32.2. [Interpolation \(From File\) User Functions \(Profile Functions\)](#)
- 32.3. [Table User Functions](#)
- 32.4. [Functional Mockup Unit Instance](#)
- 32.5. [User CEL Functions](#)
- 32.6. [GT-SUITE Patch Functions](#)
- 32.7. [GT-SUITE Turboshaft Functions](#)

## 32.1. Interpolation (Data Input) User Functions

---

A user function with the option `Interpolation (Data Input)` is defined by a set of coordinates and variable values at those coordinates.

Example uses:

- Specify an inlet temperature profile using some experimentally-measured values of temperature at various points on the inlet.
- Specify some property of a material as a function of temperature using a table of known values.

If you have data in a profile data file, then instead of creating a user function directly, you can import the profile data as described in [Profile Boundary Conditions in the CFX-Solver Modeling Guide](#). More details can be found in [Interpolation \(From File\) User Functions \(Profile Functions\)](#) (p. 423).

The following topics are discussed:

[32.1.1. One Dimensional Interpolation](#)

[32.1.2. Three Dimensional Interpolation](#)

[32.1.3. Importing Data from a File](#)

[32.1.4. Using Interpolation \(Data Input\) Functions in CEL Expressions](#)

### 32.1.1. One Dimensional Interpolation

1D interpolation functions can be used to specify any quantity in CFX-Pre for which a standard CEL function (such as `sin`, `cos`, `step`, and so on) can be used. The function is created by interpolating from a list of points and a list of values at those points.

For a 1D interpolation using the `Interpolation (Data Input)` option, you should set a single coordinate value and a single value associated with the coordinate. The coordinate and the value are interpreted in the local coordinate frame, which will depend on where the function is used. For example, if the function is used to set a boundary condition value, the coordinate frame selected for that boundary condition will apply. For details, see [Coordinate Frames in the CFX-Solver Modeling Guide](#).

Data for 1D interpolation can be entered manually or read from a file. If the file contains data in the profile data format, then it can be imported as described in [Profile Boundary Conditions in the CFX-Solver Modeling Guide](#). Otherwise, numerical values for the 1D interpolation can be imported directly into CFX-Pre using the instructions below. Also see [Importing Data from a File](#) (p. 422) for more information on how to import the numerical values from a file.

1. Set **Option** to `Interpolation (Data Input)`.

Additional information on the other function type is available in:

- [Three Dimensional Interpolation](#) (p. 422)
- [User CEL Functions](#) (p. 434).

2. Under **Argument Units**, enter a single unit.

This will usually be a coordinate axis dimension (for example, m, cm, rad, and so on), but can be any dimension. A variable using these units is passed to the interpolation function (for example,  $x$ ,  $y$ ,  $r$ ,  $z$ , and so on) when setting a quantity in CFX-Pre.

3. Under **Result Units**, enter a single unit.

This unit should be a valid unit for the quantity you will be specifying (for example, type  $m \cdot s^{-1}$  for a velocity).

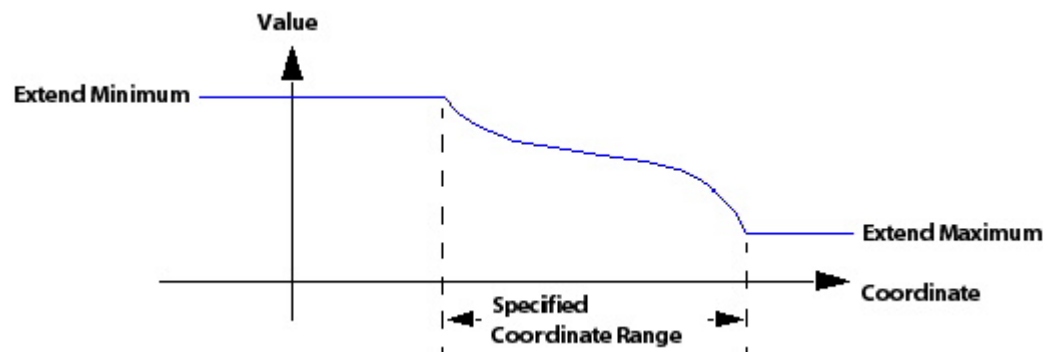
4. Set **Interpolation Data** > **Option** to One Dimensional.
5. Right-click in the window to import data from a file or delete an entry.
6. Enter a single **Coordinate** value.
7. Enter a **Value** associated with the **Coordinate**.
8. Click **Add** to add the point value to the list (or **Remove** to remove a highlighted value from the list).

For details on the Extend Min and Extend Max options, see [Extended Minimum and Maximum](#) (p. 421).

The coordinate axis to which the coordinate value relates is determined by the argument passed when calling the interpolation function. For example, for a Cartesian Velocity Component specified inlet, the U component could be set to the expression `MyInterpFunction(r)`, where `MyInterpFunction` is the function name of the 1D interpolation function, and `r` is the CFX radius system variable. The coordinate values you specify in this details view will then refer to values of `r` on the inlet boundary, and the value would be the velocity value at each `r` location.

### 32.1.1.1. Extended Minimum and Maximum

The **Extend Min** and **Extend Max** options enable you to increase the valid range of the interpolation function beyond the maximum or minimum specified coordinate values. The value the function will take at coordinate values lower than the minimum specified coordinate, which is equal to the value at the minimum specified coordinate. Similarly, the value at the maximum specified coordinate is extended for higher coordinate values.



### 32.1.2. Three Dimensional Interpolation

3D interpolation functions can be used to specify any quantity in CFX-Pre for which a standard CEL function (for example, `sin`, `cos`, `step`, and so on) can be used. The function is created by interpolating values between a "cloud of points" using a distance weighted average based on the closest three points. Common applications include setting an initial guess or a profile boundary condition from experimental data values.

For a 3D interpolation using the `Interpolation (Data Input)` option, you should set X, Y, and Z coordinate values and a single value associated with the coordinate. The coordinates and the value are interpreted in the local coordinate frame, which will depend on where the function is used. For example, if the function is used to set a boundary condition value, the coordinate frame selected for that boundary condition will apply. For details, see [Coordinate Frames \(p. 371\)](#) and [Coordinate Frames in the CFX-Solver Modeling Guide](#). If the local coordinate frame is cylindrical, the units for the **Argument List** should still be those of a Cartesian frame.

Data for 3D interpolation can be entered manually or read from a file. If the file contains data in the profile data format, then it can be imported as described in [Profile Boundary Conditions in the CFX-Solver Modeling Guide](#). Otherwise, numerical values for the 3D interpolation can be imported directly into CFX-Pre using the instructions below. Also see [Importing Data from a File \(p. 422\)](#) for more information on how to import the numerical values from a file.

1. Enter a unique function name that you will use when setting the value of a quantity using an expression.
2. **Argument Units:** Enter a comma separated list of the units used for the coordinates.  
These will usually be coordinate axis dimensions (for example, [m], [cm], and so on).
3. Under **Result Units**, enter a single unit.  
This unit should be a valid unit for the quantity you will be specifying (for example, [m s<sup>-1</sup>] for a velocity).
4. Right-click in the window to import data from a file or delete an entry.
5. Enter a comma, separating X, Y, Z coordinate values.  
The coordinates are interpreted in the local coordinate frame, which will depend on where the function is used.
6. Enter a **Value** associated with the **Coordinate**.
7. Click **Add** to add the point to the list (or **Remove** to remove a highlighted value from the list).

### 32.1.3. Importing Data from a File

After you create a user function (**Insert > Expressions, Functions and Variables > User Function**), you see the function in the details view with **Option** set to `Interpolation (Data Input)`.

To import data from a file, set **Interpolation Data > Option** as appropriate, then right-click in the **Interpolation Data** list and select **Import Data...** from the shortcut menu. This displays the **Import Interpolation Data** dialog box, where you specify a file name.

- For 1D data, **Column Selection** fields appear. You can select which column of data in your import file is appropriate for the coordinates and values.
- For 3D interpolations, columns are selected in the same way, with **X**, **Y**, and **Z** data all required.
- Each line of the file must contain numbers, separated by a space or comma.
- Once the data is read into CFX-Pre, the values will be written directly to the solver input file and the data file will not be read again. If its contents are changed, then you must return to CFX-Pre and explicitly import the data again to update the stored values.

### 32.1.4. Using Interpolation (Data Input) Functions in CEL Expressions

A user function can be used in CEL expressions in any place that a standard CEL function (for example, `sin`, `cos`, `step`, and so on) can be used. For a user function with the `Interpolation (Data Input)` option selected, you can enter an expression using the following notation:

- 1D interpolation

```
<Function Name>(arg1[units])
```

- 3D interpolation

```
<Function Name>(arg1[units], arg2[units], arg3[units])
```

When using a system variable, an expression, or a value, you do not need to specify units. For example, a pipe inlet velocity profile might be set by entering:

```
inletvelocity( r )
```

or

```
inletvelocity( x, y, z )
```

## 32.2. Interpolation (From File) User Functions (Profile Functions)

An interpolation (from file) function, or profile function, is a type of user function that provides access, from a CEL expression, to profile data based on a set of spatial coordinates.

Profile data is imported from an external data file (typically named with file extension `.csv`). You have the option of storing the data with the case or continuing to reference the external data file, in which case that file is read directly by the CFX-Solver each time the solver is started or restarted.

Applications of profile functions include:

- Profile Boundary Conditions

For details, see [Profile Boundary Conditions in the CFX-Solver Modeling Guide](#).

- Injection Regions

For details, see [Injection Regions \(p. 253\)](#).



The following topics are discussed:

[32.2.1. Establishing Profile Data and Creating Profile Functions](#)

[32.2.2. Using Interpolation \(From File\) Functions in CEL Expressions](#)

## 32.2.1. Establishing Profile Data and Creating Profile Functions

For information on creating profile data files, see:

- [Type in the \*CFD-Post User's Guide\*](#) (about exporting profile files from CFD-Post)
- [Edit Profile Data \(p. 99\)](#) (about editing profile data in CFX-Pre)

Once you have a file that contains data in the profile data format (see [Profile Data Format \(p. 424\)](#)), you can import it as described in [Initialize Profile Data \(p. 106\)](#).

When the profile file is initialized, a user function is created in the **Outline** tree view under `Simulation > Expressions, Functions and Variables > User Functions`. The user function has a details view where you can complete the profile function setup by:

- Making associations between function data and CEL parameters.
- Controlling visibility options.

The following topics are discussed:

[32.2.1.1. Profile Data Format](#)

[32.2.1.2. Profile Function Details View](#)

### 32.2.1.1. Profile Data Format

A profile data file is formatted as follows:

```
# Comment line
# The following section (beginning with [Name] and ending with
# [Data]) represents one profile, which can be repeated
# to define multiple profiles.
[Name]
My Boundary

[Parameters]
Coefficient ABC = 12.8
Coefficient2 = -5500
Parameter123 = 123.5 [K]

[Spatial Fields]
r, theta, z
.
.
.
[Data]
X [ m ], Y [ m ], Z [ m ], Area [ m^2 ], Density [ kg m^-3 ]
-1.773e-02, -5.382e-02, 6.000e-02, 7.121e-06, 1.231e+00
-1.773e-02, -5.796e-02, 5.999e-02, 5.063e-06, 1.231e+00
.
.
.
# ----- end of first profile 'My Boundary'-----
[Name]
Plane 2
.
```

Notes about the profile data format:

- The name of each locator is listed under the [Name] heading.
- The names of the fields are case-insensitive (that is, [data] and [Data] are acceptable).
- The names of variables used in the data fields *are* case sensitive.

For example, u [m] is a valid x velocity component, whereas U [m] is an unrecognized field name. You have to map this unrecognized field name with a valid variable name when loading into CFX-Pre. This is consistent with the use of CEL elsewhere.

- Comments in the file are preceded by # (or ## for the CFX polyline format) and can appear anywhere in the file.
- Commas must separate all fields in the profile. Any trailing commas at the end of a line are ignored. Any additional commas within a line of data will be a syntax error.
- Blank lines are ignored and can appear anywhere in the file (except between the [<data>] and first data line, where <data> is one of the key words in square brackets shown in the data format).
- If any lines with text are included above the keyword [Name], a syntax error will occur. Such lines should be preceded by # character to convert them into comments.
- Multiple data sets are permitted within the same file; each profile begins with keyword [Name].
- Point coordinates and the corresponding variable values are stored in the [Data] section.
- [Spatial Fields] can contain 1, 2, or 3 values, corresponding to 1D, 2D, or 3D data.
- Within the [Data] section, the Spatial Fields must be the first entries in each row of data, before the other variables.
- The data file has a .csv extension for compatibility with other software packages.

When this data file is read in, it is checked for any format violations; physics errors are shown for such situations.

---

**Note:**

A file exported from CFD-Post in a user-specified coordinate system contains a coordinate frame ([CoordFrame]) section. The coordinate frame definition is written to the profile file; CFX-Pre defines the coordinate frame when you initialize the data.

---

Additional information on profile data is available:

- [Physics Message Window \(p. 45\)](#)
- [RULES and VARIABLES Files in the CFX-Solver Manager User's Guide](#)

- [Profile Boundary Conditions in the CFX-Solver Modeling Guide](#)

### 32.2.1.2. Profile Function Details View

When you import data from a profile data file, a corresponding user function is created in the **Outline** tree view under *Simulation > Expressions, Functions and Variables > User Functions*. Right-click the user function and select **Edit** to view and edit the **Basic Settings**.

Some of the settings on the **Basic Settings** tab are read-only, providing information about the profile function.

The following topics are discussed:

[32.2.1.2.1. Associating Profile Data with CEL Parameters](#)

[32.2.1.2.2. Coordinate System Definition](#)


[32.2.1.2.3. Visualization Options for Profile Data](#)

#### 32.2.1.2.1. Associating Profile Data with CEL Parameters

In the **Basic Settings** tab, you can optionally associate the value fields of the profile data with CEL parameters. If you are using the profile data in a profile boundary condition or an injection region, this association can facilitate the automatic configuration of related settings (via the **Generate Values** button).

