CFD EXPERTS

Simulate the Future

WWW.CFDEXPERTS.NET

Ansys 2021 R2

Engineering What's Ahead.

Ansys Fluent in Ansys Workbench User's Guide

Ansys

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

Release 2021 R2
July 2021

Published in the U.S.A.

Table of Contents

List of Figures

List of Tables

Using This Manual

This preface is divided into the following sections:

1. The Contents of This Manual

This document provides information about using the Fluent application within Workbench.

A brief description of what is in each chapter follows:

- Getting Started With Fluent in Workbench (p. 21), describes an overview of Fluent within Workbench.
- Working With Fluent in Workbench (p. 49), describes the details of using Fluent within Workbench.
- Appendix A: The Fluent Ribbon Tab Under Workbench (p. 595), describes the differences in the Fluent **File** ribbon tab within Workbench.
- Appendix B: The Fluent Meshing Menu under Workbench (p. 601), describes the differences in the Fluent Meshing **File** menu within Workbench.
- Appendix C: The **Workbench Tools** Toolbar Commands (p. 603), describes the **Workbench Tools** toolbar that is available in Fluent when running within Workbench.

2. Typographical Conventions

Several typographical conventions are used in this manual's text to help you find commands in the user interface.

- Different type styles are used to indicate graphical user interface items and text interface items. For example:

  Iso-Surface dialog box

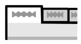
  `surface/iso-surface` text command

- The text interface type style is also used when illustrating exactly what appears on the screen to distinguish it from the narrative text. In this context, user inputs are typically shown in boldface. For example,

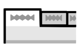
```
solve/initialize/set-fmg-initialization

 Customize your FMG initialization:
   set the number of multigrid levels [5]

   set FMG parameters on levels ..

    residual reduction on level 1 is:  [0.001]
    number of cycles on level 1 is:  [10]  100

    residual reduction on level 2 is:  [0.001]
    number of cycles on level 2 is:  [50]  100
```

- Mini flow charts are used to guide you through the ribbon or the tree, leading you to a specific option, dialog box, or task page. The following tables list the meaning of each symbol in the mini flow charts.

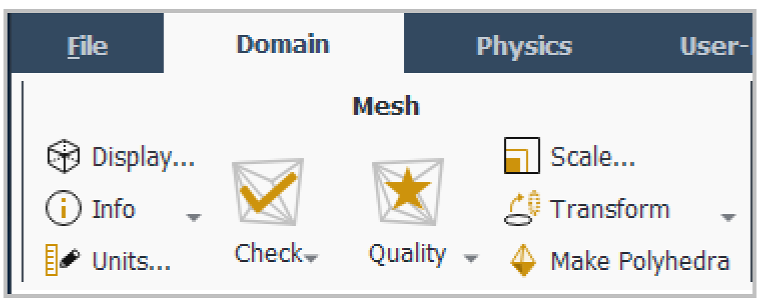
Table 1: Mini Flow Chart Symbol Descriptions

Symbol	Indicated Action
	Look at the ribbon
	Look at the tree
	Double-click to open task page
	Select from task page
	Right-click the preceding item

For example,

 **Setting Up Domain** → **Mesh** → **Transform** → **Translate...**

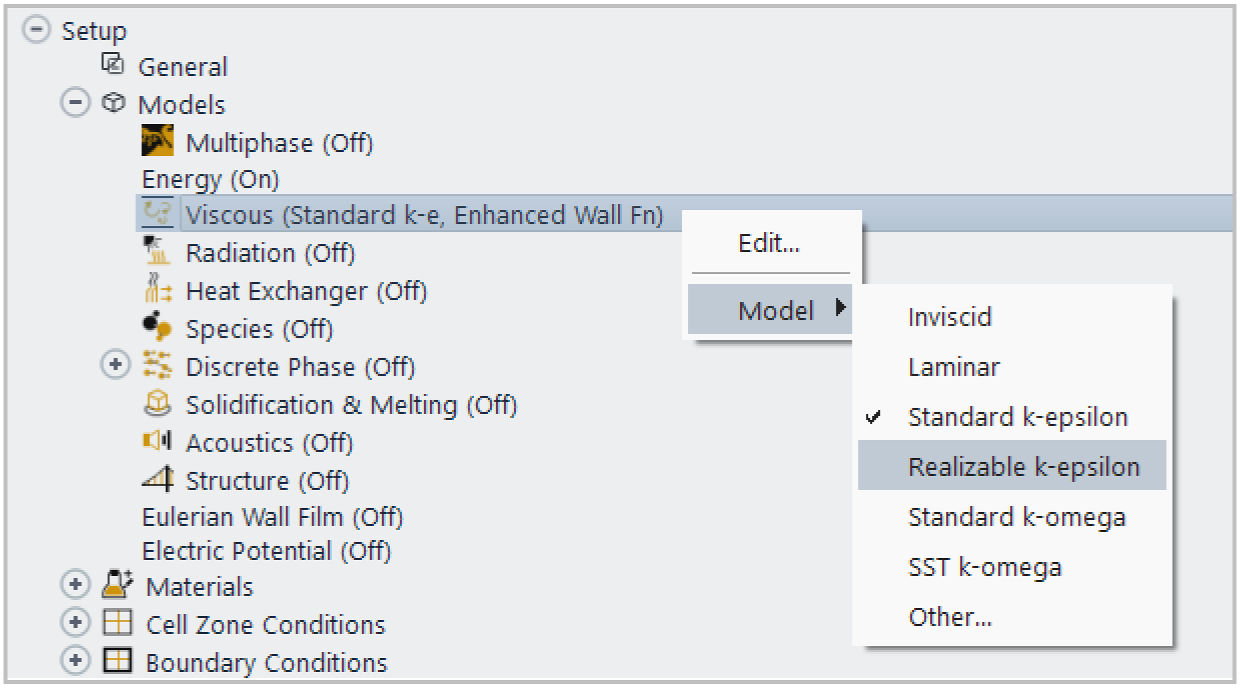
indicates selecting the **Setting Up Domain** ribbon tab, clicking **Transform** (in the **Mesh** group box) and selecting **Translate...**, as indicated in the figure below:



And

 **Setup** → **Models** → **Viscous**  **Model** → **Realizable k-epsilon**

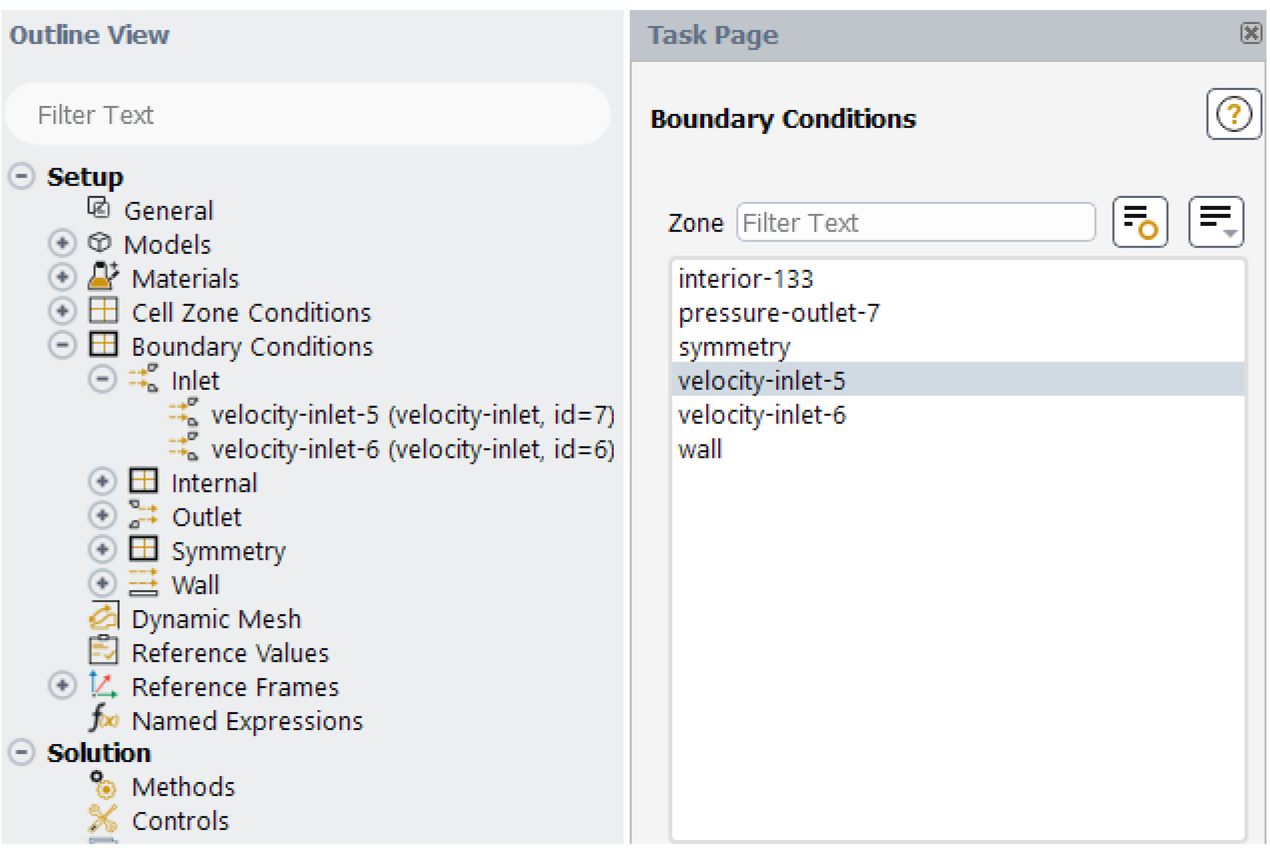
indicates expanding the **Setup** and **Models** branches, right-clicking **Viscous**, and selecting **Realizable k-epsilon** from the **Model** sub-menu, as shown in the following figure:



And

Setup → **Boundary Conditions** → **velocity-inlet-5**

indicates opening the task page as shown below:



In this manual, mini flow charts usually accompany a description of a dialog box or command, or a screen illustration showing how to use the dialog box or command. They show you how to quickly access a command or dialog box without having to search the surrounding material.

- In-text references to **File** ribbon tab selections can be indicated using a "/". For example **File/Write/Case...** indicates clicking the **File** ribbon tab and selecting **Case...** from the **Write** submenu (which opens the **Select File** dialog box).

Part I: General Applications

Chapter 1: Getting Started With Fluent in Workbench

This document is designed to provide information about using Fluent within Ansys Workbench. Some basic information about using Workbench is provided here, but the majority of the information about using Workbench can be found in the Workbench online documentation.

This chapter provides some basic instructions for getting started with using Fluent in Workbench.

1.1. Introduction to Workbench

Ansys Workbench combines access to Ansys applications with utilities that manage the product workflow.

Applications that can be accessed from Workbench include: Ansys DesignModeler and Ansys SpaceClaim Direct Modeler (for geometry creation); Ansys Meshing (for mesh generation); Ansys Fluent or CFX (for setting up and solving fluid dynamics analyses); and Ansys CFD-Post (for postprocessing the results). In Workbench, a project is composed of a group of systems. The project is driven by a schematic workflow that manages the connections between the systems. From the schematic, you can interact with workspaces that are native to Workbench, such as Design Exploration (parameters and design points), and you can launch applications that are data-integrated with Workbench (such as Ansys Fluent or CFX). Data-integrated applications have separate interfaces, but their data is part of the Workbench project and is automatically saved and shared with other applications as needed. This makes the process of creating and running a CFD simulation more streamlined and efficient.

Workbench allows you to construct projects composed of multiple dependent systems that can be updated sequentially based on a workflow defined by the project schematic. For instance, you can construct a project using two connected Fluent-based systems where the two systems share the same geometry and mesh; and the second system uses data from the first system as its initial solution data. When you have two systems connected in this way, you can modify the shared geometry once and then update the results for both systems with a single mouse click without having to open the Ansys Meshing application or Fluent. Some examples of when this is useful include: performing a reacting flow analysis starting from the solution obtained from a cold flow analysis; performing a second order analysis starting from the solution obtained from a first order analysis; and performing a transient simulation starting from the solution obtained from a steady-state analysis.

In addition, Workbench also allows you to copy systems in order to efficiently perform and compare multiple similar analyses. Workbench also provides a parametric modeling capabilities in conjunction with optimization techniques to allow you to efficiently investigate the effects of input parameters on selected output parameters.

For more information, see the following section:

1.1.1. Limitations

The following limitations are known when using Fluent in Workbench:

- Fluent design point studies using the **Remote Solve Manager** may not completely update due to errors. Should this happen, you can rerun the affected design points and they will update as expected (you may have to rerun failed design points multiple times).

- Workbench units and options are not passed to Fluent.

- The text user interface (TUI) shortcuts, or aliases, for reading case and data files for Fluent in Workbench are disabled by design. For example, `file read-case`, `f read-case`, and `f r-c` can be used, however `f rc` cannot be used.

- The version of Fluent used under Workbench must always be the version of Fluent that was packaged and installed with that version of Ansys Workbench. It is not possible to use previous versions of Fluent under Workbench even through a `Fluent Inc.` environment variable.

- Graphical user interface (GUI) journal files in Fluent 2021 R2 are not backwards compatible. That is, Workbench journal files (`*.wbjn` files) created in version 12.0 or version 12.1 that contain Fluent GUI commands may need to be recreated if they fail due to a change in the Fluent graphical user interface or a string used therein.

- For older Fluent in Workbench projects (prior to version 2021 R2), when making any changes to the mesh using Ansys Meshing (for example, renaming a surface), you should first open Fluent using the **Setup** cell of the Fluid Flow (Fluent) analysis system in order for Fluent to be aware of the upstream mesh changes (for example, when detecting upstream zone name changes).

- Changing the meshing method to "Assembly Meshing" methods (for example, CutCell) from other meshing methods may not preserve the previously defined contact regions. Currently, the Fluent simulation workflow does not support this type of change.

Note:

Ansys Fluent only allows a period to be used as a decimal separator. If your system is set to a European locale that uses a comma separator (for example, Germany), fields that accept numeric input may accept a comma, but may ignore everything after the comma. If your system is set to a non-European locale, numeric fields will not accept a comma at all.

Ansys Workbench accepts commas as decimal delimiters. These are translated into periods when data is passed to Ansys Fluent.

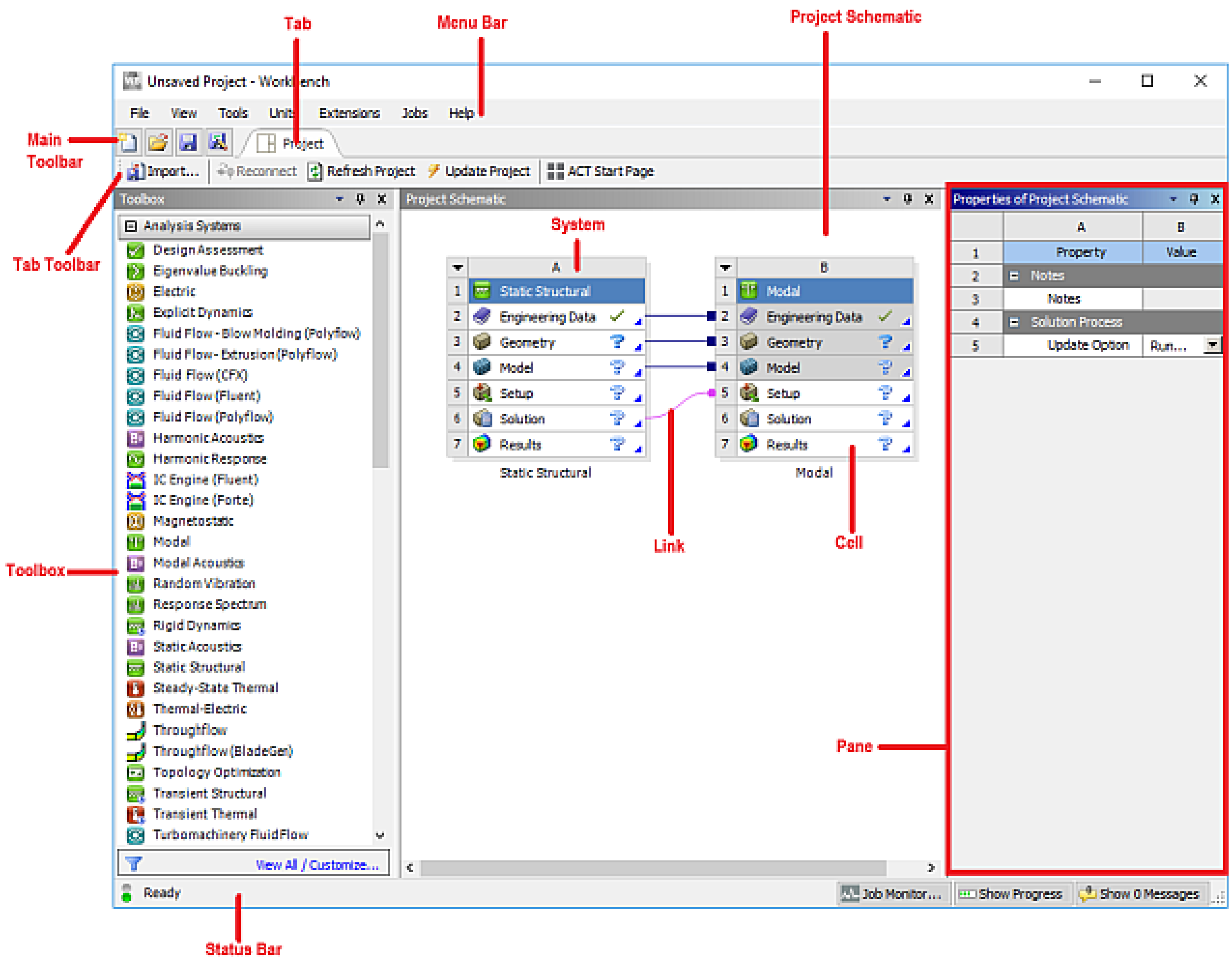
1.2. The Workbench Graphical User Interface

The Workbench graphical user interface (Figure 1.1: The Workbench Graphical User Interface (p. 24)) consists of the Toolbox, the Project Schematic, the Toolbar, and the Menu bar. The most common way to begin work in Workbench is to drag an item, such as a component system (application) or an analysis system, from the Toolbox to the Project Schematic, or to double-click an item to initiate the default action. You will view your component and/or analysis systems – the pieces that make up your analysis – in the Project Schematic, including all connections between the systems. The individual applications in which you work will display separately from the Workbench graphical interface, but the actions you take in the applications will be reflected in the Project Schematic.

Important:

Note that Fluent can be accessed in Workbench as either a component system or as an analysis system. Details for using both are described throughout this document.

Figure 1.1: The Workbench Graphical User Interface



Important:

Note that Fluent in Workbench uses informational, question, and warning dialog boxes that are designed to guide you in various ways as you work through your CFD analysis. Informational dialog boxes display messages that assist you in a specific task, or provide additional information relating to the task at hand. Question dialog boxes present questions concerning a task that is about to be performed, displaying an **OK** and a **Cancel** button in order to enable you to choose from one of two options (to proceed or not to proceed). Warning dialog boxes contain only an **OK** button and are designed to display a cautionary message indicating that you need to be aware that the application is about to change something or has internally changed something to maintain the consistency.

Note:

You can set various Fluent–specific preferences in Workbench (for example, Launcher settings). For more information, see Configuring Workbench in the Workbench User's Guide.

1.3. Creating Fluent-Based Systems

There are two basic types of systems: analysis systems and component systems. The following Fluent-based systems are available in Workbench:

- The **Fluid Flow (Fluent)** analysis system enables you to perform a complete CFD analysis and contains cells that allow you to create geometry, generate a mesh, specify settings in Fluent, run the Fluent solver, and visualize the results in CFD-Post.
- The **Fluent** component system enables you to access the Fluent application from within Workbench and contains only the cells needed to specify settings in Fluent and run the Fluent solver. When using a Fluent component system, a mesh must be imported into the system or provided through a connection from an upstream system.
- The **Fluent (with Fluent Meshing)** component system enables you to generate a mesh, specify settings in Fluent, and run the Fluent solver. For this system, you can provide a geometry through the upstream connection and use a journal file to automatically generate a mesh for your application.

Note:

A separate cell for results visualization is only needed when using CFD-Post. The postprocessing capabilities in Fluent can be accessed from both the **Fluid Flow (Fluent)** analysis system and the Fluent component system.

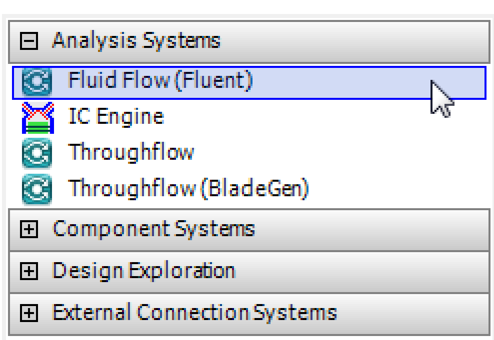
To create an analysis or component system in Workbench: In the **Toolbox**, double-clicking the desired system under **Analysis Systems** or **Component Systems**, respectively. Or you can click the system and drag it onto the **Project Schematic**. Notice that when you hover over systems in the **Toolbox**, a tool tip appears.

For more information, see the following sections:

1.3.1. Fluent-Based Analysis Systems

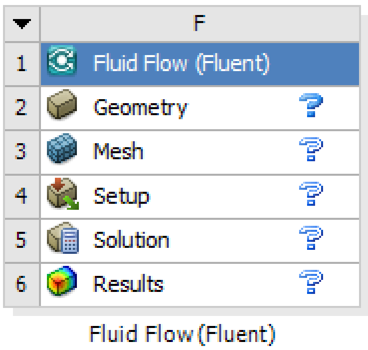
The **Fluid Flow (Fluent)** analysis system is found in the **Toolbox** under **Analysis Systems**.

Figure 1.2: Selecting the Fluid Flow (Fluent) Analysis System in Workbench



When you create the new **Fluid Flow (Fluent)** analysis system in Workbench (as described above), it appears in the **Project Schematic** as a box containing several cells (Figure 1.3: A **Fluid Flow (Fluent)** Analysis System (p. 26)). Each cell corresponds to a typical task you would perform to complete a CFD analysis.

Figure 1.3: A Fluid Flow (Fluent) Analysis System



The following cells are available in a **Fluid Flow (Fluent)** analysis system:

Geometry

allows you to define the geometrical constraints of your analysis. You can perform the following operations:

- Import a pre-existing geometry into your system by using the context menu accessible through right-clicking the cell.
- Create a new geometry either in Ansys SpaceClaim Direct Modeler (Windows only) or Ansys DesignModeler. The geometry editor is launched by either double-clicking the **Geometry** cell or right-clicking the cell and selecting an appropriate geometry option from the cell context menu. On Windows machines, the default geometry editor is SpaceClaim. You can change the preferred geometry editor to Ansys DesignModeler in the **Options** dialog box (**Tools** → **Options** → **Geometry Import** → **Geometry Import** → **Preferred Geometry Editor**).
- Modify an existing geometry. Note that double-clicking a **Geometry** cell opens the geometry in the most recently used editor. You can also launch the editor of your choice by selecting the corresponding context menu option.

Mesh

allows you to define and generate a computational mesh for your analysis. Double-clicking the **Mesh** cell opens Ansys Meshing and loads the current mesh database (or the geometry defined by the **Geometry** cell) if you have not yet begun working on the mesh. Alternatively, you can use the context menu (by right-clicking the **Mesh** cell) to import a pre-existing Fluent mesh into the system.

Important:

- Importing a Fluent mesh file into the **Mesh** cell results in the **Mesh** cell becoming the starting point for your analysis (and the name of the **Mesh** cell changes to **Im-**

ported Mesh). Therefore, the **Geometry** cell (and data it contains) will be deleted from the system. The deleted **Geometry** cell can be retrieved by selecting **Reset** from the context menu of the **Imported Mesh** cell.

- Fluent meshes imported into the **Mesh** cell cannot be modified by the Ansys Meshing application.

Setup

allows you to define the boundary conditions, physical models and solver settings for the Fluent analysis. Double-clicking the **Setup** cell opens Fluent and loads the mesh defined by the **Mesh** cell as well as any Fluent settings that have already been specified. Alternatively, you can use the context menu (by right-clicking the **Setup** cell) to import a pre-existing Fluent case or mesh file into the system. After you specify the file you want to import, Fluent will open and load the file.

Important:

- If you open Fluent before defining a mesh, Fluent will open without loading any files. You can then choose to import files from the **File** ribbon tab in Fluent.
- Importing a Fluent case or mesh file into the **Setup** cell or the Fluent application results in the **Setup** cell becoming the starting point for your analysis. Therefore, the **Geometry** and **Mesh** cells (and any data they contain) will be deleted from the system.

Solution

allows you to calculate a solution in Fluent. Double-clicking the **Solution** cell opens Fluent and loads the current Fluent case and data files. If you have not yet performed any calculations, Fluent will load the mesh file as well as any settings that have been specified.

Important:

You can also use the **Solution** cell context menu to import a pre-existing Fluent data file to use for initial solution data. If you have not yet performed any calculations, Fluent will load this data file in addition to the mesh and settings.

Results

allows you to display and analyze the results of the CFD analysis. Double-clicking the **Results** cell opens CFD-Post and loads the current Fluent case and data files as well as the current CFD-Post state file.

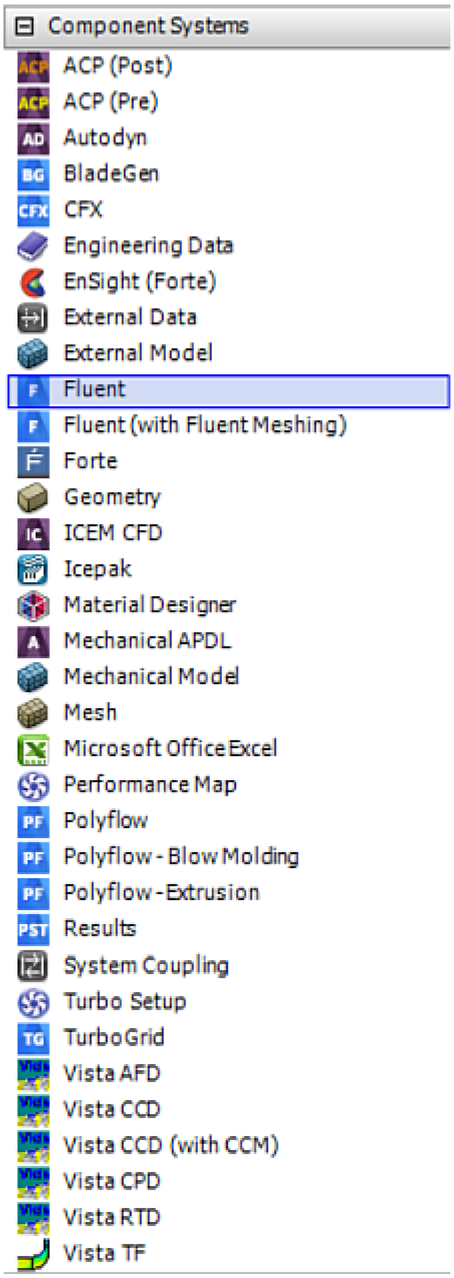
Note:

While it is possible to apply different names for the **Setup** or the **Solution** cells by right-clicking either cell, and selecting the **Rename** command in the context menu, it is not generally recommended to do so.

1.3.2. Fluent-Based Component Systems

The **Fluent** and **Fluent (with Fluent Meshing)** component systems are found in the **Toolbox** under **Component Systems**.

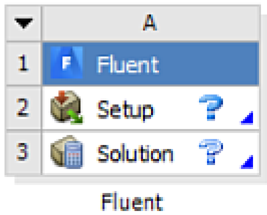
Figure 1.4: Selecting the Fluent Component System in Workbench



Fluent Component System

When you create a new **Fluent** component system in Workbench (as described above), it appears in the **Project Schematic** as a box containing two cells: **Setup** and **Solution** (Figure 1.5: A **Fluent** Component System (p. 29)).

Figure 1.5: A Fluent Component System

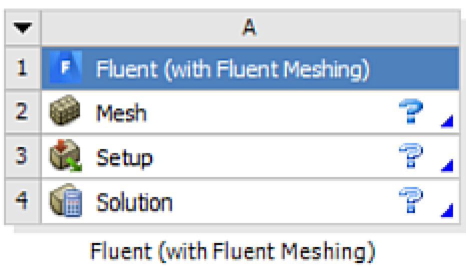


The **Setup** and **Solution** cells in a **Fluent** component system work in the same manner as described above for the **Fluid Flow (Fluent)** analysis system (see Fluent-Based Analysis Systems (p. 25)). The only difference is that the mesh must originate from a file imported into the **Setup** cell or the Fluent application, or it must be provided through a connection from an upstream system.

Fluent (with Fluent Meshing) Component System

When you create a new **Fluent (with Fluent Meshing)** component system in Workbench, it appears in the **Project Schematic** as a box comprising three cells: **Mesh**, **Setup**, and **Solution**.

Figure 1.6: A Fluent (with Fluent Meshing) Component System



By double-clicking the **Mesh** cell, you can access Fluent Meshing, and define and generate a computational mesh for your problem. Upon opening, Fluent Meshing will automatically load either the current mesh data or, if the mesh data is not available, the geometry defined in the upstream **Geometry** cell. Otherwise, you can import input data file(s) directly into Fluent Meshing.

The **Setup** and **Solution** cells work in the same way as in the **Fluid Flow (Fluent)** analysis system (see Fluent-Based Analysis Systems (p. 25)).

1.4. Understanding Cell States with Fluent in Workbench

Workbench integrates multiple data-integrated (for example, Fluent) and native applications into a single, seamless project flow, where individual cells can obtain data from and provide data to other cells. Workbench provides visual indications of a cell's state via icons on the right side of each cell. Brief descriptions of the each possible state are provided below. For more information about cell states, see the Workbench online help:

- Unfulfilled ( ) indicates that required upstream data does not exist. For example, when you first create a new **Fluid Flow (Fluent)** analysis system, all cells downstream of the **Geometry** cell appear as **Unfulfilled** because you have not yet specified a geometry for the system.

- Refresh Required ( ) indicates that upstream data has changed since the last refresh or update. For example, after you assign a geometry to the **Geometry** cell in your new **Fluid Flow (Fluent)** analysis system, the **Mesh** cell appears as **Refresh Required** since the geometry data has not yet been passed from the **Geometry** cell to the **Mesh** cell.

- Attention Required ( ) indicates that the current upstream data has been passed to the cell, however, you must take some action to proceed. For example, after you launch Fluent from the **Setup** cell in a **Fluid Flow (Fluent)** analysis system that has a valid mesh, the **Setup** cell appears as **Attention Required** because additional data must be entered in Fluent before you can calculate a solution.

- Update Required ( ) indicates that local data has changed and the output of the cell must be regenerated. For example, after you launch Ansys Meshing from the **Mesh** cell in a **Fluid Flow (Fluent)** analysis system that has a valid geometry, the **Mesh** cell appears as **Update Required** because the **Mesh** cell has all the data it needs to generate a Fluent mesh file, but the Fluent mesh file has not yet been generated.

- Up-To-Date ( ) indicates that an update has been performed on the cell and no failures have occurred (or an interactive calculation has been completed successfully). For example, after Fluent finishes performing the number of iterations that you request, the **Solution** cell appears as **Up-to-Date**.

- Interrupted, Update Required ( ) indicates that you have interrupted an update (or canceled an interactive calculation that is in progress). For example, if you select the **Cancel** button in Fluent while it is iterating, Fluent completes the current iteration and then the **Solution** cell appears as **Interrupted, Update Required**.

- Input Changes Pending ( ) indicates that the cell is locally up-to-date, but may change when next updated as a result of changes made to upstream cells. For example, if you change the **Mesh** in an **Up-to-Date Fluid Flow (Fluent)** analysis system, the **Setup** cell appears as **Refresh Required**, and the **Solution** and **Results** cells appear as **Input Changes Pending**.

- Pending ( ) indicates that a batch or asynchronous solution is in progress. When a cell enters the **Pending** state, you can interact with the project to exit Workbench or work with other parts of the project. If you make changes to the project that are upstream of the updating cell, then the cell will not be in an up-to-date state when the solution completes.

If a particular action fails, Workbench provides a visual indication as well. Brief descriptions of the failure states are described below.

- Refresh Failed, Refresh Required ( ) indicates that the last attempt to refresh cell input data failed, and so the cell must be refreshed.

- Update Failed, Update Required ( ) indicates that the last attempt to update the cell and calculate output data failed, and so the cell must be updated.

- Update Failed, Attention Required ( ) indicates that the last attempt to update the cell and calculate output data failed, and so the cell requires attention.

If an action results in a failure state, you can view any related error messages in the **Messages** pane by clicking the **Show Messages** button on the lower right portion of Workbench.

1.5. Starting Fluent in Workbench

This section describes how to start Fluent in Workbench using Fluent-based systems and how you can use Fluent Launcher within Workbench.

For more information, see the following sections:

1.5.1. Starting Fluent from a Fluent-Based System

You can start Ansys Fluent by double-clicking the **Setup** cell in a **Fluid Flow (Fluent)** analysis system or a Fluent component system. Fluent launches and loads the **Setup** cell's input data (for example, mesh) and the **Setup** cell's local data, if it exists (for example, Fluent settings or setup output case file). If no mesh has been specified, Fluent launches and waits for your input.

You can also start Ansys Fluent by double-clicking the **Solution** cell in a **Fluid Flow (Fluent)** analysis system or a Fluent component system. Fluent launches and loads the current case and data files, as well as the **Setup** cell's input data (for example, mesh), the **Setup** cell's local data, if it exists (for example, Fluent settings), and the **Solution** cell's initial data, if it exists. If no mesh has been specified, Fluent launches and waits for your input.

Important:

When Fluent is launched from the **Setup** cell, it loads only the mesh and settings that served as the starting point for your analysis and are associated with the **Setup** cell. In order to load the current case and data files or the initial data file, you must launch Fluent from the **Solution** cell.

1.5.2. Specifying Fluent Launcher Settings Within Workbench

You can use Fluent in Workbench on either Windows or Linux machines, both interactively and in batch mode. You can also start Fluent on Linux from a Workbench session running on Windows and you can use the same Fluent-specific project (and related files) in Workbench using both Linux and Windows hardware interchangeably. The information is this section is the same for both Windows and Linux, except where noted.

When you start Ansys Fluent from either type of Fluent-based system within Workbench, Fluent Launcher will appear by default. Most Fluent Launcher settings are available, except for the following options:

- **Version** (disabled)

- **Working Directory** (disabled)

Note:

In some instances, the working directory shown in the Fluent Launcher may not match the true working directory. You can confirm the working directory in Fluent by entering `pwd` into the Fluent console.

The **Do not show this panel again** option allows you to bypass Fluent Launcher for subsequent Fluent sessions. This option is only available when running Fluent in Workbench.

Important:

When using LSF to schedule a Fluent run from Workbench, by default the working directory is used for checkpointing. You can specify an alternate directory for checkpointing using the `LSB_CHKPNT_DIR` environment variable.

Important:

Note that, when using Fluent with Workbench on Linux, the **Fluent Root Path** option is disabled in the **General** tab of Fluent Launcher.

To start your Fluent simulation on a Linux cluster from a Workbench session running on Windows, use the **Use Remote Linux Nodes** option in Fluent Launcher. This option is available in the **Remote** tab (visible when you click the **Show More Options** button).

Important:

The **Remote** tab of Fluent Launcher can only be used for 64-bit Linux machines.

Important:

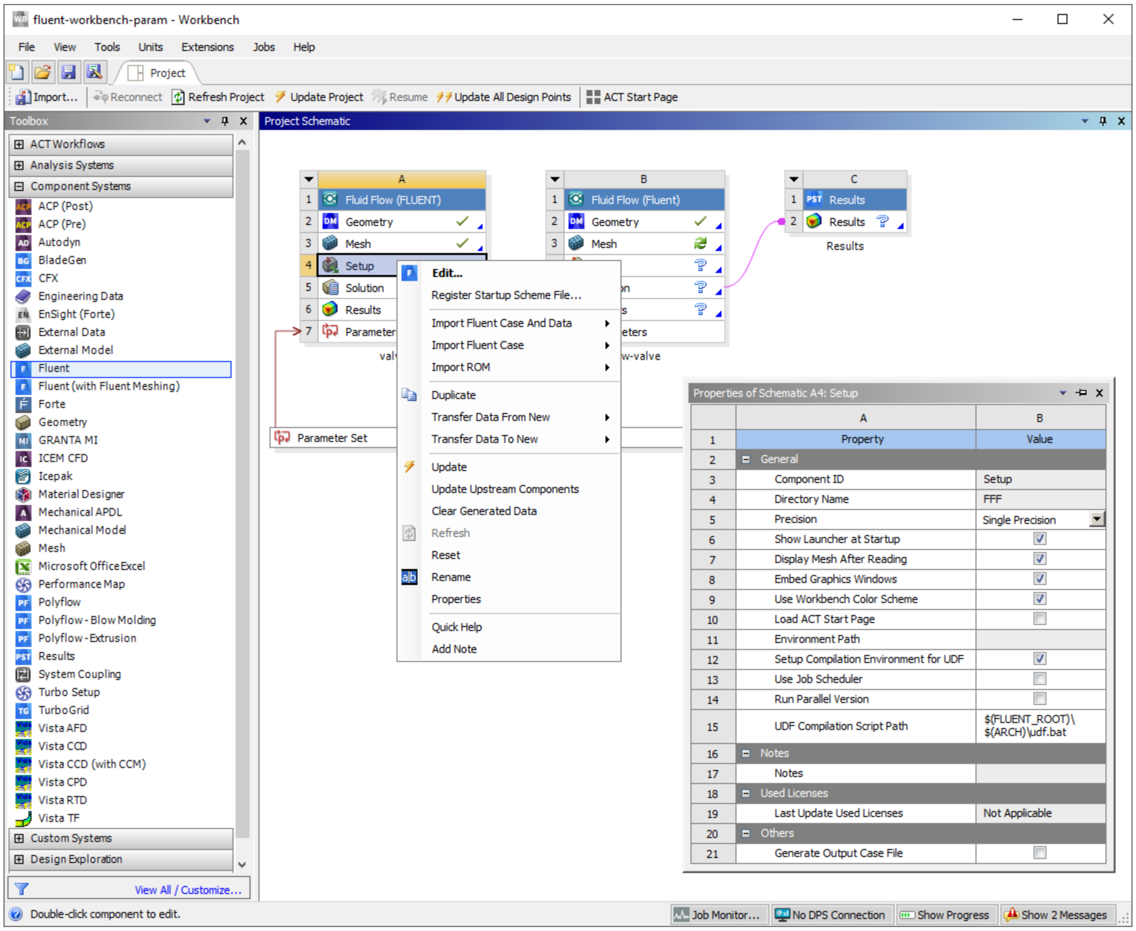
If you are using the **Select IP Interface** option in the **Parallel** tab of Fluent Launcher, the selected interface address is not saved. As a workaround, you can define the `FLUENT_HOST_IP=host:ip-address` environment variable.

For more information about Fluent Launcher, see Starting Ansys Fluent Using Fluent Launcher. For more information about using Fluent Launcher to access remote Linux clusters, see Setting Remote Options in Fluent Launcher.

1.5.2.1. Specifying Fluent Launcher Settings Using Cell Properties

You can view the properties of a selected cell in Workbench by selecting the **Properties** option under the **View** menu, or by right-clicking a cell and selecting **Properties** from the context menu. The properties of the selected cell are displayed in the **Properties** pane in Workbench.

Figure 1.7: Properties for Fluent-Based Systems in Workbench



The Fluent-based system **Setup** and **Solution** cells have the following properties that you can set for Fluent Launcher:

Use Setup Launcher Settings

(for **Solution** cells only) allows you to specify that the current systems **Solution** cell should use the Fluent Launcher property settings from the current system's **Setup** cell. By default, the Fluent-based system **Setup** and **Solution** cells share Fluent Launcher settings, however, if required, these settings can be set autonomously using this property.

Precision

allows you to choose either the single-precision or the double-precision solver

Show Launcher at Startup

allows you to show or hide Fluent Launcher when Fluent starts.

Display Mesh After Reading

allows you to show or hide the mesh after the mesh or case/data is read into Fluent.

Embed Graphics Windows

allows you to embed the graphics windows in the Fluent application window, or to have them free-standing.

Set up Compilation Environment for UDF

allows you to specify compiler settings for compiling user-defined functions (UDFs) with Fluent.

Use Job Scheduler

allows you to specify settings for running Fluent with the available job scheduler. On Linux, you can use either LSF, SGE, PBS Pro or Slurm as the job scheduler. On Windows, you can use the Microsoft Job Scheduler, or, when **Run Parallel Version** and **Use Remote Linux Nodes** are selected, you can use the LSF, SGE, PBS Pro or Slurm job schedulers.

Run Parallel Version

allows you to choose to run the parallel version of Fluent or not.

UDF Compilation Script Path

(if **Set up Compilation Environment for UDF** is selected) allows you to specify the path to the UDF compilation script.

Initialization Method

allows you to initialize your Fluent simulation. Available options are:

- **Program Controlled** (default) allows Fluent to use any existing results (solution data) to initialize the solution.
  - For a new calculation, if no results are available, then the system initializes the solution using the settings specified in Fluent.
  - For existing projects, if the system is *not* connected to another upstream system through the **Solution** cell, then the system uses results available from the previous calculation. If the results are not compatible with the mesh, then the system initializes the solution using the settings specified in Fluent.
  - For existing projects, if the system is connected to another upstream system through the **Solution** cell, then the **Initialization Method** is not available for the downstream **Solution** cell. In this case, the downstream **Solution** cell always uses results coming from the upstream **Solution** cell to initialize its solution for the initial design point (DP0) as well as any other design points. If the solution data file from the upstream **Solution** cell is not available or not compatible with the mesh, then Workbench uses the upstream data interpolation (`*.ip`) file for solution initialization. If the `*.ip` file is also not available, or the results are not compatible with the mesh, then the system initializes the solution using the settings specified in Fluent.
- **Solver Controlled** allows Fluent to always initialize the solution using settings specified in Fluent, enforcing the initialization method available in Fluent for new and existing projects. This option is not available for **Solution** cells connected with upstream **Solution** cells.

- **Use Solution Data from File** allows you to import a solution data file (`*.dat`) or data interpolation file (`*.ip`) that Fluent will use to initialize the solution. If selected, you are prompted to provide an **Initial Data File**. For new and existing calculations, the **Solution** cell uses a registered file as the initial data in Workbench.

For more information, see Using the Update Command (p. 55).

Use Remote Linux Nodes

(available only when **Run Parallel Version** is selected) allows you to run your Fluent simulation on 64-bit Linux machines.

If **Run Parallel Version** is selected, the following additional properties are available:

Number of Processors

allows you to set the number of processors you want to use for the parallel calculations (for example, 2, 4, and so on).

Interconnect

allows you to set the interconnects you want to use for the parallel calculations (for example, infiniband).

MPI Type

allows you to set the MPI type you want to use for the parallel calculations (for example, **intel**, **msmpi**, and so on).

Use Shared Memory

allows you to specify if shared memory is to be used or not.

Machine Specification

(if **Use Shared Memory** is not selected) allows you to specify a list of machine names, or a file that contains machine names.

Machine List

(if **Use Shared Memory** is not selected and **Machine List** is selected as the **Machine Specification**) allows you to specify a list of machine names to run the parallel job.

Machine Filename

(if **Use Shared Memory** is not selected and **File Containing Machine List** is selected as the **Machine Specification**) allows you to specify the name of the file that contains a list of machines to run the parallel job.

If **Use Remote Linux Nodes** is selected, the following additional properties are available (for 64-bit Linux machines only):

Remote Fluent Root Path

allows you to specify the remote Fluent Linux installation root path.

Use Specified Remote Working Directory

allows you to specify a directory other than `temp` directory as the working directory for the remote Linux nodes. When this property is selected, you can specify the directory in the **Remote Working Directory** property that appears.

Use Remote Cluster Head Node

allows you to specify the remote node that Fluent will connect to for spawning (for example, via `ssh`). When this property is selected, you can specify the remote node in the **Remote Host Name** property that appears.

Important:

When the **PBS Pro** option is selected for the **Job Scheduler**, the host specified in the **Use Remote Cluster Head Node** field should be the PBS Pro submission host.

If **Use Job Scheduler** is selected, the following additional properties are available:

Computer Cluster Head Node Name

allows you to specify the name of the compute cluster head node.

Job Template

(available only when running Microsoft HPC Pack 2008 or newer) allows you to create a custom submission policy to define the job parameters for an application. The cluster administrator can use job templates to manage job submission and optimize cluster usage.

Node Group

(available only when running Microsoft HPC Pack 2008 or newer) allows you to specify a collection of nodes. Cluster administrators can create groups and assign nodes to one or more groups.

Processor Unit

(available only when running Microsoft HPC Pack 2008 or newer) allows you to choose the following:

- **Core** refers to a single, named host in the cluster.
- **Socket** refers to a set of processors with a dedicated memory bus. This is also known as a non-uniform memory access (NUMA) node.
- **Node** refers to an individual CPU on a node. For example, a dual-core processor is considered two cores.

Start When Resources Are Available

allows you to start the job when resources are available.

Create Job Submission XML

allows you to create the job submission XML file.

Job Submission XML File

(if **Create Job Submission XML** is selected) allows you to specify the name of the job submission XML file.

If **Use Job Scheduler** and **Use Remote Linux Nodes** are selected, the following additional properties are available:

Job Scheduler

allow you to specify the job scheduler for the remote Linux nodes. Available options are **LSF** (the default), **SGE**, **PBS Pro**, or **Slurm**.

If **LSF** is selected as the job scheduler, the following additional properties are available:

Use LSF Queue

allows you to use an LSF queue.

LSF Queue

(when **Use LSF Queue** is selected) allows you to specify the name of the LSF queue.

Use Checkpointing

allows you to use checkpointing with LSF.

Enable Automatic Checkpointing

(when **Use Checkpointing** is selected) allows you to automatically checkpoint within a specific period of time.

Checkpointing Period

(when **Use Checkpointing** and **Enable Automatic Checkpointing** are selected) allows you to specify a time period for automatic checkpointing.

Important:

When using LSF to schedule a Fluent run from Workbench, by default the working directory is used for checkpointing. You can specify an alternate directory for checkpointing using the `LSB_CHKPNT_DIR` environment variable.

If **SGE** is selected as the job scheduler, the following additional properties are available:

SGE Qmaster

allows you to set the machine in the SGE job submission host list.

SGE Queue

allows you to set the queue where you want to submit your Fluent jobs. Leave this field blank if you want to use the default queue.

SGE PE

allows you to set the parallel environment where you want to submit your Fluent jobs. Leave this field blank if you want to use the default parallel environment.

Use SGE Settings

allows you to use SGE settings.

SGE Settings

(when **Use SGE Settings** is selected) allows you to specify SGE settings.

Important:

When running Fluent in Workbench via SGE, there may be instances when not enough time is allotted for Fluent to start. By default, Workbench performs two steps: starting the Fluent process and establishing initial contact, with a default wait time of one minute; completing the Fluent start up process and establishing a final connection (for example, to spawn parallel nodes, to checkout licenses, and so on), with a default wait time of five minutes. You can change the waiting time increments for each step by defining the `Fluent_WB_MAX_STARTUP_WAIT` environment variable before starting Workbench. Note that you *cannot* define this environment variable within Fluent Launcher. The value of this environment variable is equivalent to the wait time in minutes for each start up step. So, the overall wait time is double the value specified for the environment variable. Fractions are allowed as values for the environment variable, and any value less than or equal to zero will let Workbench wait indefinitely for Fluent to start. Note that while Workbench is waiting for Fluent to start, the Workbench interface is locked, and if, for some reason Fluent fails to start, you will have to manually kill the Workbench session.

If **PBS Pro** is selected as the job scheduler, the following additional property is available (on Linux only):

PBS Submission Host

(optional) allows you to specify the name of the PBS Pro submission host if the machine you are using to run Workbench cannot submit a job to PBS Pro.

If **Slurm** is selected as the job scheduler, the following additional properties are available:

Slurm Submission Host

allows you to specify a machine to submit jobs to Slurm if the machine you are using to run Workbench cannot submit to Slurm.

Slurm Partition

allows you to request a specific partition for the resource allocation.

Slurm Account

specify the Slurm account.

Important:

When running Fluent in Workbench via Slurm, there may be instances when not enough time is allotted for Fluent to start. By default, Workbench performs two steps: starting the Fluent process and establishing initial contact, with a default wait time of one minute; completing the Fluent start up process and establishing a final connection (for example, to spawn parallel nodes, to checkout licenses, and so on), with a default wait time of five minutes. You can change the waiting time increments for each step by defining the `Fluent_WB_MAX_STARTUP_WAIT` environment variable before starting Workbench. Note that you *cannot* define this environment variable within Fluent Launcher. The value of this environment variable is equivalent to the wait time in minutes for each start up step. So, the overall wait time is double the value specified for the environment variable. Fractions are allowed as values for the environment variable, and any value less than or equal to zero will let Workbench wait indefinitely for Fluent to start. Note that while Workbench is waiting for Fluent to start, the Workbench interface is locked, and if, for some reason Fluent fails to start, you will have to manually kill the Workbench session.

These properties are the same as the settings within Fluent Launcher. For more information about Fluent Launcher, see the Fluent Getting Started Guide.

Properties for **Setup** and **Solution** cells related to problem setup and solution processes are discussed in Specifying Other Setup and Solution Cell Settings (p. 40).

1.5.2.2. Copying Fluent Launcher Property Settings

Typically, Fluent Launcher settings are specified in the Fluent Launcher dialog box when Fluent is launched from the **Setup** cell.

By default, the **Solution** cell uses the same Fluent Launcher property settings as the **Setup** cell. If you want the **Solution** cell's Fluent Launcher settings to be different than those specified for the **Setup** cell, you can disable the **Use Setup Launcher Settings** property setting (see Specifying Fluent Launcher Settings Within Workbench (p. 31)) and then set the **Solution** cell's Fluent Launcher property settings without impacting the settings for the **Setup** cell.

You can copy the **Setup** cell's Fluent Launcher settings to the **Solution** cell by selecting the **Copy Launcher settings to Solution cell** command from the **Setup** cell's context menu (available only when the **Use Setup Launcher Settings** property setting is disabled). Note that the values that are written are retained, even if you later disable this setting. Likewise, you can also copy the **Solution** cell's Fluent Launcher settings to the **Setup** cell by selecting the **Copy Launcher settings to Setup cell** command from the **Solution** cell's context menu (available only when the **Use Setup Launcher Settings** property setting is disabled).

1.5.3. Specifying Other Setup and Solution Cell Settings

The **Setup** cell has the following property related to the problem setup process:

- **Others**
  - **Generate Output Case File**: enables or disables the generation of the setup output case file, *name*-Setup-Output.cas.h5. If the state for the **Setup** cell is Update Required and the upstream mesh has not been modified, Fluent will read the output case file (*name*-Setup-Output.cas.h5) as opposed to the mesh and settings files when editing the **Setup** cell. This way you can improve the Fluent run time speed provided the upstream mesh data has not changed.

    The output case file will be generated when

    → recorded mesh operations (Recording Mesh Manipulation Operations and Resolving Mesh Incompatibility in Fluent (p. 76)) have been performed prior to running the simulation in Fluent, or

    → Fluent session is started with a mesh file only

    This setting overrides the **Enable Generation of Setup Output Case File** preference in the **Options** settings.

The **Solution** cell contains additional general properties and some specific properties related to the solution process:

- **General**
  - **Component ID**: the name of the cell (not editable).
  - **Directory Name**: the name of the directory ("FFF") where solution files are located (not editable).
  - **Use Setup Launcher Settings**: enables or disables the use of the **Setup** cell's Fluent Launcher settings (see above).
  - **Solution Monitoring**: allows you to be able to graphically view Fluent solution convergence and monitor data without having Fluent open. When this option is enabled, you can use the **Show Solution Monitoring** option in the **Solution** cell context menu to display a convergence and monitor charts. This property overrides the **Enable Solution Monitoring** preference in the **Options** settings.

Note:

→ A report definition will only be included in a solution monitor chart if it is included in a report plot.

→ Each report plot will be shown as a separate solution monitor chart.

→ A single report definition will only appear once in the solution monitor charts, even if that report definition is included in multiple report plots.

- **Generate Solution Monitor Plots for Report**: when selected, Fluent will automatically generate the report images during a **Solution** cell update or project report export. See Generating Fluent Project Reports (p. 68) for information about exporting project reports and the types of report images. This option is disabled by default.

- **Data Interpolation**: enables or disables the interpolation file generation at the end of the Fluent solver run. The solver will automatically create an `*.ip` file that can be used when restarting the Fluent session, even if the project mesh or case file has been modified and is no longer compatible with the existing data file (if available). This setting overrides the **Enable Generation of Interpolation File** preference in the **Options** settings. Note, that you can use the **Clear Generated Data** command to delete the generated interpolation file (see Using the Clear Generated Data Command (p. 61) for details).

- **Solution Process**

  - **Update Option**: enables the solution process to be either **Run in Foreground** (solutions are run within the current Workbench session), **Run in Background** (solutions are run in the background on the local machine), or **Submit to Remote Solve Manager** (solutions are run in the background by submitting the solution to Remote Solve Manager (RSM)). For more information about these options and the Remote Solve Manager, refer to Submitting Solutions to Remote Solve Manager in the Ansys Workbench User's Guide.

1.6. Registering and Unregistering Startup Scheme Files

You can register or unregister a customized Scheme file through your Fluent setup using the context menu for the Fluent **Setup** cell. Fluent loads this registered scheme file at startup. To register a scheme file, use the **Register Startup Scheme File...** command from the context menu for the Fluent **Setup** cell. Likewise, to unregister a scheme file, use the **Unregister Startup Scheme File...** command. If Fluent is already started, these menu commands are disabled and grayed out.

1.7. Saving Your Work in Fluent with Workbench

Data that is read into and written by Fluent when it is run within Workbench is split into two parts:

- Setup data
- Solution data

Setup data is the data used to start a simulation over from the beginning. This data is associated with the **Setup** cell and includes the mesh (`.msh.h5`) file and the settings (`.set`) file.

Important:

- The settings file is a file used when Fluent is run under Workbench. It contains the case settings but does not contain mesh data. The settings file and the mesh file are read by Fluent whenever Fluent is launched from the **Setup** cell.

- Note that sometimes, instead of a mesh file, a case file (`.cas.h5`) is used to represent the mesh. In this situation, Fluent reads the case file first and then reads the settings file if it exists. Therefore, the settings stored in the settings file will overwrite any settings that might be contained in the case file.

Solution data is the data that results from performing a calculation and is used to restart a simulation from existing data. This data is associated with the **Solution** cell and includes the current case (`.cas.h5`) file and the current data (`.dat.h5`) file.

Important:

The case file and the data file are read by Fluent whenever Fluent is launched from the **Solution** cell.

When working in Workbench, your work in Fluent is automatically saved as needed. For example, whenever you close Fluent or save your Workbench project, your unsaved data is automatically saved.

You can save your Workbench project directly from Fluent by selecting **Save Project** command under the **File** ribbon tab.

File → Save Project

Alternatively, you can save your Workbench project by selecting the **Save** command under the **File** menu within Workbench or by clicking the **Save Project** icon () from the Workbench toolbar.

1.8. Exiting Fluent and Workbench

You can end your Fluent session by using the **Close Fluent** command under the **File** ribbon tab.

File → Close Fluent

If you want to end your Fluent session without saving your work, you can use the **Close Without Save** command under the **File** ribbon tab. You can also use the corresponding text command (`file/close-without-save`).

File → Close Without Save

Alternatively, to discard unsaved changes in Fluent and load the setup files into the current Fluent session, you can use the **Reload** command in the Fluent **File** ribbon tab (see Reloading Data and Synchronizing Fluent with Workbench (p. 60) for details).

File → Reload

You can end your Workbench session by using the **Exit** command under the Workbench **File** menu.

File → Exit

All open applications that are associated with your Workbench session, including any open instances of Ansys Fluent, are automatically closed upon exiting Workbench. If there is any unsaved data in

Workbench or any of the open applications associated with your Workbench session, you will be prompted to save your project before exiting Workbench.

Important:

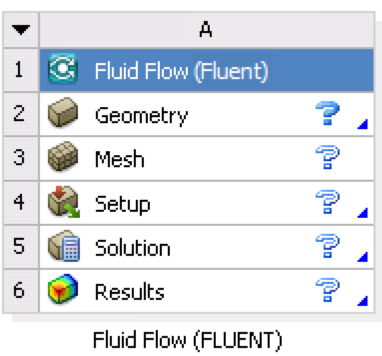
There are several other circumstances in which open instances of Fluent as well as other applications are automatically closed:

- Whenever you close the current project in Workbench, all open applications are automatically closed.
- Whenever you open a different project in Ansys Workbench, all open applications associated with the original project are automatically closed.
- Whenever a system is deleted, all open applications associated with that system are automatically closed.
- Whenever data is reset or cleared from a cell, all open applications associated with that cell are automatically closed.

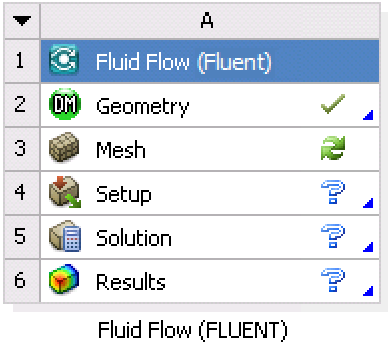
1.9. An Example of a Fluent Analysis in Workbench

This example describes when the files that are generated and used by Fluent are written and how the cell states change as you work with a Fluent-based system in Workbench.

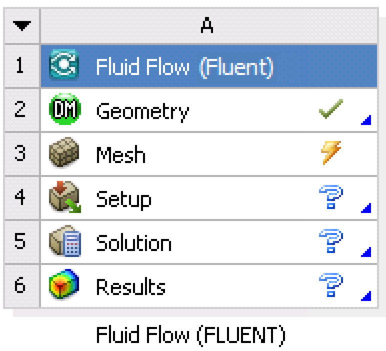
1. Add a new **Fluid Flow (Fluent)** analysis system to the Project Schematic. The state of the **Geometry** cell is **Attention Required** and that the states for the **Mesh**, **Setup**, **Solution**, and **Results** cells are **Unfulfilled**.



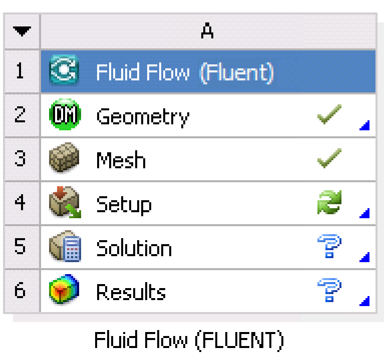
2. Import a geometry file by using the context menu on the **Geometry** cell. The state of the **Geometry** cell becomes **Up-to-Date** and the state of the **Mesh** cell becomes **Refresh Required**.



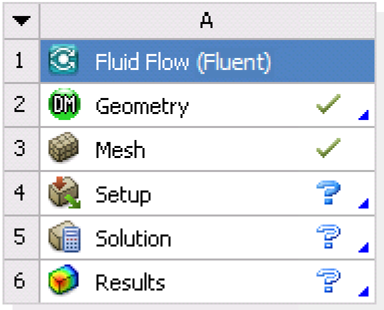
3. Double-click the **Mesh** cell. The Ansys Meshing application launches and loads the geometry file. The state of the **Mesh** cell becomes **Update Required**.



4. In the Ansys Meshing application, specify settings for the mesh, then select the **Update** command. The mesh is generated, the mesh (`.msh`) file is written, the state of the **Mesh** cell becomes **Up-to-Date**, and the state of the **Setup** cell becomes **Refresh Required**.

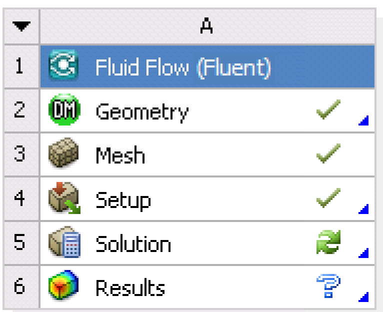


5. Double-click the **Setup** cell. Fluent launches and loads the mesh file. The state of the **Setup** cell becomes **Attention Required**.



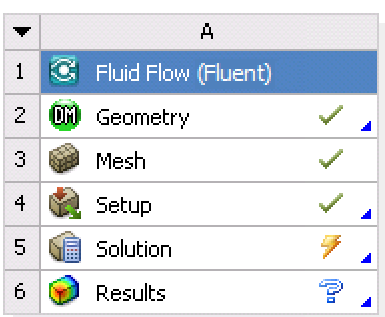
Fluid Flow (FLUENT)

6. In Fluent, specify boundary conditions, initialize the solution, and enter a nonzero number of iterations on the **Run Calculation** task page. The state of the **Setup** cell becomes **Up-to-Date**, and the state of the **Solution** cell becomes **Refresh Required**.



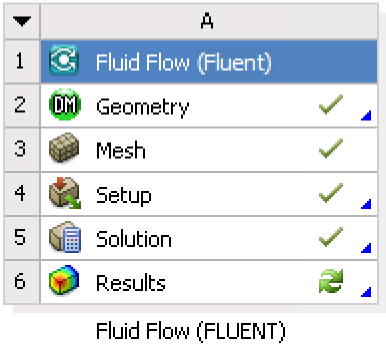
Fluid Flow (FLUENT)

7. In the Fluent application, select the **Calculate** button. The settings (`.set`) file is written and iterations begin. The state of the **Solution** cell becomes **Update Required**.



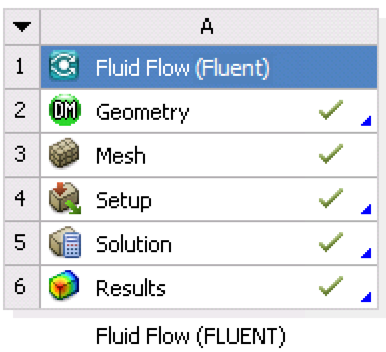
Fluid Flow (FLUENT)

Iterations are completed, or the solution meets the convergence criteria. The state of the **Solution** cell becomes **Up-to-Date** and the state of the **Results** cell becomes **Refresh Required**.



8. Double-click the **Results** cell. CFD-Post launches.

   The case (`.cas.h5`) and data (`.dat.h5`) files are written, CFD-Post loads the case and data files, and the state of the **Results** cell becomes **Up-to-Date**.

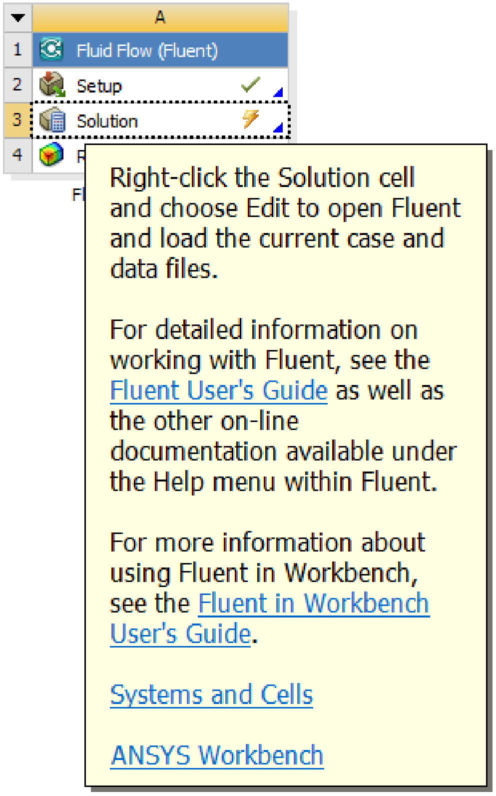


1.10. Getting Help for Fluent in Workbench

Accessing help from within Ansys Workbench:

Workbench offers three levels of help:

- Quick help - available for most cells in a system. Click the blue triangle in the bottom right corner of the cell to see a brief help description on that cell. For Fluent-based systems, Fluent-specific quick help is available for the **Setup** and **Solution** cells, providing you with instructions for proceeding further. For example:



- Sidebar or context-sensitive help - available at any time by clicking **F1**.
- Online help - available from the **Help** menu, or from any of the links in the quick help or sidebar help.

For more information about Workbench help, see the online documentation.

Accessing help from within Fluent:

Fluent documentation and help can be accessed by clicking once the Fluent application is running. The documentation is automatically installed when you install Workbench.

Chapter 2: Working With Fluent in Workbench

This chapter provides instructions for using Fluent in Workbench.

2.1. Importing Fluent files in Workbench

In Ansys Workbench, you can import Fluent mesh, case, and data files into a Fluent-based system in one of the following ways:

- from the Workbench **Project Schematic**
- from within the active Fluent session

The information about importing Fluent files in Workbench is described in details in the following sections:

2.1.1. Importing files from Workbench Project Schematic

The details of how you can import various file types into a Fluent-based system from Workbench **Project Schematic** are provided in the following sections:

2.1.1.1. Importing Mesh and Case Files

You can import Fluent mesh (`*.msh*`) and case (*.cas*) files into Fluent-based system using the **Import Fluent Case** command in **Setup** cell context menu as follows:

1. Right-click the **Setup** cell.
2. From the cell's context menu, point to **Import Fluent Case** and then click **Browse...**.
3. (For mesh files only) In the **Open** dialog box, select the **Fluent Mesh File** type from the drop-down menu next to **File name**.
4. Browse to the location of a specific case or mesh file and select the file.

Expected Outcome:

- **Mesh file import**: Once the import operation is complete, the **Setup** cell status becomes **Attention Required** ( ).

  You can launch Fluent to commence the setup procedure.

- **Case file import**: Once the import operation is complete, the **Setup** cell status becomes one of the following:

  – **Attention Required** ( ) – if the iterations or time step of the imported case is set to 0.

    You can launch Fluent to commence the setup procedure.

  – **Up-To-Date** ( ) – if the case iterations or time step of the imported case is set to a positive value.

    You can either proceed directly to the solution stage, or you can first alter the setup settings by editing the **Setup** cell.

If the **Setup** cell already has setup information associated with it, the setup information is deleted.

If your Fluent-based system contains a **Mesh** cell, Ansys Workbench informs you that the imported information will not be compatible with the information coming from the **Mesh** cell and asks whether the **Mesh** cell (and **Geometry** cell, if it also exists) should be deleted and replaced by the mesh from the imported file.

If your Fluent-based system has a connection to a **Mesh** cell in an upstream system, Ansys Workbench informs you that the mesh is being provided to the current system from an upstream system and asks whether the connection should be deleted and the mesh replaced with the mesh from the imported file.

2.1.1.2. Importing Solution Files

You can import Fluent solution files as start-up files or as final results as described below.

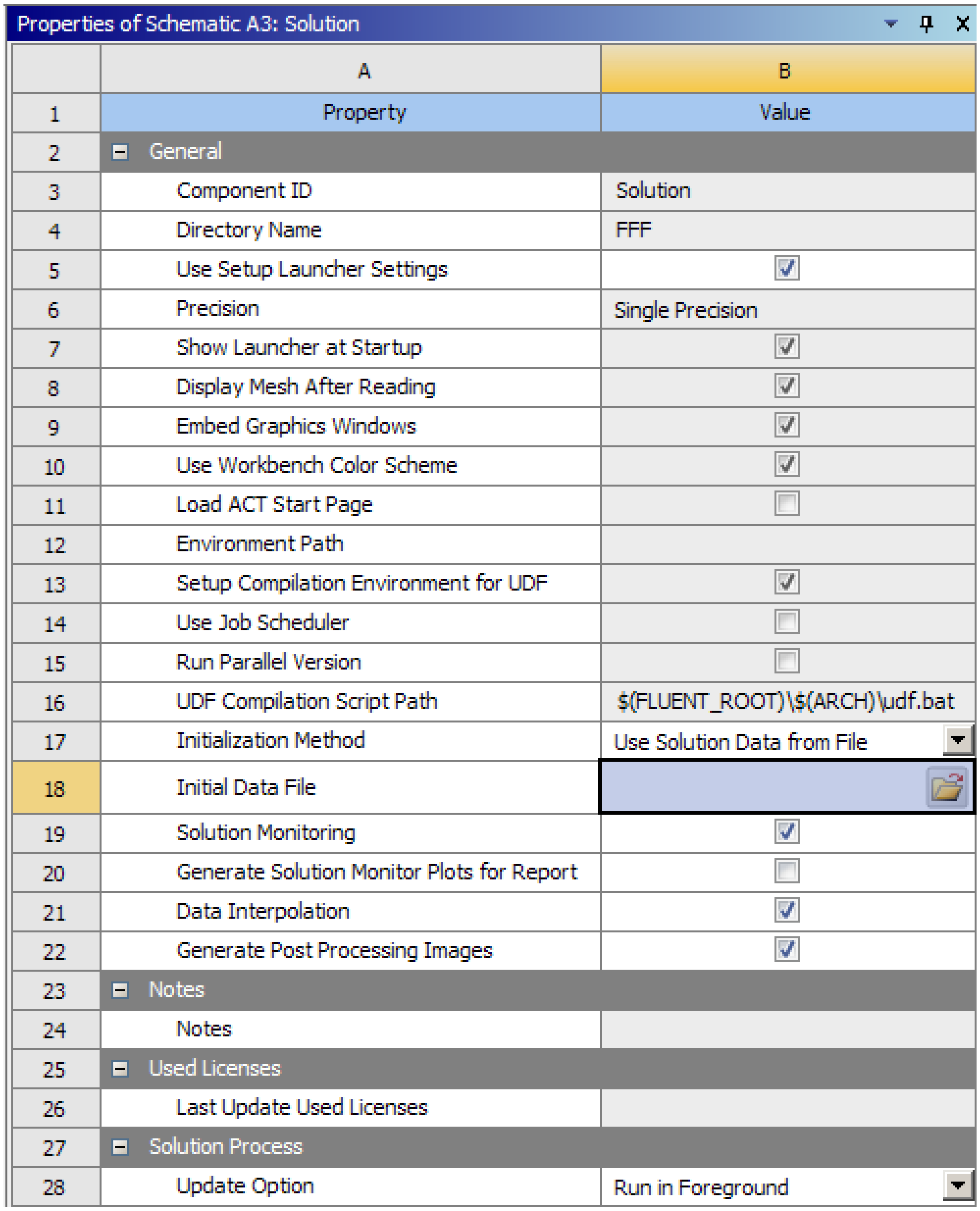
2.1.1.2.1. Importing Fluent Solution Files for the Purpose of Initialization

If you want to use the solution file as a start-up file, you can import the following files:

- `*.dat` or `*.dat.gz` – if imported results match the mesh used at the setup stage
- `*.ip` (interpolation file) – if imported results do not match the mesh used at the setup stage

To import the solution file as a start-up file into the Fluent-based system:

1. In the **Properties** pane for the **Solution** cell, select **Use Solution Data from File** from the **Initialization Method** drop-down list.



Properties of Schematic A3: Solution

	A	B
1	Property	Value
2	⊟ General	
3	Component ID	Solution
4	Directory Name	FFF
5	Use Setup Launcher Settings	☑
6	Precision	Single Precision
7	Show Launcher at Startup	☑
8	Display Mesh After Reading	☑
9	Embed Graphics Windows	☑
10	Use Workbench Color Scheme	☑
11	Load ACT Start Page	☐
12	Environment Path	
13	Setup Compilation Environment for UDF	☑
14	Use Job Scheduler	☐
15	Run Parallel Version	☐
16	UDF Compilation Script Path	$(FLUENT_ROOT)\$(ARCH)\udf.bat
17	Initialization Method	Use Solution Data from File
18	Initial Data File	
19	Solution Monitoring	☑
20	Generate Solution Monitor Plots for Report	☐
21	Data Interpolation	☑
22	Generate Post Processing Images	☑
23	⊟ Notes	
24	Notes	
25	⊟ Used Licenses	
26	Last Update Used Licenses	
27	⊟ Solution Process	
28	Update Option	Run in Foreground

2. In the **Initial Data File** cell that appears below **Initialization Method** enter the filename (including path) or navigate to the file location and select the file.

For the alternative procedure on importing initial results within Fluent, see Importing Files Directly in Fluent (p. 54).

Expected Outcome:

Importing an initial data file does not affect the state of the system. When the case and "initial" data files are imported into Fluent, Ansys Workbench treats them as start-up files and not results files. Therefore, the state of the **Solution** cell becomes **Refresh Required** ( ) and not **Up-to-Date** ( ).

Once the **Solution** cell is in the **Refresh Required** state, the old data generated in previous runs will be deleted when you **Refresh** or **Update** the cell (including autosaved data files).

Important:

The **Use Solution Data from File** option is not available if the **Solution** cell is connected to the up-stream component.

2.1.1.2.2. Importing Data Files as Final Results

You can also import a Fluent final data set (for example, a legacy Fluent data or a large data set solved on an external cluster) into a Fluent-based system. Using this approach, you can:

- Immediately perform postprocessing on the results using Ansys CFD-Post, or
- Directly transfer the data to another system in Ansys Workbench (for example, to perform one-way fluid-structure interaction (FSI) analysis).

To import the solution file as final results into the Fluent-based system:

1. Right-click the **Solution** cell.
2. From the cell's context menu, point to the **Import Final Data** and then click **Browse...**.
3. When prompted, browse to the location of a specific data file and select the file.
4. If the **Data Interpolation** option is enabled for the **Solution** cell, then you will be prompted about `.ip` file generation.

   If you click **Yes** *in the dialog box, then the* **Solution** *cell becomes* **Update Required***, otherwise it will remain* **Up-to-Date**.

Expected Outcome:

The state of the **Solution** cell becomes **Up-to-Date** (✓).

For unsteady cases only:

- If the last time-step data file is imported as a final data set, then Ansys CFD-Post can load all the previous time-step data as well, provided:
  - All case/data file pairs are originally generated by Fluent and their base names are the same.
  - All of the previous time-step data files are copied into the same Workbench Fluent system directory (you must do this manually) where the last data file is located (for example, `dp0/FLU/Fluent`). *You can see the location of the final data file by enabling* ***Files*** *from the* ***View*** *drop-down menu. Here you can also confirm that the previous time-step files are also located in the same folder.*
  - The case file imported into the **Setup** cell is the same as that of original case file (including the filename).

- If you want to use Ansys CFD-Post (the **Results** cell or component system) downstream of your Fluent system into which you have imported a pair of case and data files with different names, then you need to perform at least one iteration in Fluent in Workbench. When you close Fluent or save the Workbench project, Ansys Workbench automatically saves the new data file with the same name as the case file. Alternatively, you can manually rename the case and data files to have the same base filename prior to importing the final data set.

- The **Solution** cell does not extract any of the solution history information from the imported data file, and further calculations will result in loss of this information. Performing additional calculations changes the solution history, however, it does *not* affect the data that has been loaded into Ansys CFD-Post.

2.1.1.3. Importing Case and Data Files

You can also import Fluent final result files with corresponding case files into Fluent-based system in a single step using the **Import Fluent Case And Data** command in the **Setup** cell context menu:

1. Right-click the **Setup** cell.

2. From the cell's context menu, point to **Import Fluent Case And Data** and then click **Browse...**.

3. When prompted, browse to the location of a specific case or mesh file and select the file.

4. If the **Data Interpolation** option is enabled for the **Solution** cell, then you will be prompted about `.ip` file generation.

   If you click **Yes** *in the dialog box, then the* **Solution** *cell becomes* **Update Required**, *otherwise it will remain* **Up-to-Date**.

Expected Outcome:

The states of the **Setup** and **Solution** cells become **Up-to-Date** (✓).

Important:

Imported case and data files must share the same name for the importing operation to work properly.

2.1.2. Importing Files Directly in Fluent

Alternatively, you can also import pre-existing mesh, case, and data files from within Fluent by using one of the following ribbon tab items:

File → Import → Case...

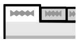
File → Import → Data...

File → Import → Case and Data...

When mesh and case files are imported from within Fluent using these commands, the behavior is exactly the same as when files are imported from the **Project Schematic**.

When data file is imported directly into Fluent using these commands, they are treated as initialization data. In this case, the behavior is the same as when using the **Use Solution Data from File** option in the **Properties** pane for the **Solution** cell (for details, see Importing Fluent Solution Files for the Purpose of Initialization (p. 51)).

In addition, you can also import the mesh from pre-existing mesh and case files from within Fluent by using the following ribbon tab item:

 **File → Import → Mesh...**

When using this command, Fluent gives you the following two options:

- **Discard Case, Read New Mesh**: discards any settings information currently in Fluent and imports the specified file. If the specified file is a case file, the settings information from that case file is also imported.
- **Replace Mesh**: preserves the settings information currently in Fluent and imports only the mesh from the specified file.

2.2. Using the Update Command

The **Update** command is available from:

- the context menu of all cells
- the context menu for the system
- the Workbench toolbar
- the Workbench **Tools** menu
- the context menu for the **Project Schematic**
- the **Workbench Tools** toolbar in Fluent () (not available for **Fluent (with Fluent Meshing)** component systems) (For more information, see Appendix C: The **Workbench Tools** Toolbar Commands (p. 603).)

When selected from a cell, the **Update** command updates the current cell and all upstream cells that must be updated to bring the current cell **Up-to-Date**. When a cell is updated, any new upstream data is passed to it before performing the update command.

When selected from the system, the **Update** command updates all of the out-of-date cells in the current system, as well as any cells in upstream systems that must be updated to bring the current system **Up-to-Date**.

When selected from the Workbench Toolbar, the Workbench **Tools** menu, or the context menu for the **Project Schematic**, the **Update** command updates all out-of-date cells in the project.

When updating the **Solution** cell in a Fluent-based system, the following steps take place:

1. Fluent launches in the background.

2. Fluent performs either the number of iterations (or time-steps) specified in the settings or case file or the number of iterations required to reach convergence.
3. Fluent writes the following files:
   - case file
   - data file
   - data interpolation file, `name.ip`

     The data interpolation file will be generated when **Data Interpolation** is selected in the **Solution** cell properties. Fluent will use this file as a restart data file in future sessions, but only if the solution data file becomes incompatible with the mesh.