To associate the value fields of the profile data with CEL parameters:

1. Select a value field in the **Value Fields** list.
2. Select **Parameter List**.
3. Select one or more CEL parameters to associate with the value field.

You can do this by clicking the drop-down arrow beside **Parameter List** and selecting (or **Ctrl**-selecting) parameters. If a parameter that you want to select does not appear, click *Multi-select from extended list*  to see all of the parameters that are available.

#### 32.2.1.2.2. Coordinate System Definition

You can create a custom coordinate system using the profile's data, if the latter defines a mapping between the spatial coordinates and custom coordinates (such as turbomachinery-related coordinates).

For example, a profile containing  $x$ ,  $y$ , and  $z$  as spatial fields and  $u$ , and  $v$  as value fields could be used to define a coordinate system with  $u$  and  $v$  as the spatial fields. If such a coordinate system were to be used for a user point definition, then the user point coordinates would be entered as a  $(u,v)$  pair instead of as  $(x,y,z)$ . CFX-Pre uses the provided profile data to interpolate the given  $(u,v)$  pair back to  $(x,y,z)$  for visualisation and for use in the CFX-Solver.

A custom coordinate system created from profile data could be used:

- As a basis for defining user locations such as user lines and user points. For details, see [User Locations](#) (p. 377).

- As a basis for producing a transformed profile. For example, using the mapping provided by a profile's custom (u,v) coordinates, you could transform another profile having spatial fields of the same custom coordinates to yield a transformed version having Cartesian spatial fields. For details on profile transformations based on a custom coordinate system, see [Modifying Spatial Fields in Profile Data](#) (p. 104).

A profile file used to define a custom coordinate system must meet the following requirements:

- There must be surface connectivity data.
- The profile must have 3 Cartesian spatial fields.
- There must be an unambiguous mapping between the custom coordinates and Cartesian coordinates. For example, a custom two dimensional coordinate system intended to fit a turbine blade surface might need to be defined for only one side of the blade in order to avoid having any single point in custom coordinates map to multiple locations in Cartesian coordinates.

If the profile function contains values of custom coordinates, and if the profile function meets the requirements described above, then you can define a custom coordinate system as follows:

1. Select **Coordinate System Definition**.
2. For **Coord. Variables**, multi-select profile variables to indicate the axes for the custom coordinate system.

### 32.2.1.2.3. Visualization Options for Profile Data

If the imported profile data has three Cartesian spatial fields (i.e. x, y, and z), you can display the profile data in the viewer by editing the corresponding user function. To display the profile data in the viewer:

1. Select the **Plot Options > Visibility** check box.
2. Optionally select the **Color by Profile Variable** check box and select a **Profile Variable**.
3. Optionally select the **Profile Vector** check box to visualize vectors defined from the variables in the profile file. There are two types of vector quantities available for selection in the drop-down list:

- **Standard Vectors**

These vector quantities are automatically constructed from the variables in the profile file, wherever CFX-Pre is able to recognize the available variables as components of a vector quantity. You cannot modify the settings for these vectors.

- **User Defined**

This selection enables you to manually specify the vector components individually. You can specify the **Vector Type** as either *Cartesian* or *Cylindrical*. The **Profile Variable** drop-down boxes enable you to select any variable defined in the file for each component. All components must have the same dimensions.

4. Optionally change the rendering options (such as the type and size of the symbol used to mark each profile data point) by doing one of the following:
  - Right-click the profile in the tree view and select **Render > Properties**.
  - Right-click the profile in the viewer and select **Render > Properties**.

You can also control the visibility of the profile data by the visibility check box in the tree view, and by the **Hide** command in the menu that appears when you right-click the profile in the viewer.

### 32.2.2. Using Interpolation (From File) Functions in CEL Expressions

A user function can be used in CEL expressions in any place in which a standard CEL function (for example, sin, cos, step, and so on) can be used. For a profile data function or an Interpolation (Data Input) function, you can enter an expression using the notation:

```
<Function Name>.<Value Field Name>(arg1[units], arg2[units], ...)
```

or

```
<Function Name>.<Parameter Name>()
```

where:

- <Value Field Name> identifies which column of data from the profile data file should be used, and
- <Parameter Name> identifies which parameter from the profile data file should be used.

The allowed values are shown when editing the user function. When using a system variable, an expression, or a value, you do not need to specify units. For example, a pipe pressure profile might be set by entering:

```
inletvelocity.Total Pressure(r)
```

if the profile data file contains a profile labelled "inletvelocity" with a data column labelled "Total Pressure".

## 32.3. Table User Functions

A table function is a type of user function that provides access, from a CEL expression, to tabular data based on an index value (or set of index values, depending on the number of defined index fields in the table data). Table data can be created locally or imported from an external data file (typically named with file extension `.CSV`). If the table data is imported from an external data file, you have the option of storing the data with the case or continuing to reference the external data file, in which case that file is read directly by the CFX-Solver each time the solver is started or restarted.

Applications of table user functions include:

- Operating Point Input Parameter Tables

For details, see [Operating Points in the CFX-Pre User's Guide \(p. 447\)](#).

The following topics are discussed:

[32.3.1. Establishing Table Data and Creating Table Functions](#)

[32.3.2. Using Table Data User Functions in CEL Expressions](#)

### 32.3.1. Establishing Table Data and Creating Table Functions

To create a 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. Specify a name for the table function and click **OK**.

The table editor appears.

3. Use the table editor to establish (create/import/edit) the table data.

You can optionally use the **Auto Fill Values** dialog box, which is available from the table editor, to automatically generate table data that is suitable for providing input parameter values for operating points.

4. Optionally export the table data to a file.

5. If a table data file is involved, optionally specify, via the **Table Data Storage** setting, whether the simulation should refer to the table data file (`Use External File`) for the latest definition of the table data, or whether the simulation should use the internally stored copy of the table data (`Copy Table Data Into Setup`).

6. Click **OK** to close the table and initialize the table data.

Table data initialization results in the creation of a table function, which is listed in the **Outline** tree view under `Simulation > Expressions, Functions and Variables > User Functions`.

If required, you can subsequently 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, etc.

If you have table data in a `.CSV` file, you can alternatively skip the above procedure and create a table function without using the table editor by using the **Tools > Initialize Profile Data** command. In this case, the table function is named according to the table name in the table data file. For details, see [Initialize Profile Data \(p. 106\)](#).

The following topics are discussed:

[32.3.1.1. Table Data Format](#)

[32.3.1.2. Table Editor](#)

### 32.3.1.3. Table Function Details View

#### 32.3.1.1. Table Data Format

Table data is formatted the same as profile data except that:

- Table data contains index fields instead of spatial fields.
- Table parameters (fixed values that would appear under the heading [Parameters]) can be present but they are not supported by the table editor. Table files containing table parameters must be initialized from the **Tools > Initialize Profile Data** command.

A table data file is formatted as follows:

```
[Name]
  Input Parameter Values

[Parameters]
Coefficient ABC = 12.8
Coefficient2 = -5500
Parameter123 = 123.5 [K]

[Index Fields]
Table Index

[Data]
Table Index, massFlowOut2 [kg s^-1], InletTemp [K]
1, 1.23, 345.0
3, 1.24, 345.0
5, 1.23, 350.0
7, 1.24, 350.0
```

Notes about the table data format:

- The "Index Fields" section is optional; if it is omitted, the table data is indexed by row number.
- Any fields used as index fields must contain only integer values.
- If the "Index Fields" section is specified then each row in the table must correspond to a different set of index field values.

Notes pertaining to the profile data format also apply to the table data format. For details, see [Profile Data Format \(p. 424\)](#).

#### 32.3.1.2. Table Editor

The table editor provides the following functions:

- View table data
- Create table data:
  - Click table cells and enter values directly.
- Import table data from a file:
  - Click **Import**, specify a file name, and click **Open**.

- Edit table data:
  - Click and type in any cell to edit the cell value.
  - Right-click a row/column label to add/delete/move a row/column.
  - Right-click a column label to rename it, and to assign units for the data in the column, via the **Edit Column Header** command.
  - Right-click a row/column label to hide/show the row. Hidden rows/columns do not appear in the saved table file.
  - Click **Revert** to re-read the table data from the file from/to which you last loaded/saved.
- Export table data to a file:
  - Click **Export**, specify a file name, and click **Save**.

Table data is written to a text file with extension `.csv`.

- For a new table function, the **Table Data Storage** setting is available. The options are:
  - Use External File

With this option, the simulation gets table data directly from the table data file. Any changes to numerical values in the file take effect when the simulation is run. More substantial changes (new or renamed columns, etc.) require re-initialization in CFX-Pre, either via the table editor or via the **Tools > Initialize Profile Data** command.

- Copy Table Data Into Setup

With this option, the table data is copied into the current case definition and the table data file is no longer referenced. Any changes to the table data file require re-initialization in CFX-Pre in order to take effect; re-initialization can be done either via the table editor or via the **Tools > Initialize Profile Data** command.

The **Table Data Storage** setting is not available after you create the table function. In this case, a change to this setting can be made by re-creating the table function.

When you click **OK** in the table editor, the table data is initialized, resulting in a table function that can be used in your setup via CEL expressions. The table function is listed in the **Outline** tree view under `Simulation > Expressions, Functions and Variables > User Functions`.

#### Buttons of the table editor:

- The **Apply** button effectively saves the table data.
- The **Close** button closes the table editor without saving the data at that time.
- The **OK** button is equivalent to **Apply** followed by **Close**.



**Toolbar icons of the table editor:**

- *Import* — Select a `.csv` file from which to import data. The `.csv` file must contain a single profile of type `Table`. Note that, when you create a table using the table editor, the table name in the table data file is ignored.
- *Clear Values* — Remove all the table values, leaving the index values and column headings unchanged.
- *Reset* — Remove all of the table data, resetting the table to a blank state.
- *Revert* — If changes to the table data have been made manually but have not been applied, revert to the data that was last saved.
- *Export* — Export the table data to the selected `.csv` file.
- *Auto Fill* — Show the **Auto Fill Values** dialog box, which enables you to automatically generate table data that is suitable for providing input parameter values for operating points. For details, see [Auto Fill Values Dialog Box \(p. 432\)](#).

**32.3.1.2.1. Auto Fill Values Dialog Box**

The **Auto Fill Values** dialog box is available from the table editor. Using this dialog box, you can automatically generate table data that is suitable for providing input parameter values for operating points. The feature uses "Design of Experiments" algorithms (see [DOE Types in the DesignXplorer User's Guide](#)) to provide suitable sampling points, given ranges for the input parameters and the target number of points.

To use the **Auto Fill Values** dialog box:

1. Leave **Method** set to the only option: `Optimal Space Filling`.
2. Set **Number of Samples** to the number of table rows to generate.
3. Fill in the ranges of values to generate.
4. Click **OK** or **Apply** to generate the table values, or click **Cancel** to do nothing.

---

**Note:**

After generating table values, the lower and upper bounds (range limits) that you specified in the **Auto Fill Values** dialog box are stored with the table as parameters, accessible as usual for table parameters; for details, see [Using Table Data User Functions in CEL Expressions \(p. 433\)](#). For example, the bounds for table column `MassFlowRate` would be named `MassFlowRate Generation Lower Bound` and `MassFlowRate Generation Upper Bound`.

---

**32.3.1.3. Table Function Details View**

The table function details view shows basic information about the table function.

### 32.3.2. Using Table Data User Functions in CEL Expressions

A user function can be used in CEL expressions in any place in which a standard CEL function (for example, sin, cos, step, and so on) can be used. For a table data function, you can enter an expression using the notation:

```
<Table Name>.<Value Field Name>(index1, index2, ...)
```

or

```
<Table Name>.<Parameter Name>()
```

where:

- <Value Field Name> identifies which column of data from the table data file should be used, and
- <Parameter Name> identifies which parameter from the table data file should be used.

The allowed values are shown when editing the user function. The index argument must be an integer or integers that identify which row of the table should be used to look up the result. If the table data file does not contain the "Index Fields" section, then a single index that corresponds to the table row number must be used as the argument — if there are  $n$  rows of numerical data in the file, then the valid arguments are from 1 to  $n$ . If the table data file does contain the "Index Fields" section, then the index arguments supplied must correspond to the index field values from a single row in the table.

## 32.4. Functional Mockup Unit Instance

If you have an FMU file, it can be initialized as described in [Initialize FMU Instance \(p. 108\)](#). When the FMU file is initialized, a user function is created in the **Outline** tree view under **Simulation > Expressions, Functions and Variables > User Functions** with the **Functional Mockup Unit Instance** option selected. The input variables, input parameters, and output variables that are available in the function are those that were selected during initialization.

To complete the function definition:

- Right-click the applicable function under **Simulation > Expressions, Functions and Variables > User Functions** and select **Edit**.

The function details view appears.

- Ensure that **FMU File Name** is set to the name of the FMU file.
- Specify Coordinate Frame as appropriate for correct interpretation of the function inputs and outputs.
- Under **Argument Values (FMU Input Variables)**, select each variable listed, one-at-a-time, and specify a value (typically via a CEL expression) under **Expression Value**.
- Under **Argument Constants (FMU Parameter Variables)**, select each parameter listed, one-at-a-time, and specify a value (not via a CEL expression) under **Value**.

- Under **Result Values (FMU Output Variables)**, select each variable listed, one-at-a-time, and review the units.

Note that, in the details view:

- The lists show the CFX names.
- The **Value Name** fields show the names that are used in communication with the FMU binary.

- Click **OK**.

Function usage syntax:

```
<function name>.<variable or parameter name>()
```

For example, if you have a function named `WaterTank Control`, and the FMU output variable is `valve` (a Boolean value that is returned as 0 (zero) for off, or 1 (one) for on), then you could implement a conditional connection control (see [Conditional Connection Control \(p. 212\)](#)) of a domain interface (that acts as a valve), using the **Option Specified Open State with Logical Expression**:

```
WaterTank Control.valve() > 0
```

For modeling details, see [Coupling CFX to an External Solver: Functional Mock-up Interface \(FMI\) Co-simulation in the CFX-Solver Modeling Guide](#).

## 32.5. User CEL Functions

---

User CEL functions are used in conjunction with user CEL routines. A user CEL function must be created after a user CEL routine. For details, see [User CEL Routines \(p. 439\)](#). User CEL functions set the name of the user CEL routine associated with the function, the input arguments to pass to the routine and the expected return arguments from the routine.

1. Select the user CEL routine name (user routine name) from the drop-down list that the function will apply to. For details, see [Function Name \(p. 435\)](#).
2. Enter the input **Argument Units** list to pass to the subroutine.

For details, see [Argument Units \(p. 435\)](#).

3. Enter the **Result Units** list output from the subroutine.

For details, see [Result Units \(p. 435\)](#).

The following topics are discussed:

[32.5.1. Function Name](#)

[32.5.2. Argument Units](#)

[32.5.3. Result Units](#)

### 32.5.1. Function Name

The function name is assigned when you create a new user CEL function, and is equivalent to the name you would set for an expression. You use this name, together with the input arguments, when setting the expression for the quantity of interest. For details, see [Defining Quantities with User CEL Functions \(p. 435\)](#). The function name should follow usual naming rules (it may contain spaces but should not include underscores).

#### 32.5.1.1. Defining Quantities with User CEL Functions

After you have created a user CEL function, you can use it to specify any quantity in CFX-Pre for which a standard CEL function (for example, `sin`, `cos`, `step`, and so on) can be used. You can enter an expression using the notation:

```
<Function Name>(arg1[units], arg2[units], ...)
```

When using a system variable, an expression, or a value, you do not need to specify units. For example, a pipe inlet velocity profile might be set by entering:

```
inletvelocity(MaxVel, r, 0.2[m])
```

where `inletvelocity` is the function name of the user CEL function, `MaxVel` is an existing expression or value, `r` is a system variable, and `0.2[m]` corresponds to the pipe diameter.

You would enter the above expression as one of the velocity components at the inlet boundary condition (you may also want to use it as a velocity component of the initial guess).

### 32.5.2. Argument Units

You should enter the units of each argument that you will be passing to the subroutine. Units should be comma separated and correspond to the order used when setting the expression for a quantity in CFX-Pre. For example, enter `[m]`, `[m s-1]`, `[Pa]` if you are passing a length, velocity, and pressure value to the subroutine. The values of arguments passed to a subroutine are specified when you set an expression for a quantity in CFX-Pre. For details, see [Defining Quantities with User CEL Functions \(p. 435\)](#).

### 32.5.3. Result Units

The result argument units are the units of the return arguments from the subroutine. Units should be comma separated and correspond to a valid unit for the quantity that you are specifying.

## 32.6. GT-SUITE Patch Functions

For details about using GT-SUITE with CFX, see [Coupling CFX to an External Solver: GT-SUITE Coupling Simulations in the CFX-Solver Modeling Guide](#).

The **Function** details view for a GT-SUITE Patch function has the following settings:

- **Option** is set to `GT-SUITE Patch`.

- **GT-SUITE Model** specifies the name of the GT-SUITE model that, as part of its setup, associates a GT-SUITE interface with the present GT-SUITE Patch function.

This association was made when the GT-SUITE model was being set up for initialization, in the **Initialize GT-SUITE Coupling** dialog box, on the **Function Names** tab. On that tab, the column named "GT-SUITE Interface or Turboshaft Name" lists the GT-SUITE patches and turboshafts, while the column named "CFX Function Name" lists the names of the planned GT-SUITE Patch functions and GT-SUITE Turboshaft functions. For details, see [Function Names Tab \(p. 114\)](#).

- **GT-SUITE Patch**

**GT-SUITE Patch** shows the name of the associated GT-SUITE interface, as known in GT-SUITE. This name is often the same as the function name, but is different if the name of the GT-SUITE interface contains characters that are not valid to use in function names.

- **Boundary**

Specify the CFX boundary to associate with the GT-SUITE patch. When the simulation is run, values from this CFX boundary will be sent to GT-SUITE to provide the input values for the corresponding GT-SUITE interface (GT-SUITE patch). You should also be sure that the specified CFX boundary is set to use values from the same GT-SUITE patch by using the relevant GT-SUITE Patch function.

- **Transfer Variables**

A transfer variable is a variable that will be exchanged with GT-SUITE. You can select each listed transfer variable to see the name (**Variable Name**) and units (**Variable Units**) of the corresponding CFX variable.

You can access a transfer variable using the following CEL syntax:

```
<function name>.<transfer variable name>()
```

## 32.7. GT-SUITE Turboshaft Functions

---

For details about using GT-SUITE with CFX, see [Coupling CFX to an External Solver: GT-SUITE Coupling Simulations in the CFX-Solver Modeling Guide](#).

The **Function** details view for a GT-SUITE Turboshaft function has the following settings:

- **Option** is set to GT-SUITE Turboshaft.
- **GT-SUITE Name** shows the name of the associated GT-SUITE turboshaft, as known in GT-SUITE. This name is often the same as the function name, but is different if the name of the GT-SUITE turboshaft contains characters that are not valid to use in function names.
- **GT-SUITE Model** specifies the name of the GT-SUITE model that includes the associated GT-SUITE turboshaft.

This association was made when the GT-SUITE model was being set up for initialization, in the **Initialize GT-SUITE Coupling** dialog box, on the **Function Names** tab. On that tab, the column named "GT-SUITE Interface or Turboshaft Name" lists the GT-SUITE patches and turboshafts, while the column named "CFX Function Name" lists the names of the planned GT-SUITE Patch functions and GT-SUITE Turboshaft functions. For details, see [Function Names Tab \(p. 114\)](#).

- **GT-SUITE Motion**

The **GT-SUITE Motion > Axis Definition** settings in the function details view are the same as the **Axis Definition** settings in the **Initialize GT-SUITE Coupling** dialog box, on the **Turboshaft Setup** tab, which is described in [Turboshaft Setup Tab \(p. 112\)](#).

- **GT-SUITE Torque**

The **GT-SUITE Torque** settings in the function details view are the same as the **Torque** settings (excluding the **Torque** check box itself) in the **Initialize GT-SUITE Coupling** dialog box, on the **Turboshaft Setup** tab, which is described in [Turboshaft Setup Tab \(p. 112\)](#).

- **Transfer Variables**

A transfer variable is a variable that will be exchanged with GT-SUITE. You can select each listed transfer variable to see the name (**Variable Name**) and units (**Variable Units**) of the corresponding CFX variable.

You can access a transfer variable using the following CEL syntax:

```
<function name>.<transfer variable name>()
```



---

# Chapter 33: User Routines

---

*User routines* are Fortran subroutines (User CEL Functions, Junction Box Routines, and Particle User Routines) that you write for the CFX-Solver. You use the **User Routine** details view to create objects that link your Fortran subroutines to the CFX-Solver. To access the **User Routine** details view, either

select **Insert > Expressions, Functions and Variables > User Routines** or click *Insert User Routine*  in the main toolbar.

Once your User Routines have been created and linked, they appear in the tree view under **Simulation > Expressions, Functions and Variables > User Routines**.

## User Routine Details View

The **User Routine** details view creates objects that link your Fortran subroutines to the CFX-Solver.

There are four types of user routines:

- [User CEL Routines \(p. 439\)](#)
- [Junction Box Routines \(p. 441\)](#)
- [Particle User Routines \(p. 441\)](#)
- For details on the fourth user-routine option, `Transient Particle Diagnostics Routine`, refer to [User Diagnostics Routine in the CFX-Solver Modeling Guide](#).

### 33.1. User CEL Routines

---

Available when **Option** is set to `User CEL Function`, User CEL Routines are used in conjunction with User CEL Functions to define quantities in CFX-Pre based on a Fortran subroutine. The User CEL Routine is used to set the calling name, the location of the subroutine, and the location of the Shared Libraries associated with the subroutine.

1. Enter the [Calling Name \(p. 440\)](#) of the subroutine within the subroutine file.

You should always use lowercase letters for this even when the subroutine name in the Fortran file is in uppercase.

2. Enter the **Library Name** and **Library Path**.

The library name will usually be your subroutine filename (without any extensions). The library path is the path to the directory containing the system dependent directories and supports lists of paths. For details, see [Library Name and Library Path \(p. 440\)](#).

User CEL Functions are created after the associated routine has been defined. For details, see:



- [User CEL Functions \(p. 434\)](#)
- [User CEL Functions and Routines in the CFX-Solver Modeling Guide.](#)

### 33.1.1. Calling Name

The **Calling Name** is the name of the subroutine within a Fortran file. This name appears immediately after the `SUBROUTINE` statement in a Fortran file. It is usually in upper case in the Fortran file, but should always be entered in lower case. It must not include spaces but may contain underscores (this is different from the usual naming rules).

### 33.1.2. Library Name and Library Path

This is the name of the shared library. The **Library Name** will be the name of the file containing the subroutine, ignoring any file extensions (for example, `InletProfile` for the file named `InletProfile.F`). The filename depends on how you ran the `cfx5mkext` command:

- If you *did not* specify a `-name` option when running the `cfx5mkext` command, the filename will be the name of your shared library. For details, see [Shared Libraries in the CFX-Solver Modeling Guide](#).
- If you *did* specify the `-name` option when running the `cfx5mkext` command, the filename will be the name you specified.

Note that if you look at the actual filename of the shared library, it will have a `lib` prefix (UNIX only) and either a `.so`, `.sl`, or `.dll` suffix depending on your platform. Do not include the prefix or suffix in the **Library Name**.

The **Library Path** is the absolute path to the directory that contains the shared libraries in subdirectories for each platform. The pathname depends on how you ran the `cfx5mkext` command:

- If you *did not* specify a `-dest` option when running the `cfx5mkext` command, this will be the path to the directory in which the `cfx5mkext` command was executed. For details, see [Shared Libraries in the CFX-Solver Modeling Guide](#).
- If you *did* specify the `"-dest"` option when running the `cfx5mkext` command, the pathname will be the name you specified.

On UNIX platforms, the **Library Path** will look like:

```
/home/user/SharedLibraries
```

On Windows systems, the **Library Path** will look like:

```
F:\user\SharedLibraries
```

If you are running in parallel and specify only a single library path, then each machine should be able to locate the shared libraries using the specified **Library Path**. On Windows systems, you may have to map network drives so that the path to the libraries is the same on each machine. However, you can also specify the **Library Path** as a list. Ansys CFX will try to locate your shared libraries on each machine in the parallel run using the list of paths provided. Comma ( , ), colon ( : ) and semi-colon ( ; )

separated lists are valid. For example, when running in parallel across Windows and UNIX machines, a valid path may look like:

```
/home/user/SharedLibraries,C:\Shared Libraries
```

The colon used after a Windows drive letter is treated correctly and is not interpreted as a list separator.

## 33.2. Junction Box Routines

---

Junction box routines are used to call your own Fortran subroutines during execution of the CFX-Solver. You must create a junction box routine object so that the CFX-Solver knows the location of the subroutine, the location of the shared libraries associated with the subroutine, and when to call the subroutine. Each of these items is specified in the details view. For details, see [User Junction Box Routines in the CFX-Solver Modeling Guide](#).

To complete the Junction Box Routine details:

1. The first three parameters are identical to those described for the `User Function` option, under [Calling Name \(p. 440\)](#) and [Library Name and Library Path \(p. 440\)](#).
2. Enter the **Junction Box Location** at which to call the subroutine. For details, see [Junction Box Routine Options and Parameters in the CFX-Solver Modeling Guide](#).

Junction box routines appear in the `LIBRARY` section of a CCL file. You can create many junction box routines in CFX-Pre, but only call the required routines during execution of the CFX-Solver. This enables you to read in a CCL file containing a list of junction box routines and then select only those that you want to call. This selection is made on the **Solver Control** tab. For details, see [Basic Settings Tab \(p. 289\)](#).

## 33.3. Particle User Routines

---

Particle user routines are used to create user defined injection regions and particle user sources. Creating a user routine with the `Particle User Routine` option selected is identical to creating a routine with the `User CEL Function` option selected. For details, see:

- [User CEL Routines \(p. 439\)](#)
- [Particle Injection Regions Tab \(p. 199\)](#)
- [Particle User Sources in the CFX-Solver Modeling Guide](#).



---

# Chapter 34: GT-SUITE Model Object

---

CFX can run simulations that interact with GT-SUITE software. For details about using GT-SUITE with CFX, see [Coupling CFX to an External Solver: GT-SUITE Coupling Simulations in the CFX-Solver Modeling Guide](#).

After initializing a GT-SUITE model, model objects and patch functions are created.

The GT-SUITE Model object in CFX-Pre contains settings that prepare CFX to interact with GT-SUITE. For details, see [GT-SUITE Model Details View \(p. 443\)](#).

Patch functions enable you to make boundary conditions that are interactive between CFX and GT-SUITE. For details, see [GT-SUITE Patch Functions \(p. 435\)](#).

## 34.1. GT-SUITE Model Details View

---

GT-SUITE Model objects appear in the **Outline** tree view under **Simulation > Expressions, Functions and Variables**. To access the **GT-SUITE Model** details view, right-click an existing GT-SUITE Model object and select **Edit**. You use the **GT-SUITE Model** details view to review and adjust settings for the object.

- **File Name** shows the name of the `.gtm` file that defines some of the inputs for the GT-SUITE software.

---

### Note:

The file extension `.gtm` is used by files written by GT-SUITE software and by mesh files written by some Ansys software. In this section of the documentation, `.gtm` refers to files written by GT-SUITE software.

---

- **Version** is the version of GT-SUITE that needs to be run, and should match the version that was used to produce the specified `.gtm` file.

The versions that are available under **Version** are controlled by a preference. For details, see [User Interface \(p. 86\)](#).