     If you select the **Data Interpolation** property after completing calculations, the state of the **Solution** cell will change to **Update Required**. Upon the next update, Fluent will only write the `.ip` file.

     Note:

     If the data interpolation file is not available and the upstream mesh has not been modified, Fluent will use the regular data file (`.dat.h5`) as a restart file.

   - output case file, `name-Setup-Output.cas.h5`

     The output case file will be generated when

     – the **Setup** cell property **Generate Output Case File** is enabled, and

     – mesh operations have been performed prior to running the simulation in Fluent.
   - solution transcript file `Solution.trn`
   - residual file `SolutionMonitor.gz` (if **Solution Monitoring** is enabled in the **Properties** pane for the **Solution** cell) and miscellaneous files for generating reports (see Generating Fluent Project Reports (p. 68) for details).
   - image files for any graphics objects that are defined, such as contours and vectors. You can turn this feature on/off by toggling the **Generate Post Processing Images** option in the properties of the **Solution** cell in Workbench (right-click the **Solution** cell and select **Properties**).
4. If **Data Interpolation** is cleared in the **Solution** cell properties, Fluent removes the existing `.ip` file (if available).
5. Fluent exits.

If Fluent is already open, you can use the **Update** toolbar command from the **Update Mesh/Setup/Solution** tool () drop-down menu to bring the state of the system cells **Up-To-Date**.

While updating the **Solution** cell, you can visually monitor the solution convergence data for your Fluent simulations in Workbench using the solution monitoring charts, if **Solution Monitoring** is enabled in the **Properties** pane for the **Solution** cell. See Monitoring Fluent Solutions in Workbench (p. 64) for more information.

Important:

When an update is performed on a **Solution** cell and a Fluent session from the same system as that **Solution** cell is open, the calculation will be performed in that open Fluent session. This may not result in the action that you intended, therefore it is recommended that you close any open Fluent sessions before executing an **Update** command. For example, if you have specified initial data using either a connection from an upstream solution cell or by importing initial data into the **Solution** cell and you open Fluent from the **Setup** cell of the system, the initial data is not loaded (see Starting Fluent from a Fluent-Based System (p. 31)). If you subsequently perform an **Update**, the calculation will be performed in the open Fluent session and the initial data you specified will not be used as the starting point.

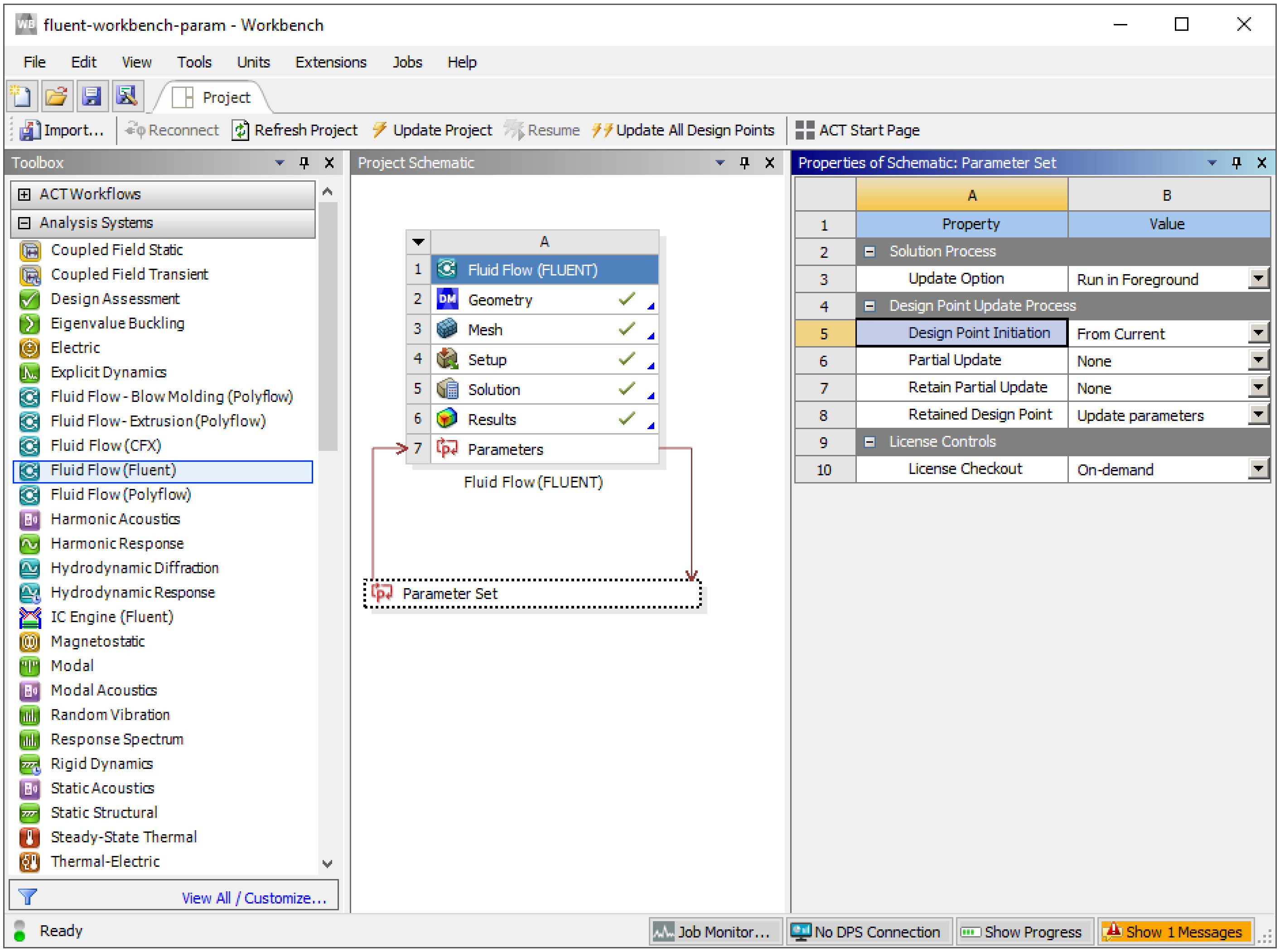
The **Update** command is particularly useful when you make changes to upstream data that impact downstream data. For example, if you start with an **Up-to-Date Fluid Flow (Fluent)** analysis system and then modify the mesh in the Ansys Meshing application, you can simply select **Update** from the system's context menu to generate the new results.

When performing an **Update**, you can specify whether the **Solution** cell in a Fluent-based system should be updated starting from existing solution data, case file data, or from an imported data file. This is specified by selecting an appropriate **Initialization Method** option from the **Solution** cell **Properties** pane (see Specifying Fluent Launcher Settings Using Cell Properties (p. 32)).

The **Initialization Method** options for the **Solution** cell in a Fluent-based system will always have the highest priority.

If you perform parametric studies, you can use the following **Design Point Initiation** options in **Properties of Schematic** for **Parameter Set**:

Figure 2.1: The Properties Pane of the Parameter Set



- **From Current**: Fluent uses the **Initialization Method** specified in the **Solution** cell properties (see Specifying Fluent Launcher Settings Using Cell Properties (p. 32)). Additionally, if **Program Controlled** or **Use Solution Data from File** options are selected and the mesh compatible solution data file is available, Fluent uses this data to initialize subsequent design point studies, or

- **From Previous Updated**: Fluent uses the last computed design point's data file to initialize the solution for the next design point (that is, DP2 uses DP1's data file, DP5 uses DP4's data file, and so on).

If the solution data is not available, or if the solution data is not compatible with the updated solution mesh, Fluent initializes the solution using the methods set in the case file. The available initialization methods in Fluent, relevant to Design Point studies in Workbench, consist of the following:

- The **Initialization Method** set in the **Solution Initialization** task page in Fluent.

- Specify a case modification strategy (see Case Modification Strategies with Fluent and Workbench (p. 86)).

Initial data can also be specified by:

- Importing an initial data file into the **Solution** cell or the Fluent application (see Importing Fluent files in Workbench (p. 49)).
- Creating a connection from an upstream **Solution** cell (see Connecting Systems in Workbench (p. 69)).

If you have imported an initial data file or created a connection to specify initial data, that data will override the initialization method specified on the **Solution Initialization** task page. If you have a case modification strategy defined, the initialization step in that strategy will always be performed regardless of whether initial data is specified in any other way.

Important:

- There is also an **Update** command in the Ansys Meshing application which generates the mesh and creates the input files required by downstream cells. The **Generate Mesh** command in the Ansys Meshing application generates the mesh but does not produce any input files. If a connection is made from an up-to-date **Mesh** cell, the state of the **Mesh** cell may become **Update Required**, indicating that the Ansys Meshing application needs to generate an additional input file. This file can be generated by selecting the **Update** command from the context menu of the **Mesh** cell. If you try to open Fluent before the **Mesh** cell is updated, a warning message is displayed informing you that you must update the **Mesh** cell before you can start Fluent, since the mesh file required for Fluent does not yet exist.
- Whether you edit the project through the **Setup** or **Solution** cells, your project's postprocessing settings in Fluent (including any surface definitions that you may have created), are saved at the start of iteration or when you close Fluent. When Fluent is later opened from either cell, these new settings will be available. Changes made to postprocessing settings in Fluent do not affect cell state.

2.3. Refreshing Fluent Input Data

You can refresh the input data for a cell by right-clicking the cell and selecting the **Refresh** command from the context menu. The **Refresh** command passes modified upstream data to the cell but does not conduct any long-running operations to regenerate the cell's output data.

For example, you can refresh the mesh by right-clicking the **Setup** cell in Workbench and selecting the **Refresh** command from the context menu. The state of the **Setup** cell becomes **Update Required**. It will become **Up-to-Date** the next time you launch Fluent from the **Setup** cell, or if you select the **Update** command from the context menu of the **Setup** cell.

You can refresh the input data for the **Setup** cell in a Fluent-based system by using either the **Refresh** command from the cell's context menu or, if Fluent is already open, by selecting the **Refresh Input**

Data option in the Fluent **File** ribbon tab, or using the **Refresh Input Data of Mesh/Setup/Solution** toolbar button ( ).

Important:

- Selecting the **Update** command from the context menu performs a **Refresh** command (if needed) before performing the **Update** command. You do not need to perform a **Refresh** and an **Update** in two separate steps.
- If you open Fluent after making a modification to the mesh and without refreshing the input data, you will be asked whether you want to load the modified mesh before Fluent launches.
- If Fluent is open and you make a modification to the mesh, you will be informed that the upstream mesh has changed and asked whether you want to load the new mesh before proceeding.

2.4. Reloading Data and Synchronizing Fluent with Workbench

When performing multiple preliminary investigative analyses with Fluent in Workbench (for example, establishing the most suitable solver settings) your Fluent settings are changed multiple times. In order to quickly and safely discard recently unsaved changes in your Fluent settings and to read input data from the **Setup** cell, you can use the **Reload** command in the Fluent **File** ribbon tab:

File → **Reload**

or the corresponding **Reload** toolbar command from the drop-down menu of the Mesh ( ), Setup ( ), or Solution ( ) Cell Commands. Reloading information will discard any changes performed in the current session and will delete any corresponding generated data from the Solution cell. This command is available only if input data is present. See Appendix C: The **Workbench Tools** Toolbar Commands (p. 603) for more details.

In addition, you can directly update Workbench with the most recent Fluent changes using the **Sync Workbench** option in the Fluent **File** ribbon tab, or the corresponding **Synchronize WB cell status** toolbar button ( ). You can also use the corresponding text command (`file/sync-workbench`).

2.5. Deleting Solution and Setup Cell Data for Fluent-Based Systems

The following sections describes how to delete solution and setup data for your Fluent-based systems by using the **Clear Generated Data** command, the **Reset** command, and the **Clear Old Solution Data** command.

2.5.1. Using the Clear Generated Data Command

You can delete all past and current generated files associated with a **Mesh**, **Setup**, or **Solution** cell by right-clicking the cell and selecting the **Clear Generated Data** command from the context menu. The following actions will be performed:

Solution cell: All past and current generated case and data files associated with the cell are deleted and the Fluent application is closed if it is open. All solution monitoring data generated during the run will be cleared from the **Scene** pane (if applicable).

Setup cell: output case file *`name`*`-Setup-Output.cas.h5` (if available)

Mesh cell: generated mesh file (if available)

If the cell status is **Up-to-Date**, it will become **Update Required** when the **Clear Generated Data** command is executed.

If Fluent is already open, you can use the corresponding **Clear Generated Data** command from the drop-down menu of the Mesh ( ), Setup ( ), or Solution ( ) Cell Commands. See Appendix C: The **Workbench Tools** Toolbar Commands (p. 603) for more details.

2.5.2. Using the Reset Command from the Setup and Solution Cells of Fluent-Based Systems

For either type of Fluent-based systems, you can delete all local and generated data from the **Setup** cell or from the **Solution** cell by right-clicking either cell and selecting the **Reset** command from the context menu.

For **Setup** cells, the **Reset** command removes the **Setup** cell's references to the mesh file, deletes the settings file associated with the **Setup** cell, sets the cell property values to their defaults, and closes the Fluent application if it is open. If the **Setup** cell is **Up-to-Date**, it will become **Refresh Required** when the **Reset** command is executed.

For **Solution** cells, the **Reset** command deletes all past and current case and data files (*not* including imported initial data files) associated with the cell, sets the cell property values to their defaults, and closes the Fluent application if it is open. If the **Solution** cell is **Up-to-Date**, it will become **Refresh Required** when the **Reset** command is executed.

If Fluent is already open, you can use the corresponding **Reset** command from the drop-down menu of the Mesh ( ), Setup ( ), or Solution ( ) Cell Commands. See Appendix C: The **Workbench Tools** Toolbar Commands (p. 603) for more details.

2.5.3. Using the Clear Old Solution Data Command from the Solution Cells of Fluent-Based Systems

For either type of Fluent-based systems, you can delete all older local and generated data from the **Solution** cell by right-clicking the cell and selecting the **Clear Old Solution** command from the context menu.

For **Solution** cells, the **Clear Old Solution Data** command retains only the most recent solution files associated with the **Solution** cell, and removes older case and data files that are not part of the current solution history. This command is only available if there are already solutions associated with the cell, in addition to the most recent solution.

Note:

- Archived projects will only contain the most current solution files, and all solution files that are associated with the current run (that is, solution history).
- While importing design points, the next design point (`dp1`) will contain only the last data file of the current solution from the last design point (`dp0`).

2.6. Interrupting, Restarting, and Continuing a Calculation

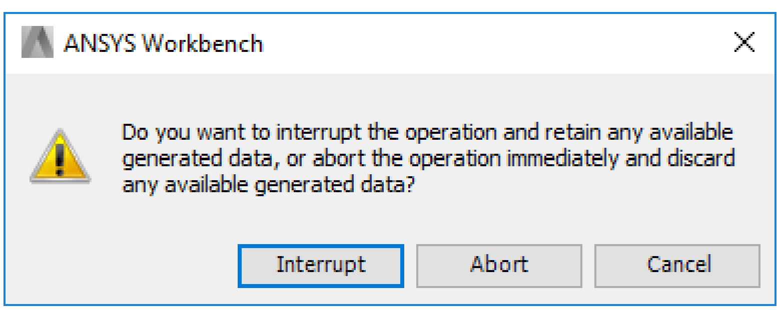
The following topics are discussed in this section:

2.6.1. Interrupting a Calculation

To interrupt the calculation in an interactive Fluent session, do one of the following:

- Click **Cancel** in the **Working** dialog box
- Press **Ctrl**+**c**

To interrupt an ongoing Fluent calculation from Workbench, click the  icon in the **Progress** pane (see Progress Pane in the *Workbench User's Guide* for more details). This displays a prompt, asking if you would like to interrupt or stop the calculations.



You can select from the following options:

- **Interrupt**: Stops the calculation at the next point where data can be safely stored for later use.

- **Abort**: Stops the calculation immediately without concern for whether data associated with the current action can be stored. Closes the associated Fluent session, if open.

Important:

When the calculation is interrupted from within an interactive Fluent session, it always stops the data at the next point where data can be safely stored for later use.

When a calculation is interrupted, the state of the **Solution** cell becomes **Interrupted, Update Required**. If a background calculation is interrupted, Fluent writes the case and data file and then closes.

If you interrupt a calculation, review the results, and decide that the solution is converged, you can force the **Solution** cell state to be **Up-to-Date** by right-clicking the **Solution** cell and selecting the **Accept Interrupted Solution as Up-to-Date** command from the context menu.

2.6.2. Continuing and Restarting a Calculation

You can continue a previously interrupted Fluent calculation from within Workbench by right-clicking the **Solution** cell and selecting **Continue Calculation** or **Update** from the context menu. This will allow you to continue the calculation from the current case and data files, performing the total number of iterations (or time-steps) specified in the case file.

If Fluent is already open, you can use the corresponding **Continue** toolbar command from the **Update Mesh/Setup/Solution** tool ( ) drop-down menu. If you want to start the interrupted simulation over, you can use the **Restart** toolbar command from the **Update Mesh/Setup/Solution** tool ( ) drop-down menu. See Appendix C: The **Workbench Tools** Toolbar Commands (p. 603) for more details.

2.6.3. Recovering Case and Data Files after an Unexpected Interruption

To recover the missing latest matching/compatible case and data files from the system folder:

1. Open Fluent from the **Solution** cell.

   Warning:

   Opening Fluent from the **Setup** cell will clear all solution data.

2. Select the **File/Solution Files...** ribbon tab item.
3. In the **Solution Files** dialog box, click **Recover Missing Solution...**
4. From the **Solution Files at** list, select the desired restart point, click **Read** and close the **Solution Files** dialog box.

For more information, see Managing Solution Files in the *Fluent User's Guide*.

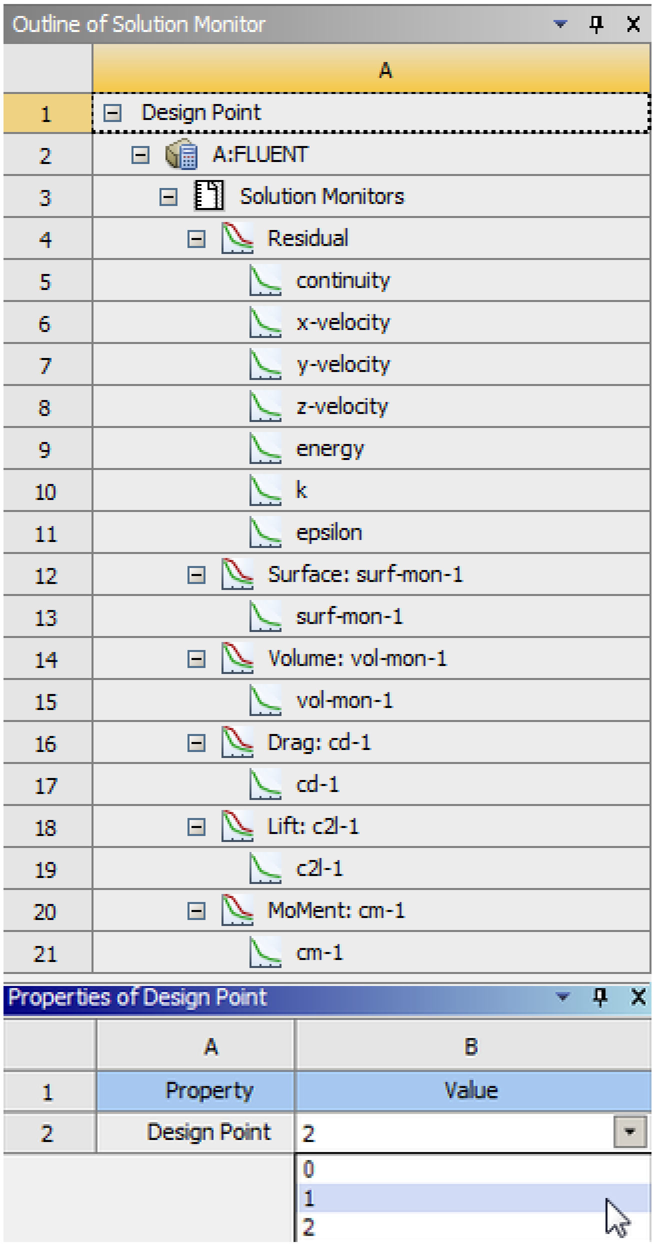
2.7. Monitoring Fluent Solutions in Workbench

You can visually monitor solution convergence data for your Fluent simulations in Workbench using solution monitoring charts. To control the display of the solution monitoring charts, select the **Solution Monitoring** option in the **Properties** pane for the **Solution** cell.

Once this option is enabled, you can open the solution monitor workspace from within Workbench by right-clicking a Fluent-based system **Solution** cell and selecting the **Show Solution Monitoring** command from the context menu.

The **Show Solution Monitoring** command opens the **Solution** tab with an **Outline** pane for the **Solution Monitors** cell (see Figure 2.2: Example of the Solution Monitors Outline Pane Within Workbench (p. 65)) and a chart for the system (see Figure 2.3: Example of a Fluent Scene Pane within Workbench (p. 67)).

Figure 2.2: Example of the Solution Monitors Outline Pane Within Workbench



The Outline Pane

The following types of monitors are available in the **Outline of Solution Monitor** pane:

- Residual convergence monitors, such as Continuity, Energy, Velocity, and so on
- User-defined monitors that have been defined in Fluent, such as Surface, Volume, Lift, Drag, and Moment based monitors.

The residual convergence monitors are grouped under the **Residual** group. User-defined monitors are listed with their names specified in Fluent under corresponding monitor groups. The default names for monitor groups have the following pattern: ***`type-name`*** , where ***`type`*** is the type of user-defined monitor and ***name*** is the monitor name defined in Fluent. You can edit the names of the monitor groups directly in the cells. The monitor names can only be edited in Fluent.

The Scene Pane

As the Fluent simulation progresses, the convergence graphs are plotted interactively in the **Scene** pane. By default, Workbench displays a fully populated residual convergence chart (for example, continuity, energy, velocity, and so on). The user-defined monitor charts (that is, force, moment, and integrals) are not displayed by default. To view user-defined monitor plots, click the corresponding cell in the **Outline of Solution Monitor** pane.

Once the solution is complete, or while the solution is progressing, right-click the cell for the appropriate monitor group and select **Remove Variable**, **Add Variables**, or **Add All Variables** (if applicable) from the context menu to add or remove available residual and/or user-defined monitors in the Fluent **Scene** pane (see Figure 2.3: Example of a Fluent Scene Pane within Workbench (p. 67)). You can create a new residual or user-defined chart by right-clicking the **Solution Monitors** cell in the **Outline** pane and selecting **Create Residual Monitor Chart** or **Create User-Defined Monitor Chart** respectively from the context menu.

The solution monitor chart is embedded in the Workbench **Scene** pane and once it is visible, it can be hidden and displayed by disabling or enabling the **View** > **Scene** menu option, respectively. You can save the chart that you are viewing as a graphic by right-clicking the background of the chart and select **Save Image As** from the context menu. You can alter the appearance of individual plots by changing **Color**, **Line Width** and **Symbol Size** in the **Properties** pane for the correspondent plot cells. You can control the settings for the monitor charts in the **Scene** pane using the **Properties** pane for the monitor groups as described in Setting Chart Properties in the *Workbench User's Guide*.

Note:

Solution monitoring in Ansys Workbench provides only basic insight into the convergence process of your simulation. To access full spectrum of solution monitoring features available in Ansys Fluent, you need to run an interactive session in Fluent.

You can also monitor the solution convergence of any design points in your simulation. When you perform an update of all design points during the calculation, the **Design Point** value (as shown in Figure 2.2: Example of the Solution Monitors Outline Pane Within Workbench (p. 65)) changes internally and will be in read-only mode. Once all of the design points are updated, you can change the **Design Point** value (using the corresponding drop-down menu) to view the data of any design point. Once the **Design Point** is selected, then the data of the selected system will be displayed in the **Scene** pane.

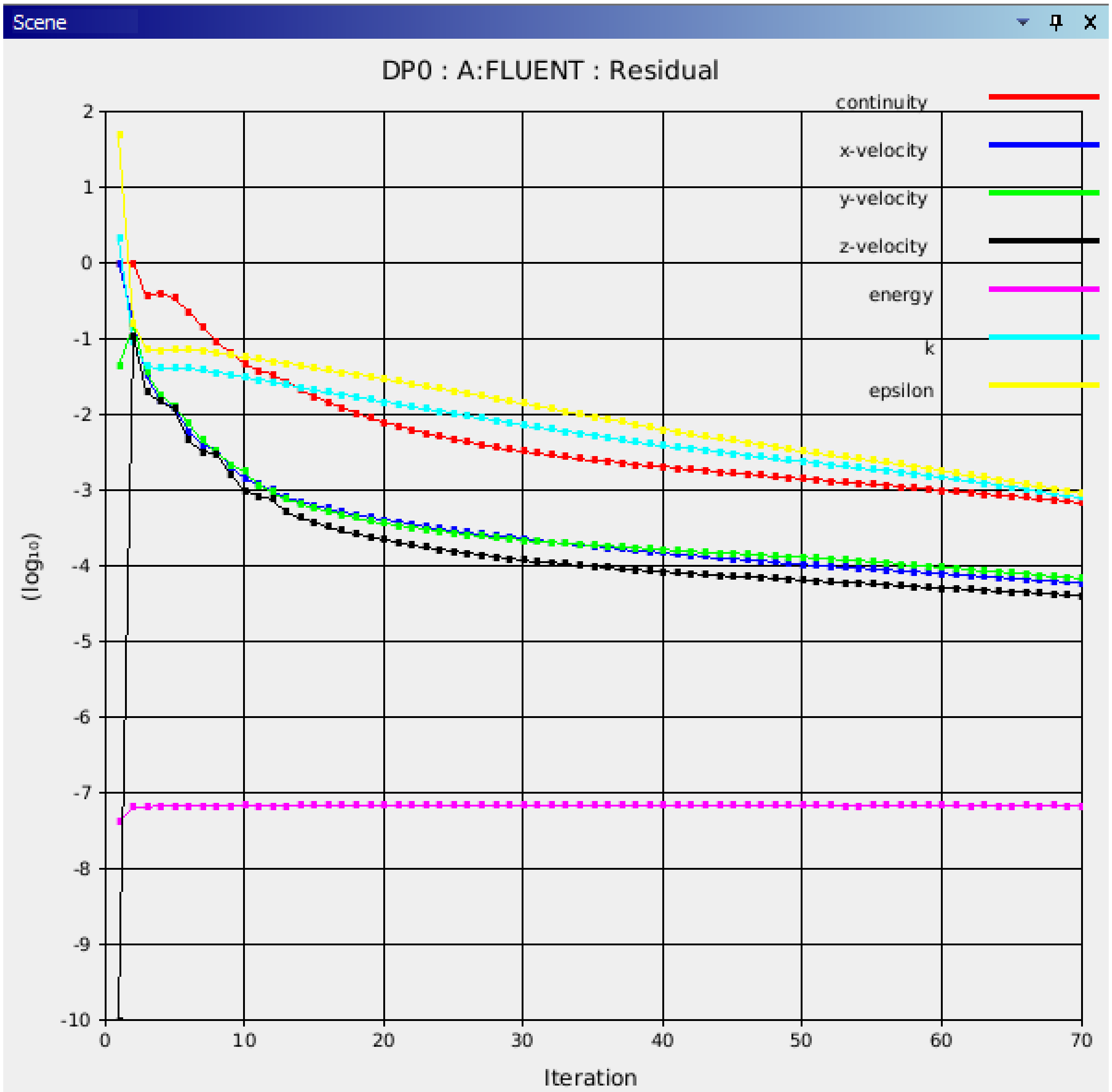
Note:

- For the following situations, solution monitoring data will become available only at the end of the calculation and the chart will not be updated during the calculation:
  - when you run a parametric study in the foreground and you update the **Solution** cell in the background, or

- when you remotely run a parametric study, for example, calculating design points using the Remote Solve Manager (RSM) for the parameter set.

- All residual monitoring data is plotted with iteration number as X axis data. User-defined monitors can be plotted with Iteration, Flow Time, or Time Step as the X axis quantity.

Figure 2.3: Example of a Fluent Scene Pane within Workbench



For more information about using charts in Workbench, see Working with the Chart Pane.

The Fluent Transcript Pane

The **Fluent Transcript** pane displays the content of the Fluent solution transcript. To close the **Fluent Transcript** pane, deselect it from the **View** menu of the **Solution** tab.

In a design point study, you can review the solution transcript for each design point (if available) by selecting the design point in the **Properties** pane for the **Design Point** cell as shown in Figure 2.2: Example of the Solution Monitors Outline Pane Within Workbench (p. 65). By default, the transcript for the current design point is displayed.

When simulating multiple systems, you can review the transcript for each system (if available) by clicking any cell relevant to the system of interest in the **Outline** pane.

During the foreground or component-level local Remote Solve Manager (RSM) update of the system, the **Fluent Transcript** pane will continue being updated with the solution progress information.

You can use the following standard keyboard short-cuts to navigate through the content displayed in the **Fluent Transcript** pane:

- **Page Up** and **Page Down** to scroll up or down one page, respectively
- **Ctrl**+**Home** and **Ctrl**+**End** to jump to the top or the bottom of the transcript, respectively

The Fluent Result Image Pane

The **Fluent Solution Images** pane displays contour and vector plots that have been previously defined in Fluent (see Creating and Using Contour Plot Definitions and Creating and Using Vector Plot Definitions in the *Fluent User's Guide* for details). To access the **Fluent Solution Images** pane, select **Fluent Solution Images** from the **View** menu of the **Solution** tab. You can use the scroll bar or standard navigation keyboard buttons (**Page Up**, **Page Down**, **Ctrl**+**Home** and **Ctrl**+**End**) to view a desired plot.

Similar to the **Fluent Transcript** pane, if you are running a design point study, you can review solution plots for each design point by selecting the desired design point in the **Properties** pane for the **Design Point** cell.

To close the **Fluent Solution Images** pane, deselect it from the **View** menu of the **Solution** tab.

To return to viewing the **Project Schematic**, click the **Project** tab above the Workbench toolbar.

Note:

Fluent Solution Images are not populated when running Fluent in the background or via the Remote Solve Manager.

2.8. Generating Fluent Project Reports

When you update a **Solution** cell or interrupt an ongoing Fluent calculation, Fluent automatically generates the following files:

- `report.xml` that contains the information about the current design point.
- `.png` images of residuals and any solution monitors that have been defined in Ansys Fluent (if **Generate Solution Monitor Plots for Report** is selected in the **Properties** pane for the **Solution** cell.

- (if available) `.png` images of contour and/or vector plots that have been previously defined in Ansys Fluent.

Fluent saves these files in appropriate file directories within the Workbench project. These files appear in the **Files** pane.

The Fluent project report XML file contains the following information in a tabular format:

`Version`

displays the information about the solver version.

`Models`

lists the Ansys Fluent models and their status.

`Material Properties`

lists all the materials defined in the case and their properties.

`Cell Zone Conditions`

displays information about cell zones defined in the problem and their setup conditions.

`Boundary Conditions`

displays information about boundary zones that exist in the problem and their specific conditions.

`Solver Settings`

shows information about solution controls, discretization schemes, and other solution-related settings specified in the simulation.

When you export the project report as HTML as described in Working with Project Reports in the *Workbench User's Guide*, the content of the Fluent project report file will be added to the end of the generated project report. In addition, the following sections will be included at the end of the project report:

`Solution Monitor Plot Images`

display the convergence history and available solution monitor plots (if **Generate Solution Monitor Plots for Report** is selected in the **Properties** pane for the **Solution** cell). Note that the solution monitor plot images can be created only if the `SolutionMonitor.gz` file has been generated during the update.

`Post Processing Graphics Object Images`

(if available) display contour and vector plots that have been previously defined in Fluent.

2.9. Connecting Systems in Workbench

In Workbench, you can create connections between multiple systems that enable the systems to access the same data. This is useful, for instance, when you want to compare the results from multiple Fluent-based systems in the same Ansys CFD-Post session. In this case, you would connect the various **Solution** cells to one **Results** cell (either in one of your Fluent-based systems or in a separate **Results** system). When you double-click that **Results** cell, the results from all connected systems will be loaded into Ansys CFD-Post at the same time.

Another example would be where you want to load multiple meshes created in the Ansys Meshing application to your Fluent-based system. See Connecting Multiple Upstream Meshes to a Setup Cell of a Fluent-Based System (p. 71) for further information on such connections.

Workbench supports two different types of connections:

- Shared data connections
  - Used when the inputs and outputs of the two connected cells are identical
  - Can only be created between two cells of the same type

  A shared data connection is represented in the **Project Schematic** by a line with a square on its right (target) side (see Figure 2.4: Connected Systems Within Workbench (p. 71)).

- Transfer data connections
  - Used when the output of one cell is used as the input to the connected cell
  - Usually created between two cells of different types. One exception is that a transfer data connection can be used between the **Solution** cells of two Fluent-based systems when you want to use the current data from one system as the initial data for the other system.

  A transfer data connection is represented in the **Project Schematic** by a line with a circle on its right (target) side (see Figure 2.4: Connected Systems Within Workbench (p. 71)).

There are four ways to create connected systems in Workbench.

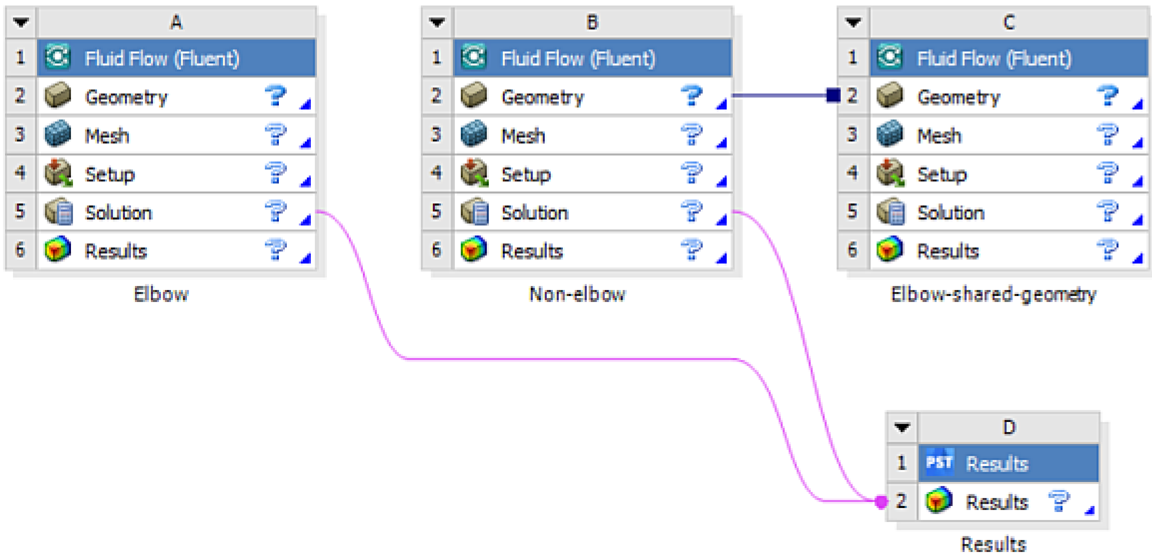
- Click a cell in one system, then drag and drop it onto a compatible cell in another system.
- Click a system in the Toolbox, then drag and drop it onto a compatible system in the **Project Schematic**.
- Create a duplicate system (see Duplicating Fluent-Based Systems (p. 75)).
- Right-click a cell and select one of the options under **Transfer Data From New...** or **Transfer Data To New...** (these options are not available for all cells). Transferring data from the **Solution** cell to a new Fluent system's **Setup** cell transfers the mesh (but not the settings) to the **Setup** cell.

When you click and hold down the mouse button at a system in the Toolbox, Workbench highlights all of the compatible drop targets in the **Project Schematic**. As you move the mouse over a drop target, it is highlighted in red and a message appears in the **Project Schematic** that informs you what the result will be if you drop the system onto that target.

There are usually several compatible drop targets on empty space in the **Project Schematic**. Dropping the system onto one of these targets will create a stand-alone system in that location.

Similarly, when you highlight a cell and begin to drag it, Workbench highlights all of the compatible drop targets in the **Project Schematic**. As you move the mouse over a drop target, it is highlighted in red and a message appears in the **Project Schematic** that informs you what the result will be if you drop the cell onto that target.

Figure 2.4: Connected Systems Within Workbench



For more information about connecting systems, see the Workbench online help, as well as the following sections:

2.9.1. Connecting Multiple Upstream Meshes to a Setup Cell of a Fluent-Based System

You can connect multiple **Mesh** cells of **Mesh** component systems to the **Setup Cell** of a Fluent-based system. The mesh data, including the mesh input parameters that you may have defined in the Ansys Meshing application, will then be transferred to the **Setup** cell. When you start Fluent from the **Setup** cell, Fluent will read the mesh data from the connected **Mesh** cells in the order in which the connections were created. If your **Fluent**-based system contains a **Mesh** cell, **Fluent** will read this data first.

Note the following limitation with these connections:

- If any upstream mesh is modified while the currently open Fluent session is running, then on refresh, Fluent will reload all mesh files.

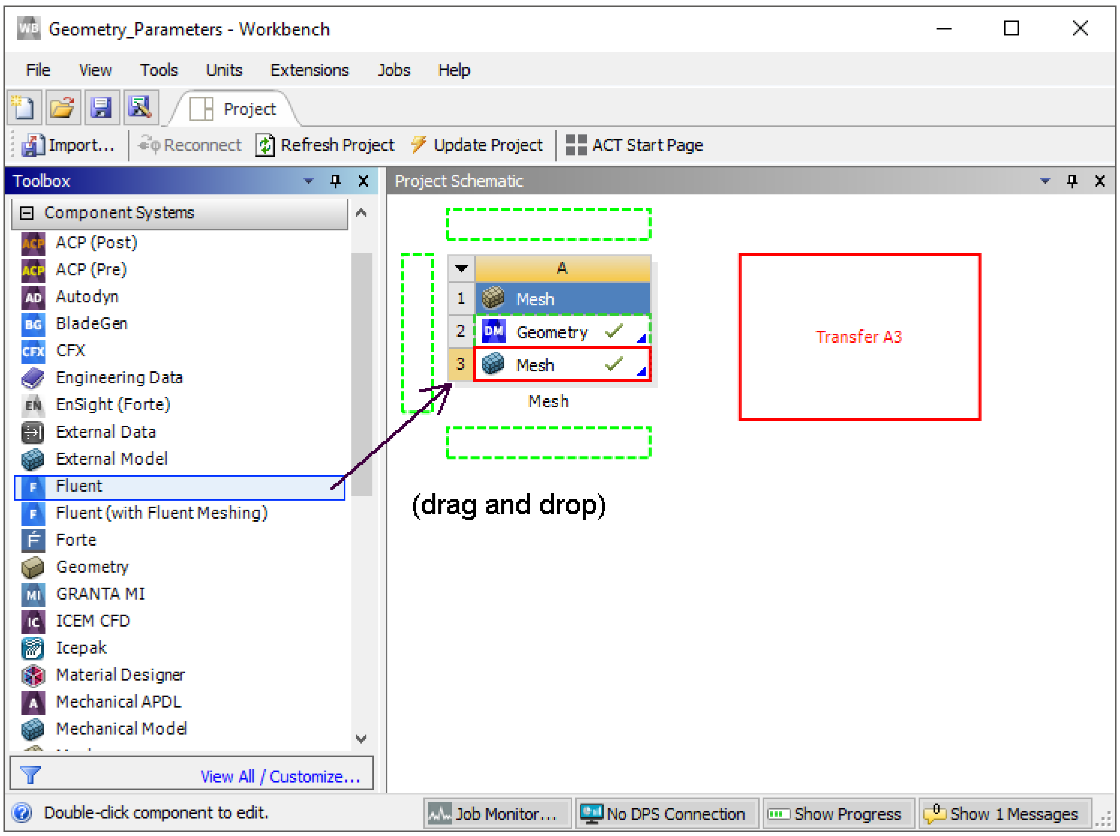
Important:

If you change the zone topology of one of the upstream meshes, Fluent may report a mesh incompatibility. You must then resolve the mesh incompatibility issue in order to proceed with the case setup.

2.9.2. Connecting Systems by Dragging and Dropping a System from the Toolbox onto Another System

The following example demonstrates the procedure for creating connected systems by dragging a system from the Toolbox and dropping it onto a compatible system in the **Project Schematic**. (See Figure 2.5: Applying the Mesh Settings to a New Fluent-Based Component System by Dragging and Dropping Systems (p. 72))

Figure 2.5: Applying the Mesh Settings to a New Fluent-Based Component System by Dragging and Dropping Systems



1. Starting from a project with an up-to-date **Mesh** component system, select the Fluent-based component system from the Toolbox; the compatible drop targets are highlighted in green.

2. Drag the system over the **Project Schematic** and pause over the **Mesh** cell of the **Mesh** component system; the **Mesh** cell target is highlighted in red and a message informs you that selecting that target will transfer the data from cell A3 to the new system.

3. Drop the system on the drop target and a transfer data connection is created between the **Mesh** cell A3 and the **Setup** cell B1.

Note that **Mesh** cell A3 becomes **Update Required**, this is because the input data for the new system has not yet been generated by the Ansys Meshing application.

Important:

If you try to open Fluent before the **Mesh** cell is updated, a warning message is displayed informing you that you must update the **Mesh** cell before you can start Fluent, since the mesh file required for Fluent does not yet exist.

---

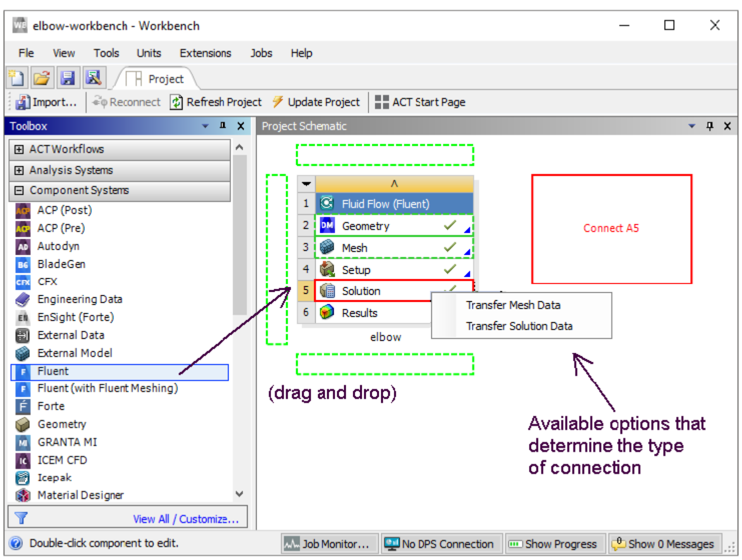
4. Right-click **Mesh** cell A3 and select **Update**.

5. Double-click **Setup** cell B1; Fluent launches and loads the mesh from cell A3.

In the previous example, a transfer data connection was created. Shared data connections can also be creating by dragging a system from the Toolbox and dropping it onto a compatible system in the **Project Schematic**. The type of connection that Workbench creates depends on which drop target you select. The red preview messages in the **Project Schematic** inform you of the type of connection(s) that will result from your action.

When a Fluent-based component system is dragged from the Toolbox onto the **Solution** cell of an existing Fluent-based analysis system, you are presented with two choices: **Transfer Solution Data** or **Transfer Mesh Data**. The connection that is made between the two systems is based on the option selected.

Figure 2.6: Transferring Solution Data or Mesh Data to a New Fluent-Based Component System by Dragging and Dropping Systems



Important:

The mesh from the case file associated with the **Solution** cell of a Fluent-based analysis system or component system can be transferred to the **Setup** cell of a Fluent-based component system only and not the **Setup** cell of a Fluent-based analysis system.

2.9.3. Connecting Systems By Dragging and Dropping Fluent-Based Solution Cells Onto Other Systems

The following figures demonstrate the procedure for creating a transfer data connection by dragging a **Solution** cell from a Fluent-based system and dropping it onto a compatible cell in another system:

Figure 2.7: An Example of Two Unconnected Systems

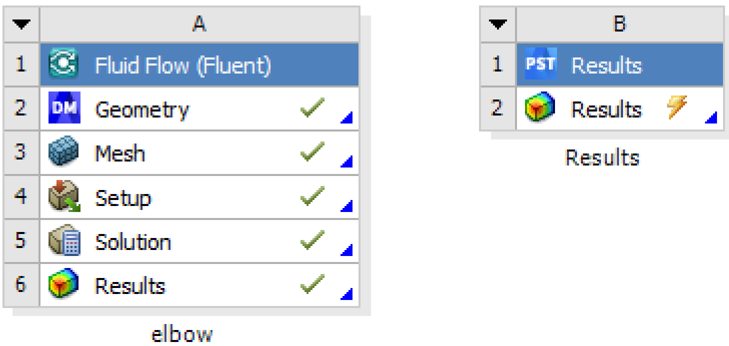


Figure 2.8: An Example of Dragging and Dropping a Solution Cell Onto Another Compatible Cell

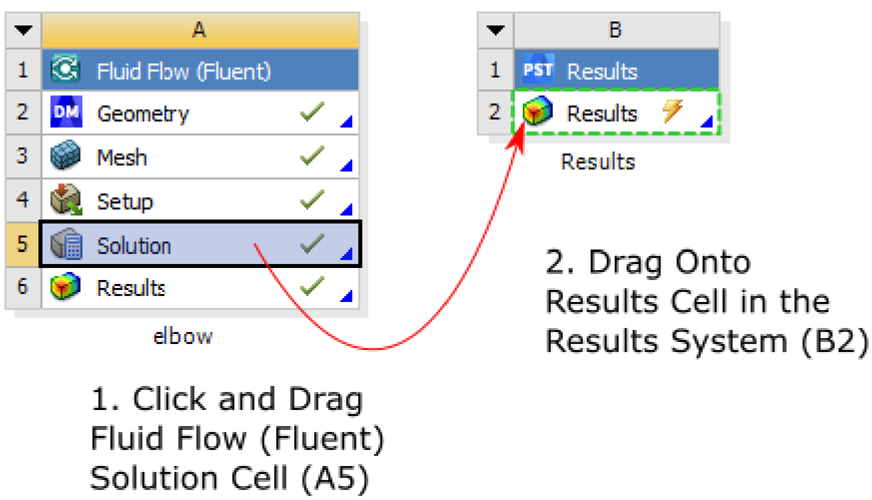
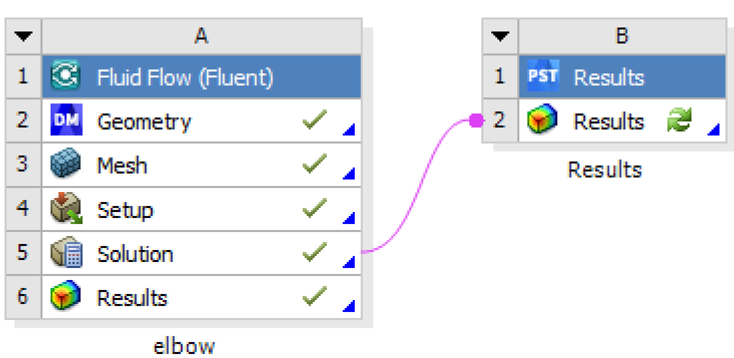


Figure 2.9: An Example of Two Connected Systems



The following table lists the compatible drop targets for the **Solution** cell from a Fluent-based system:

Table 2.1: Compatible Drop Targets for the Solution cell of a Fluent-Based System

Analysis or Component System	Cell(s)
Fluent	**Setup** **Solution**
Fluid Flow (Fluent)	**Solution** **Results**
CFX **Fluid Flow (CFX)** **Vista TF** **Results**	**Results**
Static Structural (ABAQUS) **Static Structural (Samcef)** **Shape Optimization (Ansys)** **Static Structural** **Steady-State Thermal** **Transient Structural** **Thermal-Electric** **Transient Thermal**	**Setup**

2.10. Duplicating Fluent-Based Systems

Workbench allows you to create a duplicate of a system so that you can set up multiple, similar systems and analyze them at the same time. For instance, if you would like to study the differences in the fluid flow between two slightly different geometries, then you can create, set up, and solve a single fluid flow analysis system, duplicate the entire system, change the geometry in the duplicate system and perform another fluid flow analysis on the new geometry.

You can create a duplicate of a Fluent-based system by performing the following steps:

1. In the **Project Schematic**, right-click the system header to open the system's context menu.
2. Select **Duplicate** from the context menu.

   A copy of the original Fluent-based system is created in the **Project Schematic**.

All data associated with the Fluent-based system, except for any case, data, and initial data files associated with the **Solution** cell, are copied to the new system. The states of the **Geometry**, **Mesh**, and **Setup** cells in the new system will be the same as the states of the cells in the original system. The state of the **Solution** and **Results** cells in the new system will be different than those of the original system if the original system had case and data files associated with its **Solution** cell.

In addition, you can use the **Duplicate** command to create a duplicate of a Fluent-based system in which the data in the **Geometry** cells or the data in both the **Geometry** cells and the **Mesh** cells is shared between the two systems rather than copied.

To create a duplicate system in which the geometry is shared between the original and new system:

1. In the **Project Schematic**, right-click the **Mesh** cell in the system you want to duplicate to open the context menu.

2. Select **Duplicate** from the context menu.

   A copy of the original Fluent-based fluid flow system is created in the **Project Schematic**. A shared data connection is created between the **Geometry** cell in the original system and the **Geometry** cell in the new system.

To create a duplicate system in which both the geometry and the mesh are shared between the original and new system:

1. In the **Project Schematic**, right-click the **Setup** cell or any cell below it in the system you want to duplicate to open the context menu.

2. Select **Duplicate** from the context menu.

   A copy of the original Fluent-based fluid flow system is created in the **Project Schematic**. Two shared data connections are created: one between the **Geometry** cell in the original system and the **Geometry** cell in the new system, and the other between the **Mesh** cell in the original system and the **Mesh** cell in the new system.

2.11. Recording Mesh Manipulation Operations and Resolving Mesh Incompatibility in Fluent

When you perform mesh manipulation operations in Fluent while running under Workbench, Fluent preserves some basic mesh operations listed below. These operations are saved as part of the Workbench project in the **Setup** cell such that the operations are explicitly applied by Workbench once Fluent loads the mesh (and before reading the settings file) and automatically reapplied the next time an update is performed. This enables you to more easily update the system if the upstream mesh has changed, or parameters have been modified (see Changing the Settings and Mesh in Fluent (p. 81)).

Fluent under Workbench preserves the following mesh operations:

- Scaling
- Rotation
- Translation
- Smoothing
- Swapping

- Merging
- Fusing
- Zone/Domain renaming
- Zone/Domain reordering
- Zone type changing (The final zone type is dependent on the settings file. This is recorded primarily to address occasional mesh changes triggered by a change in the zone type.)
- Creating periodic zones
- polyhedra operations (convert to polyhedra, convert skewed cell to polyhedra)

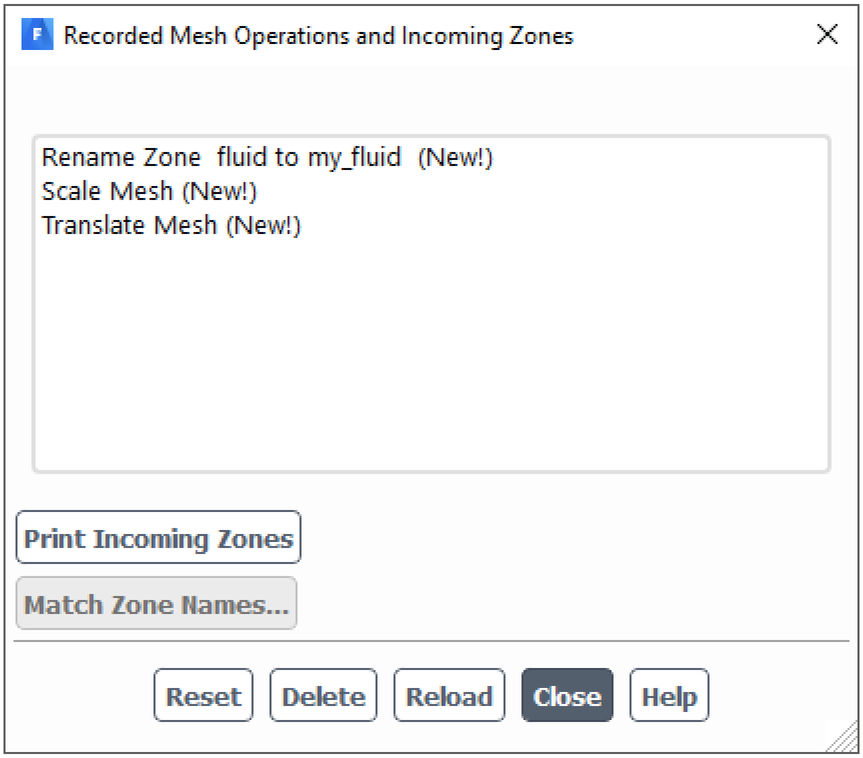
The following mesh operations are not recorded within Fluent in Workbench:

- Separating faces and cells
- Zone operations, such as activating, deactivating, deleting, and replacing zones, as well as appending case and data files
- Replacing meshes
- Adapting meshes, specifically using:
  - Boundaries
  - Gradients
  - Iso-values
  - Regions
  - Yplus/Ystar
  - Anisotropic
  - Volume

See Modifying the Mesh in the User's Guide for more information about manipulating the mesh in Fluent.

When you start Fluent from the **Setup** cell in Workbench, you can review or delete the preserved mesh manipulation operations using the **Recorded Mesh Operations and Incoming Zones** dialog box (Figure 2.10: The Recorded Mesh Operations and Incoming Zones Dialog Box (p. 78)). When Fluent is started from the **Solution** cell, the **Recorded Mesh Operations** toolbar button is not available.

Figure 2.10: The Recorded Mesh Operations and Incoming Zones Dialog Box

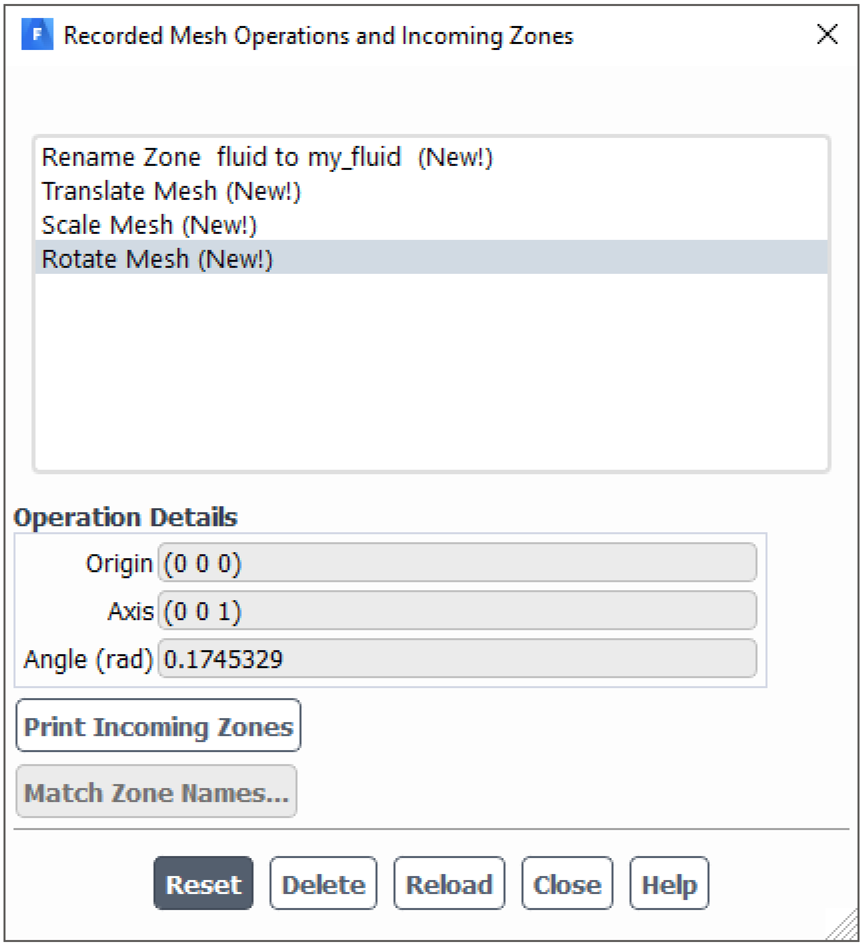


To access the **Recorded Mesh Operations and Incoming Zones** dialog box, select the **File/Recorded Mesh Operations...** ribbon tab item (or click the **Recorded Mesh Operations** toolbar button ).

In the **Recorded Mesh Operations and Incoming Zones** dialog box, you can view the mesh manipulation operations that are currently stored. You can also delete, see new operations, or see if an operation has failed due to an incompatible upstream mesh.

To view information about a mesh operation, select it from the recorded mesh operations list. The specific details about the selected mesh manipulation operation appear under **Operation Details**. In Figure 2.11: Reviewing the Details of Rotating the Mesh in the Recorded Mesh Operations Dialog Box (p. 79), you can see the details of the rotation operation applied to a mesh. Note that when rotating the mesh, the angle is displayed in radians, and when translating the mesh, the distance is displayed in meters.

Figure 2.11: Reviewing the Details of Rotating the Mesh in the Recorded Mesh Operations Dialog Box



To review information about boundary and cell zones coming from an upstream mesh, click **Print Incoming Zones**. Fluent lists the details about the incoming zones in the console.

```
Incoming zones from upstream mesh data are:
  id  name                        type               material              kind  status
----  ------------------------    -----------------  --------------------  ----  ----------
   1  wall-part_1                 wall               air                   face  regular
   2  interior-part_1             interior                                 face  regular
  20  velocity_inlet1             velocity-inlet                           face  new
   -  pressure_outlet             -                                        -     modifed
   3  part_1                      fluid              air                   cell  regular
```

The `status` column indicates whether a zone is `new`, `modified`, or `regular`, that is existing and unmodified.

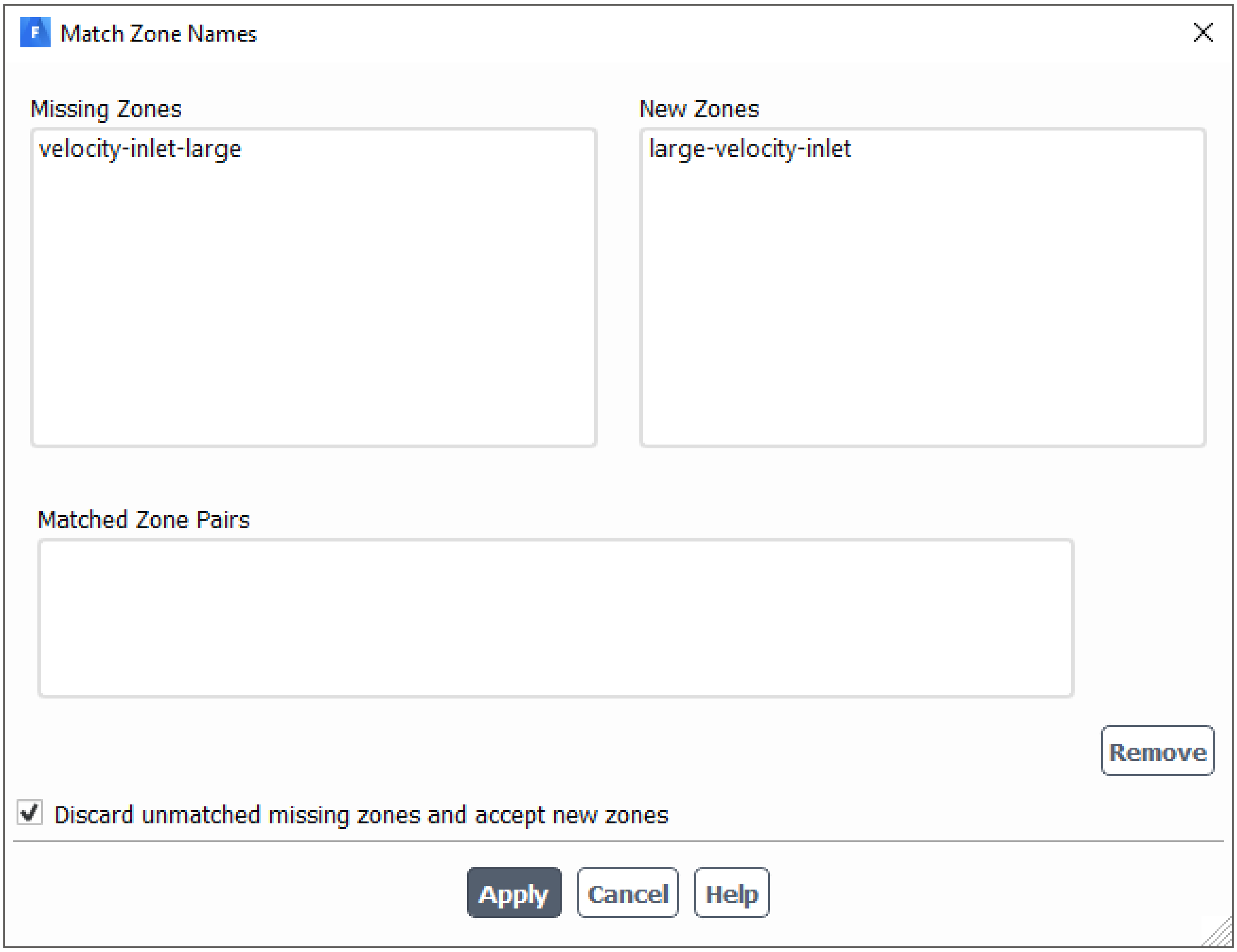
To manually manage upstream topological changes (such as zone name changes) or correct any issues with missing or new incoming zone names:

1. Click the **Match Zone Names...** button.

2. In the **Match Zone Names** dialog box, select a zone from the **Missing Zones** selection list and select a corresponding zone from the **New Zones** selection list.

The selected zones will be removed from their zone lists, and a matched zone pair will be added to the **Matched Zone Pairs** list.

Figure 2.12: The Match Zone Names Dialog Box



If you want to break the association between missing and new zones that you have created, select the corresponding matched zone pair from the **Matched Zone Pairs** list and click **Remove**. The zone pair will be removed from the **Matched Zone Pairs** list, and the zone names will reappear in their original zone lists.

3. To remove any unmatched zones in the selection **Missing Zones** list, and/or keep any additional new zones in the **New Zones** selection list, select the **Discard unmatched missing zones and accept new zones** option.

4. When you have completed matching zone names, click **Apply**.

5. When prompted whether or not you want to reload the **Setup** cell, click **Yes**.

In addition, the following controls are available in the **Recorded Mesh Operations** dialog box (see Figure 2.10: The Recorded Mesh Operations and Incoming Zones Dialog Box (p. 78)):

- **Reset** — resets the stored mesh operations, clearing out all existing mesh operations, and accepting the current zones as expected incoming zones.

- **Delete** — deletes the selected stored mesh operation.
- **Reload** — saves only the current set of mesh operations and incoming zone information, and reloads the **Setup** cell.

The **Reset**, **Delete**, and the **Reload** buttons are disabled when there are no mesh operations listed in the **Recorded Mesh Operations** dialog box.

Important:

Using the **Reset**, **Delete**, or the **Reload** buttons is not recommended for cases with topological mesh transformations because of potential mismatches between recorded operations and the settings file.

2.12. Changing the Settings and Mesh in Fluent

In order to use the **Update** command (see Using the Update Command (p. 55)) when changes are made to your project, Fluent must know which settings changes should be stored as part of the **Setup** cell's data (and therefore used during an update) and which settings changes should only be reflected in the results data that is associated with the **Solution** cell.

Important:

Note that there are certain changes to the mesh that you can perform within Fluent that are able to be recorded and reviewed and are treated as settings. In addition, there are other changes to the mesh within Fluent that are not recorded, and, as such, are not saved as settings, and cannot be automatically re-applied when the **Update** command is used. For more information about recording and reviewing mesh manipulation operations in Fluent in Workbench, see Recording Mesh Manipulation Operations and Resolving Mesh Incompatibility in Fluent (p. 76).

In order to address these issues, if you make certain changes to the mesh and/or settings in Fluent, you may be prompted when you attempt to calculate, close Fluent, or save the project from Fluent. The dialog boxes that appear (described in more detail below) allow you to select an action or to cancel the operation.

Important:

If you save the project from an application other than Fluent or from the **Project Schematic**, you will not be prompted; Fluent will automatically perform the default action for each dialog box described below.

For more information, see the following sections:

2.12.1. Changing Case and Mesh Settings Before Beginning a Calculation

---

Important:

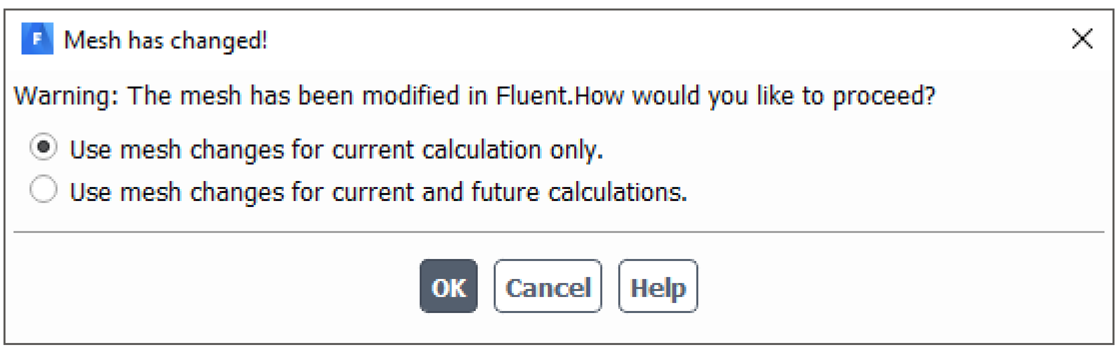
Note that in the descriptions below, mesh changes performed within Fluent only include those mesh modifications that are *not* recorded in Fluent in Workbench. In addition, changes to the settings can also include any mesh modifications that *can* be recorded in Fluent in Workbench. For more information about recording and reviewing mesh manipulation operations in Fluent in Workbench, see Recording Mesh Manipulation Operations and Resolving Mesh Incompatibility in Fluent (p. 76).

If the mesh has not been changed using tools in Fluent, any changes to the settings are automatically saved to the settings (`.set`) file.

If the mesh has been changed using tools in Fluent and the mesh was imported into the Fluent application or the system's **Setup** cell, any changes to the settings are automatically saved to the settings (`.set`) file. In addition, a case file is saved and registered to the **Setup** cell to represent the modified mesh.

If the mesh has been changed using tools in Fluent and the mesh was provided to the system's **Setup** cell by an upstream cell, the saved settings may not be compatible with the original mesh available to the **Setup** cell, and you are prompted with the following dialog box:

Figure 2.13: The Mesh has changed! Dialog Box



Since the mesh was provided to the system's **Setup** cell by an upstream cell, the original mesh cannot be replaced by the modified mesh without also making changes to the **Project Schematic**. Since there are several ways in which the schematic can be modified, Fluent does not provide a way to do this automatically.

You can choose any of the following actions:

- Select **OK** in the **Mesh has changed!** dialog box. The action you requested when you were prompted will proceed. Any changes to the settings will be saved to the settings file. If you had selected to calculate, the iterations (or time-steps) will be performed on the modified mesh. If you had selected to close Fluent or save the project, the modified mesh will be stored in the case file that is written as a result of either of those actions. The modified mesh will not be saved if Fluent is closed without initializing or generating data.

Important:

- If you open Fluent from the **Setup** cell, or update or refresh the **Setup** or **Solution** cell from the **Project Schematic**, the modified mesh will be replaced with the mesh provided by the upstream cell. To calculate using the modified mesh, either open Fluent from the **Solution** cell or select **Continue Calculation** from the **Solution** cell's context menu.
- If you specified settings before you changed the mesh, you must also verify that those settings are consistent with the modified mesh.
- If you want to use the modified mesh as the starting point for another analysis, simply create a new Fluent-based system and import the new case file that contains the modified mesh into its **Setup** cell.

- Select **Cancel** in the **Mesh has changed!** dialog box and create a case modification strategy that automatically performs the desired mesh modification steps (as well as any settings changes that are dependent on the modified mesh) before calculating (see Case Modification Strategies with Fluent and Workbench (p. 86)). This will allow you to automatically repeat the desired mesh modifications every time you perform an **Update** or restart the calculation from the **Setup** cell.

Important:

If you plan to make modifications to your mesh in Fluent (such as scaling, rotating, or converting to polyhedra) before performing any calculations, and you do not require any change made upstream of the Fluent **Setup** cell to be propagated to the mesh in the future, you should import the mesh you plan to modify directly into the **Setup** cell of a Fluent-based analysis or component system.

2.12.2. Changing Case and Mesh Settings After a Calculation Has Started

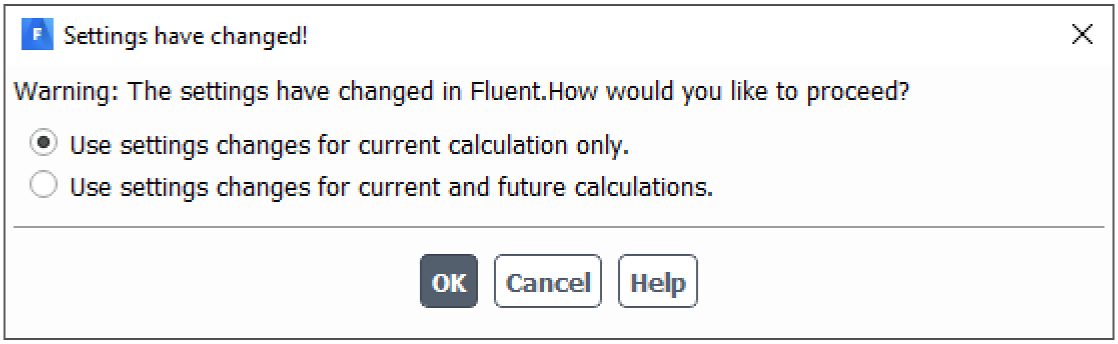
---

Important:

Note that in the descriptions below, mesh changes performed within Fluent only include those mesh modifications that are *not* recorded in Fluent in Workbench. In addition, changes to the settings can also include any mesh modifications that *can* be recorded in Fluent in Workbench. For more information about recording and reviewing mesh manipulation operations in Fluent in Workbench, see Recording Mesh Manipulation Operations and Resolving Mesh Incompatibility in Fluent (p. 76).

If changes to the settings have been made and the mesh has not been changed using tools in Fluent, you are prompted with the following dialog box:

Figure 2.14: The Settings have changed! Dialog Box



Since changes were made to the settings after you began the calculation, you have to specify whether or not the settings changes should be saved to the settings file.

You can choose any of the following actions:

- Select the **Use settings changes for current calculations only** option in the warning dialog box. The action you requested when you were prompted will proceed. The modified settings will not be saved to the settings file. If you had selected to calculate, the iterations (or time-steps) will start using the new settings. If you had selected to close Fluent, or save the project, the new settings will be stored in the case file that is written as a result of either of those actions.

  Important:

  – If you open Fluent from the **Setup** cell, or update or refresh the **Setup** or **Solution** cell from the **Project Schematic**, the settings will be replaced with the settings in the settings file that is associated with the **Setup** cell. To calculate using the new settings either open Fluent from the **Solution** cell or select **Continue Calculation** from the **Solution** cell's context menu.

  – If you want to use the new settings as the starting point for another analysis, simply create a new Fluent-based system and import the new case file that contains the new settings into its **Setup** cell.

- Select the **Use settings changes for current and future calculations** option in the warning dialog box. The action you requested when you were prompted will proceed. The modified settings will be saved to the settings file. If you had selected to calculate, the iterations (or time-steps) will start using the new settings. If you had selected to close Fluent, or save the project, the new settings will also be stored in the case file that is written as a result of either of those actions.

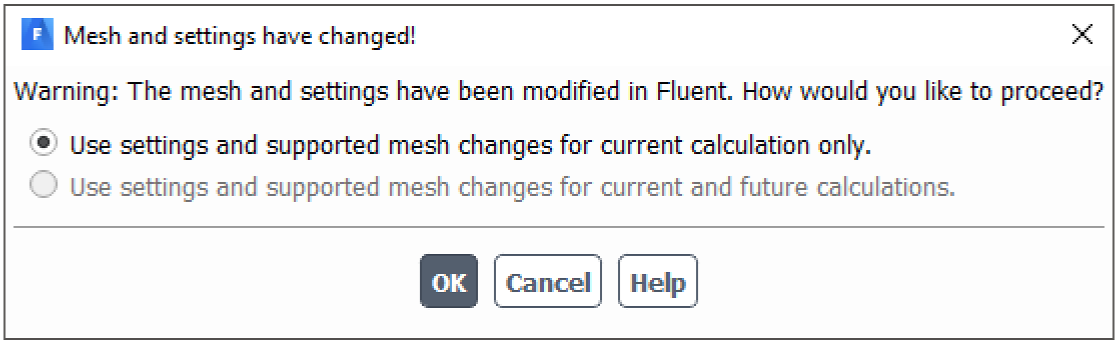
- Select **Cancel** in the warning dialog box. Create a new duplicate system (see Duplicating Fluent-Based Systems (p. 75)), modify the settings, and connect the **Solution** cells for the two systems so that the calculations will be performed in sequence.

- Select **Cancel** in the warning dialog box and create a case modification strategy that automatically performs the desired settings changes after the appropriate number of iterations (or time-steps); see Case Modification Strategies with Fluent and Workbench (p. 86) for more details.

The last two approaches allow you to automatically repeat setting changes after a specified number of iterations (or time-steps) every time you perform an **Update** or restart the calculation from the **Setup** cell.

If the mesh has been changed using tools in Fluent, and the mesh includes changes made using dynamic or sliding mesh, you are prompted with the following dialog box:

Figure 2.15: The Mesh and settings have changed! Dialog Box

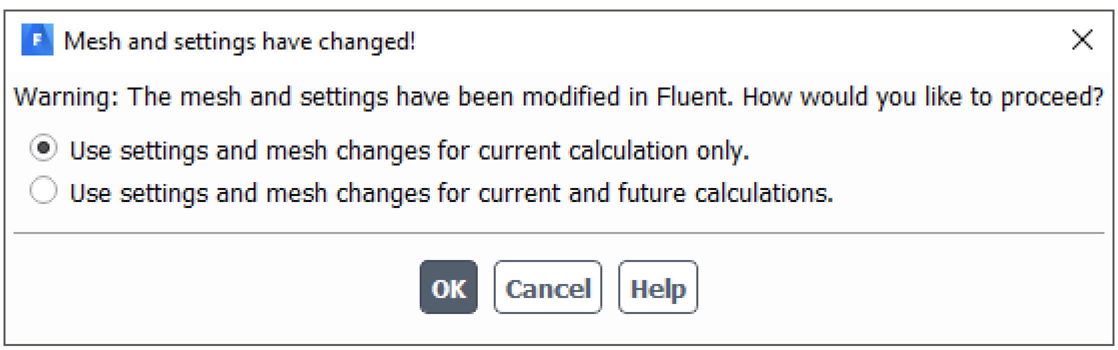


Important:

Note that when Fluent modifies the mesh, Fluent does not allow you to save the modified settings and/or mesh operations once the calculation has started.

Otherwise, if the mesh has been changed using tools in Fluent, you are prompted with the following dialog box:

Figure 2.16: The Mesh and settings have changed! Dialog Box



You need to specify whether or not any changes you made to the settings after you began the calculation should be saved to the settings file. Note that in cases where unrecorded mesh operations are performed in Fluent, those mesh operations will be ignored in future solution runs.

Important:

- Whenever you modify the mesh in Fluent, you also make changes to some settings in Fluent. Therefore, mesh modifications always result in setting changes.
- The original mesh cannot be replaced by the modified mesh after the calculation has begun.

You can choose any of the following actions:

- Select the **Use settings and mesh changes for current calculations only** option in the warning dialog box. The action you requested when you were prompted will proceed. The modified settings will not be saved to the settings file. If you had selected to calculate, the iterations (or time-steps) will start using the new settings and the modified mesh. If you had selected to close Fluent, or save the project, the new settings and the modified mesh will be stored in the case file that is written as a result of either of those actions.

  Important:

  – If you open Fluent from the **Setup** cell, or update or refresh the **Setup** or **Solution** cell from the **Project Schematic**, the settings and the modified mesh will be replaced with the mesh and settings in the files that are associated with the **Setup** cell. To calculate using the new settings and the modified mesh, either open Fluent from the **Solution** cell or select **Continue Calculation** from the **Solution** cell's context menu.

  – If you want to use the new settings and the modified mesh as the starting point for another analysis, simply create a new Fluent-based system and import the new case file that contains the new settings and the modified mesh into its **Setup** cell.

- Select the **Use settings and mesh changes for current and future calculations** option in the warning dialog box. The action you requested when you were prompted will proceed. The modified settings will be saved to the settings file. If you had selected to calculate, the iterations (or time-steps) will start using the new settings and the modified mesh. If you had selected to close Fluent, or save the project, the new settings and the modified mesh will also be stored in the case file that is written as a result of either of those actions.