- The **Installation Path** check box can normally remain cleared. Selecting the **Installation Path** check box and specifying an **Installation Path** similar to

```
C:\Program Files (x86)\GTI
```

indicates where CFX should find the GT-SUITE software. This check box and its corresponding text field override the `GTIHOME` environment variable, which also indicates an installation path to GT-SUITE.

Whether CFX finds the installation path to GT-SUITE from the environment variable or from the overriding **Installation Path** settings, it looks directly under that installation path for the version-specific installation subdirectory of GT-SUITE, such as v2019, as set by the **Version** setting.

- **Material Mapping**

The GT-SUITE Model object contains a mapping between materials in CFX and materials in GT-SUITE. When you initialize a GT-SUITE model, materials are read from the GT-SUITE definition (.gtm) file. You must set up mapping between the materials in the .gtm file and the materials available in CFX-Pre. In the **Material Mapping** list, choose a GT-SUITE material, then in the **Material Name** field below the list, choose the corresponding CFX material.

---

## Chapter 35: Simulation Control

---

Simulation controls enable you to define the execution of analyses and related tasks in the simulation, such as remeshing. Specific controls include definitions of global execution, termination controls, and the ability to have more than one configuration. Additional information regarding these topics is provided in [Execution Control \(p. 449\)](#) and [Configurations \(p. 461\)](#), respectively.



---

# Chapter 36: Operating Points

---

Operating points are used to construct operating maps. For modeling information, see [Operating Maps and Operating Point Cases in the CFX-Solver Modeling Guide](#).

You access the **Operating Points** details view in CFX-Pre by clicking **Insert > Solver > Operating Points** or by right-clicking **Simulation Control** in the **Outline** tree view and selecting **Insert > Operating Points**.

Details of the **Operating Points** tab:

## Option

The only available option is `From Table`.

## Table Name

Choose a table that provides operating point input parameter values.

In order for a table to be available for use, it must either be initialized from the **Initialize Profile Data** dialog box (see [Initialize Profile Data \(p. 106\)](#)) or be saved from the **Edit Table Data** dialog box (see [Establishing Table Data and Creating Table Functions \(p. 429\)](#)) within the current session.

## Operating Map Definitions

Define one or more operating maps by specifying a pair of independent variables (from the set of operating point input parameters) for each map.





---

# Chapter 37: Execution and Termination Control

---

This chapter describes:

- 37.1. Execution Control
- 37.2. Termination Control

## 37.1. Execution Control

---

The **Execution Control** settings described below apply to all configurations in the simulation. You can override these settings for any configuration from the **Run Definition** tab on the details view of the **Configuration** (for details, see [Run Definition Tab \(p. 467\)](#)) or the **Define Run** dialog box in the CFX-Solver Manager (for details, see [The Define Run Dialog Box in the CFX-Solver Manager User's Guide](#)).

This section describes:

- 37.1.1. Overview of Defining CFX-Solver Startup
- 37.1.2. The Details View for Execution Control

### 37.1.1. Overview of Defining CFX-Solver Startup

To define how a CFX-Solver can be started, the number of settings that you need to set up for **Execution Control** depends on the case:

- In some cases, you need to specify only the name of a CFX-Solver input file (\* .def or \* .mdef). For cases that require initialization from previous results, you also need to specify the name of a results file (\* .res).
- You can configure runs in serial or parallel:
  - *Serial run* is the default way of running a CFD case. During a serial run, all computation is done by a single process running on one processor.
  - *Parallel run* partitions the computation into more than one process and is done on more than one processor in a single machine (local parallel processing) or on more than one machine (distributed parallel processing). You also have the option of specifying how the computation is partitioned for a parallel run.
- You can optionally select the system priority for the interpolator and solver computation as well as settings such as precision and memory allocation.

When you have finished defining how CFX-Solver will start, click **OK** or **Apply** to save the settings.

Details of the above steps are described in the next section.

## 37.1.2. The Details View for Execution Control

You access the **Execution Control** details view in CFX-Pre by clicking **Insert > Solver > Execution Control** or by right-clicking **Simulation Control** in the **Outline** tree view and selecting **Insert > Execution Control**.

The tabs presented in the details view for **Execution Control** are described in the following sections:

- [Run Definition Tab \(p. 450\)](#)
- [Initial Values Tab \(p. 451\)](#)
- [Partitioner Tab \(p. 453\)](#)
- [Solver Tab \(p. 455\)](#)
- [Interpolator Tab \(p. 457\)](#)

### 37.1.2.1. Run Definition Tab

To configure the **Run Definition** tab:

Under **Input File Settings > Solver Input File**, ensure that the name of a CFX-Solver input file (extension `.def` or `.mdef`) is specified.

Note that CFX-Solver input file names must not contain spaces when run with an associated Ansys input file (`.inp`).

- In the **Execution Control** details view of CFX-Pre, there is an option to change the order of the nodes in the mesh. Depending on the case, this reordering may result in a reduction in the run time for the CFX-Solver. From **Mesh Options > Node Reordering > Options**, you can select `None`, `Cuthill McKee` and `Reverse Cuthill McKee`. If either `Cuthill McKee` or `Reverse Cuthill McKee` is selected, you can choose `None`, `Ascending` or `Descending` for **Sorting Within Levels**. Additional information is in [Initial Condition Modeling in the CFX-Solver Modeling Guide](#).
- Set **Type of Run** to `Full` or `Partitioner Only`.
  - `Full` runs the partitioner if applicable, and then runs the solver.
  - `Partitioner Only` is used for parallel runs only and does not run the solver. This writes a `.par` file.
- You can select or clear **Double Precision**. This setting will determine the default (single or double) precision of the partitioner, solver, and interpolator executables. For details on the precision of executables, see [Double-Precision Executables in the CFX-Solver Manager User's Guide](#). The precisions of the partitioner, solver, and interpolator executables can be optionally overridden individually on the **Partitioner**, **Solver**, and **Interpolator** tabs.
- You can select or clear **Large Problem**. This setting will determine the default ("large problem" or not) problem size capability of the partitioner, solver, and interpolator executables. For details on the problem size capability of the executables, see [Large Problem Executables in the CFX-Solver Manager User's Guide](#). The problem size capability of the partitioner, solver, and interpolator

executables can be optionally overridden individually on the **Partitioner**, **Solver**, and **Interpolator** tabs.

- You can configure the **Parallel Environment** as required (see [Parallel Environment \(p. 451\)](#)).
- System Coupling input files are always written if there are boundary conditions set to use the `System Coupling` option. Additionally, you can request that a System Coupling Input (`.scp`) file is written regardless of the boundary condition options, by selecting **Always write System Coupling Input File**. The System Coupling Input file is written alongside the CFX-Solver input (`.def`) file and has the same name except for the file extension (`.scp`).

For details on System Coupling, see:

- [Coupling CFX to an External Solver: System Coupling Simulations in the CFX-Solver Modeling Guide](#)
- [System Coupling User's Guide](#)
- [System Coupling in the Workbench User's Guide](#)
- You can elect to have CFX-Pre quit upon writing CFX-Solver Input file by selecting **Quit CFX-Pre when Solver Input File Written > Quit**.

### 37.1.2.1.1. Parallel Environment

Under **Parallel Environment**, select a **Run Mode** option. The run mode determines whether the run is serial (the default when defining a run in which a problem is solved as one process), or parallel (problem is split into partitions).

- A serial run (the default) requires no additional configuration.
- For a local parallel run, specify the number of partitions. This value is the number of separate processes that will be executed by the host.

Note that distributed parallel runs cannot be set up in CFX-Pre. For more information on *Parallel Processing*, see [Parallel Run in the CFX-Solver Manager User's Guide](#).

### 37.1.2.2. Operating Points Tab

If an **Operating Points** object is specified in the **Outline** tree view, the **Operating Points** tab is available in the **Execution Control** details view. This tab is similar to the **Operating Points** tab in CFX-Solver Manager, described in [Operating Points Tab in the CFX-Solver Manager User's Guide](#).

For modeling information, see [Operating Maps and Operating Point Cases in the CFX-Solver Modeling Guide](#).

### 37.1.2.3. Initial Values Tab

The **Initial Values** tab is used for specifying initial values via results files.

If you do not specify initial values via results files, the software will use initialization data from (in order of precedence):

- In CFX-Pre on the **Initialization** tab in the **Domain** details view
- In CFX-Pre on the **Initialization** tab in the **Global Initialization** details view
- Automatically generated initial values

However, automatically generated initial values may not be suitable for some steady-state cases and are insufficient for all transient cases.

If you do specify initial values via results files, those values will override any initial values listed above, as applicable.


You can set initial values via results files in a variety of places as listed here (in order of precedence):

- In CFX-Solver Manager in the **Define Run** dialog box on the **Initial Values** tab
- In CFX-Pre in the **Configuration** details view on the **Initial Values** tab

Note that for cases with multiple configurations, initial values cannot be set globally (that is, in CFX-Solver Manager or in CFX-Pre in the **Execution Control** details view); they can only be set per configuration (that is, in CFX-Pre in the **Configuration** details view).

- In CFX-Pre in the **Execution Control** details view on the **Initial Values** tab

You can set initial values via results files as follows:

1. On the **Initial Values** tab, select the **Initial Values Specification** check box to show the initial values settings.
2. Click **New**  to create an initial values object.
3. Select an initial values object from the list.
4. Choose the source of initial values for the initial values object.

The **Results File** option is the only option available. Specify the name of the results file from which initial values should be obtained.

5. Optionally specify **Interpolation Mapping** settings in order to select, position, and/or replicate the data. For details, see [Interpolation Mapping in the CFX-Solver Modeling Guide](#).
6. Select **Continue History From** if you want to continue the run history (convergence history, monitor plots, time and time step counters, and so on) and use the smoothest restart possible from the selected Initial Values File. The run will continue from the one contained in the specified initial values object. Note that, in CFX-Pre, the **Continue History From** option is made visible by selecting **Initial Values Control**. Note that the run history will reset if **Continue History From** is not selected.
7. The **Use Mesh From** setting determines which file provides the mesh that is used for the analysis: the Solver Input file or the Initial Values file. The mesh from the Initial Values file can be used in a limited set of circumstances. See [Using the Mesh from the Initial Values File in the CFX-Solver Modeling Guide](#) for details. Note that, in CFX-Pre, the **Use Mesh From** option is made visible by selecting **Initial Values Control**.

Full details for Initial Values Files can be found in [Reading the Initial Conditions from a File in the CFX-Solver Modeling Guide](#).

### 37.1.2.4. Partitioner Tab


Use the **Partitioner** tab to configure mesh partitioning options for parallel runs.

---

#### Note:

Once started, the run progresses through partitioning, and then into the solution of the CFD problem. Extra information is stored in the CFX-Solver Output file for a parallel run. For details, see [Partitioning Information in the CFX-Solver Manager User's Guide](#).

---

You can select a partition file to load by selecting **Initial Partition File** and clicking *Browse* . The \*.par file is only available if a model has already been partitioned. The number of partitions in the partitioning file must be the same as on the **Run Definition** tab.

---

#### Note:

An existing partition file cannot be used if the simulation involves either the Monte Carlo or Discrete Transfer radiation model. Partitions may be viewed prior to running CFX-Solver. For details, see [CFX Partition File in the CFX-Solver Manager User's Guide](#).

---

**Run Priority** may be set to *Idle*, *Low*, *Standard* or *High* (*Standard* is selected by default). For a discussion of these priorities, see [The cfx5control Application in the CFX-Solver Manager User's Guide](#).

#### 37.1.2.4.1. Executable Settings

You can override the precision set on the **Run Definition** tab by selecting **Override Default Precision** and then setting the precision. For details on the precision of executables, see [Double-Precision Executables in the CFX-Solver Manager User's Guide](#).

You can override the problem size capability ("large problem" or not) set on the **Run Definition** tab by selecting **Override Default Large Problem Setting** and then setting the problem size capability (via the **Large Problem** check box). For details on the problem size capability of the executables, see [Large Problem Executables in the CFX-Solver Manager User's Guide](#).

#### 37.1.2.4.2. Partitioning Detail

Under **Partitioning Detail**, you can specify various partition method options.

##### 37.1.2.4.2.1. Partition Type

Configure the base partition method. Available **Partition Type** selections include:

- [Multilevel Graph Partitioning Software - MeTiS in the CFX-Solver Modeling Guide](#). When first running in parallel, it is recommended that **Partition Type** be set to **MeTiS**.
- [Optimized Recursive Coordinate Bisection in the CFX-Solver Modeling Guide](#)

- [Recursive Coordinate Bisection in the CFX-Solver Modeling Guide](#)
- [Directional Recursive Coordinate Bisection in the CFX-Solver Modeling Guide](#)
- [User Specified Direction in the CFX-Solver Modeling Guide](#)
- [Radial in the CFX-Solver Modeling Guide](#)
- [Circumferential in the CFX-Solver Modeling Guide](#)
- [Simple Assignment in the CFX-Solver Modeling Guide](#)

#### 37.1.2.4.2.2. Partition Weighting

Configure how partitions are weighted between machines. Available **Partition Weighting** selections include:

- `Automatic` (default): Calculates partition sizes based on the **Relative Speed** entry specified for each machine in the `hostinfo.ccl` file. Machines with a faster relative speed than others are assigned proportionally larger partition sizes.

---

**Note:**

The entry of relative speed values is usually carried out during the CFX installation process. Parallel performance can be optimized by setting accurate entries for relative speed.

---

- `Uniform`: Assigns equal-sized partitions to each process.

---

**Note:**

Both `Uniform` and `Automatic` give the same results for local parallel runs; it is only for distributed runs that they differ.

---

- `Specified`: Only available in CFX-Solver Manager. For details, see [Partitioner Tab in the CFX-Solver Manager User's Guide](#).

#### 37.1.2.4.2.3. Multidomain Option

Configure how domains are partitioned. Available **Multidomain Option** selections include:

- `Automatic` (default): If the case does not involve particle transport, this is the same as the `Coupled Partitioning` option; otherwise it is the same as the `Independent Partitioning` option.
- `Independent Partitioning`: Each domain is partitioned independently into the specified number of partitions.