  Important:

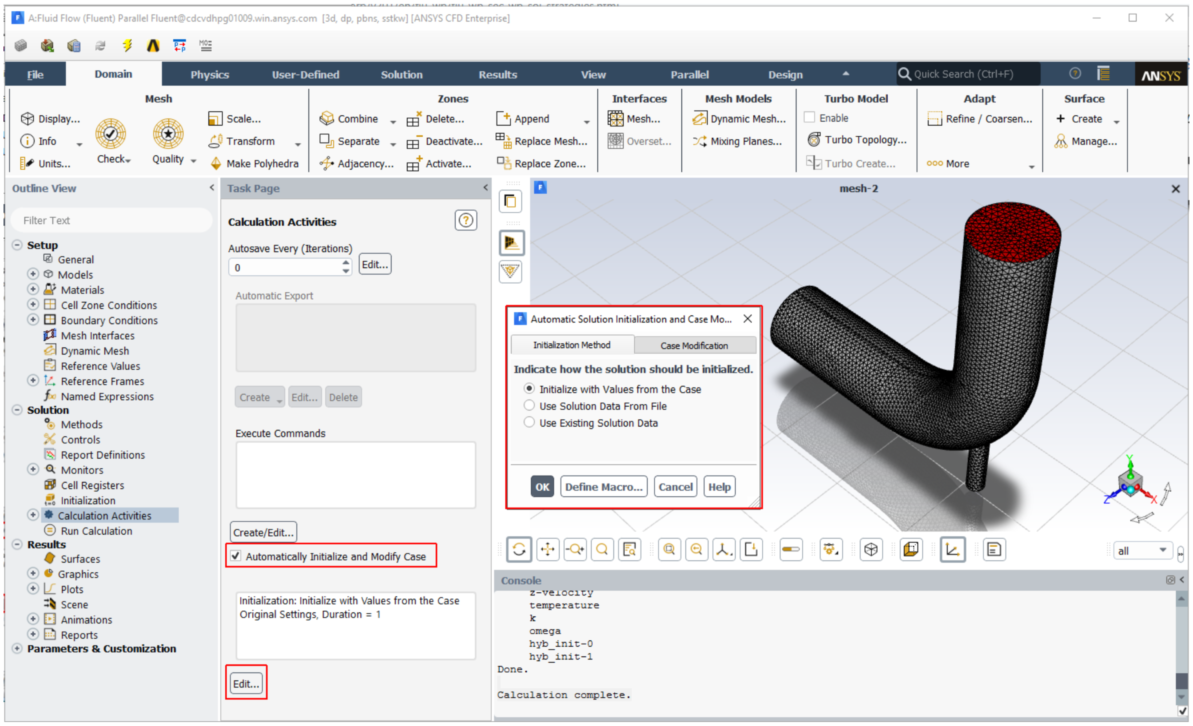
  If you open Fluent from the **Setup** cell, or update or refresh the **Setup** or **Solution** cell from the **Project Schematic**, the new settings will be used in conjunction with the mesh in the file that is associated with the **Setup** cell. Since those settings may have been specified after the mesh was modified, you must verify that the new settings are consistent with the original mesh.

- Select **Cancel** in the warning dialog box and create a case modification strategy that automatically performs the desired settings changes and mesh modification steps after the appropriate number of iterations (or time-steps); see Case Modification Strategies with Fluent and Workbench (p. 86) for more details. This allows you to automatically repeat the desired setting changes and mesh modification steps after a specified number of iterations (or time-steps) every time you perform an **Update** or restart the calculation from the **Setup** cell.

2.13. Case Modification Strategies with Fluent and Workbench

Fluent allows you to specify your own initialization and start-up routines using automatically executed strategies that you can define using the **Automatic Solution Initialization and Case Modification** option on the **Calculation Activities** task page within Fluent. Selecting the option and clicking the **Edit...** button opens the **Automatic Solution Initialization and Case Modification** dialog box.

Figure 2.17: Accessing Solution Strategies in Fluent



Using the **Automatic Solution Initialization and Case Modification** dialog box, you are able to specify your initialization method on the **Initialization Method** tab and use any text user interface (TUI) command to modify the case after a specified number of time steps or iterations in the **Case Modification** tab. This option replaces the need for some simple journal files, especially for cases where a prescribed start-up and solution routine is used (start with 1st order, switch to 2nd order, turn on reactions, and so on)

When a case modification strategy is defined within Fluent, it will be used when a system is updated from the **Project Schematic** in Workbench.

As mentioned in Changing the Settings and Mesh in Fluent (p. 81), mesh manipulations steps executed from within Fluent are not repeated when an update is performed from the **Project Schematic** or you start Fluent from the **Setup** cell. However, for mesh manipulation, you can specify steps that will be executed at the beginning of your calculation if you incorporate them into a case modification strategy.

Solution strategies can also be useful when you need to perform mesh manipulation steps after a specified number of iterations during a calculation.

Important:

Whenever you run a case modification strategy interactively, make sure to reload the original mesh and revert to the original settings before re-running the case modification strategy (see the separate Fluent User's Guide for information on how the original settings can be reset automatically as part of a case modification strategy).

Important:

When running Fluent in Workbench, the behavior when continuing the calculation from the **Solution** cell, when a solution strategy is defined, is different than when additional iterations or time steps are specified in the **Run Calculations** task page. When a solution strategy is defined, Fluent will complete the iterations or time steps specified by the solution strategy. Once the solution is **Up-to-Date**, continuing the calculation has no effect. On the other hand, when additional iterations or time steps are specified in the **Run Calculations** task page, every time you choose to continue the calculation, Fluent will try to perform the specified number of iterations or time-steps.

For more information about using these features in Fluent, see the Fluent User's Guide.

2.14. Working With Input and Output Parameters in Workbench

Workbench uses parameters and design points to allow you to run optimization and what-if scenarios. You can define both input and output parameters in Fluent that can be used in your Workbench project. You can also define parameters in other applications including Ansys SpaceClaim Direct Modeler, Ansys DesignModeler and Ansys CFD-Post. Once you have defined parameters for your system, a **Parameters** cell is added to the system and the **Parameter Set** bus bar is added to your project. Arrows representing input and output parameters connect the bus bar to each system in which you have defined parameters.

Double-click the **Parameter Set** bus bar to open the **Parameters Set** tab. The **Parameters Set** tab includes the **Outline of All Parameters** table that lists all of the parameters in your project as well as the **Table of Design Points** table in which you can specify design points.

To create a new design point, enter the input parameter values that you want to use for that design point in the **Table of Design Points** in the row with an asterisk (*) in the first column. You can create several design points. Once you have finished specifying design points, you can right-click the row for one design point and select the **Update Selected Design Point** command from the context menu to compute the output parameters for that design point. Alternatively, you can select **Update All Design Points** from the Toolbar to update all of your design points in sequence.

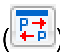
Important:

By default, Workbench only saves the calculated data for the design point in the row labeled **Current**. If you want to save the data for a design point other than **Current** within the project (so you can postprocess the results from a different design point in either Ansys Fluent or Ansys CFD-Post), enable the option in the **Retain** column for that design point before you update it. After the calculation is complete, you can then right-click the design point in the **Table of Design Points** and select **Set as Current** to access the data. Alternatively, you can export design points to separate projects. For details, see Retaining Design Point Data and Exporting Design Points to New Projects in the separate Workbench User's Guide.

Important:

Note that you cannot create, edit, delete, or rename parameters in Fluent if any iterations (or time-steps) have been performed. If you want to create, edit, delete, or rename parameters in Fluent for a case with an existing solution, you must first initialize the solution. Alternatively, you can disable the **Use as parameter** option in the **Expression** dialog box in Fluent to edit the expression, then re-enable the **Use as parameter** option.

Note:

In Fluent, you can directly access the **Parameters** dialog box using the **Parameters & Customization**/**Parameters** tree item in Fluent, or the corresponding **Parameter System** toolbar command ().

For more information about using input and output parameters in Fluent (along with using them in Scheme functions and UDFs), see Defining and Viewing Parameters and Creating Output Parameters in the Fluent User's Guide.

For more information about parameters, design points, what-if scenarios and optimization studies in Workbench, see the Workbench online documentation.

2.15. Reduced Order Model (ROM) Evaluation in Fluent

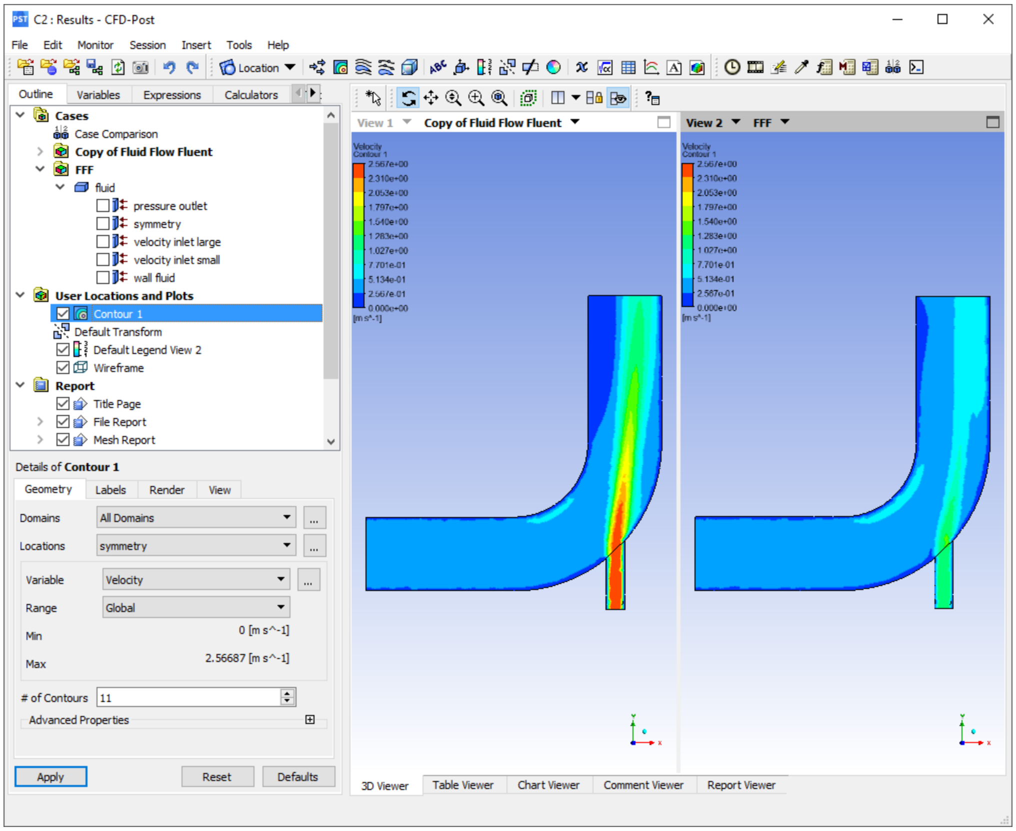
In addition to reviewing solved reduced order models (ROMs) in the ROM Viewer (Using ROMs in the *DesignXplorer User's Guide*), you can view and evaluate the results in Ansys Fluent. For additional information about this functionality, refer to Reduced Order Model (ROM) Evaluation in Fluent in the *Fluent User's Guide*.

2.16. Viewing Your Fluent Data Using Ansys CFD-Post

Ansys CFD-Post is a postprocessing application that helps you visualize the results of your Ansys Fluent CFD simulations. For Fluent-based analysis systems, the **Results** cell provides access to the Ansys CFD-Post application. In addition, the Toolbox contains a separate **Results** component system (that is, Ansys CFD-Post) that you can add to the **Project Schematic** and connect to Fluent-based systems.

When a **Results** cell is connected to a Fluent-based system's **Solution** cell and the **Results** cell's state is either **Refresh Required** or **Update Changes Pending**, you can view the results of your Fluent calculation in Ansys CFD-Post by double-clicking the **Results** cell. This will load the Fluent results into Ansys CFD-Post.

Figure 2.18: Multiple Fluent Results Loaded Into Ansys CFD-Post



Important:

In addition to analyzing your results in Ansys CFD-Post, you can also view the results of your simulation using the standard Fluent postprocessing tools. For more information, see the separate Fluent User's Guide.

Important:

There are two options for exporting Fluent files for Ansys CFD-Post:

1. Fluent data files in the legacy format (`.dat` files) do not contain all variables; however, you can add additional quantities using the **Data File Quantities** dialog box in Fluent.

2. Lightweight data files are created by exporting Ansys CFD-Post compatible files. These files can be used to save just the variables of interest.

When you edit a **Results** cell that is connected to the **Solution** cell in a Fluent-based system, Ansys CFD-Post always loads the standard Fluent case and data files. For more information on exporting Fluent files for Ansys CFD-Post, see the separate Fluent User's Guide.

2.17. Understanding the File Structure for Fluent in Workbench

When you save a Workbench project (for example, `my-project`), the project is saved with a `.wbpj` extension (for example, `my-project.wbpj`). Other files associated with the project (through other Workbench applications such as Ansys Fluent or CFX) are located in the `dp0` folder within a `_files` folder (for example, `my-project_files`).

Each system in the **Project Schematic** has its own directory under the `dp0` directory. The directory is named using the corresponding system identifier (for example, `FFF` represents a Fluent-based analysis system; `FLU` represents a Fluent-based component system; `Post` represents a **Results** component system, and so on). The directory name is appended with a number to distinguish it from the directories for other systems of the same type (with the exception of the directory name for the first system of a specific type which has no number appended to it).

Within each system directory is a folder for each application that is part of the system. This folder is used to store the files generated and used by the application.

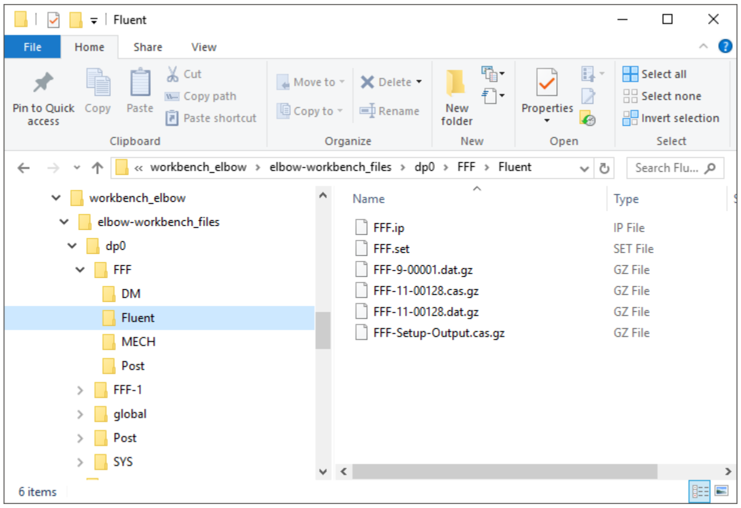
In addition to the settings, case and data files, the following files are managed by Fluent in Workbench:

- monitor files
- flamelet files
- view factor files
- non-premixed files (`.pdf`)
- interpreted user-defined (UDF) files
- compiled UDF libraries
- Discrete Transfer Radiation (DTRM) `.ray` files
- mechanism files (`.che` or `.inp`)
- property database files (`thermo.db` and `transport.db`)

You may use other types of files with Fluent in Workbench, however, you are responsible for making sure that they are located in the appropriate folder within the project file structure.

The following figure represents an example of the directory structure for Workbench project with two **Fluid Flow (Fluent)** systems and one **Results** system:

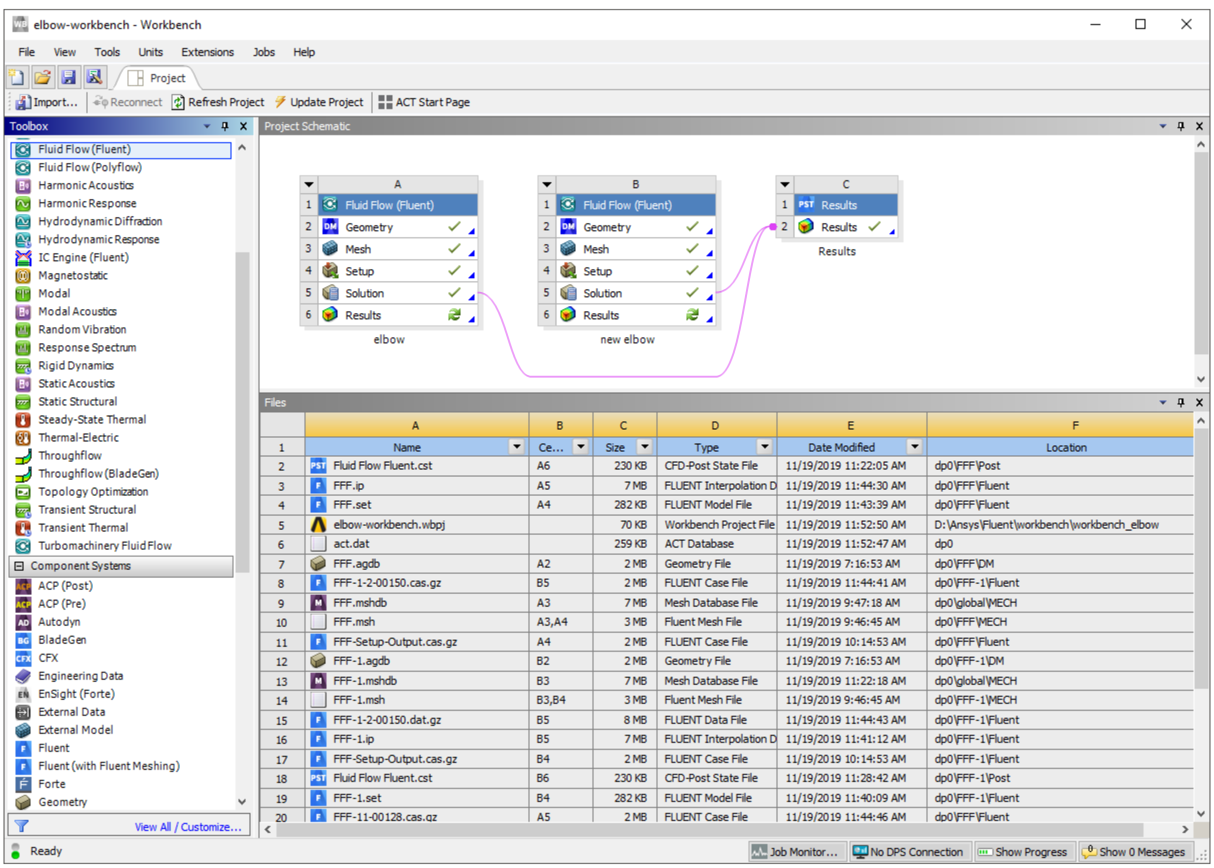
Figure 2.19: Example of the Directory Structure for a Fluent-Based Project in Workbench



You can view the files associated with your Workbench project by selecting the **Files** option under the **View** menu.

View → Files

Figure 2.20: The Files Pane for a Project in Workbench



If data is shared between two systems, then files are also shared between the two systems. The shared file will exist in either the directory of the first system that used it, or in a global directory in the design point directory (depending on the type of system that generated the file).

The `_files` folder also contains a `user_files` directory. This directory should be used for any files you create or reference that you would like to store with the project.

Important:

In general, you should not modify the structure of the project directories or delete or modify any of the files that Workbench applications have stored in the project directories. However, you may delete Fluent case and data files that are stored with the project but are no longer needed. You should close Fluent before deleting Fluent case and data files from the project directories.

Important:

Monitor files are automatically written to the appropriate working directory within the Workbench project files directory. You will not be able to specify a different path for monitor files from within Fluent. When reading older case files in which a different location is specified for monitor files, a warning message will inform you that the path will be modified.

Important:

If you intend to use a Fluent journal file that reads or writes files while running under Workbench, the journal file and the files it references should be moved to the appropriate Fluent folder in the appropriate system folder in the Workbench project working directory. File paths in the journal files should use relative paths to point to the new locations for the files.

Important:

If you use models that generate mesh-dependent information the first time they are set up (for example, DTRM, S2S) and then you change the mesh used in your system, the mesh dependent information (for example, `.dtrm` file, `.s2s` file) may be incompatible with the new mesh. To resolve this issue, you can:

- Open Fluent from your system, compute/write the `.dtrm` or `.s2s` output files, and then update the system.
- Use a solution strategy and write the `.s2s` file as a pre-initialization method using the appropriate command (for example, `(write-sglobs "2d.s2s.gz")`).

Important:

If your Fluent setup involves compiling and loading a user-defined function (UDF), it is recommended that you copy the UDF source code files to the same location as your mesh and settings files before compiling the UDF libraries. This location is available in the **Files** pane in Workbench. You may need to save the project first to create the appropriate files folder within the Workbench project (for example `project name\dp0\FFF\Fluent\`). If you need to unload a UDF library for any reason during your simulation, it is recommended that you save the project soon after unloading the library. If you archive a project that includes compiled user defined functions, you will need to recompile the libraries after opening the archived project. To do so, you will need to open Fluent from the **Setup** cell, go to the **UDF Library Manager** dialog box (**User Defined/User Defined/Functions/Manage...**) and unload the existing UDF library. Next, rebuild the UDF library locally by going to the **Compiled UDFs** dialog box (**User Defined/User Defined/Functions/Compiled...**), selecting the archived `.c` and `.h` files, compiling, and loading the new UDF library. Finally, save the project.

For more information, see the following section:

2.17.1. Fluent File Naming in Workbench

When running under Workbench, Fluent automatically saves the settings, case, and data files for your system as needed.

Fluent names these files using the base name of the mesh file. If a mesh (or case) file is imported, the base name of the imported file is used as the base name for the settings, case, and data files. If the mesh is created in Workbench, the mesh file uses the system's directory name as its base name.

When Fluent saves a case file, it appends the base name with the run number. When Fluent next writes a case file, the run number is incremented in order to avoid overwriting the previous case file. When Fluent saves a data file, it appends the base name with the same run number as the associated case file and with the iteration (or time-step) number at which it was saved. A new case file is not written every time that a data file is written. The case file is only written when the settings and/or mesh is changed.

The following example shows how file naming works in Fluent under Workbench. In this example, you have a single **Fluid Flow (Fluent)** analysis system and you have created the mesh in Workbench.

1. Double-click **Setup** cell. Fluent launches and loads `FFF.msh.h5`.
2. In Ansys Fluent, specify boundary conditions, initialize the solution and enter 10 iterations on the **Run Calculation** task page.
3. In the Fluent application, select the **Calculate** button.

   `FFF.set` file is written and 10 iterations are completed.
4. Save the project.

   `FFF-1.cas.h5` and `FFF-1-00010.dat.h5` are written.
5. In the Fluent application, select the **Calculate** button.

   10 additional iterations are completed.
6. Save the project.

   `FFF-1-00020.dat.h5` is written.

7. In the Fluent application, change the value of the inlet velocity, select the **Calculate** button. When prompted, select the **Store the modified settings for a future calculation and use them this time** option.

   `FFF.set` is written (overwriting the existing file) and 10 additional iterations are completed.

8. Save the project.

   `FFF-2.cas.h5` and `FFF-2-00030.dat.h5` are written.

> **Important:**
>
> The same naming convention is used when autosaving with Fluent under Workbench.

2.18. Working with Ansys Licensing

When working with Workbench, you have the option to share a single license between applications that use the same license or the option for each application to check out its own license.

For more information, see the following section:

2.18.1. Shared Licensing Mode

When using shared licensing, although a single license is shared between multiple applications, only one application can actively use the license at a time. Therefore, with just a single license, you can have both Ansys Fluent and Ansys CFD-Post open at the same time. If iterations are being performed in the Fluent session, you cannot do anything in the Ansys CFD-Post session. However, if the Fluent session is open and idle, you can work in Ansys CFD-Post without closing Fluent. Similarly, you can have multiple sessions of Fluent open at the same time using just a single license.

If you open an application, it will first check to see if it can use a license that is already checked out. If it can, and that license is available, it will use that license. If the license is not available because it is being used by another application, you will be informed that the required license is not available. You will not be able to use the new application until that license becomes available. If there is not a license checked out that is compatible with the new application, the new application will check out an additional license.

In shared mode, you can have multiple licenses of each type of non-shareable licenses checked out at a time. For example, you can have 1 `acfd` license and 1 `acfd prepost` license checked out at the same time but you cannot have 2 `acfd` licenses checked out at the same time.

In shared mode, you can have multiple licenses of each type of non-shareable licenses checked out at a time. Non-shareable licenses include solver-only licenses and parallel licenses.

For more information about licensing and shared license mode, see the Workbench online documentation.

2.19. Using Fluent With the Remote Solve Manager (RSM)

The Remote Solve Manager (RSM) is a job queuing system used to distribute tasks that require computing resources. RSM schedules and manages tasks within Workbench and allows tasks to be sent *only* to the local machine to run in background mode, or they can be broken into a series of jobs for parallel processing.

For more information about using Fluent in Workbench and RSM (with associated limitations), see Submitting Solutions to Remote Solve Manager and Submitting Fluent Jobs to Remote Solve Manager.

2.20. Using Custom Systems

Workbench allows you to add custom templates and includes some pre-defined custom templates. To use a custom template, double-click the template to add it to the **Project Schematic**.

Under **Custom Systems** in the Toolbox, there is a predefined Fluent-based custom system (**FSI: Fluid Flow (Fluent)** −> **Static Structural** – the single arrow is meant to convey that the interaction is one-way). When you double-click the template to add it to the **Project Schematic**, Workbench automatically creates a link between the **Geometry** cell in the Fluent system and the **Geometry** cell in the **Structural** system and between the **Solution** cell in the Fluent system and the **Setup** cell in the **Structural** system.

You can create your own system template and then save it as a **Custom System** by performing the following steps:

1. Manually create the desired system diagram in the **Project Schematic**.
2. Right-click the white space in the **Project Schematic** and select **Add to Custom** from the context menu.

Important:

When you use Fluent to perform a one-way fluid-structure interaction (FSI) analysis using the approach demonstrated in the predefined Fluent-based custom system, you can only exchange surface data for force and thermal loads.

Important:

When you use Fluent to perform a one-way fluid-structure interaction (FSI) analysis using the approach demonstrated in the predefined Fluent-based custom system, and you are performing a multiphase simulation in Fluent, you cannot exchange thermal load data.

2.21. Using Journaling and Scripting with Fluent in Workbench

You can keep a history of your interactions within Workbench, that can also include your interactions within Fluent, by recording your interactions with the program(s) in session journals (also referred to as *journaling*). This is done using the **Scripting** submenu in the Workbench **File** menu:

File → **Scripting** → **Record Journal...**

In addition, since the journal files are Python-based scripts, you can edit and/or play back previously recorded journal files, or create new journals manually (also known as *scripting*), that include your interactions within Workbench and, if applicable, any interactions within Fluent:

File → Scripting → Run Script File...

For more information about recording and using session journals in Workbench, as well as reference documentation containing available commands and properties, see the separate Ansys Workbench Scripting Guide.

Important:

- When using the **Send Command** method to directly call a Fluent text user interface (TUI) command, the TUI command will not be recognized unless you use double quotes around it (for example, `setup.SendCommand("define model energy no")`). If a string is included in the TUI command, then a backslash is required before the quotes around the string. For example:

```
setup1.SendCommand(Command="(cx-gui-do cx-activate-item \"MenuBar*FileMenu*Close Fluent\") ")
```

- Fluent internal Scheme commands may not work properly when called directly using the **Send Command** method. Therefore, you should use Fluent text user interface (TUI) commands instead.

2.22. Performing Fluent and Maxwell Coupling in Workbench

Fluent can be coupled with Ansys Maxwell within Workbench in order to perform either a one-way or a two-way electromagnetic-thermal interaction problem. Coupling between Maxwell and Fluent applications within Workbench can be used for applications such as simulating fluid flow around or inside electromechanical (EM) devices, when the temperature of the device is influenced by electromagnetic or electrical losses.

Maxwell is supported for one–way and two-way coupling between Maxwell and Fluent in Workbench.

Additional information can be found in the following sections:

2.22.1. One–Way Coupling Between Maxwell and Fluent Within Workbench

The coupling involves solving an electromagnetic problem in Maxwell, and mapping the resulting volumetric and/or surface loss information into Fluent. Volumetric loss is mapped onto the cell zones as a heat source (load) at the cell centroids that is then added to the energy equation. Surface loss is applied to the adjacent cells of the solid zones that contribute to the source terms of these cells.

Note:

Surface loss is highly concentrated near the surface of the solid zone, so it is recommended that you have a layer of good quality hexahedral or prism mesh elements located where surface loss occurs.

Note:

You can analyze the results of volumetric/surface losses using the following postprocessing variables:

- Volumetric loss: **Temperature EM Source** (W/m^3)
- Surface loss: **Temperature EM Surface Source** (W/m^2)

The overall workflow for an Maxwell-Fluent analysis is as follows:

1. Create and solve the electromagnetic application using Maxwell.
2. Drag and drop a Fluent–based system and import a case or mesh file into Fluent, and then double-click the **Setup** cell to start Fluent.

   Note:

   If you import a mesh into Fluent that was generated using non-SI units, then you should scale and/or translate the mesh before you perform any coupling with Maxwell.

3. Drag and drop the **Solution** cell of the Maxwell system onto the **Setup** cell of Fluent system to enable the data transfer.
4. Update the **Solution** cell of the Maxwell system.
5. Refresh the **Setup** of Fluent system.
6. Specify the EM settings in Fluent (**File/EM Mapping**).
7. Set up the Fluent analysis as you normally would (for example, specify boundary conditions, solution settings, and so on).
8. Solve the Fluent analysis.

During EM mapping, Fluent automatically turns on the energy model, if you have not already done so.

Note:

In 2D axisymmetrical cases only, Maxwell uses the RZ (or XZ) plane while Fluent uses the XY plane by default. Therefore, when you export a geometry from Maxwell, you need to rotate the geometry in order for Fluent to mesh the geometry before EM mapping can take place properly. This is not a concern for regular 2D or 3D cases. If you use Ansys DesignModeler, rotate the geometry exported from Maxwell by performing the following steps:

1. Define a new plane based on the XY plane, applying a 90 degree rotation along the X axis, and a 90 degree rotation along the Y axis.
2. Move the geometry from the XY plane to the new plane.

If you use SpaceClaim (Windows only), follow the steps below:

1. Make sure that the world origin is displayed (**Display** → **Display** → **Show** → **World Origin**).
2. In the **Design** ribbon tab, select the **Orient** tool (**Assembly** group).
3. Select an edge of the geometry you want to rotate and then select a global axis.

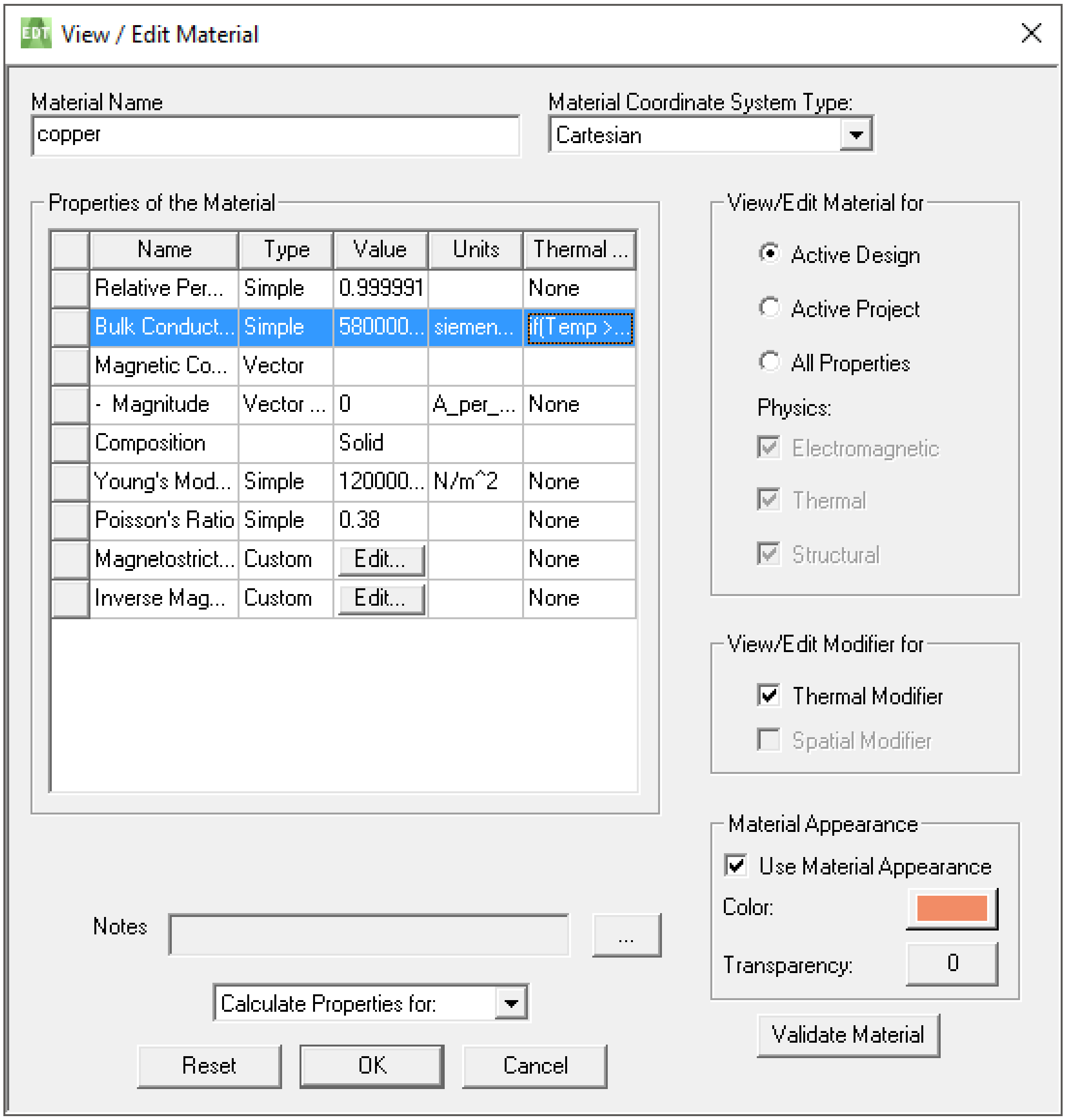
   The geometry will be rotated so that the selected edge is aligned along the selected global axis.

2.22.2. Two–Way Coupling Between Maxwell and Fluent Within Workbench

Two-way coupling enables thermal feedback to be provided between the systems so that you can exchange temperature data from Fluent to Maxwell within Workbench.

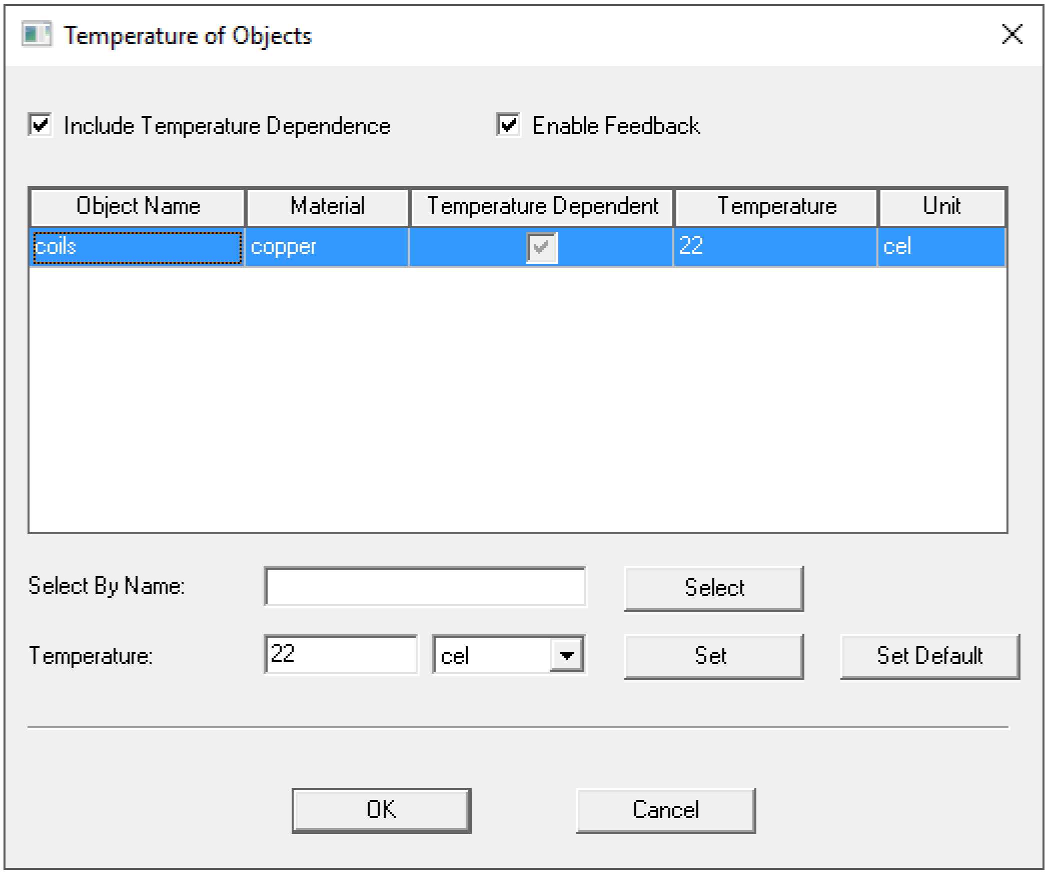
Aside from the basic workflow described in Performing Fluent and Maxwell Coupling in Workbench (p. 97), to enable two–way coupling between Fluent and Maxwell, you should perform the following steps:

1. Open the Maxwell project and define temperature–dependent material properties (enabling the **Thermal Modifier** field in the **View/Edit Material** dialog box and editing the material's thermal property definition). This makes sure that the Maxwell and Fluent applications generate the temperature–dependent data.



2. Select the **Enable Feedback** option in the **Temperature of Objects** dialog box.

Maxwell 3D → Set Object Temperature...



Once Maxwell generates the centroid file, Fluent detects the centroid file, and generates the temperature file as feedback data and places the file in the appropriate location. An example of what Fluent displays in the console window is presented below:

```
Starting Thermal feedback for EM ...

Reading centroid file... Number of Nodes = 4031
Writing mechanical.ths...
```

Note:

In Maxwell, you can have different losses (Ohmic loss, Core loss, Hysteresis loss) depending on the type of simulation. Maxwell can also compute the "total loss" which is the sum of all the losses when appropriate. All the different losses mentioned above including the "total loss" are a function of space and they are also a function of time in the Maxwell transient solver. When doing the coupling simulation, the "total loss" is mapped to Fluent for the temperature calculation. For transient simulations, the total loss is time averaged between two times that you specify before it is mapped to Fluent.

2.22.3. Handling Coupling Iterations Between Maxwell and Fluent

You can perform automatic system updates (coupling iterations) using the Maxwell Feedback Iterator. Refer to the Maxwell help documentation for more information on Feedback Iterator.

You can also perform manual cyclic updates of individual system components until the solution stops changing within a desired level of tolerance. For example:

- Coupling Iteration 1
  - Update Maxwell **Solution** cell
  - Perform EM Mapping on solid zones in Fluent
  - Update Fluent **Solution** cell
  - Perform **Enable Update** on Maxwell **Solution** cell
  - Update Maxwell **Solution** cell
- Coupling Iteration 2
  - Update Fluent **Setup** cell
  - Update Fluent **Solution** cell
  - Perform **Enable Update** on Maxwell **Solution** cell
  - Update Maxwell **Solution** cell
- Coupling Iteration 3
  - Update Fluent **Setup** cell
  - Update Fluent **Solution** cell
  - Perform **Enable Update** on Maxwell **Solution** cell
  - Update Maxwell **Solution** cell
- Coupling Iteration 4
  - Update Fluent **Setup** cell
  - Update Fluent **Solution** cell
  - Perform **Enable Update** on Maxwell **Solution** cell
  - Update Maxwell **Solution** cell
- and so on

Within the Workbench environment, you can create a script to accomplish these tasks. Workbench scripts are written in the Python programming language and can be created, modified and executed with tools available in Workbench. For more information on writing scripts in Workbench, see Workbench Scripting Guide in the *Workbench Scripting Guide*. You can contact your support representative to obtain a sample script.