- **Coupled Partitioning:** All connected domains are partitioned together, provided they are the same type (i.e. solid domains are still partitioned separately from fluid/porous domains). For details, see [Optimizing Mesh Partitioning in the CFX-Solver Modeling Guide](#).

---

**Note:**

Coupled partitioning is often more scalable, more robust and less memory expensive than independent partitioning because fewer partition boundaries are created. However, coupled partitioning may worsen the performance of particle transport calculations.

---

#### 37.1.2.4.2.4. Multipass Partitioning

When **Coupled Partitioning** is activated, you can further choose to set the **Multipass Partitioning** option. The **Transient Rotor Stator** option is relevant only for simulations having transient rotor stator interfaces. It uses a special multipass algorithm to further optimize the partition boundaries. This approach generates circumferentially-banded partitions adjacent to each transient rotor stator interface, which ensures that interface nodes remain in the same partition as the two domains slide relative to each other. Away from the interface, the partitioning is handled using whichever method is specified for the **Partition Type**.

#### 37.1.2.4.3. Partition Smoothing

Partition smoothing attempts to minimize the surface area of partition boundaries by swapping vertices between partitions. Partition smoothing reduces communication overhead and improves solver robustness. Smoothing is enabled by default, but may be disabled by changing **Partition Smoothing > Option**.

If smoothing is enabled, the algorithm will, by default, perform a maximum of 100 sweeps. The maximum number of smoothing sweeps can be specified by changing the value of **Partition Smoothing > Max. Smooth. Sweeps**. The smoothing algorithm will stop before this value is reached if it finds no improvement between successive sweeps.

For further details on partition smoothing, see [Optimizing Mesh Partitioning in the CFX-Solver Modeling Guide](#).

#### 37.1.2.4.4. Partitioner Memory

If required, you can adjust the memory configuration under **Partitioner Memory**. For details, see [Configuring Memory for the CFX-Solver in the CFX-Solver Manager User's Guide](#).

### 37.1.2.5. Solver Tab

Select the **Solver** tab to configure solver settings.

- Under **Run Priority**, you can select *Idle*, *Low*, *Standard* or *High*. For a discussion of these priorities as well as how you can change them after the execution of the solver has started, see [The cfx5control Application in the CFX-Solver Manager User's Guide](#).



- You can override the precision set on the **Run Definition** tab. From **Executable Settings** > **Override Default Precision**, select or clear **Double Precision**. For details on the precision of executables, see [Double-Precision Executables in the CFX-Solver Manager User's Guide](#).
- You can override the problem size capability ("large problem" or not) set on the **Run Definition** tab by selecting **Override Default Large Problem Setting** and then setting the problem size capability (via the **Large Problem** check box). For details on the problem size capability of the executables, see [Large Problem Executables in the CFX-Solver Manager User's Guide](#).
- If required, you can adjust the memory configuration under **Solver Memory**. For details, see [Configuring Memory for the CFX-Solver in the CFX-Solver Manager User's Guide](#).
- If your case has user locations that are generated in CFX-Pre and stored in the CFX-Solver input file (see [User Locations \(p. 377\)](#)), select **User Location Regeneration** to control whether or not these user locations are regenerated (recalculated) as part of the solver run. The options are:

- Automatic

This is the default. If the stored user location data is out-of-date, this option runs CFX-Pre temporarily in the background to regenerate the data at the start of the solver run. A user location is considered to be out-of-date if the settings for the user location have changed since the CFX-Solver input file was written (for example, if the command language in the CFX-Solver input file was edited) or if the user location depends on expressions and there has been *any* change to the definition of *any* CEL expression, including the index for an operating point input parameter table.

- Validate

Stops the run if and when there is any change to the definition of any user location, in which case the user location data is out-of-date. If the user location depends on a CEL expression, and any expression has changed, the run continues but with a warning written to the CFX-Solver Output file.

- None

No user locations are updated during the run; the run proceeds using the user location data stored in the CFX-Solver input file regardless of whether there is any change to the definitions of the user locations or expressions. This option can be used to suppress user location updates if, for example, a user location depends on expressions, and an expression has been changed, but it is known that the expression change will not affect the user location data.

- All

All user locations are forced to be recalculated at the start of the run regardless of whether there is any change to the definitions of the user locations or expressions.

---

**Note:**

This option (All) is effectively selected, regardless of which option is actually selected, when running a design points study in Ansys Workbench.

---

- Selected Locations

The selected user locations are forced to be recalculated at the start of the run regardless of whether there is any change to the definitions of the user locations or expressions. Any other user locations will not be recalculated.

### 37.1.2.6. Interpolator Tab

Select the **Interpolator** tab.

- Under **Run Priority**, you can select `Idle`, `Low`, `Standard` or `High`. For a discussion of these priorities, see [The cfx5control Application in the CFX-Solver Manager User's Guide](#).
- You can override the precision set on the **Run Definition** tab by selecting **Override Default Precision** and then setting the precision. For details on the precision of executables, see [Double-Precision Executables in the CFX-Solver Manager User's Guide](#).
- You can override the problem size capability ("large problem" or not) set on the **Run Definition** tab by selecting **Override Default Large Problem Setting** and then setting the problem size capability (via the **Large Problem** check box). For details on the problem size capability of the executables, see [Large Problem Executables in the CFX-Solver Manager User's Guide](#).
- If required, under **Interpolator Memory**, you can adjust the memory configuration. For details, see [Configuring Memory for the CFX-Solver in the CFX-Solver Manager User's Guide](#).
- You can select **Domain Search Control** to access the **Bounding Box Tolerance** setting (see below).
- You can select **Interpolation Model Control** to access the **Enforce Strict Name Mapping for Phases**, **Particle Relocalization Tolerance**, and **Mesh Deformation Option** settings (see below).

The **Bounding Box Tolerance** and **Particle Relocalization Tolerance** settings are described in [Adjusting the Bounding Box Tolerance in the CFX-Solver Modeling Guide](#).

The **Enforce Strict Name Mapping for Phases** setting controls how fluids are mapped between the Initial Values files and the Solver Input File. By default, when interpolating from multiple Initial Values files onto a single Solver Input File, if each Initial Values file contains just one fluid, then the CFX-Interpolator regards all of these fluids as being the same fluid, regardless of the Fluid Definition name. You can change this default behavior by turning on the **Enforce Strict Name Mapping for Phases** setting. This forces the CFX-Interpolator to use the Fluid Definition names to match each fluid from each Initial Values file with the appropriate fluid in the Solver Input File.

The **Mesh Deformation Option** setting controls which mesh in the Initial Values File is used to check the mesh displacements: the initial mesh or the final mesh. The options are `Automatic`, `Use Latest Mesh`, and `Use Initial Mesh`. When the `Automatic` option is selected, the initial mesh will be used if the **Continue History From** check box is selected, and the latest mesh will be used if the **Continue History From** check box is cleared.

## 37.2. Termination Control

The Termination Control settings apply to the simulations with one or more configurations. This section describes:

### 37.2.1. Overview of Configuration Termination

### 37.2.2. Details View for Termination Control

#### 37.2.1. Overview of Configuration Termination

Many simulations with one or more configurations will terminate naturally without explicitly introducing **Termination Control**. However, in some cases explicit **Termination Control** is required. For example, consider a case where a simulation has a sequence of configurations that are set up to run one after the other with the sequence to then return to the first configuration in the sequence (this could occur when modeling an internal combustion engine where there could be a configuration for each of the intake, compression, power and exhaust strokes and the simulation repeatedly cycles through each of the four configurations). There can be one or more conditions for terminating a simulation. Each termination condition can be based on the number of times a configuration has been executed or whenever a CFX-Solver interrupt condition for a configuration has been satisfied.

To define the conditions under which a simulation should be terminated you need to:

1. If required, create one or more termination **Control Conditions**.
2. For each **Control Condition**, select the appropriate termination control **Option**.
3. For each **Control Condition** set the **Configuration Name** appropriate for the termination control condition.
4. For each **Control Condition**, set the appropriate **Number of Steps** or **Condition Name(s)**.


When you have finished defining how the simulation will terminate, click **OK** or **Apply** to save the settings. Details of the above steps are described in the next section.

#### 37.2.2. Details View for Termination Control

You access the **Termination Control** details view in CFX-Pre by clicking **Insert > Configurations > Termination Control** or by right-clicking **Simulation Control** in the **Outline** tree view and selecting **Insert > Termination Control**.

Details of the **Termination Control** tab:

##### Control Conditions

A list displaying the available termination control conditions. Click *Add new item*  to add a new termination control condition. To change its settings, the termination control condition must be highlighted. You can highlight a condition by selecting it from the displayed list. Click *Delete*



to delete a highlighted termination control condition.

##### Control Condition: Options

The options for termination control are either **Max. Configuration Steps** or **Solver Interrupt Conditions**. The former setting is used to terminate a simulation after a selected configuration has been executed the prescribed number of times. The latter setting is used to terminate a simulation whenever the named CFX-Solver interrupt conditions for the selected configurations have been satisfied. Note that the latter option is only valid if CFX-Solver interrupt conditions have been defined. For details, see [Interrupt Control \(p. 290\)](#).

**Control Condition: Configuration Name**

Choose the configuration for which the termination control is to be applied.

**Control Condition: Number of Steps**

This setting is used to set to the maximum number of times the specified configuration is to be executed in the course of the simulation.

**Control Condition: Condition Name(s)**

This setting is used to identify the names of the solver interrupt condition for the specified configuration to be used to terminate the simulation.



---

# Chapter 38: Configurations

---

This chapter describes how to control the sequencing of configurations for a simulation, how to define when remeshing is required, and procedures for defining how CFX-Solver can be started for a configuration. The **Configuration** settings described below apply to a specified flow analyses in the simulation.

This chapter describes:

- 38.1. Overview of Defining a Configuration
- 38.2. The Details View for Configuration

## 38.1. Overview of Defining a Configuration

---

To set up the sequencing of configurations in a simulation, you need to define a configuration for each step in the simulation. Typically, there is a configuration for each analysis in the simulation. You are required to define at least one activation condition for each configuration. You set up the desired sequencing of configurations in a simulation by your choice of activation conditions. For each required configuration:

1. Create the configuration.
2. Set the name of the analysis to be associated with the configuration.
3. Create the required number of activation controls for the configuration.
4. Set the activation control option to activate the configuration at the start of the simulation or following the completion of another configuration.

Note that it is possible to have more than one configuration activated at the start of the simulation. You also have the option of specifying more than one activation condition for a configuration (for example, a configuration can be activated at the start of the simulation as well as at the completion of another configuration).

To control when remeshing is to occur, you are required to:

1. Create a remeshing definition.
2. Select the appropriate remeshing option.
3. Set up the remeshing activation condition.
4. Identify the location where the remeshing is to occur.
5. Supply any additional information required by the selected remeshing option.

To define how a CFX-Solver can be started, the number of settings that you need to define for **Configuration** depends on the case:

- In some cases, you need only to specify the name of a CFX-Solver input file (\*.def or \*.mdef). For cases that require initialization from previous results, you also need to specify the name of a results file (\*.res).
- You can configure runs in serial or parallel:
  - *Serial run* is the default way of running a CFD case. During a serial run, all computation is done by a single process running on one processor.
  - *Parallel run* partitions the computation into more than one process and is done on more than one processor in a single machine (local parallel processing) or on more than one machine (distributed parallel processing). You also have the option of specifying how the computation is partitioned for a parallel run.
- You can optionally select the system priority for the interpolator and solver computation as well as settings such as precision and memory allocation.

When you have finished setting the parameters for the configuration, click **OK** or **Apply** to save the settings.

Details of the above steps are described in the next section.

## 38.2. The Details View for Configuration

---

You access the details view for a configuration in CFX-Pre by double-clicking its **Simulation Control > Configurations** entry in the **Outline** tree view or by right-clicking its **Simulation Control > Configurations** entry in the **Outline** tree view and selecting **Edit**.

The tabs presented in the details view for the configurations are described in the following sections:



- [General Settings Tab \(p. 462\)](#)
- [Remeshing Tab \(p. 463\)](#)
- [Run Definition Tab \(p. 467\)](#)
- [Partitioner Tab \(p. 468\)](#)
- [Solver Tab \(p. 468\)](#)
- [Interpolator Tab \(p. 468\)](#)

### 38.2.1. General Settings Tab


The **General Settings** tab for a configurations requires that you:

1. Select the **Flow Analysis** for the configuration.
2. Define at least one **Activation Condition**. There are two **Option** values available for each **Activation Condition**:
  - **Start of Simulation** to activate the configuration at the start of the simulation.

- **End of Configuration** to activate the configuration whenever any one of a prescribed configuration completes.

One activation condition is automatically generated for you and the default **Option** is set to **Start of Simulation**. If required, create additional activation conditions by clicking **New** . To change the settings for an activation condition or to delete a condition (by clicking **Delete** ), you must highlight a condition by selecting it from the displayed list.

### 38.2.2. Remeshing Tab

The **Remeshing** tab allows you to introduce one or more remeshing definitions within the configuration being edited. To create a remeshing definition, click **New** . For additional details, see [Remeshing Guide in the CFX Reference Guide](#).

Each remeshing definition requires that you:

1. Select either the **User Defined** or **ICEM CFD Replay** value for the **Option** setting. Additional settings, which depend on the option selected, are described in the sections [User Defined Remeshing \(p. 464\)](#) and [Ansys ICEM CFD Replay Remeshing \(p. 465\)](#), presented below.
2. Select one or more activation condition(s) to be used to activate the remeshing object during the configuration's execution. This selection is made from a list of the solver **Interrupt Control** conditions (for details, see [Interrupt Control \(p. 290\)](#)) that were defined for the **Flow Analysis** specified in the **General Settings** tab.
3. Select the mesh **Location** that will be replaced by remeshing. This selection is made from a list of the 3D mesh regions that are used in the **Flow Analysis** specified in the **General Settings** tab.