Chapter 3: Getting Started with Fluent Meshing in Workbench

This chapter provides some basic instructions for getting started with using Fluent Meshing in Workbench. It describes how to use **Fluent (with Fluent Meshing)** component systems within Ansys Workbench. Since working with a **Fluent (with Fluent Meshing)** component system is similar to working with a **Fluent** component system, only differences in these two systems are covered in this chapter. For general information about using Fluent in Ansys Workbench, see Getting Started With Fluent in Workbench (p. 21). For details about using component systems in Workbench, refer to Working With Fluent in Workbench (p. 49).

The chapter discusses:

See the Fluent User's Guide for more details about Fluent Meshing.

3.1. Limitations

The following limitations exists when using the **Fluent (with Fluent Meshing)** component system:

- The **Fluent (with Fluent Meshing)** component system only supports mesh changes derived from using the Fluent Meshing guided workflows. Mesh changes outside of the guided workflows (such as from another connected Fluent component system, for example) are currently not supported. In such cases, for an otherwise up-to-date component system, the **Setup** and **Solution** cells remain as up-to-date even though the mesh has been changed.

- When using the **Watertight Geometry** workflow, all CAD bodies should have unique names.

- Saving a project and exiting Workbench (or creating a new project) while Fluent Meshing is still open may not register the generated mesh file with the **Mesh** cell of **Fluent (with Fluent Meshing)** component. To resolve this, launch Fluent Meshing by editing a **Mesh** cell, read the generated mesh file from the project directory, close the Fluent Meshing session, and save the project.
- You should open Fluent Meshing and confirm that all settings are correct.
- In the **Mesh** cell context menu, mesh import options are not available. (You can only import a geometry, mesh, or case file from within Fluent Meshing).
- Solver-to-mesh-solver workflows in Workbench (that is, returning to meshing mode after switching to solver mode) are currently not supported.
- 2D simulations are currently not supported.
- There are no limitations on the **Set Up** or **Solution** cells, including Fluent parameters.
- Fluent Meshing parameters are currently not supported, however, you can parameterize real arguments using the appropriate scheme commands in the registered update journal file (`*.jou`).

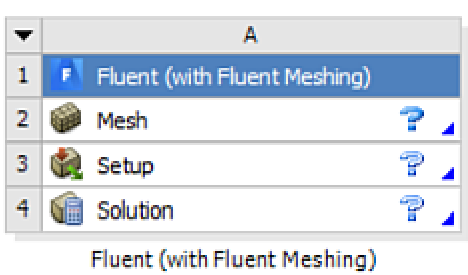
3.2. Starting Fluent (in Meshing Mode or Solution Mode) in Workbench

Once you create a **Fluent (with Fluent Meshing)** component system (as described in Fluent-Based Component Systems (p. 28)), you can start the Fluent application in

- meshing mode by double-clicking a **Mesh** cell
- setup mode by double-clicking a **Setup** cell
- solution mode by double-clicking a **Solution** cell

Note that you can also click **Edit...** in the corresponding cells context menu. When Fluent Launcher opens, click **OK** to start Fluent. You are able to start the Fluent application in either meshing mode or solution mode under certain conditions, depending on the state of the cells within the **Fluent (with Fluent Meshing)** component system.

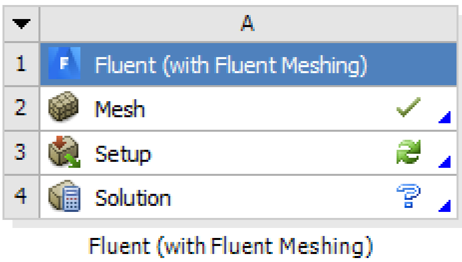
- If you are working with a new **Fluent (with Fluent Meshing)** component system (where all cells are not **Up-To-Date**):



  – Double-clicking the **Mesh** cell opens Fluent in meshing mode.

  – Double-clicking either the **Setup** cell or the **Solution** cell opens Fluent in solution setup mode where you are expected to import the mesh or case file, which deletes the **Mesh** cell from the **Fluent (with Fluent Meshing)** component system.

- If you are working with a preexisting **Fluent (with Fluent Meshing)** component system where the **Mesh** cell is **Up-To-Date**:

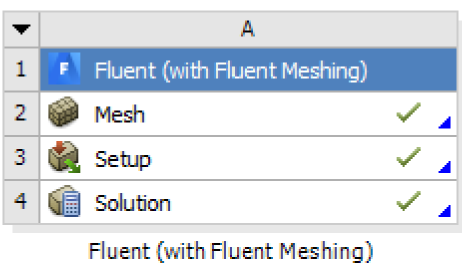


  – Double-clicking the **Mesh** cell loads available mesh in Fluent Meshing.

  – Double-clicking the **Setup** cell opens Fluent in setup mode and loads the mesh and settings files.

  – Double-clicking the **Solution** cell opens Fluent in solution mode and loads the case and data files.

- If you are working with a preexisting **Fluent (with Fluent Meshing)** component system where all cells are **Up-To-Date**:



  – Double-clicking the **Mesh** cell loads available mesh in Fluent Meshing.

  – Double-clicking the **Setup** cell opens Fluent in setup mode and loads the mesh and settings files.

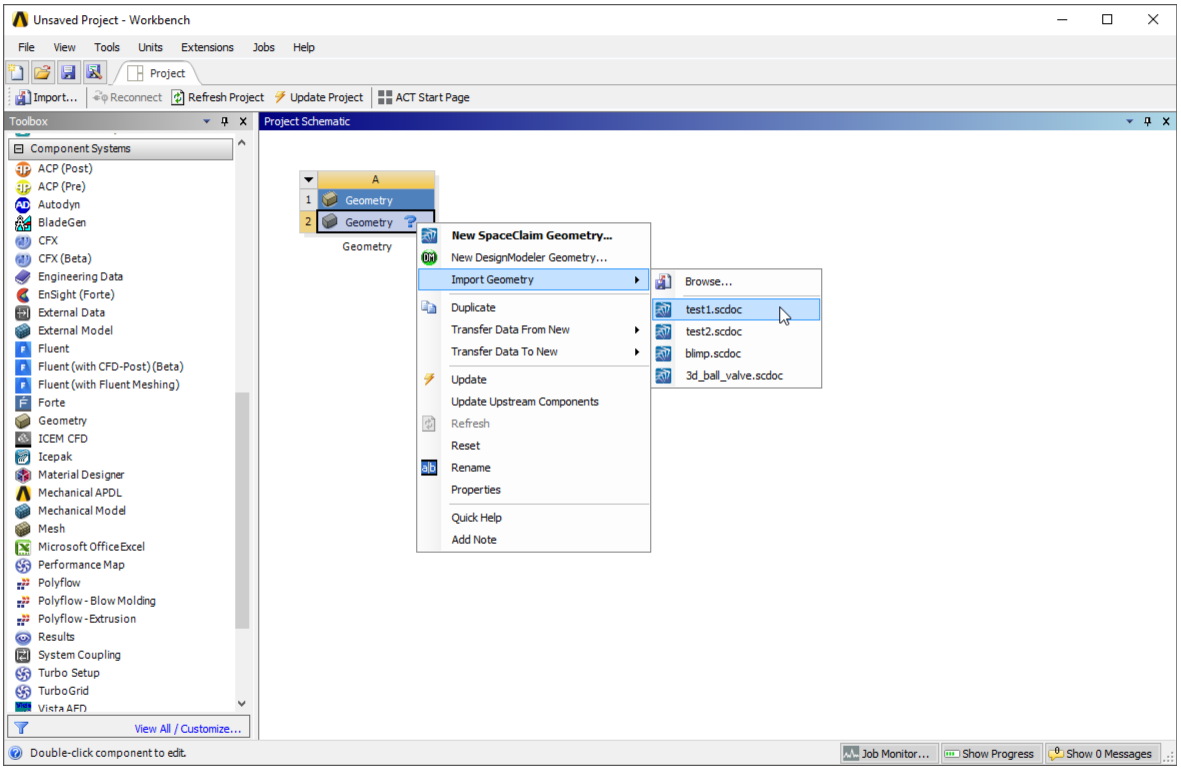
  – Double-clicking the **Solution** cell opens Fluent in solution mode and loads the case and data files.

3.3. Using the Fluent Guided Meshing Workflows

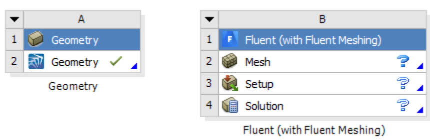
To access and use the **Fluent** meshing workflows in Workbench, perform the following:

1. Open the Workbench environment.

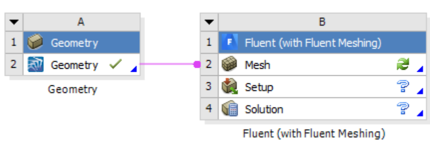
2. Add a **Geometry** component system to your Workbench project, and import a CAD geometry file into the **Geometry** cell.



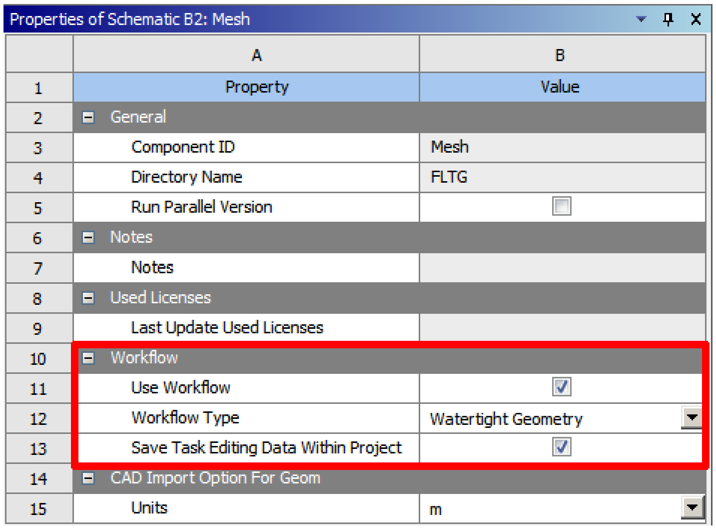
3. Add a **Fluent (with Fluent Meshing)** component system to your project.



4. Connect the **Geometry** cell of the **Geometry** component system to the **Mesh** cell of the **Fluent (with Fluent Meshing)** component system.



5. Select the **Mesh** cell and view it's properties.



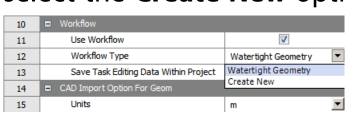
Properties of Schematic B2: Mesh

	A	B
1	Property	Value
2	General	
3	Component ID	Mesh
4	Directory Name	FLTG
5	Run Parallel Version	☐
6	Notes	
7	Notes	
8	Used Licenses	
9	Last Update Used Licenses	
10	Workflow	
11	Use Workflow	☑
12	Workflow Type	Watertight Geometry
13	Save Task Editing Data Within Project	☑
14	CAD Import Option For Geom	
15	Units	m

Note the **Use Workflow** check box in the properties of the **Mesh** cell. By default, the **Use Workflow** check box is enabled, and you are able to select a workflow from the **Workflow Type** field. When the check box is disabled, you are still able to set CAD faceting and other options within the properties of the cell.

The **Save Task Editing Data Within Project** option (enabled by default) allows you to save relevant task-related data files (such as mesh information and task state) directly in your Workbench project, making it easier to use the workflow, for example, when reverting and editing tasks. This property is available for standalone **Fluent (with Fluent Meshing)** component systems, whether or not there is a connected upstream **Geometry** system. If this property is enabled, the **Saving data for editing tasks** workflow preference will use the **Write mesh files** option. See Setting Preferences for Workflows in the *Fluent User's Guide* for more information on workflow preferences.

6. In the **Workflow Type** field, select the **Watertight Geometry** option to use that workflow, or select the **Create New** option to create your own workflow.

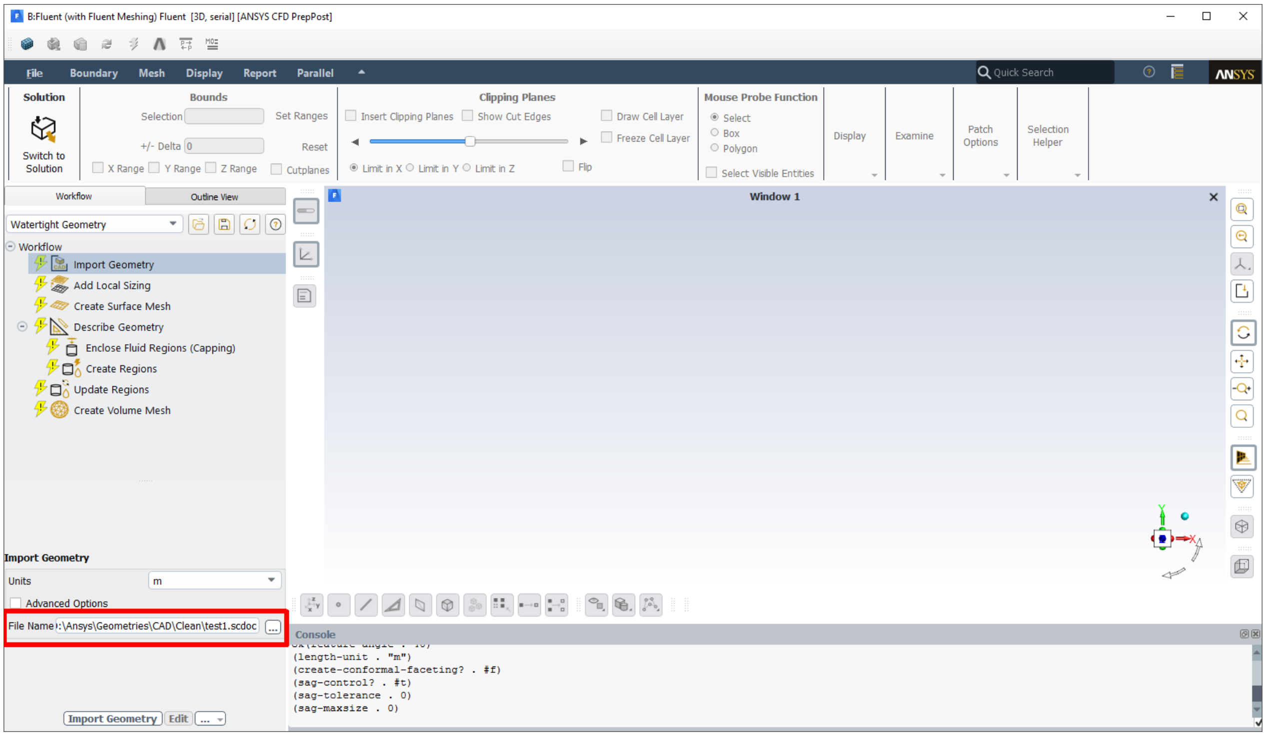


10	Workflow	
11	Use Workflow	☑
12	Workflow Type	Watertight Geometry
13	Save Task Editing Data Within Project	Watertight Geometry Create New
14	CAD Import Option For Geom	
15	Units	m

7. Start **Fluent** in Meshing mode.

   Double-click the **Mesh** cell in the **Fluent (with Fluent Meshing)** component, and make any appropriate selections in the **Fluent Launcher**, then proceed to open **Fluent** in Meshing mode to start working with a workflow.

   The Workflow tab is automatically opened in **Fluent** and the corresponding workflow is initialized with the imported CAD geometry from the **Geometry** cell from your Workbench project.

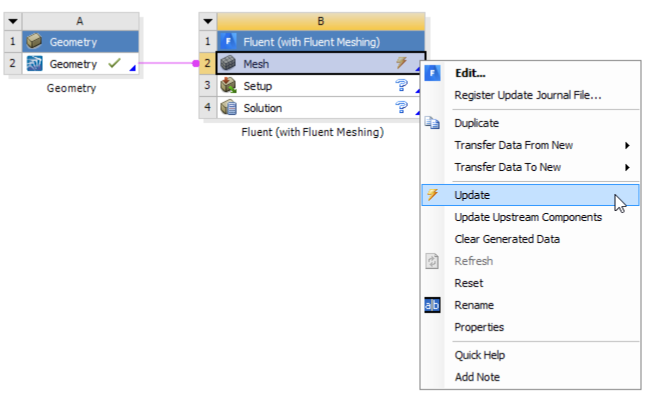


8. Proceed with setting up the various tasks of the workflow (for example, defining local sizing, capping inlets and outlets, extracting fluid regions, and generating the volume mesh).

   Once all workflow tasks are completed (and marked as up-to-date), and a volume mesh is generated, the state of the **Mesh** cell becomes **Up-to-Date**.

   Tasks can be added, removed, or otherwise customized to your needs, and the configuration will be retained when you return to **Fluent** from your Workbench project. See Working With Fluent Guided Workflows in the *Fluent User's Guide* for more information.

   When you have completed setting up the workflow, you can return to your Workbench project and update the **Mesh** cell in the **Fluent (with Fluent Meshing)** component.



9. Proceed, as usual, with completing the CFD simulation through the **Setup** and **Solution** cells in the **Fluent (with Fluent Meshing)** component.

   This functionality can be especially useful when you have similar geometries that use the same object labels (for example, different, but similar, manifold geometries that each contain three inlets named `in1`, `in2`, and `in3` and an outlet named `out1`). These similar geometries can be imported using a separate **Geometry** component that is connected to a single **Fluent (with Fluent Meshing)** component that already has a workflow defined to generate a compatible volume mesh for the same geometric configuration. The geometries can be swapped out, and once the **Fluent (with Fluent Meshing)** component is updated, the defined workflow will automatically update itself to generate a corresponding volume mesh.

3.4. Saving Your Work in Fluent Meshing with Workbench

You can save your Workbench project directly from Fluent Meshing by selecting **Save Project** under the **File** menu.

File → **Save Project**

Alternatively, you can also save your Workbench project by selecting the **Save** option under the **File** menu within Workbench or by clicking the **Save Project** icon ( ) from the Workbench toolbar.

For more information, see Saving Your Work in Fluent with Workbench (p. 41).

3.5. Exiting Fluent Meshing and Workbench

You can end your Fluent Meshing session by using the **Close Fluent** option under the **File** menu.

File → **Close Fluent**

You can end your Workbench session by using the **Exit** option under the **File** menu.

File → **Exit**

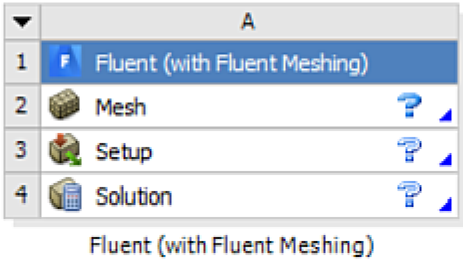
For more information, see Exiting Fluent and Workbench (p. 42).

3.6. An Example of a Fluent Meshing Analysis in Workbench

This example describes when the files that are generated and used by Fluent Meshing are written and how the cell states change as you work with a **Fluent (with Fluent Meshing)** component system in Workbench.

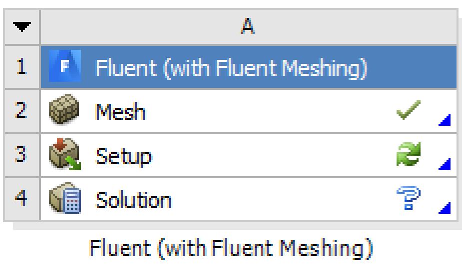
1. Add a new **Fluent (with Fluent Meshing)** component system to the **Project Schematic**.

   The state of the **Mesh** cell is **Attention Required** and the states for the **Setup**, and **Solution** cells are **Unfulfilled**.

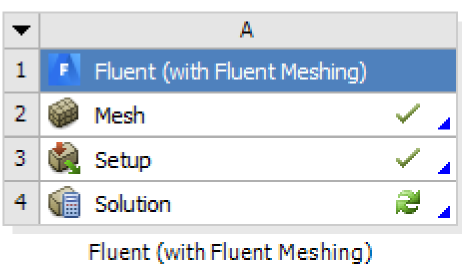


2. Double-click the **Mesh** cell. The Fluent Meshing application launches (the **Meshing Mode** option is automatically enabled in Fluent Launcher). In the Fluent Meshing application, import a geometry and/or create a mesh file, then perform your mesh operations. You can also import a Fluent case file into the Fluent Meshing application.

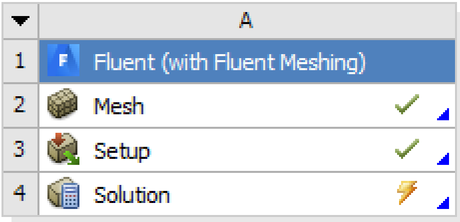
3. Switch to the solver (solution) mode (by using the **Switch to Solution** button in the **Solution** group box of the ribbon, or by using the `switch-to-meshing-mode` text command). The state of the **Mesh** cell becomes **Up-to-Date**, and the **Setup** cell becomes **Attention Required**. If you save the project at this point, the mesh (`.msh.h5`) and the settings (`.set`) files are written.



4. In Fluent, specify boundary conditions, initialize the solution, and enter a nonzero number of iterations on the **Run Calculation** task page. The state of the **Setup** cell becomes **Up-to-Date**, and the state of the **Solution** cell becomes **Refresh Required**.

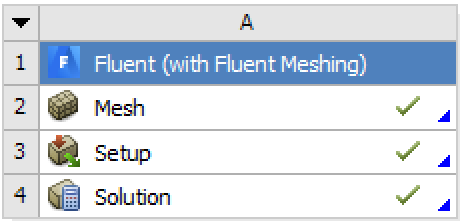


5. Start the simulation in Fluent by clicking **Calculate** on the **Run Calculation** task page. The settings (`.set`) file is written and iterations begin. The state of the **Solution** cell becomes **Update Required**.



Fluent (with Fluent Meshing)

When iterations are completed, or the solution meets the convergence criteria, the state of the **Solution** cell becomes **Up-to-Date**.



Fluent (with Fluent Meshing)

3.7. Using Parallel Fluent Meshing

If you want to run Fluent Meshing in parallel, you can specify the value for **Number of Parallel Processors** in the **Properties** pane for the **Mesh** cell.

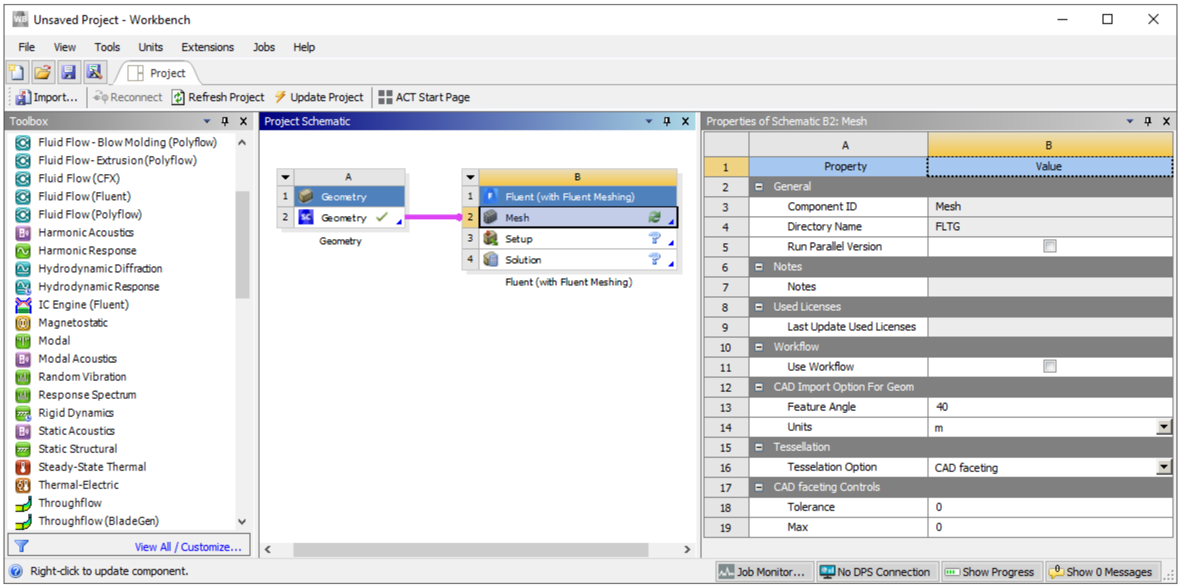
1	Property	Value
2	⊟ General	
3	Component ID	Mesh
4	Directory Name	FLTG
5	Run Parallel Version	☑
6	Number of Parallel Processors	2

You can override this parallel-process setting by setting the **Meshing Processes** under the **Parallel** processing options in Fluent Launcher.

3.8. Connecting to Upstream Geometry

You can connect an upstream **Geometry** cell to a **Mesh** cell of a **Fluent (with Fluent Meshing)** component system to transfer geometry data to Fluent Meshing.

Figure 3.1: Connecting Upstream Geometry and Mesh Cells



Once the cells have been connected, and once the **Use Workflow** property has been *disabled*, the following CAD import options and controls become available in the **Properties** for the **Mesh** cell:

Feature Angle

specifies the feature angle to determine the features to be imported. A smaller value will result in importing more features. The default value is 40.

Unit

specifies the length units for scaling the mesh. Models created in different units will be scaled accordingly. The default is meters (m).

Tessellation Option

contains options that enable you to control the tessellation (faceting).

CAD faceting

enables you to control the tessellation (faceting) refinement during the file import.

CFD surface mesh

enables conformal tessellation (faceting) that uses the curvature size function based on the minimum and maximum facet sizes, **Min Size** and **Max Size**, the specified facet **Curvature Normal Angle**, and the edge proximity size function **Edge Proximity**.

CAD faceting Controls

(when **CAD faceting** is selected for **Tesselation**)

Tolerance

specifies the tolerance for the tessellation (faceting) refinement. The default value of 0 implies no tessellation (faceting).

Max

specifies a maximum facet size for the imported model to avoid very large facets.

CFD surface mesh Controls

(when **CFD Surface Mesh** is selected for **Tesselation**)

Use Size Field File

if enabled, you can provide a size field file.

Min Size, Max Size

specify the minimum and maximum facet sizes, respectively.

Curvature Normal Angle

specifies the curvature normal angle based on the underlying curve and surface curvature.

Edge Proximity

enables the use of the edge proximity size function based on the number of cells per gap specified.

Cells Per Gap

specifies the number of element layers to be generated in a gap for the edge proximity size function.

Save Size Field

enables you to save the size field in a file, which is computed based on the specified parameters (**Min Size**, **Max Size**, **Curvature Normal Angle**, and **Cells Per Gap**).

Size Field File Location, Size Field File Name

specifies the file name and the location for the size field file, respectively.

3.9. Support for the Remote Solve Manager (RSM) in Fluent (with Fluent Meshing) Systems

You can run Workbench parametric studies with the Remote Solve Manager (RSM) when using the **Geometry** and the **Fluent (with Fluent Meshing)** component systems. For more information about RSM in general, see Remote Solve Manager User's Guide.

RSM Host on Linux

When the **Geometry** component system is connected to a **Fluent (with Fluent Meshing)** component system, and the geometry type is set to `*.scdoc`, you can run a parametric study on both Windows and Linux. Since Ansys SpaceClaim Direct Modeler files (`*.scdoc`) CAD geometry files are not supported on Linux, Fluent internally creates a compatible `.pmdb` file at the same location to be used for the parametric study, irrespective of whether the RSM host's operating system is Linux or Windows. These generated `.pmdb` files are used only when the RSM host's operating system is Linux. If the RSM host's operating system is Windows, these `.pmdb` files are still generated, however, they are not explicitly used.

Additionally, if the geometry type is set to `*.scdoc`, make sure that the **Pre-RSM Foreground Update** option is set to **Geometry** in the **Parameter Set**'s **Properties** panel.

3.10. Using a Journal File for the Mesh Cell

For **Fluent (with Fluent Meshing)** component systems, you can use a journal file (`*.jou`) to implement the text user interface (TUI) commands and/or Scheme expressions related to mesh generation.

To associate the journal file with the **Mesh** cell:

1. Right-click the **Mesh** cell and from the context menu, select **Register Update Journal File**.

2. Browse to the journal file location and select the file.

   Once the journal file has been associated with the **Mesh** cell, the **Register Update Journal File** command is replaced by **Unregister Startup Journal File**. You can execute this command if you want to disassociate the assigned journal file from the **Mesh** cell.

On performing **Update** of the **Mesh** cell, the registered journal file associated with the cell will be executed in Fluent Meshing in the background.

3.11. Using a Journal File to Parameterize Fluent Meshing Inputs

As discussed in Limitations (p. 103), parameters are not currently supported by Fluent Meshing; however, you can parameterize real arguments using the appropriate scheme commands in the registered update journal file (`*.jou`).

A journal file that parameterizes some inputs for Fluent Meshing is shown below:

```
;=================================================================================
;==Define scheme variable connected to input parameters exposed in Workbench======
(define tg-input-parameter-data '(
   (p1 (value . 0.0025)) ;thickness of the first prism layer
   (p2 (value . 5))      ;number of prism layers
   (p3 (value . 0.05))   ;max element size for global size settings
))
(create-scheme-input-parameters tg-input-parameter-data)
; (print-scheme-input-parameters);check the list for current value in fluent-meshing side
;==================================================================================
;===Add required TUI/scheme commands to generate volume mesh
;===Mesh Cell will be marked up-to-date on exit of Fluent or changing mode to solver===========

/size-functions set-global-controls 0.005 p3 1.2
/size-functions create-defaults
```

```
/objects/improve-object-quality (*) surface-remesh
/mesh/prism/controls/zone-specific-growth apply-growth (wall) last-ratio p2 p1 40 no
/mesh/auto-mesh "fluid" () yes zone-specific pyramids tet
/mesh/clean-up yes

/boundary/manage/type (inlet-cold) velocity-inlet
/boundary/manage/type (inlet-hot) velocity-inlet
/boundary/manage/type (outlet) pressure-outlet
/boundary/manage/name interior* interior
;===============================================================================


/boundary/manage/name interior* interior
```

The above journal file was created by modifying (that is, parameterizing) a journal file that was recorded during an earlier Fluent Meshing session.

The recorded journal file was edited to remove all of the commands that are not necessary for the desired parametric study, such as:

- Commands related to graphics operations (usually these commands start with "**`cx-`**")
- Commands that manage lists

Next, commands were added at the top of the journal file in order to create the parameters that will be exposed in Workbench. These commands are as follows:

- The **`define`** statement creates the user-defined scheme list **`tg-input-parameter-data`** with three sub-lists (**`p1`**, **`p2`**, and **`p3`**). Each of the sub-lists consists of a variable symbol-name (**`p1`**, **`p2`**, or **`p3`**) and an initial **`value`**.
- The **`create-scheme-input-parameters`** statement is the Fluent-specific scheme API function that creates input parameters for the name-value pairs in **`tg-input-parameter-data`** and connects it to the input parameters exposed in Workbench. After executing this command, Workbench will list **`p1`**, **`p2`**, and **`p3`** under **Input Parameters**.
- The **`print-scheme-input-parameters`** is the Fluent-specific scheme API function that prints the current input parameter values in the Fluent console. In the above example, it is commented out, but you can use this statement to check these values.

Lastly the specific values entered for the Fluent Meshing inputs during the recorded session were replaced by corresponding parameters (**`p1`**, **`p2`**, or **`p3`**).

3.12. Getting Help for Fluent Meshing in Workbench

Accessing help from within Ansys Workbench:

For information on how to access help for Ansys Workbench, see Getting Help for Fluent in Workbench (p. 46).

Accessing help from within Fluent Meshing:

Fluent Meshing documentation and help can be accessed by clicking once the Fluent Meshing application is running. The documentation is automatically installed when you install Workbench.

Part II: Internal Combustion Engine Applications

Chapter 1: Introduction to Internal Combustion Engines

The IC Engine (Fluent) system is being deprecated and will be removed in a future release. Consider using Forte in Component Systems for conducting simulations of internal combustion engines in Workbench.

The design and manufacture of Internal Combustion (IC) Engines is under significant pressure for improvement. The next generation of engines needs to be compact, light, powerful, and flexible, yet produce less pollution and use less fuel. Innovative engine designs will be needed to meet these competing requirements. The ability to accurately and rapidly analyze the performance of multiple engine designs is critical. Information in this chapter is organized into the following sections:

1.1. Engine Performance

The performance of an IC Engine depends upon complex interactions between mechanical, fluid, chemical, and electronic systems. However, the central challenge in design is the complex fluid dynamics of turbulent reacting flows with moving parts through the intake/exhaust manifolds, valves, cylinder, and piston. The time scales of the intake air flow, fuel injection, liquid vaporization, turbulent mixing, species transport, chemistry, and pollutant formation all overlap, and need to be considered simultaneously.

Computational Fluid Dynamics (CFD) has emerged as a useful tool in understanding the fluid dynamics of IC Engines for design purposes. This is because, unlike analytical, experimental, or lower dimensional computational methods, multidimensional CFD modeling allows designers to simulate and visualize the complex fluid dynamics by solving the governing physical laws for mass, momentum, and energy transport on a 3D geometry, with sub-models for critical phenomena like turbulence and fuel chemistry. Insight provided by CFD analysis helps guide the geometry design of parts, such as ports, valves, and pistons; as well as engine parameters like valve timing and fuel injection.

Engine analysis using CFD software has always been hampered due to the inherent complexity in

- Specifying the motion of the parts.
- Decomposition of the geometry into a topology that can successfully duplicate that motion.
- Creating a computational mesh in both the moving and non-moving portions of the domain.
- Solving the unsteady equations for flow, turbulence, energy, and chemistry.
- Postprocessing of results and extracting useful information from the very large data sets.

This is a time consuming and error prone process, creating a significant impediment to rapid engine analysis and design feedback.

The solution to this problem is an integrated environment specifically tailored to the needs of modeling the internal combustion engine. The environment requirements are as follows:

- It should have the necessary tools to automatically perform a problem setup.
- It should require minimal inputs from the user.
- It should be able to transfer information rapidly between the different stages of the CFD analysis.
- It should significantly compress the setup and analysis process.
- There should be no loss in the accuracy.
- The potential for errors should be reduced.

The IC Engine Analysis System provides such an integrated environment with the capabilities integrated to set up most IC Engine designs.

The IC Engine System includes:

- Bidirectional CAD connectivity to mainstream CAD systems.
- Powerful geometry modeling tools in Design Modeler.
- Flexible meshing using Ansys Meshing.
- Solution using Ansys Fluent.
- Powerful postprocessing in CFD-Post.
- In addition, persistent parameterization and design exploration (DX) allow users to modify geometry or problem setup parameters and to automatically regenerate analysis results.

The time taken for geometry, meshing, and solution setup has been reduced from several hours of work to minutes, with reduced potential for error. The user specifies the engine parameters and geometry at the beginning of setup, instead of at the solution stage, to guide and automate the entire setup process.

The next few chapters will introduce the automation tools in the IC Engine Analysis System and how to use them. A deeper examination of the fluid dynamics issues in IC Engines and the CFD modeling process will be conducted first, followed by details of the IC Engine Analysis System.

1.2. Engine Design

IC engine design involves several critical decisions which impact and interact with the fluid dynamics. The primary design decisions are

- The specifications for engine type.
- Peak power at a specified speed or RPM.

- The number of cylinders.
- Fuel and emissions characteristics.
- The total volume of the engine.
- Overall "packaging" of the engine including all the sub-systems.

Mechanical and electronics systems may also be specified at this stage, such as using different cam configurations. There may be additional specifications, such as the engine power at idle speed or low RPM.

These design decisions impact the computation of the amount of air and fuel needed by the engine and lead to a cascade of design decisions to maximize the overall efficiency of the engine. This efficiency is given by the following equation for engine brake power:

$$P_b = \frac{\eta_i \eta_c \eta_m \eta_v \rho_i V_d N}{XFQ} \tag{1.1}$$

where

η_i is the indicated (brake) efficiency

η_c is the combustion efficiency

η_m is the mechanical efficiency

η_v is the volumetric efficiency of the engine

ρ_i is the density of the air at the intake

V_d is the engine displacement volume

N is the rotational speed

X is the number of revolutions per power stroke

F is the fuel air ratio

Q is the calorific value of the fuel per unit mass

- The primary goal of engine design is to maximize each efficiency factor, in order to extract the most power from the least amount of fuel. In terms of fluid dynamics, the volumetric and combustion efficiency are dependent on the fluid dynamics in the engine manifolds and cylinders.
- The second goal of engine design is to meet emissions requirements, which are always specified by regulations. The pollutants include oxides of nitrogen, sulfur oxides (SOx), CO (carbon monoxide), unburned hydrocarbons (HC), and Poly Aromatic Hydrocarbons (PAH or “soot”), which are all products of the combustion process. Pollutants are formed by a variety of interactions of the mechanical and chemical processes inside the engine and are intimately tied to fluid dynamics in the cylinder. Though the pollutants in the exhaust stream can be reduced utilizing after-treatments, often these technologies add considerable cost to the engine. Therefore, it is desirable to minimize the pollutant formation at the source.

1.3. Fluid Dynamics During the Four Cycles

The volumetric efficiency of the engine depends on several fluid dynamic phenomena in the intake and exhaust tracts leading to the combustion chamber.

- When the air is pumped into the combustion chamber during the intake cycle, it passes through the gap between the valve and the valve seat. As it squeezes through the gap, the flow separates from the walls of the port and valve surfaces, forming a tangential jet. The jet from the valves impinges on the cylinder walls and tumbles into the space between the valves and the piston. This jet imparts angular momentum, known as swirl and tumble, to the fresh charge. The gross motions of the fresh charge are recirculation regions that promote mixing. If there is strong swirl (usually described by a normalized angular momentum value about the vertical axis through which the piston motion is constrained) the flow may develop stratification with regions of high and low velocity. The intake port may be designed to impart additional angular momentum to the air; and multiple intake valves or any partially open exhaust valves may have flow interaction.

- When the piston travels back up towards the top during the compression stroke, most of the energy contained in the tumble (or angular momentum orthogonal to the swirl axis) of the jet is converted to turbulence as the available space in the vertical direction is reduced significantly. The swirl will become stronger as the air is squeezed out to the side. If there is a narrow region between the piston and the cylinder head, the air may be squeezed (or "squished") from the sides of the cylinder into the combustion chamber, converting the energy in the swirl into turbulence. Flow phenomena, which affect volumetric efficiency include

  - separation, jet formation, and reattachment on the cylinder head
  - swirl and tumble in the cylinder volume to promote mixing
  - turbulence production during the compression of air due to squeezing of the main flow
  - flow stratification in the cylinder

  Engines that utilize port fuel injection (PFI) or carburetion are known as premixed engines. For a port fuel injected engine, the fuel is sprayed into the ports normally onto the back of the intake valve, where it vaporizes and mixes with the intake air. An engine that uses a carburetor mixes the air and fuel as the air enters the intake manifold. In this type of premixed engine, at least from the fluid mechanical point of view, a mixture of fresh air and fuel is inducted into the engine, through the intake port.