Each remeshing definition also allows you to specify a comma separated list of **Mesh Reload Options** that control how the new mesh replaces the previous one. The new mesh could, for example, be reloaded as a .gtm file using [mm] length units and all relevant mesh transformations by specifying:

```
Mesh Reload Options = "replacetype=GTM,replaceunits=mm,notransform=false"
```

These and other options are summarized in the table below.

**Table 38.1: Reload Options**

Reload Option	Description and Values
notransform	True (default) ensures mesh transformations are not performed on mesh reload.  True or False
replacetype	Type of replacement mesh file.  Ansys: cdb and inp files  CFX4: geo files



Reload Option	Description and Values
	<p>CFX5: CFX 5.1 files</p> <p>CGNS: cgns and cgn files</p> <p>GEM: Tfc files</p> <p>GTM: gtm files</p> <p>GtmDirect: def and res files</p> <p>GTM_DSDB: Ansys cmdb and dsdb files</p> <p>Def: def files that are older than CFX 5.6 (or if duplicate node removal is required)</p> <p>Fluent: cas and msh files</p> <p>Generic: ICEM CFD mesh (cfx, cfx5, msh) files</p> <p>GRD: CFX-TASCflow (grd) files</p> <p>IDEAS: unv files</p> <p>MSC: Patran out and neu files</p> <p>PDC: GridPro files</p>
replacegenargs	<p>Generic mesh import options (as space separated list):</p> <p>-g: Ignore degenerate element errors</p> <p>-n: Do not do duplicate node removal</p> <p>-T: specify duplicate node removal tolerance (float).</p> <p>-D: Primitive naming strategy ; either Standard Naming Strategy or Derived Naming Strategy.</p>
replacespecargs	<p>Space-separated list of type-specific import arguments, as discussed in <a href="#">Supported Mesh File Types (p. 123)</a>.</p>
replaceunits	<p>Length units of the replacement mesh.</p> <p>micron, mm, cm, m, in, ft</p>

### 38.2.2.1. User Defined Remeshing

Full control over how the replacement mesh is generated is provided by the **User Defined** remeshing option. When this option is used, a user-defined command is required to gather all input data needed for remeshing and for create the replacement mesh file. The CFX-Solver, however, automatically executes the following tasks:

- Import the new mesh into the problem definition

- Interpolate solution data from the previous mesh onto the new mesh
- Repartition the new mesh if a parallel run mode is used
- Restart the equation solution process.

In addition to the required and optional general settings described above, the **User Defined** option requires specification of:

- An **External Command** that is responsible for generating a replacement mesh file
- The name of the **Replacement File**.

The **External Command** is submitted to the operating system for execution. This may be a command to start a mesh (re)generation executable directly with certain inputs, or a shell script that executes several commands. It is important to note that this command is submitted from the current run directory (for example `case_001.dir`), so care is required when using relative paths to files during remeshing.

Useful inputs to the remeshing process may be extracted from the most recently generated CFX-Solver Results file. For details, see [Remeshing Guide in the CFX Reference Guide](#). This file is located in the run directory, and is simply called `res` (no prefix or suffix) at the time of submitting the **External Command** to the operating system.

For additional details, see [User Defined Remeshing in the CFX Reference Guide](#).

### 38.2.2.2. Ansys ICEM CFD Replay Remeshing

Remeshing using the Ansys ICEM CFD mesh generator is highly automated when the **ICEM CFD Replay** remeshing option is used. This is accomplished by combining settings made in the **Flow Analysis** (specified on the configuration's **General Settings** tab) with a batch run of the Ansys ICEM CFD mesh generator using replay (session) files.

When this option is used the CFX-Solver automatically executes the following tasks:

- Compile a comprehensive remeshing replay file from a combination of provided and user-specified replay files
- Execute the Ansys ICEM CFD mesh generator in batch mode, using the remeshing replay file
- Import the new mesh into the problem definition
- Interpolate solution data from the previous mesh onto the new mesh
- Repartition the new mesh if a parallel run mode is used
- Restart the equation solution process.

In addition to the required and optional general settings described above, the **ICEM CFD Replay** option requires specification of:

- An Ansys ICEM CFD Geometry File (with a `.tin` extension) that contains the reference geometry

- A **Mesh Replay File** (with an `rpl` extension) that contains a recording of the steps (that is, the commands) used to generate the mesh in the Ansys ICEM CFD application.

Additional, optional settings include:

- **ICEM CFD Geometry Control** definitions
- **ICEM CFD Mesh Control** definitions
- **Scalar Parameter** definitions.

For additional details, see [ICEM CFD Replay Remeshing in the CFX Reference Guide](#).

### 38.2.2.2.1. ICEM CFD Geometry Control

**Option** settings for **ICEM CFD Geometry Control** other than **None** are used to modify the reference geometry contained in the ICEM CFD Geometry File according to the mesh motion specifications defined in the CFX case setup. If the geometry control option is set to **Automatic**, then one or more **ICEM CFD Part Map** definitions may be defined. Each definition provides a mapping between:

- An **ICEM CFD Parts List**, which is a list of parts (or families) defined in the referenced **Geometry File**
- The translation of the centroid of a **Boundary** defined in the **Flow Analysis**.

These definitions are applied, in conjunction with the default geometry control replay file (`icemcfd_GeomCtrl.rpl` contained the `<CFXROOT>/etc/Remeshing` directory), to modify the reference geometry prior to regenerating the mesh. If the geometry control option is set to **User Defined Replay File**, then a **File Name** is required and the specified file is used instead of the default geometry control replay file.

### 38.2.2.2.2. ICEM CFD Mesh Control

**Options** settings for **ICEM CFD Mesh Control** other than **None** are used to set values of some pre-defined parameters used by Ansys ICEM CFD during remeshing. If the mesh control option is set to **Automatic**, then one or more **ICEM CFD Part Parameter** definitions may be defined. Each definition provides a mapping for an **ICEM CFD Parameter** that governs mesh attributes like the maximum element size (`emax`) or the maximum element height (`ehgt`), between:

- An **ICEM CFD Parts List**, which is a list of parts (or families) defined in the referenced **Geometry File**
- A **Monitor Point** defined in the **Flow Analysis**.

These definitions are applied in conjunction with the default mesh control replay file (`icemcfd_MeshCtrl.rpl` contained the `<CFXROOT>/etc/Remeshing` directory), to modify the reference geometry prior to regenerating the mesh. If the mesh control option is set to **User Defined Replay File**, then a **File Name** is required and the specified file is used instead of the default mesh control replay file.

### 38.2.2.2.3. Scalar Parameter

**Scalar Parameter** definitions are used to set values of additional pre- or user-defined parameters referenced in any of the replay files used by Ansys ICEM CFD during remeshing. Each definition provides a mapping between a scalar parameter used during remeshing (with the same name as the **Scalar Parameter** definition) and a **Monitor Point** defined in the **Flow Analysis**.

The parameters listed in the table below are used in the default geometry control replay file, and become relevant if a **Scalar Parameter** definition is created with the same name.

**Table 38.2: Scalar Parameters**

Scalar Parameter	Description
ICEM CFD Geometry Scale	The specified scale is used to address length unit differences between the geometry contained in the specified Ansys ICEM CFD <b>Geometry File</b> and the mesh contained in the CFX-Solver Input file. For example, if the length unit is [mm] in the Ansys ICEM CFD geometry and [m] in the CFX-Solver InputCFX-Solver Input file, then the geometry scale should be set to 0.001.
OFFSET X PartName	The specified offset values are added to the centroid displacements (see the discussion on <b>ICEM CFD Geometry Control</b> presented above) that are applied for the part (or family) named PartName. Note that the <b>ICEM CFD Geometry Scale</b> is also applied to the offset specified offset.
OFFSET Y PartName	
OFFSET Z PartName	

### 38.2.3. Run Definition Tab

Select the **Configuration Execution Control** option to override the global Execution Control settings for the configuration. Note that, when this is selected, the configuration will not inherit any settings from the global Execution Control settings. Once the **Configuration Execution Control** option is selected, settings on the **Run Definition**, **Initial Values**, **Partitioner**, **Solver**, and **Interpolator** tabs become available.

The remaining settings on the **Run Definition** tab are the same as the equivalent settings described in [Run Definition Tab \(p. 450\)](#) (once the **Configuration Execution Control** setting is selected), although only the settings that are relevant for **Configuration Execution Control** are available on the tab. You can override these settings for the specific configuration from the **Define Run** dialog box in the CFX-Solver Manager (for details, see [The Define Run Dialog Box in the CFX-Solver Manager User's Guide](#)).

### 38.2.4. Initial Values Tab

These settings are essentially the same as those described in [Initial Values Tab \(p. 451\)](#).

For configurations, when choosing the source of initial values for the initial values object, an additional option is available: `Configuration Results`. To use this option, specify the name of the configuration from which initial values should be obtained. This option allows the results (which become available at run time) of another configuration to be used as initial values for the configuration being defined.

### **38.2.5. Partitioner Tab**

These settings are the same as those described in [Partitioner Tab \(p. 453\)](#) except that the option to specify an initial partition file is not available for configurations.

### **38.2.6. Solver Tab**

These settings are the same as those described in [Solver Tab \(p. 455\)](#).

### **38.2.7. Interpolator Tab**

These settings are the same as those described in [Interpolator Tab \(p. 457\)](#).

---

# Chapter 39: Quick Setup Mode

---

Quick Setup mode is a mode of operation for CFX-Pre that greatly simplifies the physics setup for a case. More complex physics, such as multiphase, combustion, radiation, advanced turbulence models, and so on are not available in Quick Setup mode. You can, however, use Quick Setup mode to get started, and then add more physics details later.

---

## Note:

You can switch to Quick Setup mode by selecting **Tools > Quick Setup Mode** from the main menu.

---

This chapter describes:

- 39.1. Starting a New Case in Quick Setup Mode
- 39.2. Simulation Definition Tab
- 39.3. Physics Definition
- 39.4. Boundary Definition
- 39.5. Final Operations

## 39.1. Starting a New Case in Quick Setup Mode

---

To start a new simulation in Quick Setup mode:

1. Start CFX-Pre.
2. Select **File > New Case** from the main menu.

The **New Case** dialog box appears.

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

The **Simulation Definition** area opens.

## 39.2. Simulation Definition Tab

---

Under **Simulation Definition**, you can set the analysis type data, the fluid data, and import a mesh file.

- [Simulation Data \(p. 470\)](#)
- [Working Fluid \(p. 470\)](#)
- [Mesh Data \(p. 470\)](#)

## 39.2.1. Simulation Data

A short description of each of the **Problem Type** options is displayed in CFX-Pre.

### 39.2.1.1. Problem Type

**Problem Type** can be set to any of the following:

- `Single Phase`

Only one fluid is present in a single phase simulation, and it is usually a pure substance.

- `Multi-Component`

If this option is selected, the simulation is used to model the average properties of a mixture of chemical species.

- `Multi-Phase`

This option contains more than one fluid, each of which is modeled separately. In general, unlike multi-component simulations, the fluids are of different chemical species.

## 39.2.2. Working Fluid

Under **Working Fluid**, you select the fluids for use in the domain.

- **Fluid(s)**

If **Analysis Type** is set to `Single Phase`, you may select only one fluid for the domain. If `Multi-Phase` was chosen, you may select at least two fluids.

- **Mixture**

If you are defining a multi-component simulation, you must provide a name for your custom material, which is defined by the fluids specified under **Components**.

- **Components**

Select the fluids you plan to use in the simulation from this drop-down menu. At least two fluids are required.

## 39.2.3. Mesh Data

- **Mesh File**

Click *Browse*  to open the **Import Mesh** dialog box and search for the mesh file to import.

The most common mesh file formats can be imported in Quick Setup mode. If other mesh formats, advanced options or user import methods are required, General mode should be used.

For details, see [Importing and Transforming Meshes \(p. 121\)](#).

- **Available Volumes > 3D Regions**

Using the drop-down menu, select the 3D region you want to use for the domain. By default, all 3D regions of mesh from the selected mesh file will be selected.

## 39.3. Physics Definition

---

Under Physics Definition, you will set the type of simulation and specify model data such as the pressure, heat transfer, and turbulence options.

- [Analysis Type \(p. 471\)](#)
- [Model Data \(p. 471\)](#)

### 39.3.1. Analysis Type

Select the analysis type: `Steady State` or `Transient`.

- `Steady State`

No further settings are required.

- `Transient`

If `Transient` is selected, set the **Total Time** and **Time Step** values for the transient simulation.

### 39.3.2. Model Data

- `Reference Pressure`

Set a value for the reference pressure. For details, see [Setting a Reference Pressure in the CFX-Solver Modeling Guide](#).

- `Heat Transfer`

Select the heat transfer model. For details, see [Heat Transfer: Option \(p. 176\)](#).

- `Turbulence`

Select the turbulence model. For details, see [Turbulence: Option \(p. 178\)](#).

## 39.4. Boundary Definition

---

CFX-Pre will automatically create boundary conditions based on name matching criteria, but you can define your own as follows:

1. Right-click in the blank area and select **New** to create a new boundary condition.  
A dialog box will pop up and ask for the name of the boundary condition you want to create.
2. Accept the default name or enter a new name.



3. Click **OK**.
4. Set **Boundary Type** to one of: Inlet, Outlet, Symmetry, or Wall.

Opening type boundary conditions are not available using Quick Setup mode.

5. Set the location (2D Region) for the boundary condition.

You may use the **Ctrl** key to multi-select a group of 2D regions for the single boundary condition.

6. Fill in information specific to the type of boundary condition.

Additional information for a particular setting is available. For details, see [Boundary Details Tab \(p. 221\)](#).

7. Repeat the preceding steps for each remaining boundary condition you want to create.

You may delete any boundary conditions that are not required for the simulation simply by right-clicking the boundary condition in the list and selecting **Delete** from the shortcut menu.