  For Direct Injection (DI) and all modern diesel engines, high pressure fuel is injected directly into the combustion chamber as the piston nears the end of the compression stroke. The liquid fuel spray breaks up into smaller droplets and vaporizes into the surrounding air. High levels of turbulence in the cylinder enhance the mixing and high pressure of the fuel spray enhance breakup. In any engine, charge motion at the start of combustion is necessary for efficient burning of the mixture. However, often some compromises need to be made in the range of speeds over which the engine operates.

- In a spark ignited (SI) engine, a flame front is formed which moves outwards from the ignition point, consuming the available fuel air mixture. Turbulence again plays a significant role in flame propagation, since the flame moves at the turbulent flame speed. Hence, if the turbulence levels are high, the flame front will move more rapidly to all parts of the combustion chamber. For SI engines, the rapid flame propagation avoids knock due to autoignition of fuel air ahead of the flame. The flame speed depends on the air fuel ratio of the mixture. If the mixture is outside of the flammability limits, usually between equivalence ratios of 0.5 and 4, the flame will not propagate and the engine will misfire. Similarly, if regions exist inside the cylinder that are outside of the flammability limits, these regions will not burn and will most likely be pushed out through the exhaust and into the atmosphere.

For compression ignition engines, air is compressed to a high temperature and pressure and fuel is injected directly into the combustion chamber. After some time for spray breakup, mixing, and low temperature chemical breakdown to occur, the mixed air and fuel in the spray plume ignites and begins to burn, usually forming a stratified, or diffusion flame. Compression ignition engines have no knock limit, however are limited by the amount of mixing in the cylinder and the material limits of the components.

Combustion produces a rise in pressure and temperature as the energy contained in the fuel is released and the chemical reaction is completed. The fuel combustion produces a spike in pressure and temperature as the energy contained in the fuel is released, with the production of exhaust gases. Some of the energy is radiated and convected to the cylinder walls, cylinder head, piston and the valves; and is lost. Most of the energy goes into the power stroke, where the exhaust gases expand under high pressure and push the piston down to the bottom center position. A thermodynamic energy balance shows that the energy produced due to combustion is used for work done due to expansion, while the thermal losses includes heat losses through the walls and the enthalpy of the exhaust gases at high temperature.

- During the subsequent exhaust stroke, the exhaust gases are pushed out through the exhaust valves, which start opening towards the end of the power stroke. This process involves formation of a high speed, high temperature jet in the gap between the exhaust valves and ports.

  During combustion, the fuel, which is a long chain hydrocarbon, breaks up into smaller molecules. The carbon and hydrogen contained in these molecules combine with the oxygen in the air in exothermic reactions. If the fuel air ratio is stoichiometric at each location in the combustion chamber, carbon dioxide and water are formed. However, if the fuel air ratio is rich at particular locations due to inadequate mixing, the oxygen molecules are not sufficient and the combustion will be incomplete. Here carbon monoxide (CO) and unburnt hydrocarbon molecules will be produced.

  Some of the unburnt hydrocarbons will be polycyclic or poly-aromatic as the carbon chains wrap around each other and form solid particles, which is called soot. If the carbon monoxide and unburnt hydrocarbons are then transported to a region with adequate oxygen, then the combustion may still reach completion. If not, they will leave with the exhaust gases and represent a loss of energy. Due to high temperatures, the nitrogen molecules contained in the air break up under intense heat and the nitrogen ions combine with the available oxygen radicals to form nitrous oxides or NOx. If the fuel contains nitrogen or sulfur atoms, they will also form NOx and sulfur oxides (SOx).

Thus, the combustion efficiency of the engine and pollutant formation depends on the fluid dynamics of swirl, tumble, mixing, and turbulence production during the intake and compression strokes, losses due to incomplete combustion, the heat transfer losses to the wall, and the exhaust losses. Engine emissions include carbon dioxide, water, carbon monoxide, NOx, SOx, unburnt hydrocarbons, and soot.

1.4. Designing High Efficiency IC Engines

The design of high efficiency IC engines has to take into account the complex fluid dynamics that occurs in the manifolds and cylinders. Several design issues come to the forefront here.

1.4.1. Port Flow Design

The air flow rate through the intake manifold ports, depends on the pressure difference between the cylinder and the manifold, as well as the throttle position. A critical consideration here is the packaging; i.e. the engine and its supporting systems have to fit in a certain amount of space and still allow easy access for future maintenance. This means that the intake manifolds and engine ports might be routed around other parts; which introduces an additional resistance to the air flow and affects the swirl and tumble in the cylinder. Port flow design to achieve a given air flow rate and desired levels of swirl/tumble within a certain packaging layout to maximize volumetric efficiency, thus becomes a critical fluid dynamics design problem.

1.4.2. Combustion Chamber and Piston Shape

A critical design issue is the size and shape of the combustion chamber, the piston crown shape, and the layout of the valves. Here, the chamber can be flat, a hemispheric dome, or a penta-head, while the piston crown can be flat, domed, or a bowl. The valves can be positioned as "straight", i.e. the valves are aligned with the cylinder axis as shown in Figure 1.1: Straight Valve Engine (p. 124), or they can be "canted", i.e. they are at an angle to the cylinder axis and normal to the surfaces of the combustion chamber (Figure 1.2: Canted Valve Engine (p. 125)).

Figure 1.1: Straight Valve Engine

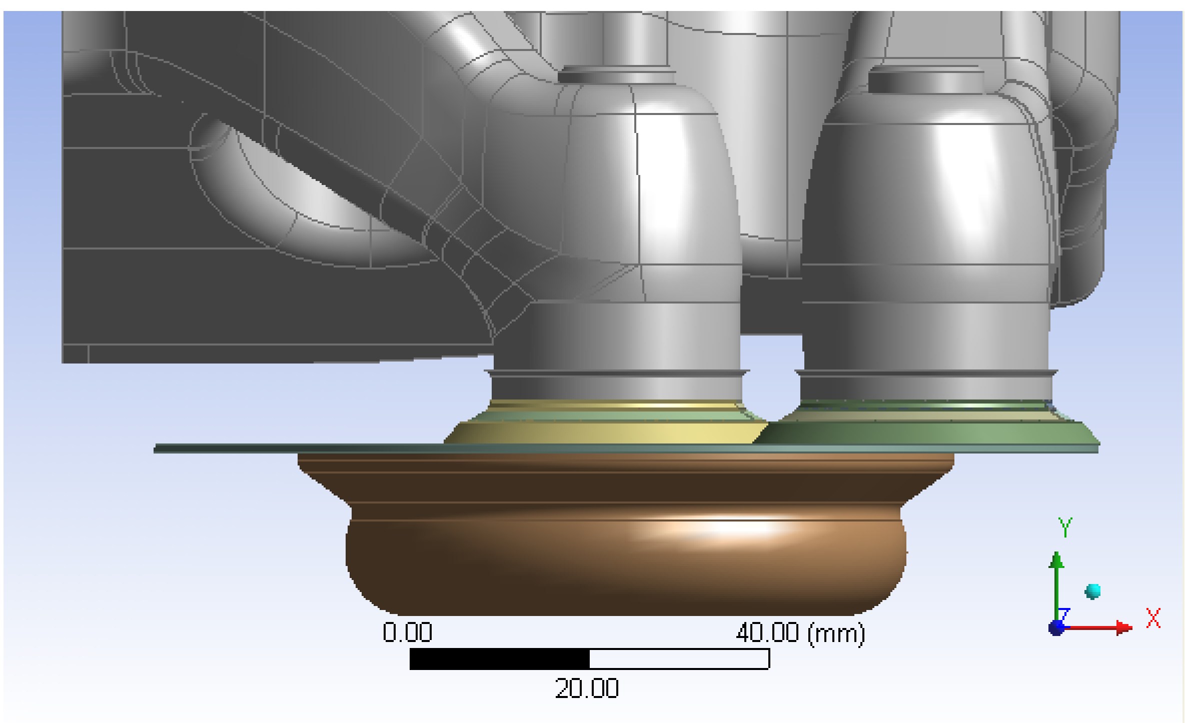
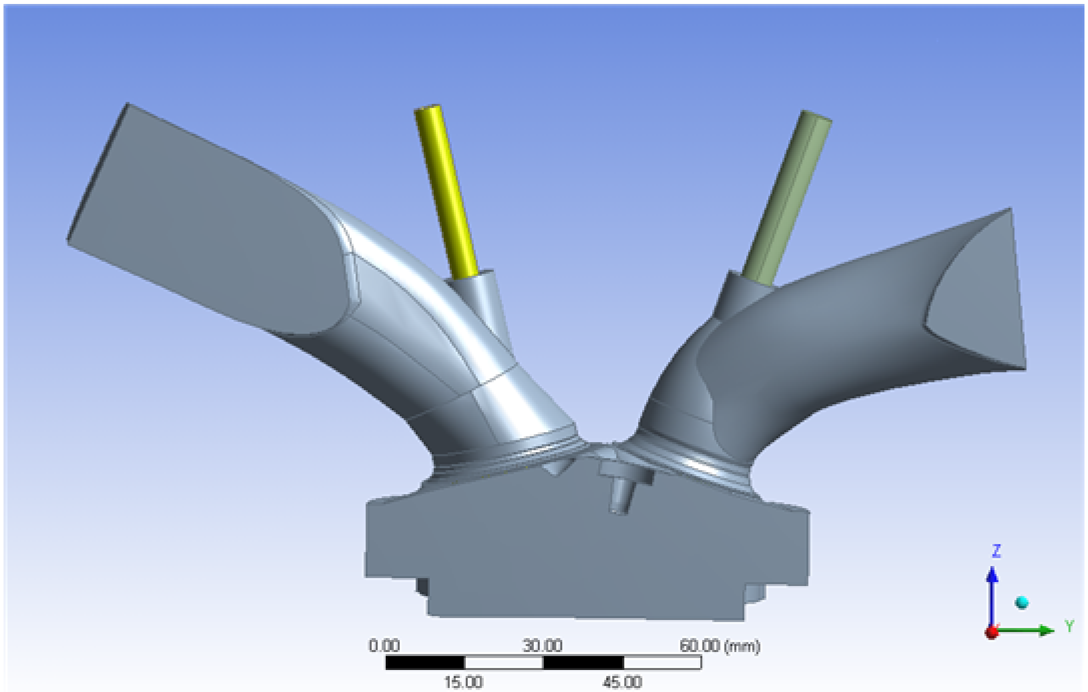


Figure 1.2: Canted Valve Engine

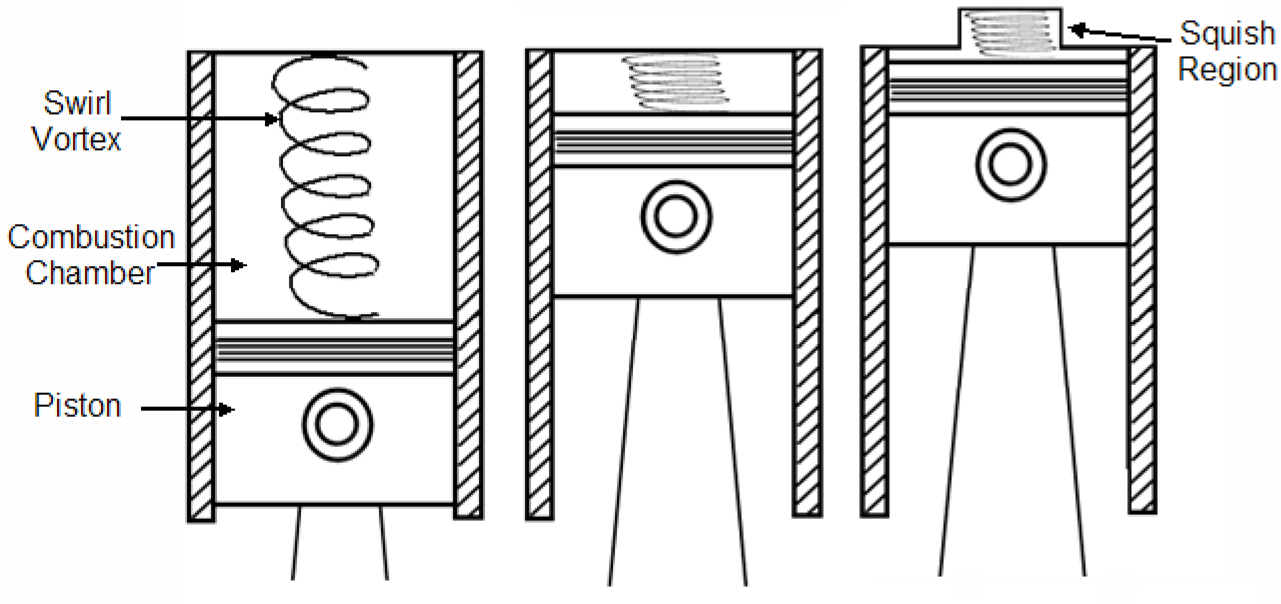


It has been shown that the volumetric efficiency and the amount of air that makes it into the cylinder is dependent on the ratio of the intake valve area relative to the cross section area of the cylinder. Hence it is desirable that the intake valves be as large as possible relative to the bore. However, if the combustion chamber is flat, it limits the surface area available for the valve layout to just the cross section. If the combustion chamber is hemispheric or penta-headed, it opens up more surface area for the intake and exhaust valves, allowing them to be larger and more efficient. However, this means that the combustion chamber has a larger volume and surface area, which implies that the flame front for combustion has a longer distance to travel, increasing the chance of incomplete combustion. Also the compression ratio will be decreased since there is a larger volume at the top center. In addition, a larger wall surface area increases the heat losses during combustion. Thus, there is adverse impact on combustion efficiency.

This may be counteracted by changing the piston shape from the flat shape to a domed shape to reduce the volume. But this means that the flame front has to travel around the piston dome to reach all parts of the combustion chamber volume, thus increasing the time taken for complete combustion, raising the possibility of knocking in SI engines. The piston could then be made to have a bowl in the center, which would reduce the flame travel time, but reduce the compression ratio.

1.4.3. Squish

An additional geometric design consideration is the "squish" region, which is the region around the perimeter of the combustion chamber with the smallest clearance volume between the piston and the cylinder head at top center. As noted before, a small or low squish region causes the swirling air flow to get squeezed out into the combustion chamber and form a turbulent jet, which converts the mean flow energy in the swirl into turbulence, increasing the turbulent flame speed and the compression ratio and the combustion efficiency.

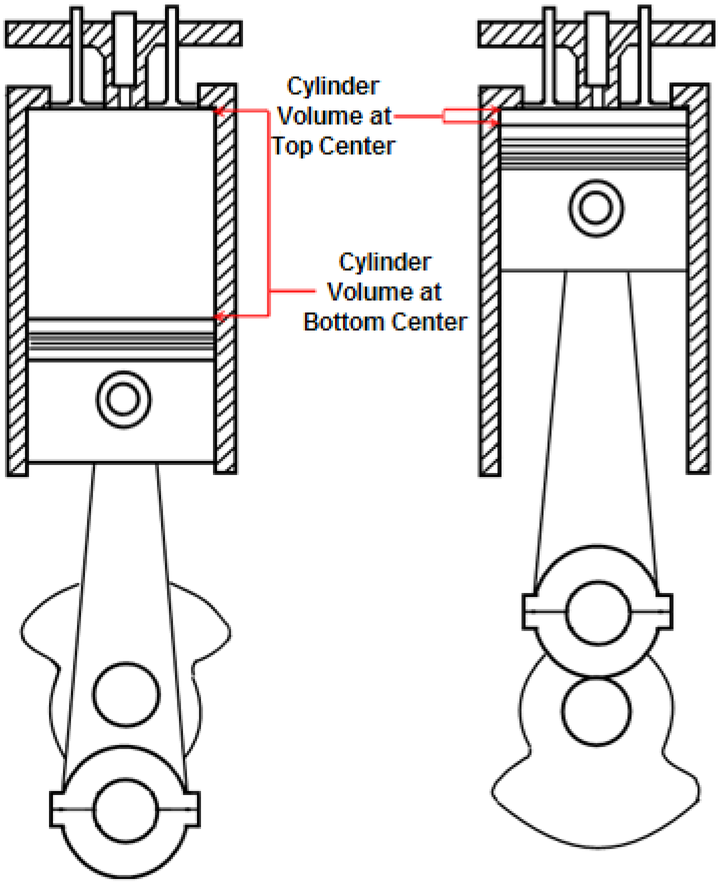


Swirl Vortex First Compressed and then Squished

But if the squish is too low, there is a chance of collision between the piston and the cylinder head when the engine material expands at high temperature. In addition, mass manufacturing of engines requires that there should be room for production and manufacturing tolerances. Hence the squish needs to be low enough to allow higher turbulence production, but high enough to allow room for variability due to thermal expansion and manufacturing tolerances.

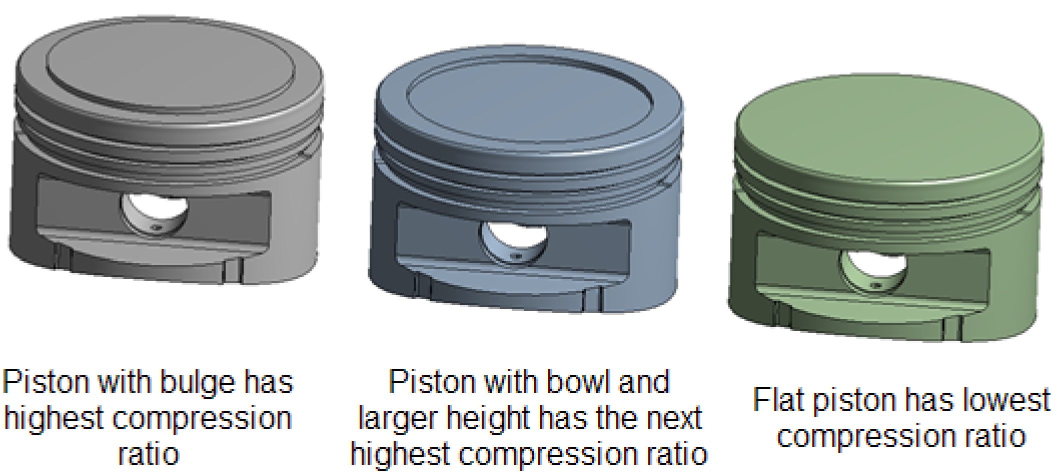
1.4.4. Compression Ratio

The compression ratio, which is defined as the ratio of the cylinder volume at the bottom center to the volume at the top center, is a critical factor in combustion efficiency and pollutant formation. A high compression ratio enhances the combustion efficiency, but the higher temperatures cause more NOx to form, thus increasing the emissions.



Automotive engines in the 1970s had much higher compression ratios since the emissions standards were much lower. With stricter environmental regulations on emissions, the compression ratios were reduced to meet the new standards. In the 1990s, technological improvements in catalytic converters and improvements in combustion efficiency allowed higher compression ratios and improved fuel economy. An additional consideration, especially for diesel engines, is that the materials used for the piston and combustion chamber must be able to withstand the peak temperatures and pressures encountered with high compression ratios and high boost levels.

Figure 1.3: Different Piston Shapes Used to Achieve Desired Compression Ratio for an Engine

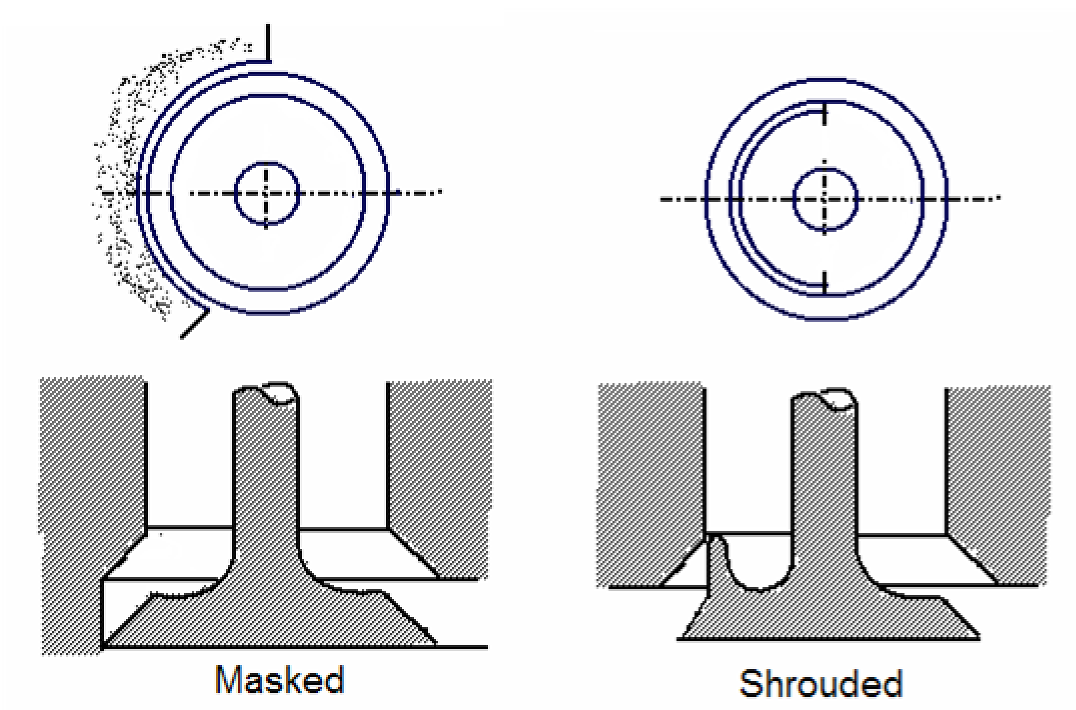


Thus the geometric design of intake/exhaust ports, cylinder heads, valves and pistons involves interplay and trade-offs between the volumetric and combustion efficiency, pollutant formation, packaging considerations, materials choices and manufacturing tolerances. The ability to accurately analyze the engine fluid dynamics plays a key role in optimizing the engine to efficiently deliver the power needed, while meeting emissions standards and respect packaging and manufacturing constraints.

1.4.5. Design for Low Speed and Idle

The performance of engines at idle or low speed is a critical design consideration in many cases. For passenger car engines, the engine is designed for peak power at a specific speed, which is typically high. However, the engine will still have to perform well at lower speeds and at idle. Variable valve timing, which allows the valves to have different lift profiles and opening and closing events for different engine speeds, is used in many modern engines. Here the goal is to maximize the volumetric and combustion efficiency by producing optimal levels of swirl, tumble and turbulence at both low and high speeds so that the combustion charge, i.e. the air/fuel mixture, is well mixed and the turbulent flame speed is high enough for complete combustion. There are additional geometric design changes to the ports and valves that can be made for low speed or idle conditions. For example, valve masking and valve shrouding is used to impart additional swirl to the air flow and increase the jet velocity.

Figure 1.4: Inlet Valves Used to Induce High Swirl at Low Engine Speeds and Low Valve Lift



1.4.6. Spark and Injection Timing

During the engine cycle, the spark timing and start of injection has been optimized to provide the desired power or torque with minimum pollutant formation. Strategies, such as Exhaust Gas Recirculation (EGR) are used to minimize the peak temperature in the engine by increasing the thermal mass of the intake air, which reduces the NOx production, which is strongly correlated with higher temperatures. For diesel engines, there is tradeoff between soot production and NOx, since soot production decreases with temperature increase and NOx increases with temperature increase. Thus designing high efficiency engines to meet performance and emissions standards requires tradeoffs which take engine fluid dynamics into account.

Chapter 2: Modeling CFD in IC Engine Design

This chapter emphasizes the role of CFD modeling in IC Engine design. The information in this chapter is divided into the following sections:

2.1. The Role of CFD Analysis in Engine Design

As described in Introduction to Internal Combustion Engines (p. 119), IC engines involve complex fluid dynamic interactions between air flow, fuel injection, moving geometries, and combustion. Fluid dynamics phenomena like jet formation, wall impingement with swirl and tumble, and turbulence production are critical for high efficiency engine performance and meeting emissions criteria. The design problems that are encountered include port-flow design, combustion chamber shape design, variable valve timing, injection and ignition timing, and design for low or idle speeds.

There are several tools which are used in practice during the design process. These include experimental investigation using test or flow bench setups, 1D codes, analytical models, empirical/historical data, and finally, computational fluid dynamics (CFD). Of these, CFD has the potential for providing detailed and useful information and insights that can be fed back into the design process. This is because in CFD analysis, the fundamental equations that describe fluid flow are being solved directly on a mesh that describes the 3D geometry, with sub-models for turbulence, fuel injection, chemistry, and combustion. Several techniques and sub-models are used for modeling moving geometry motion and its effect on fluid flow.

Using CFD results, the flow phenomena can be visualized on 3D geometry and analyzed numerically, providing tremendous insight into the complex interactions that occur inside the engine. This allows you to compare different designs and quantify the trade-offs such as soot vs NOx, swirl vs tumble and impact on turbulence production, combustion efficiency vs pollutant formation, which helps determine optimal designs. Hence CFD analysis is used extensively as part of the design process in automotive engineering, power generation, and transportation. With the rise of modern and inexpensive computing power and 3D CAD systems, it has become much easier for analysts to perform CFD analysis. In increasing order of complexity, the CFD analyses performed can be classified into

- **Port Flow Analysis** (p. 132): Quantification of flow rate, swirl and tumble, with static engine geometry at different locations during the engine cycle.

- **Cold Flow Analysis** (p. 133): Engine cycle with moving geometry, air flow, and no fuel injection or reactions.

- **In-Cylinder Combustion Simulation** (p. 133): Power and exhaust strokes with fuel injection, ignition, reactions, and pollutant prediction on moving geometry.

- **Full Cycle Simulation** (p. 135): Simulation of the entire engine cycle with air flow, fuel injection, combustion, and reactions.

However, the CFD analysis process for engines has been complex, time consuming, and error-prone. Typically, the analyst has to go through several steps to set up the problem, and any minor error can lead to failure of the simulation. Once the analysis has been set up, it takes many hours or days of computation to get the solution and evaluate the results. The results are fairly complex, with large data sets, which require time and effort to analyze and get useful information, which can be fed back to the design stage.

Automation and process compression thus becomes a critical need. In the next section, you will further evaluate the practical issues facing engineers in conducting a CFD analysis on IC Engines. Following that, you will explore the solutions that are available in an integrated environment like the Ansys Workbench and explain the rationale for the IC Engine Analysis System.

2.2. Types of CFD Analysis for IC Engines

The following sections present some practical issues in conducting a CFD analysis for IC Engines.

2.2.1. Port Flow Analysis

In port flow analysis, the geometry of the ports-valves and cylinders is "frozen" at critical points during the engine cycle and the air flow through the ports is analyzed using CFD. The flow rate through the engine volume, swirl and tumble in the cylinder and turbulence levels are determined. Fluid dynamics phenomena like separation, jet formation, valve choking, wall impingement, and reattachment, as well as the secondary motions, can be visualized and analyzed.

The results provide snapshots of the fluid dynamics throughout the engine cycle; and are used to modify the port geometry to produce desired behavior of the air flow. Simulation validation can be performed using the real geometry mounted on a flow bench with measurement of flow rates, velocities, and turbulence levels using techniques like LDV (Laser Doppler Velocimetry). The results do not capture dynamic phenomena such as expansion and compression of air due to piston movement and turbulence production from swirl and tumble.

In practice, conducting port flow analysis at a single point is relatively straightforward because of the static geometry, which fits well with the workflow and capabilities in CFD software. You start with the port, valve and cylinder geometry at a particular position, create a mesh, specify the mass flow rate or pressure drop for the compressible flow and a turbulence model and compute the results. The RANS approach based turbulence models are used to compute the effect of turbulence. Since the turbulent flow interactions with the walls are critical, mesh refinement in the near wall region is necessary using inflation or boundary layers. Experimental data provides validation to develop confidence and best practices for model setup and accuracy.

However, when the number of critical positions and hence the number of cases increases, the problem complexity increases significantly. Setting up large numbers of static cases with identical mesh and flow settings is time consuming, with scope for error.

2.2.2. Cold Flow Analysis

Cold flow analysis involves modeling the airflow and possibly the fuel injection in the transient engine cycle without reactions. The goal is to capture the mixture formation process by accurately accounting for the interaction of moving geometry with the fluid dynamics of the induction process. The changing characteristics of the air flow jet that tumbles into the cylinder with swirl via intake valves and the exhaust jet through the exhaust valves as they open and close can be determined, along with the turbulence production from swirl and tumble due to compression and squish.

This information is very useful to ensure that the conditions in the cylinder at the end of the compression stroke are right for combustion and flame propagation. High turbulence levels facilitate rapid flame propagation and complete combustion during the power stroke. A well mixed and highly turbulent air flow is critical to ensure the right air/fuel ratio throughout the combustion. CFD can also assess the level of charge stratification.

However, since cold flow simulations do not include the significant thermodynamic changes that accompany combustion, the flow characteristics during the power and exhaust strokes do not reflect reality. In terms of validation, experimental PIV (Particle Image Velocimetry) or LDV data in cycling engines is not easy to obtain as with port flow analysis, but transparent pistons and cylinders have been used to gather velocity information for some engine configurations.

Setting up the CFD model for cold flow analysis involves additional work in specifying the necessary information to compute the motion of the valves and piston in addition to the boundary conditions, turbulence models and other parameters. This includes specifying valve and piston geometry, along with the lift curves and engine geometric characteristics in order to calculate their position as a function of crank angle. Since the volume in the cylinder is changing shape throughout the engine cycle, the mesh has to change accordingly. Different approaches to automatically modify the mesh during motion also need to be specified. The CFD calculation is an inherently transient computational problem when involved with moving deforming or dynamic mesh. All the geometric motion is a function of a single parameter, the position of the crankshaft in its rotation, or crank angle.

The preprocessing from geometry to solver setup is typically time consuming and challenging. Here, you have to separate or decompose the geometry into moving and stationary parts. Typically, the intake ports are split off from the cylinder and valves. The region between the valve margin and valve seat, which opens and closes during valve motion may be separated. The combustion chamber and piston region may be also decomposed or separated into smaller parts. Then each part can be meshed accordingly for the solver setup. Any errors at this stage can lead to failures downstream during the solution process.

The run times for solver runs can be fairly long since the motion is typically resolved with small time steps (approximately 0.25 crank angle) to get accurate results and the simulation is run for two or three cycles to remove the initial transients. Finally, the large volume of transient data that results from the CFD solution needs to be postprocessed to obtain useful insight and information. Thus cold flow analysis would also benefit from design automation and process compression.

2.2.3. In-Cylinder Combustion Simulation

Combustion simulation involves simulation of the power stroke during the engine cycle, starting from closing of valves to the end of the compression stroke. Since the valves are closed or in the process of closing, the combustion chamber is the chief flow domain, and the piston the sole moving part. These simulations are also known as "in-cylinder combustion" and though multi-dimensional, are less complicated geometrically than a port flow simulation. In addition, if the geometry is rotationally symmetric and has a single feature like a very high pressure spray that dominates the flow in the calculation, the entire domain can be modeled as a sector to speed up the calculation.

Typically, the initial flow field at this stage is obtained from

- a cold flow simulation if the full geometry is used
- patching-in based on a cold flow analysis
- running the piston without combustion to obtain charge compression

As with cold flow, a moving deforming mesh model is used for the piston motion. Geometric decomposition is not required here, since only the piston motion is included in the simulation. Hence, in-cylinder combustion simulations typically do not include the modeling of the fluid dynamics in the valve port region and their effect on combustion.

Models are used to account for the fuel spray, combustion and pollutant formation. For direct injection engines, the fuel spray from the tip of the nozzle injector is introduced at the specific crank angle and duration using a spray model. For port fueled engines, it is assumed that the combustion charge is well mixed. A chemical mechanism describing the reaction of vapor fuel with air is used to describe combustion, and models for turbulence-chemistry interaction are specified. Sub-models for NOx and soot formation are used to calculate pollutant formation, which can be coupled with the combustion calculation or calculated as a postprocessing step.

With in-cylinder combustion, the main challenge lies in the physics for spray modeling and combustion. The spray is composed of a column of liquid entering the domain at high speed which subsequently breaks into droplets due to aerodynamic forces. These droplets can coalesce into larger droplets or break into even smaller droplets, all while exchanging mass with the surrounding gases. Sub-models for coalescence and breakup, as well as heat and mass transfer, are used to capture spray dynamics. The CFD mesh has to be sufficiently resolved to capture the coupling between the liquid droplets and the gases in the cylinder accurately. If liquid spray impinges upon the cylinder walls, it is possible to form a thin liquid film which undergoes its own processes of movement and vaporization and requires a separate treatment.

To calculate combustion, detailed chemical mechanisms for pure fuels that constitute the components of diesel fuel and gasoline involve hundreds of species and thousands of reactions. These reactions are coupled with the fluid dynamics due to the similar time scales of fluid mechanical motions and chemical reactions. The energy release from combustion increases pressures and temperatures for the fluid flow, which affects the fluid motions inside the cylinder. A direct computation of this coupled interaction without submodels while including detailed chemistry is staggeringly expensive in terms of computation time and is impractical for complex geometries.

Reduced order mechanisms capture most of the essential chemistry in a narrower range of temperature and equivalence ratio, and are used along with a submodel for turbulence-chemistry interaction. One such model is the Probability Density Function (PDF) approach which allows an efficient computation of turbulence-chemistry interaction. Flame propagation is modeled using a progress variable based approach such as the Zimont model, which calculates the transient flame front speed and location. These approaches allow computation of the combustion process on large meshes in complex geometries with a reasonable computational power.

Simplified mechanisms are used to compute the NOx formation due to

- high temperatures (thermal NOx)

- nitrogen in the fuel (fuel NOx)
- fuel reactions in the flame front (prompt NOx)
- sulfur oxides in the fuel (SOx)
- soot formation

Since these pollutants are generally a very small percentage of the total mass in the cylinder, these calculations can be decoupled from the calculation of the main energy release. In some cases, this is done as a postprocessing operation at the end of the simulation, but it is more accurate to include the pollutant formation in the simulation, especially for the pollutants arising from incomplete combustion that oxidize later in the cycle. In terms of automation and process compression, the problem setup at the solver stage can benefit from automation.

2.2.4. Full Cycle Simulations

As the name indicates, full cycle simulations essentially involve all the elements from cold flow analysis and in-cylinder combustion to complete simulations of the entire engine cycle. Thus this type of simulation is a transient computation of turbulent airflow, spray and combustion, and exhaust with moving valves and pistons. The initial flow field is obtained from a cold flow simulation or by running the engine without combustion for a cycle before turning on spray and combustion.

The advantage of full cycle simulations is that they provide the full picture of engine performance, including intake and exhaust valve fluid dynamics, mixing, turbulence production, spray, combustion and flame propagation, and pollutant formation. However, they are extremely complex to set up and expensive to run.

The geometry preparation can include geometry from the throttle body, ports, valves, combustion chamber, cylinder and the piston, making it difficult to perform cleanup, decomposition and meshing. The solver setup has to include moving mesh, airflow, turbulence, spray, turbulence-chemistry and flame propagation, and pollutant formation.

Here, the need is for process compression and automation all the way from geometry to postprocessing to reduce the time needed for problem setup and postprocessing. In addition, accurate and efficient models for chemistry, spray and combustion, as well as efficient solver techniques, are required to get the solution in the shortest time possible.

2.3. The IC Engine Analysis System: Process Compression in the Ansys Workbench

As demonstrated in the previous section, IC Engine simulations require process compression tools and automation to reduce problem setup time, automate solution runs, and postprocessing of large data sets. In the past, geometry and meshing, solution, and postprocessing were performed in different software running independently, with no interaction between them. This meant that each simulation had to be set up completely from the beginning, even when simple design changes were made. With a complex problem setup, any simple user error at any stage has the potential to derail the entire simulation. Thus the previous process is inherently time consuming and error prone.

Process compression and automation can only be accomplished in an integrated environment where the software at each step is aware of the overall goals of the simulation and shares a common problem description. Ansys Workbench provides an ideal integrated environment with powerful tools for geometry, meshing, CFD solvers, and postprocessing available on a common platform.

In Ansys Workbench,

- Bidirectional CAD connectivity ensures that design changes from CAD are automatically propagated into the simulation.
- The geometry tool (Design Modeler) can be linked to the meshing tool (Ansys Meshing or Forte Sector Mesh Generator).
- Ansys Meshing in turn can be linked to Ansys Fluent or Forte and Forte Sector Mesh Generator is linked to Forte.
- The results can be automatically sent to CFD-Post, a postprocessing tool.
- The data generated at each stage is stored in an organized structure and can be easily exchanged between different tools.

All of these tools can be linked together in "systems" in Ansys Workbench and provide a built-in pathway for simulation automation. In addition, each tool has built-in technological capabilities for creating process compression tools to automate repeated tasks, such as geometry decomposition and cleanup, meshing, solution setup and solver runs; and postprocessing. Thus Ansys Workbench has tremendous potential as a platform for process compression and solution automation.

The IC Engine Analysis System exploits these capabilities to create process compression for performing IC Engine simulations.

1. An **Input Manager** allows you to specify input parameters related to DesignModeler at the first step, with minimum possible information. This information is used to perform automatic decomposition and animation of engine motion in DesignModeler.
2. The geometry model is sent to Ansys Meshing or Forte Forte Sector Mesh Generator, where the meshing parameters are setup automatically with user input and the mesh generated.
3. The mesh is sent to Ansys Fluent or Forte, which performs automatic CFD problem setup and runs the calculation.
4. Once the results are ready, the solution data is automatically analyzed in CFD-Post and a report generated.

You provide minimal input at each stage and have the ability to modify the inputs. Thus the IC Engine Analysis System allows engine designers to rapidly progress from geometry to problem setup in a very short time with attenuated scope for error. In the next few chapters, the essential elements of the system and their working will be described.

Chapter 3: Getting Started With ICE

This chapter provides information about using ICE within Ansys Workbench.

Some basic information about using Workbench is provided here, but the majority of the information about using Workbench can be found in the Workbench on-line documentation (Workbench User's Guide).

3.1. Introduction to Workbench

Ansys Workbench combines access to Ansys applications with utilities that manage the product workflow.

This chapter provides some basic instructions for using ICE in Workbench.

Applications that can be accessed from Workbench include: Ansys DesignModeler (for geometry creation); Ansys Meshing (for mesh generation); Ansys Fluent (for setting up and solving computational fluid dynamics (CFD) simulations); Ansys Ansys Forte (an additional solver system); Ansys CFD-Post (for postprocessing the results). In Workbench, a project is composed of a group of systems. The project is driven by a schematic workflow that manages the connections between the systems. From the schematic, you can interact with workspaces that are native to Workbench, such as Design Exploration (parameters and design points), and you can launch applications that are data-integrated with Workbench. Data-integrated applications have separate interfaces, but their data is part of the Workbench project and is automatically saved and shared with other applications as needed. This makes the process of creating and running a CFD simulation more streamlined and efficient.

In addition, Workbench also allows you to copy systems in order to efficiently perform and compare multiple similar analyses. Workbench also provides parametric modeling capabilities in conjunction with optimization techniques to allow you to efficiently investigate the effects of input parameters on selected output parameters.