A default boundary condition will be created automatically for any 2D regions on the boundary of the domain, which have not been assigned a boundary condition. By default, the default boundary condition is a no-slip adiabatic wall.

8. Click **Next** to continue.

## 39.5. Final Operations

---

The final step enables you to select from various options.

---

### Important:

If there are additional settings you need to address that are not covered in Quick Setup mode, you must select `Enter General Mode`. The other two options will automatically write a solver (`.def`) file based on the settings defined in Quick Setup mode.

---

1. Select one of these options:
  - `Start Solver` enters General mode, writes the solver (`.def`) file, and passes it to the CFX-Solver Manager.
  - `Start Solver and Quit` (available in stand-alone mode) writes the solver (`.def`) file, passes it to the CFX-Solver Manager, and then shuts down CFX-Pre.
  - `Enter General Mode` enters General mode in CFX-Pre.
2. Click **Finish** to exit Quick Setup mode.

---

# Chapter 40: Turbomachinery Mode

---

Turbo mode in CFX-Pre is a specialist mode enabling you to set up turbomachinery simulations, such as compressors or turbines, in a simple manner. Each component of a rotating machine can be simply defined by selecting a mesh file and some basic parameters and then the boundary conditions and interfaces between the components are automatically generated. In addition to the quick setup, existing turbomachinery simulations can be easily modified to use alternative meshes or to add extra components with minimum effort. Turbo mode is designed to complement Ansys TurboGrid but supports all the common mesh file formats that are supported in the General mode.

Consider an axial turbine made up of three stages (three stators and three rotors). In a design process you may first want to construct six individual cases in order to check the flow around each individual component. Next you might want to analyze some of the "stator-rotor-stator" portions of the machine, creating three more analysis cases. Once these analyses are complete, you might want to re-design one of the rotors, and re-create the stator-rotor-stator analysis with the new rotor. Other cases to investigate might include multiple blade passages for some/all components, and ultimately you will want to analyze the entire three stage (six component) machine. Turbo mode is designed to handle all of these cases.

---

## Note:

- You can switch to Turbo mode when working on a general simulation by selecting **Tools > Turbo Mode** at any time during the simulation.
- **Turbo Mode** is designed specifically for the setup of Turbo cases, so if it is used for unsuitable cases some data may be lost.
- After setting up a turbomachinery simulation in CFX-Pre, the simulation setup can become invalid if you make changes to the mesh or mesh topology, including changes that affect the number of blade passages. Re-running Turbo mode (available from the **Tools** menu in CFX-Pre) might fully or partially correct the setup. If necessary, any remaining boundary inconsistencies can be corrected in General mode.

---

This chapter describes:

- [40.1. Starting a New Case in Turbo Mode](#)
- [40.2. Navigation through Turbo Mode](#)
- [40.3. Basic Settings](#)
- [40.4. Component Definition](#)
- [40.5. Physics Definition](#)
- [40.6. Interface Definition](#)
- [40.7. Disturbance Definition](#)
- [40.8. Boundary Definition](#)
- [40.9. Final Operations](#)

## 40.1. Starting a New Case in Turbo Mode

---

To start a new case in Turbo mode:

1. Launch CFX-Pre from the Ansys CFX Launcher.
2. Select **File > New Case** from the main menu.
3. Select **Turbomachinery** and click **OK**.

---

### Note:

You do not need to specify a filename until the end of Turbo mode, as you will either specify the name for the `.def` file on the **Final Operations** panel, or you will return to General mode and make any further required changes to your simulation definition.

---

## 40.2. Navigation through Turbo Mode

---

There are a number of buttons available in Turbo mode to navigate through the wizard. In the most part, these behave in the standard manner (that is, **Next** moves to the following page, **Back** moves to the previous page, and so on).

The **Cancel** button:

- Will close Turbo mode (revert back to General mode)
- Will not cause data to be committed, except for any changed component definitions
- Will not delete from the simulation any meshes that have already been imported while in Turbo mode.

## 40.3. Basic Settings


---

When Turbo mode is first entered, the **Basic Settings** panel appears.


### 40.3.1. Machine Type

The machine type can be any one of Pump, Fan, Axial Compressor, Centrifugal Compressor, Axial Turbine, Radial Turbine, or Other. In all cases, as part of the Turbo System functionality, the machine type will be part of the data passed between CFX-Pre and CFD-Post in order to aid workflow.

### 40.3.2. Axes

The axis of rotation for the turbo component is set relative to the Global Coordinate frame (Coord 0) by default. You can choose a user-specified coordinate frame from the drop-down list or create a new one by selecting . For details, see [Coordinate Frames \(p. 371\)](#). For details, see [Coordinate Frames in the CFX-Solver Modeling Guide](#).

### 40.3.2.1. Coordinate Frame

Choose a Coordinate Frame or click the  to create a new coordinate frame. For details, see [Coordinate Frames \(p. 371\)](#). For details, see [Coordinate Frames in the CFX-Solver Modeling Guide](#).

### 40.3.2.2. Rotational Axis

Select the rotational axis relative to the chosen coordinate frame.

### 40.3.2.3. Axis Visibility

Toggle to select the visibility of the axis.

## 40.3.3. Analysis Type

The analysis type determines whether your problem is `Steady State`, `Transient`, or `Transient Blade Row`. If the problem is transient, set `Total Time` to the total simulation time and **Time Steps** to the timestep increment. For example, to study 10 timesteps over 0.1 seconds, set **Total Time** to 0.1 [s] and **Time Steps** to 0.01 [s]. If you choose the `Transient Blade Row` option under `Analysis type`, you have the option of choosing between the `Time Transformation` and `Fourier Transformation` methods under **Methods**.

For details, see [Analysis Type \(p. 161\)](#).

## 40.4. Component Definition

The **Component Definition** panel is used to set up the component names, and import and/or transform the meshes used in the simulation.

### 40.4.1. Component Selector

Displays the components currently being used in the simulation. Components can be added, removed and altered using the commands available through right-clicking in the component selector area.

Command	Action
New Component	Creates a new component.
Delete	Deletes an existing component. Does not delete the mesh.
Delete Component and Mesh	Deletes both an existing component and the component's mesh. This command is only available in stand-alone mode; it is not available in Workbench.
Move Component Up	Moves the selected component up in the component list.
Move Component Down	Moves the selected component down in the component list.


Command	Action
Axial Alignment	Automatically moves meshes, along the axis of rotation, in order to align the inlet and outlet regions.

You must ensure that the components are ordered correctly in the selector from top (inlet end) to bottom (outlet end).

### 40.4.2. Component Type

If the component is rotating, the angular speed is required under **Value**. The rotation axis is defined on the **Basic Settings** panel. For details, see [Rotational Axis \(p. 475\)](#).

### 40.4.3. Mesh File

Click **Browse**  to assign a mesh file to the selected component. The **Import Mesh** dialog box will appear requesting the filename, location, and file type. For some file types, the mesh units must be specified; this is indicated by the presence of a **Mesh Units** drop-down menu. If a mesh file has previously been specified, selecting a new mesh file will result in the original file being deleted and the new mesh being imported. In addition to this, the **Reload Mesh Files** feature is available in General mode that allows multiple mesh files to be replaced at once. For details, see [Reload Mesh Files Command \(p. 69\)](#).

---

#### Note:

The **Reload Mesh Files** feature is not required (and is not available) in CFX-Pre launched from Ansys Workbench.

---

If you want to use a mesh volume that has already been imported as part of another component, do not specify a mesh file here and set **Available Volumes** > **Volumes** to the appropriate region.

### 40.4.4. Passages and Alignment

This section shows the details of the mesh geometry. In the default state, it will display the number of blades in the mesh file and the number in 360 degrees. To change these values click the **Edit** button.

For quick response, the **Preview** button provides immediate feedback of the changes made. Once the correct settings are selected, clicking **Done** will apply the transformation. You can select **Cancel** to discard your latest change or select **Reset** to return all parameters to their default values.

#### 40.4.4.1. Passages/Mesh

**Passages Per Mesh** is an indication of the number of blade passages that exist in the selected mesh file. The value will normally be 1.

### 40.4.4.2. Passages to Model


This parameter is used to specify the number of passages in the section being modeled. This parameter is optional for most cases except for Transient Blade Row using the Fourier Transformation method. For the Fourier Transformation method, you have to specify at least two passages. This value is used in CFD-Post. For details, see [Fourier Transformation in the CFX-Solver Modeling Guide](#).

### 40.4.4.3. Passages in 360

This parameter is optional and is used to specify the number of passages in the machine. This value is used in CFD-Post.

If this value is not specified it is automatically calculated based on how many copies of the mesh are required for modeling a full 360 degree section.

### 40.4.4.4. Theta Offset

Rotates the selected mesh, about the rotational axis, through an angle theta. The offset can be a single value or set to an expression by clicking .

## 40.4.5. Available Volumes

Set **Volumes** to the 3D region(s) that apply for the selected component.

Normally this will not be required—it simply contains all of the mesh volumes for the mesh file specified above. However, if you want to set up a case where a single mesh file contains the meshes for multiple components, select the appropriate mesh volume here.

## 40.4.6. Region Information

This section shows the names of the mesh regions that have been recognized as potential boundaries and interfaces. This name matching is done using the template names provided, which can be configured for your particular mesh files as appropriate. If the list of names that is shown is incorrect or incomplete, these can be modified accordingly. In the default case, this data should not need changing.

Turbo mode in CFX-Pre does not fully support meshes generated using the **Split Mesh Regions at Trailing Edge** setting in Ansys TurboGrid. For meshes created using this setting, it may be necessary to manually adjust the regions selected under **Region Information** in order to obtain the correct interfaces and boundaries.

## 40.4.7. Wall Configuration

The **Wall Configuration** settings, which are available only for rotating components, are **Tip Clearance at Shroud** and **Tip Clearance at Hub**. A setting of **Yes** causes the applicable wall (shroud or hub) to be counter-rotating in the rotating frame of reference such that it is stationary in the absolute frame of reference.

Note that each of the settings affects only whether the applicable wall is counter-rotating or not. Thus, in the case of a hub-mounted swiveling blade with a clearance at the hub, it is true that the

blade rotates with the hub, and so, for that case, the appropriate setting for **Tip Clearance at Hub** is **No**.

The default values for the **Wall Configuration** settings are **Yes** for **Tip Clearance at Shroud** and **No** for **Tip Clearance at Hub**.

## 40.5. Physics Definition

---

Properties of the fluid domain and solver parameters are specified on the **Physics Definition** panel.

### 40.5.1. Fluid

Choose a Fluid from the list. Only one is permitted.

### 40.5.2. Model Data

#### 40.5.2.1. Reference Pressure

This sets the absolute pressure level that all other relative pressure set in a simulation are measured relative to. For details, see [Setting a Reference Pressure in the CFX-Solver Modeling Guide](#).

#### 40.5.2.2. Heat Transfer

This model selection depends upon the type of fluid you have chosen. For details, see [Heat Transfer in the CFX-Solver Modeling Guide](#).

#### 40.5.2.3. Turbulence

The models available depend upon the fluid that has been chosen. For details, see [Turbulence and Near-Wall Modeling in the CFX-Solver Modeling Guide](#).

#### 40.5.2.4. Mass Flow

- Per Component  
Specify the mass flow per modeled (mesh) sector.
- Per Machine  
Specify the total mass flow (that is, for the full wheel).
- Per Passage  
Specify the mass flow per blade set (including splitters).

### 40.5.3. Boundary Templates

Select from one of the commonly used configurations of boundary conditions or choose None. The configurations are listed from the least robust option to the most robust. For details, see [Recommended Configurations of Boundary Conditions in the CFX-Solver Modeling Guide](#).

### 40.5.4. Interface

Select a default interface type that will be applied to components. This can be modified later on a per-component basis.

### 40.5.5. Solver Parameters

#### 40.5.5.1. Advection Scheme

For details, see [Advection Scheme Selection in the CFX-Solver Modeling Guide](#).

#### 40.5.5.2. Convergence Control

This option is only available for steady-state simulations and determines how the timestep size is used to aid convergence. You can select `Automatic` (timestep size controlled by the solver) or `Physical Timescale` (enter a timestep value).

#### 40.5.5.3. Time Scale Option

This option is only available for steady-state simulations. The automatic time scale algorithm can be either conservative or aggressive. For details, see [Automatic Time Scale Calculation in the CFX-Solver Theory Guide](#).

#### 40.5.5.4. Max. Coeff. Loops

This option is only available for transient simulations. With a default value of 10, this option determines the maximum number of iterations per timestep. For details, see [Max. Iter. Per Step in the CFX-Solver Modeling Guide](#).

## 40.6. Interface Definition

---

Domain interfaces are used to connect multiple assemblies together, to model frame change between domains, and to create periodic regions within and between domains. Domain interfaces are automatically generated based on the region information.

- The list box shows the existing interfaces. You can right-click and select **New** to create a new interface or **Delete** to delete an existing one.
- Clicking on an interface from the list allows for the viewing and editing of its properties, including **Side 1**, **Side 2** and **Type**. For details, see [Type \(p. 480\)](#).

CFX-Pre will automatically attempt to determine the frame change and periodic regions. For details, see [Type \(p. 480\)](#). Such interfaces are named according to the following: for two domains, A and B:



- Internal Interface connection in domain A: `A Internal Interface`

---

**Note:**

For transient blade row cases using the Fourier Transformation method, a new internal interface is created between the two blade passages.

---

- Periodic connection in domain A: `A to A Periodic`
- Periodic connection in domain B: `B to B Periodic`
- Fluid-Fluid interface between domain A and B: `A to B <type>` or `B to A <type>`

where `<type>` can be one of `Frozen Rotor`, `Stage`, or `Transient Rotor Stator`. For details, see [Type](#) (p. 480).

Turbo mode in CFX-Pre supports automatic interface detection and creation only when each side of the interface consists of matching regions. The mesh does not need to match 1:1, but each region on one side of the interface must have a corresponding region on the other side of the interface. Meshes that do not satisfy this requirement can be used in Turbo mode, but it will be necessary to inspect and modify the interfaces that are created automatically in order to ensure that the interfaces are correct.

### 40.6.1.Type

The frame change interfaces model the interface between a rotating assembly and a stationary assembly. For details, see [Frame Change/Mixing Model in the CFX-Solver Modeling Guide](#). When the analysis type is steady state, four options are available to model frame change:

- None
- Stage: For details, see [Stage \(Mixing Plane\) in the CFX-Solver Modeling Guide](#).
- Periodic
- Frozen Rotor: For details, see [Frozen Rotor in the CFX-Solver Modeling Guide](#).

For transient or Transient Blade Row simulations, the following four options are available to model frame change:

- None
- Stage: For details, see [Stage \(Mixing Plane\) in the CFX-Solver Modeling Guide](#).
- Periodic
- Transient Rotor-Stator: For details, see [Transient Rotor-Stator in the CFX-Solver Modeling Guide](#).

In addition, a frame change interface of type `None` is created for tip clearance regions and disconnected regions of mesh within a component (for example between an inlet section and a blade section). Periodic interfaces are used in regions where a portion of the flow field is repeated in many identical

regions. The flow around a single turbine blade in a rotating machine, or around a single louvre in a whole array in a heat exchanger fin are such examples.

## 40.7. Disturbance Definition

---

For a Transient Blade Row simulation, the **Disturbance Definition** panel is used to define the type of disturbance being modeled for the Time Transformation and Fourier Transformation methods.

### 40.7.1. Type

The following options are available under **Type**:

- Rotor Stator

This option is available for the Time Transformation and Fourier Transformation methods, and can be applied only when the disturbance originates from a domain interface that uses the Transient Rotor Stator frame change/mixing model.

- Rotational Flow Boundary

Use this option to characterize a disturbance that originates from a boundary condition (for example, an inlet or outlet boundary condition that is specified using one or more CEL expressions that depend on space and time).

For details, see [Transient Blade Row Models Details View \(p. 353\)](#).

## 40.8. Boundary Definition

---

The **Boundary Definition** panel is used to set boundary conditions for the remaining surfaces in the simulation.

- The list box shows the existing interfaces. You can right-click and select **New** to create a new interface or **Delete** to delete an existing one.
- The **Flow Specification** options (**Wall Influence On Flow**) vary with the boundary type. For details, see [Flow Specification/Wall Influence on Flow \(p. 482\)](#).

CFX-Pre uses the information gained from domain interfaces and region specification to automatically create the required boundary condition locations, in addition to any template boundary configuration that you have chosen. You can check the definition for each one by clicking on a boundary and viewing the properties displayed. You can change the properties of any of the automatic boundary conditions. You should ensure that you set the parameter values for the inlet and outlet boundary conditions because they will assume default values.

### 40.8.1. Boundary Data

Specify the boundary type and select the location from the drop-down list. Alternatively, while the **Location** list is active, click in the viewer to directly select the surface.

## 40.8.2. Flow Specification/Wall Influence on Flow

The options available for **Flow Specification** and **Wall Influence on Flow** will depend on the boundary type. For information on each type of boundary, refer to the relevant section in the Boundary Conditions documentation:

- [Boundary Details: Inlet \(p. 222\)](#)
- [Boundary Details: Outlet \(p. 222\)](#)
- [Boundary Details: Opening \(p. 223\)](#)
- [Boundary Details: Wall \(p. 223\)](#)
- [Symmetry Boundary Conditions \(p. 235\)](#)

## 40.9. Final Operations

---

1. Select one of the following operations:

- `Start Solver` enters General mode, writes the solver (`.def`) file with the specified name and passes it to the CFX-Solver Manager.
- `Start Solver and Quit` writes the solver (`.def`) file with the specified name, passes it to the CFX-Solver Manager and shuts down CFX-Pre. This option is not available when running CFX in Ansys Workbench.
- `Enter General Mode` simply enters General mode without writing any files.

If you are running CFX-Pre in Ansys Workbench, the only available option is `Enter General Mode`. In this case, the solver can be started from Ansys Workbench.

2. Click **Finish** to exit Turbo mode.

---

# Chapter 41: Library Objects

---

Model library templates contain CCL physics definitions for complex simulations. Loading a Model Library imports the CCL definition contained in a template file.

Once this is done, a mesh can be imported and the locations of domains, boundaries, and so on can be assigned. Template files are located in `<CFXROOT>/etc/model-templates/` directory (CFX-Pre will open this directory when a new simulation is created in Library mode).

Model libraries are designed to simplify the problem setup of simulations involving complex physics. Once the problem definition is loaded, CFX-Pre enters General mode, allowing changes to any or all parameters.

Selecting **Library Template** from the **New Case** dialog box in CFX-Pre enables you to choose one of the following simulation model templates:

- [Boiling Water \(p. 483\)](#)
- [Cavitating Water \(p. 484\)](#)
- [Coal Combustion \(p. 484\)](#)
- [Comfort Factors \(p. 484\)](#)
- [Evaporating Drops \(see \[Liquid Evaporation Model in the CFX-Solver Modeling Guide\]\(#\)\)](#)
- [Multigray Radiation \(p. 485\)](#)
- [Oil Combustion \(see \[Liquid Evaporation Model: Oil Evaporation/Combustion in the CFX-Solver Modeling Guide\]\(#\)\)](#)
- [Spray Dryer \(see \[Liquid Evaporation Model: Spray Dryer with Droplets Containing a Solid Substrate in the CFX-Solver Modeling Guide\]\(#\)\)](#)

## 41.1. Boiling Water

---

The boiling water model template contains domain settings for a simulation modeling the boiling of water. The domain `boiling_device` is specified with two fluids: `Water at 100 C` and `Water Vapour at 100 C`. `Water at 100 C` is the continuous phase and `Water Vapour at 100 C` is the dispersed phase. The inhomogeneous multiphase model is employed. For details, see [The Inhomogeneous \(Interfluid Transfer\) Model in the CFX-Solver Modeling Guide](#).

Boiling is modeled by setting the **Mass Transfer** option on the **Fluid Pair Models** tab to `Phase Change` (which uses the Thermal Phase Change model and requires a saturation temperature).

Review all settings applied to the simulation and create suitable boundary conditions. For details, see [Thermal Phase Change Model in the CFX-Solver Modeling Guide](#). Initialization data must also be set. For details, see [Initial Conditions for a Multiphase Simulation in the CFX-Solver Modeling Guide](#).

## 41.2. Cavitating Water

---

The cavitation model template contains domain settings and fluid models for a cavitating flow. The domain `cavitating_device` is specified with two fluids, `Water` at 25 C and `Water Vapour` at 25 C. The homogeneous multiphase model is employed. [The Homogeneous Model in the CFX-Solver Modeling Guide](#).

The `Rayleigh Plesset` model is used to model cavitation in the domain. For details, see [Rayleigh Plesset Model in the CFX-Solver Modeling Guide](#).

Boiling is modeled by setting the **Mass Transfer** option on the **Fluid Pair Models** tab to `Phase Change` (which uses the Thermal Phase Change model and requires a saturation temperature).

Review all settings applied to the simulation and create suitable boundary conditions. For details, see [Cavitation Model in the CFX-Solver Modeling Guide](#). Initialization data must also be set; for details, see [Initial Conditions for a Multiphase Simulation in the CFX-Solver Modeling Guide](#).

## 41.3. Coal Combustion

---

The coal combustion analysis template contains all the material definitions to perform a coal calculation using proximate/ultimate input data. Analysis data is provided for a commonly used coal. The template includes a global single-step devolatilization mechanism, together with materials and reactions for performing NO<sub>x</sub> calculations, including fuel NO<sub>x</sub> generation and NO reburn.

Further information on proximate/ultimate analysis is available. For details, see [Hydrocarbon Fuel Model Setup in the CFX-Solver Modeling Guide](#).

## 41.4. Comfort Factors

---

There are expressions for calculating Mean Radiant Temperature and Resultant Temperature for use in HVAC simulations. Resultant Temperature is a comfort factor defined in [92]. Two options are available:

- Derive the factors during postprocessing, as user scalar variables.

A CFD-Post macro is available for this purpose, and is accessed using the **Macro Calculator** from the **Tools** menu. This is the most common approach. For details, see [Comfort Factors Macro in the CFD-Post User's Guide](#).

- Compute them as run time Additional Variables.

This method is best used when the control system depends upon a derived comfort factor. The approach involves using the `comfort-factors.ccl` library file in CFX-Pre.

Most users are likely to prefer the first option, but sometimes the second option will be required, for example when the model simulates a ventilation system in which the control system depends dynamically on derived comfort factors.

The model library template creates two Additional Variables:  $PMV$  (Predicted Mean Vote) and  $PPD$  (Predicted Percentage of Dissatisfied), which are comfort factors defined in [93].

A User Fortran routine named `usr_pmvppd.F` has been developed for computing the values of  $PMV$ , and can be found in the `etc/model-templates/` directory of your Ansys CFX installation. The template contains a CCL definition for the user routine named `pmvppd`, which calls the Fortran routine. Values for  $U, V, W$ , temperature and radiation intensity are passed to the routine and the dimensionless value of  $PMV$  is returned. The value is then used to calculate  $PPD$  based on the formula:

$$PPD = 100 - 95 \times \exp(-0.03353 \times PMV^4 - 0.2179 \times PMV^2) \quad (41.1)$$

Only a fixed value for humidity for  $PMV$  and  $PPD$  can be used at the present time. The values should be supplied as partial pressure of water vapor.

Radiation and the ISO tables for metabolic rate and clothing thermal resistance are included in the template file, which can be accessed by opening the following file in a text editor:

```
<CFXROOT>/etc/model-templates/comfort-factors.ccl
```

You can also use customized values pertinent to your simulation. Full details are given in the template file itself.

Compiling the routine requires the use of the `cfx5mkext` utility. For details, see [Creating the Shared Libraries in the CFX-Solver Modeling Guide](#).

An absolute **Library Path** must be set for the User Routine. For details, see [User CEL Routines \(p. 439\)](#).

## 41.5. Multigray Radiation

The multigray radiation template contains domain settings for a simulation modeling combustion. The domain `combustor` is specified with a methane/air burning mixture. It is solved using the Eddy Dissipation combustion model, the Discrete Transfer thermal radiation model and the Multigray spectral model.

Review all settings applied to the simulation and create suitable boundary conditions. For details, see:

- [Combustion Modeling in the CFX-Solver Modeling Guide](#)
- [Radiation Modeling in the CFX-Solver Modeling Guide](#).

Additional information on multigray radiation is available; for details, see:

- [Eddy Dissipation Model in the CFX-Solver Modeling Guide](#)
- [The Discrete Transfer Model in the CFX-Solver Modeling Guide](#)
- [Spectral Model in the CFX-Solver Modeling Guide](#).



---

# Chapter 42: Command Editor Dialog Box

---

The **Command Editor** dialog box is a text interface for CFX-Pre and CFD-Post. You can use it to issue commands or to create or modify the CCL that defines the state.

This chapter describes:

- 42.1. Using the Command Editor
- 42.2. Performing Command Actions
- 42.3. Using Power Syntax

## 42.1. Using the Command Editor

---

To start the **Command Editor**:

1. Select **Tools > Command Editor**. Alternatively, right-click any object that can be modified using the **Command Editor** and select **Edit in Command Editor**.
  - If you select **Tools > Command Editor**, the **Command Editor** opens and displays the current state regardless of any selection.
    - If the **Command Editor** dialog box has not been used previously, it will be blank.
    - If the **Command Editor** dialog box has been used previously, it will contain CCL commands. If you do not want to edit the CCL that appears, click **Clear** to erase all content.
  - If you right-click an object and select **Edit in Command Editor**, the CCL definition of the specific object populates the **Command Editor** automatically. Modify or add parameters as required, then process the new object definition to apply the changes.
2. Click in the **Command Editor**.
3. Prepare the content of the **Command Editor** by adding new content, modifying the existing content, or both.

The types of content that may be prepared are CCL, action commands, and power syntax. Combinations of these types of content are allowed. For details, see:

- [CFX Command Language \(CCL\) Syntax in the CFX Reference Guide.](#)
- [Command Actions in the CFD-Post User's Guide.](#)
- [Power Syntax in Ansys CFX in the CFX Reference Guide.](#)

Right-click in the **Command Editor** to access basic editing functions. These functions include **Find**, which makes a search tool appear at the bottom of the **Command Editor** dialog box. Enter a search



term and click either **Next** or **Previous** to search upwards or downwards from the insertion point or text selection. To hide the search tool, press **Esc**.

4. Click **Process**.

The contents are processed: CCL changes will affect CCL object definitions, actions will be carried out, and power syntax will be run.

To replace the CCL currently displayed in the **Command Editor** with CCL in a file:

1. From the **Command Editor**, right-click **Import**. The **Import CCL** dialog box is displayed. For details, see [Import CCL Command \(p. 70\)](#).
2. Select the appropriate file.
3. Click **Open**. Note that independent of the **Import Method** selection on the **Import CCL** dialog, the CCL in the **Command Editor** is always replaced by the CCL loaded from the file.
4. Click **Process**.

## 42.2. Performing Command Actions

---

Many of the operations performed in the user interface can also be driven by processing a command action in the **Command Editor**. For example, you can delete a domain named `Domain1` by processing the following command:

```
>delete /FLOW/DOMAIN:Domain1
```

Command actions are preceded by a prompt consisting of a right angle bracket (>). For details, see [Command Actions in the CFD-Post User's Guide](#).

## 42.3. Using Power Syntax

---

Power Syntax (Perl) commands can be issued through the **Command Editor**. This can be useful when performing repeated operations such as creating multiple objects with similar properties. Any valid power syntax command can be entered in the editor and processed. Power Syntax commands are preceded by a prompt consisting of an exclamation mark (!). For details, see [Power Syntax in Ansys CFX in the CFX Reference Guide](#).