3.2. The Workbench Graphical User Interface

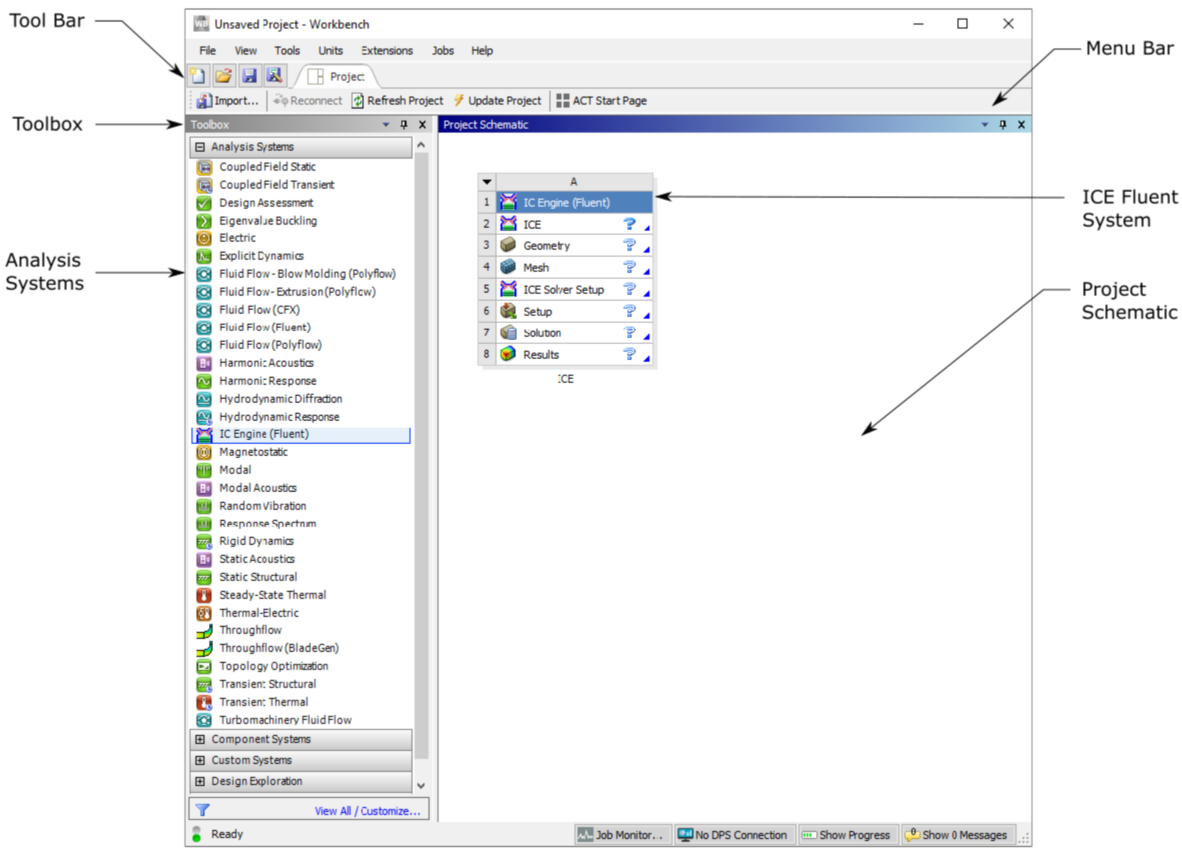
The Workbench graphical user interface consists of the Toolbox, the Project Schematic, the Toolbar, and the Menu bar. Depending on the analysis type and/or application or workspace, you may also see other windows, tables, charts, and so on. The most common way to begin work in Workbench is to drag an item, such as a component system (application) or an analysis system, from the Toolbox to the Project Schematic, or to double-click on an item to initiate the default action. You will view your component and/or analysis systems – the pieces that make up your analysis – in the Project Schematic, including all connections between the systems. The individual applications in which you work will display separately from the Workbench graphical interface, but the actions you take in the applications will be reflected in the Project Schematic.

Full details of how to use Workbench are provided in the Workbench User's Guide.

Important:

Note that IC Engine (Fluent) can be accessed in Workbench as an analysis system.

Figure 3.1: The Workbench Graphical User Interface



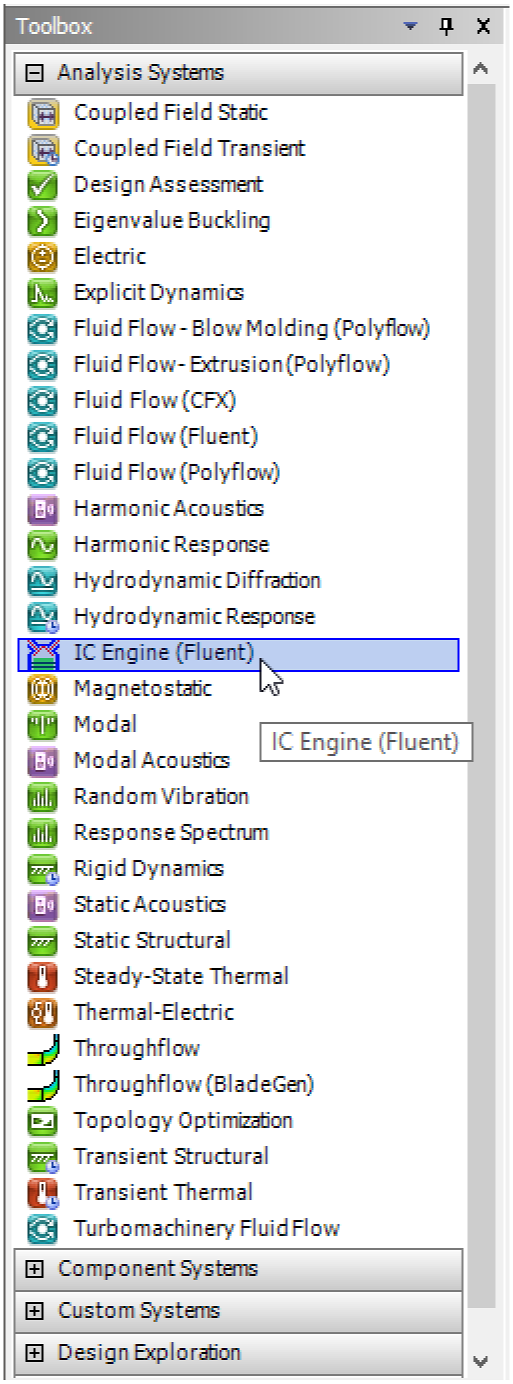
3.3. Creating an IC Engine Analysis System

The **IC Engine** system is located in the **Analysis Systems** toolbox of the Workbench user interface. You create an **IC Engine** analysis system in Workbench by double-clicking **IC Engine (Fluent)** under **Analysis Systems** in the **Toolbox**.

Important:

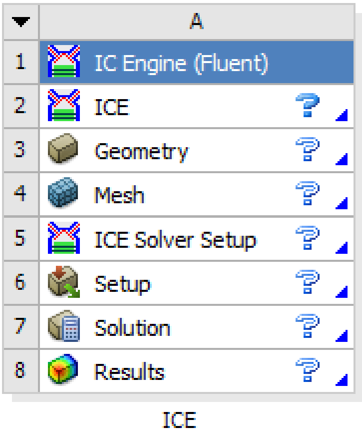
You can also create an **IC Engine** analysis system by left-clicking on **IC Engine (Fluent)** under **Analysis Systems** in the Toolbox, and then dragging it onto the Project Schematic.

Figure 3.2: Selecting the IC Engine Analysis System in Workbench



The new **IC Engine** analysis system appears in the **Project Schematic** as a box containing several cells. Each cell corresponds to a typical task you would perform to complete a CFD analysis. The following cells are available in an **ICE** analysis system.

Figure 3.3: An ICE Analysis System



ICE

Opens the **Properties** panel where you define the parameters and controls required for the analysis. It controls the behavior of the downstream components. For details on configuring the properties, refer to Setting up an IC Engine Analysis System for **IC Engine** (p. 141).

Geometry

Opens DesignModeler where you define the geometrical constraints of your analysis or import an existing geometry. You will prepare the geometry for decomposition (divide the geometry into smaller volumes before meshing). For details, refer to Cold Flow Simulation: Preparing the Geometry (p. 149), Port Flow Simulation: Preparing the Geometry in **IC Engine** (p. 271), or Combustion Simulation: Preparing the Geometry in **IC Engine** (p. 341).

Note:

To open DesignModeler from the **Geometry** cell, you should set the **Preferred Geometry Editor** to **DesignModeler** in the **Geometry Import** section in the **Options** dialog box. The **Options** dialog box can be opened from the **Tools** → **Options...**menu.

Mesh

Opens Ansys Meshing and loads the current geometry defined by the **Geometry** cell. See Cold Flow Simulation: Meshing (p. 185), Port Flow Simulation: Meshing in **IC Engine** (p. 285), or Combustion Simulation: Meshing in **IC Engine** (p. 365).

ICE Solver Setup

Allows you to change or add solver settings and also select crank angles for KeyGrid simulation. See Cold Flow Simulation: Setting Up the Analysis (p. 217), Port Flow Simulation: Setting Up the Analysis in **IC Engine** (p. 301), or Combustion Simulation: Setting Up the Analysis in **IC Engine** (p. 393).

Setup

Allows you to define the physical models, material properties, boundaries and process conditions, and solver settings for the ICE analysis in Ansys Fluent. See Cold Flow Simulation: Setting Up the

Analysis (p. 217), Port Flow Simulation: Setting Up the Analysis in **IC Engine** (p. 301), or Combustion Simulation: Setting Up the Analysis in **IC Engine** (p. 393).

Solution

Allows you to calculate a solution using Ansys Fluent. Right-click the **Solution** cell and select **Update** to run the Fluent solver using the current data file for input. If you want Ansys Fluent to run in background mode rather than the default foreground mode, before you select **Update** you must first right-click the **Solution** cell, select **Properties**, and select **Run in Background** for the **Update Option** in the **Properties of Schematic A6: Solution** pane that opens. For details on cold flow solutions, see Run Calculation (p. 265). For port flow solution details refer to Run Calculation (p. 336).

Note that you can open Ansys Fluent using the context menu (by right-clicking on the **Solution** cell), in order to postprocess the results of your ICE simulation.

Results

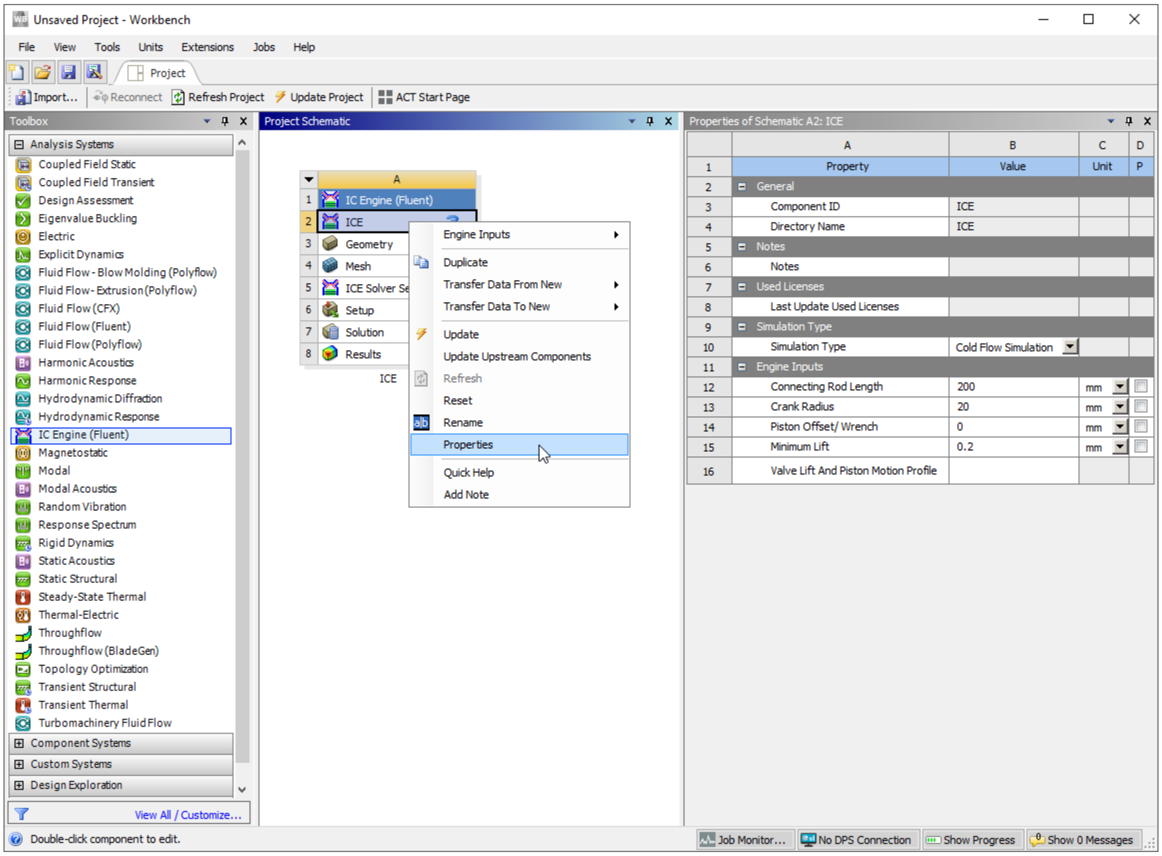
Displays and analyzes the results of the CFD analysis in Ansys CFD-Post. See Working with the Simulation Results (p. 485).

Note:

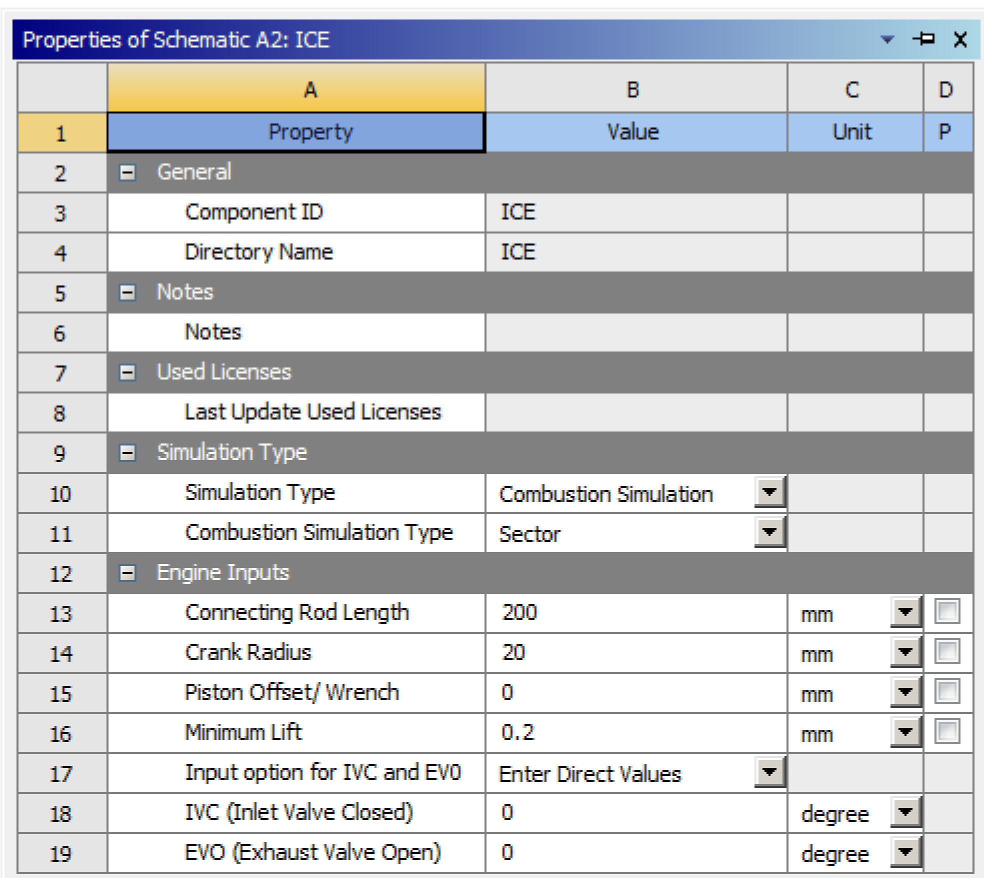
Workbench provides visual indications of a cell's state via icons on the right side of each cell. Brief descriptions of the each possible state are provided below. For more information about cell states, see Workbench User's Guide.

3.4. Setting up an IC Engine Analysis System for IC Engine

After you double-click or drag **IC Engine (Fluent)** under **Analysis Systems** in the **Toolbox**, the new **IC Engine (Fluent)** analysis system appears in the Project Schematic. Before importing the geometry, you have to define the parameters for your analysis. You can define the parameters in the **Properties** pane. To view properties, right-click **ICE** and select **Properties** from the context menu. You can also view the properties by enabling **Properties** from the **View** menu list.



Properties pane displays detailed information as shown in the figure below.



Properties of Schematic A2: ICE

	A	B	C	D
1	Property	Value	Unit	P
2	⊟ General			
3	Component ID	ICE		
4	Directory Name	ICE		
5	⊟ Notes			
6	Notes			
7	⊟ Used Licenses			
8	Last Update Used Licenses			
9	⊟ Simulation Type			
10	Simulation Type	Combustion Simulation		
11	Combustion Simulation Type	Sector		
12	⊟ Engine Inputs			
13	Connecting Rod Length	200	mm	☐
14	Crank Radius	20	mm	☐
15	Piston Offset/ Wrench	0	mm	☐
16	Minimum Lift	0.2	mm	☐
17	Input option for IVC and EVO	Enter Direct Values		
18	IVC (Inlet Valve Closed)	0	degree	
19	EVO (Exhaust Valve Open)	0	degree	

In the **Properties** table there are several sections, the most important of which are as follows.

General

The properties in this section are

Component ID

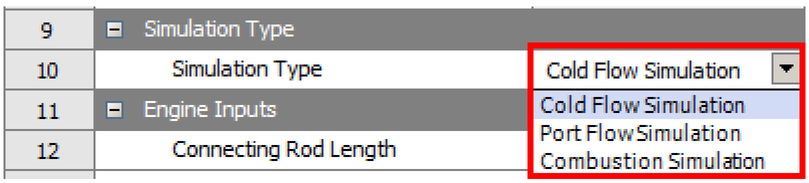
It is ICE.

Directory Name

It is the directory where temporary files are stored. It is named as ICE.

Simulation Type

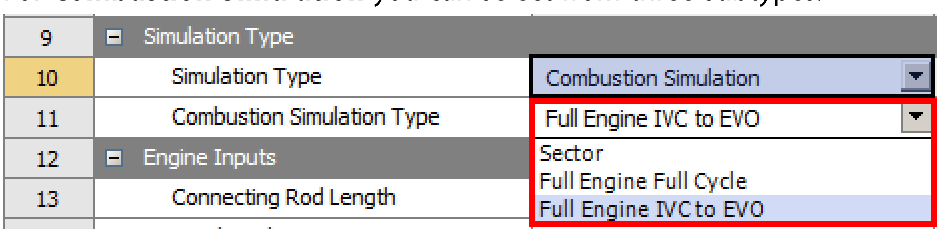
You can select the simulation type from the drop-down list. At present there are three options.



- **Cold Flow Simulation**
- **Port Flow Simulation**
- **Combustion Simulation**

Depending on the selection of the simulation type, further options change.

For **Combustion Simulation** you can select from three subtypes.



- **Sector**
- **Full Engine Full Cycle**
- **Full Engine IVC to EVO**

Engine Inputs

In this section you provide specifications of the engine model. This section is available only for **Cold Flow Simulation** and **Combustion Simulation**.

Connecting Rod Length

It is set to **200mm** by default. You can enter the connecting rod length of your engine here.

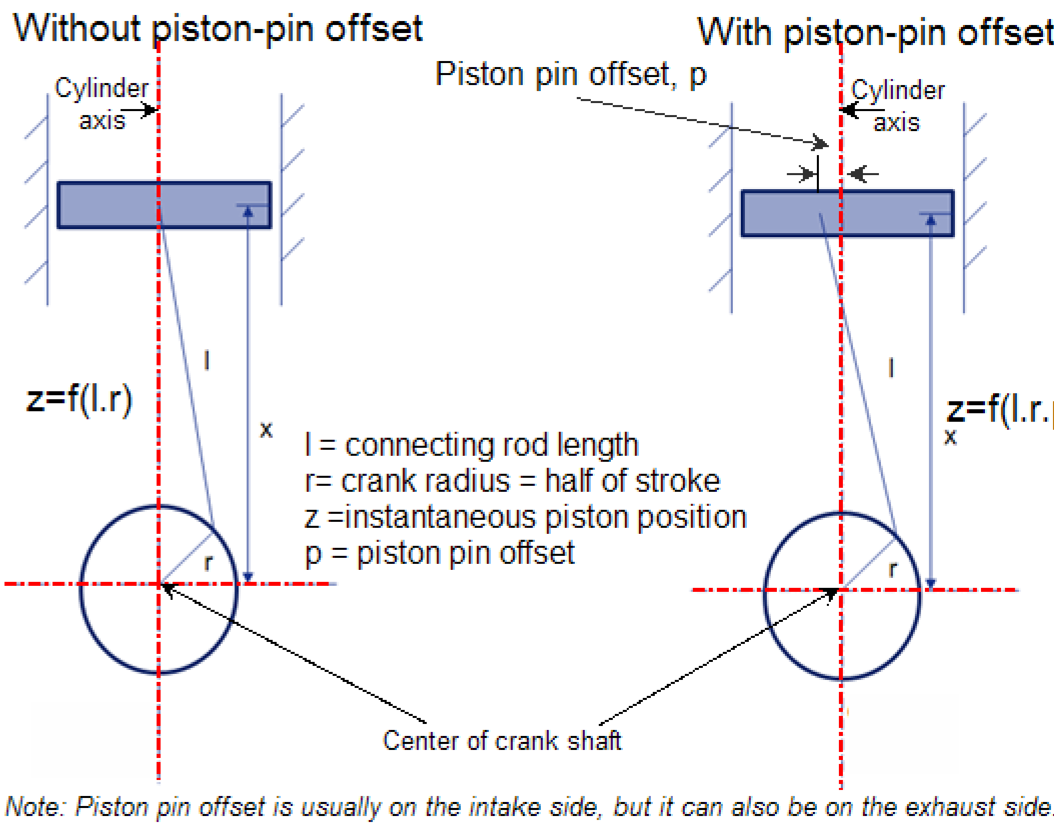
Crank Radius

It is set to **20mm** by default. You can enter the crank radius of your engine here.

Piston Offset/Wrench

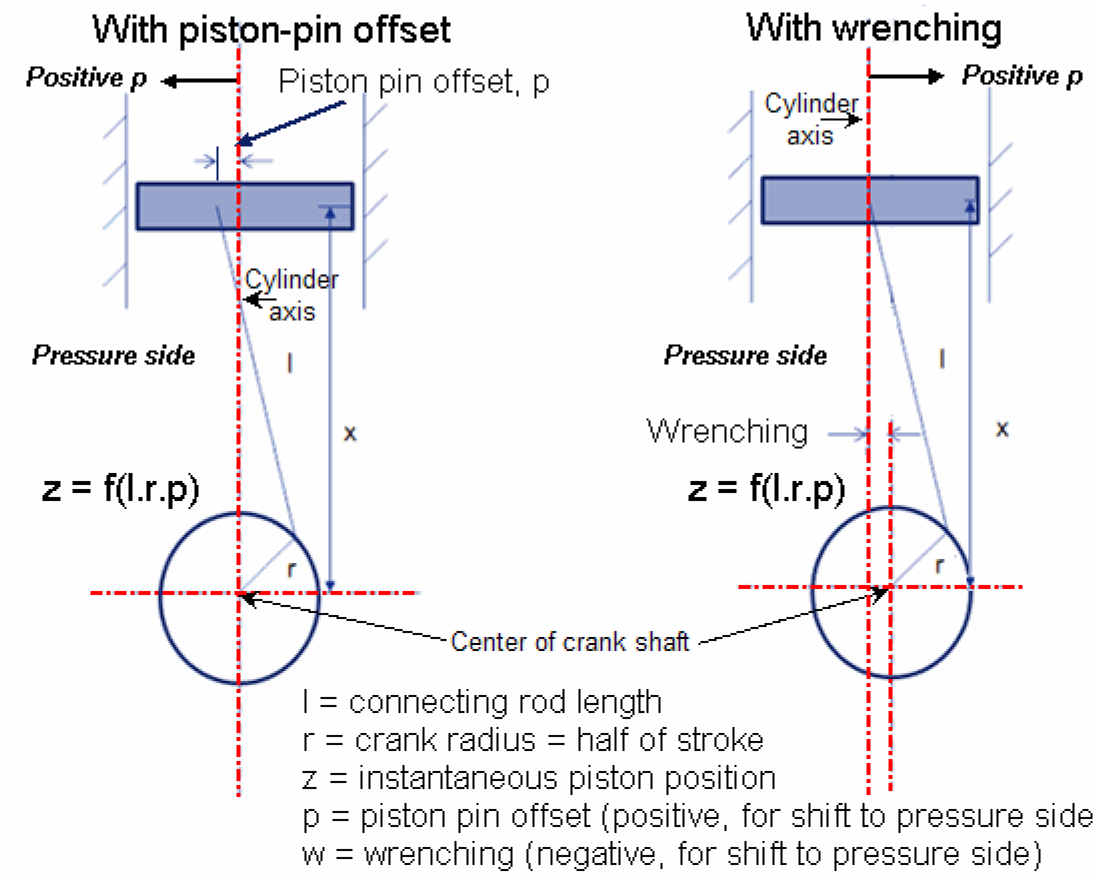
It is set to a default value of **0**.

The piston pin offset is generally used to reduce the stress on the reciprocating parts. It enables these parts to be lighter, which increases the efficiency, and decreases the power loss in the engine. It also results in higher rpm. An additional result of the piston pin offset is reduced piston slap, due to the gradual shift between major and minor thrust.



The piston pin offset is an off-centered mount of the connecting rod; piston wrenching is a nonalignment of the cylinder axis and the crank shaft axis. Both the piston offset and wrench have the same effect.

You can enter a positive or a negative value for **Piston Offset/Wrench**. This will depend upon the offset:



Minimum Lift

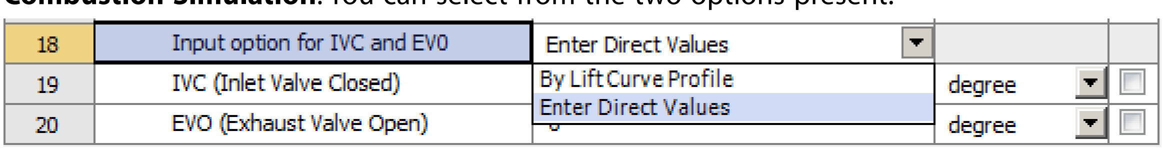
It is set to a default value of **0.2mm**. You can enter your value here. For details, see the explanation on minimum valve lift (p. ?).

Note:

If you enable **P** next to the above properties, you can create a new parameter. Also you can set the units as required for the parameters.

Input Option for IVC and EVO

This option is present only for the **Sector** and **Full Engine IVC to EVO** subtypes of **Combustion Simulation**. You can select from the two options present.



18	Input option for IVC and EVO	Enter Direct Values		
19	IVC (Inlet Valve Closed)	By Lift Curve Profile	degree	
20	EVO (Exhaust Valve Open)	Enter Direct Values	degree	

- **By Lift Curve Profile**
- **Enter Direct Values**

Valve Lift and Piston Motion Profile

This option is present for **Cold Flow Simulation**, and when you select **By Lift Curve Profile** for **Combustion Simulation**. Here you have to provide the path to the profile file. This file essentially contains the valve lift values at different crank angles. It can also include the piston motion profile. You can use this profile file to animate the movement of the valves. It is also used for simulating the valve motion.

The format of the profile file is fairly simple. The file can contain an arbitrary number of profiles. Profile names must have all lowercase letters. Here `profile-name` is the name given to the type of valve. For example, it can be `invalve1` or `exvalve2`. The types of profile are `point` and `transient`. The mandatory `field-names` are `angle` and `lift`.

Parentheses are used to delimit profiles and the fields within the profiles. Any combination of tabs, spaces, and newlines can be used to separate elements.

Important:

In the general format description below, "`...`" indicates a continuation of the list.

```
((profile-name point m n)
  (angle  a11 a12 ... a1n
          a21 a22 ... a2n
          .
          .
          .
           am1 am2 ... amn)
(lift   l11  l12 ... l1n
        l21  l22 ... l2n
        .
        .
        .
        lm1 lm2 ... lmn))
```

IVC (Inlet Valve Closed)

Here you need to enter the IVC or inlet valve closing angle of your engine.

EVO (Exhaust Valve Open)

Here you need to enter the EVO or exhaust valve opening angle of your engine.

3.5. Understanding Cell States with ICE in Workbench

Workbench integrates multiple data-integrated (for example, IC Engine) and native applications into a single, seamless project flow, where individual cells can obtain data from and provide data to other cells. Workbench provides visual indications of a cell's state via icons on the right side of each cell. Brief descriptions of the each possible state are provided below. For more information about cell states, see the Workbench on-line help (Workbench User's Guide).

- **Unfulfilled** ( ) indicates that required upstream data does not exist. For example, when you first create a new **IC Engine** analysis system, all cells downstream of the **Geometry** cell appear as **Unfulfilled** because you have not yet specified a geometry for the system.

- **Refresh Required** () indicates that upstream data has changed since the last refresh or update. For example, after you assign a geometry to the **Geometry** cell in your new **IC Engine** analysis system, the **Mesh** cell appears as **Refresh Required** since the geometry data has not yet been passed from the **Geometry** cell to the **Mesh** cell.

- **Attention Required** () indicates that the current upstream data has been passed to the cell, however, you must take some action to proceed. For example, after you launch IC Engine and then double click **Geometry** to open DesignModeler you have to load the geometry. However, if you do not load the geometry and close Design Modeler, then the **Geometry** cell appears as **Attention Required** because additional data is required before you can build or create a mesh.

- **Update Required** () indicates that local data has changed and the output of the cell needs to be regenerated. For example, after you launch DesignModeler from the **Geometry** cell in an **IC Engine** analysis system that has a valid geometry, the **Geometry** cell appears as **Update Required** until you perform decomposition.

- **Up to Date** ( ) indicates that an update has been performed on the cell and no failures have occurred (or an interactive calculation has been completed successfully). For example, after you perform decomposition, the **Geometry** cell appears as **Up to Date**.

- **Interrupted** ( ) indicates that you have interrupted an update (or canceled an interactive calculation that is in progress). For example, if you select the **Cancel** button in Ansys Fluent while it is iterating, Ansys Fluent completes the current iteration and then the **Solution** cell appears as **Interrupted**.

- **Input Changes Pending** ( ) indicates that the cell is locally up-to-date, but may change when next updated as a result of changes made to upstream cells. For example, if you change the **Mesh** in an **Up-to-Date IC Engine** analysis system, the **Setup** cell appears as **Refresh Required**, and the **Solution** and **Results** cells appear as **Input Changes Pending**.

- **Pending** () indicates that a batch or asynchronous solution is in progress. When a cell enters the **Pending** state, you can interact with the project to exit Workbench or work with other parts of the project. If you make changes to the project that are upstream of the updating cell, then the cell will not be in an up-to-date state when the solution completes.

If a particular action fails, Workbench provides a visual indication as well. Brief descriptions of the failure states are described below.

- Refresh Failed, Refresh Required () indicates that the last attempt to refresh cell input data failed, and so the cell needs to be refreshed.

- Update Failed, Update Required () indicates that the last attempt to update the cell and calculate output data failed, and so the cell needs to be updated.

- Update Failed, Attention Required () indicates that the last attempt to update the cell and calculate output data failed, and so the cell requires attention.

If an action results in a failure state, you can view any related error messages in the **Messages** pane by clicking the **Show Messages** button on the lower right portion of Workbench.

Chapter 4: Cold Flow Simulation: Preparing the Geometry

The simulation of the engine starts when you import an engine geometry. The imported geometry is divided into smaller volumes before meshing. This enables each volume to be meshed separately. Decomposition partitions a volume into sub-volumes and then the sub-volumes are meshed individually. Each volume will be meshed into hex or tet elements, depending upon the approach.

There are certain mesh topology requirements for valves and pistons. You should have pistons at TDC (top dead center) before the geometry is decomposed. With the piston at TDC, the volume is the smallest. In general it is more difficult to satisfy the mesh topology requirement at TDC, but this provides the advantage that the mesh will behave properly when the piston moves away from TDC. However, the simulation requires a minimum valve lift between the valve and valve seat so that layered cells can be placed at the region of minimum valve lift. This ensures that the gap between the valve and valve seat will not disappear. A non-conformal interface is used to completely shut the valve. Even though in theory an arbitrarily small minimum valve lift can be used, in reality a value of 0.05 mm to 0.5 mm has been successfully used to run simulations using Ansys Fluent.

This chapter provides instructions and information about preparing the IC engine geometry for simulation:

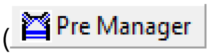
4.1. Repair Geometry Before Decomposition

You should check the geometry before decomposing so that geometrical problems can be avoided at initial level. Verify that:

- Pistons are at the TDC position.
- Valves are not extracted from the port volume.
- Each valve is correctly aligned and positioned above the valve seat.
- There are no extra bodies in the geometry other than the port, valves, or crevice.

For more information, refer to Geometry Check (p. 517).

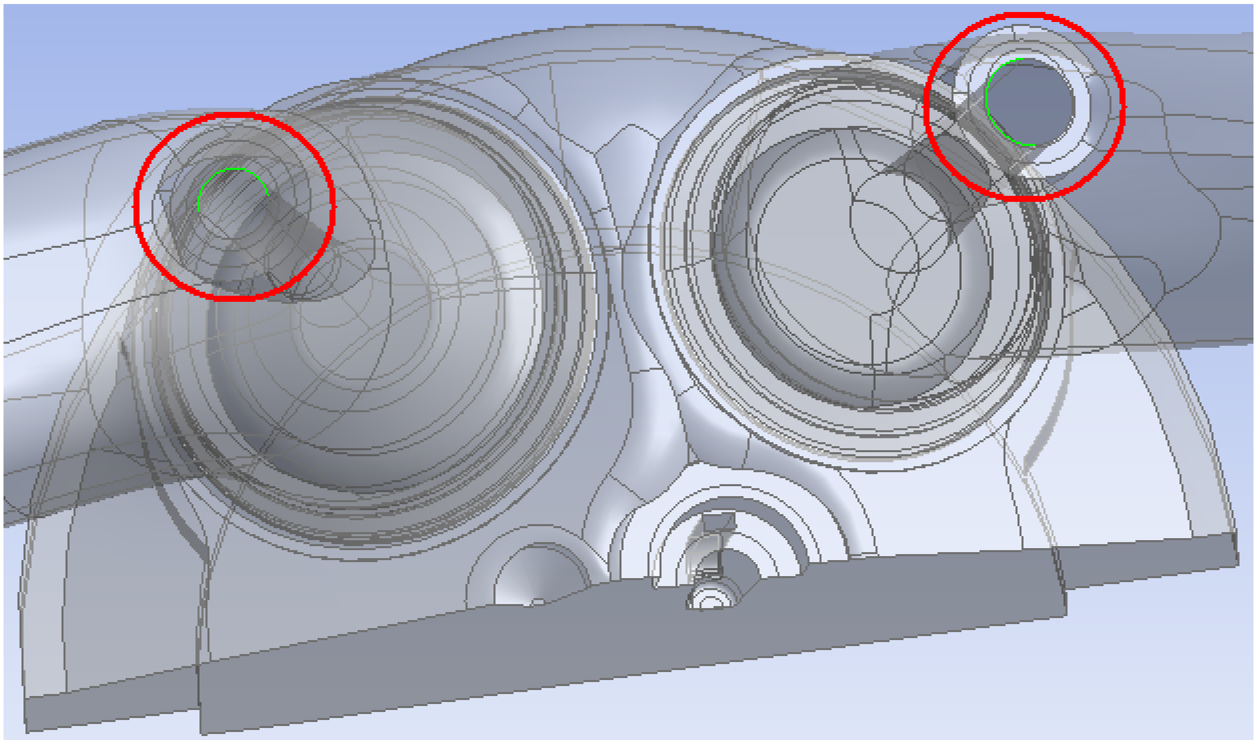
If the valves are extracted you need to create valves before decomposition.

1. Click **Pre Manager** (

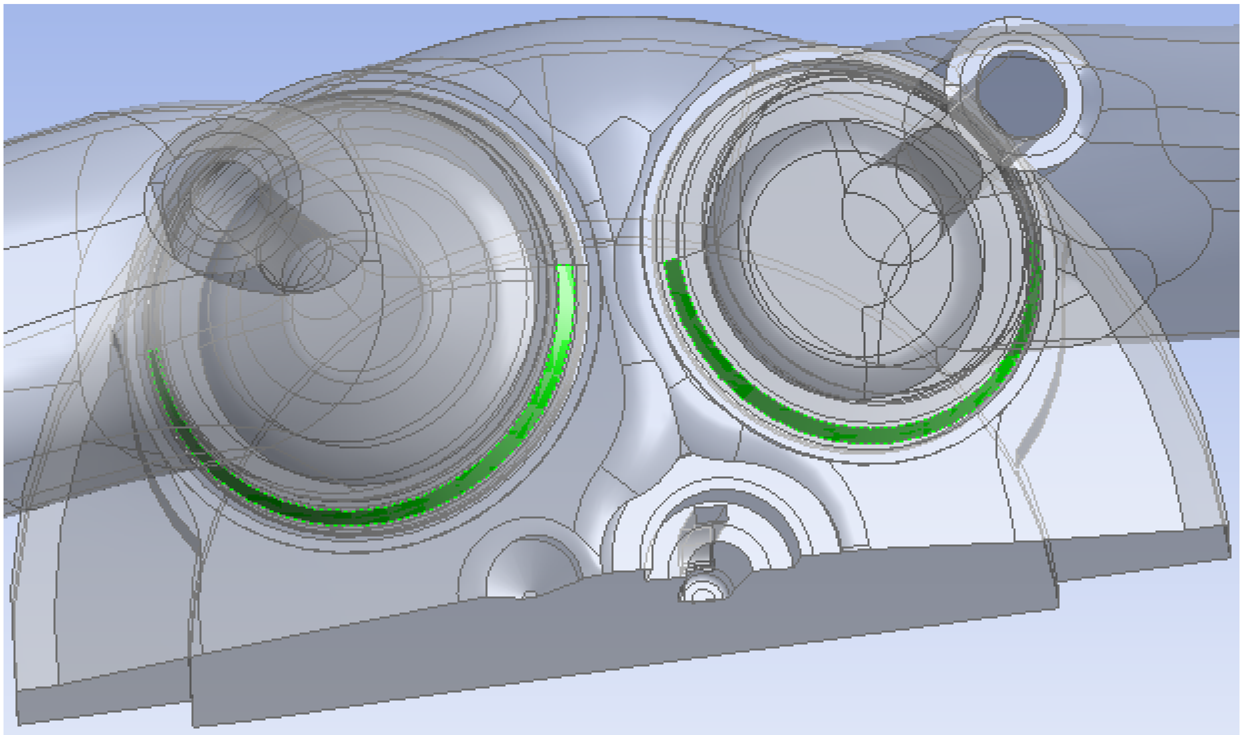
located in the **IC Engine** toolbar).

   a. In the **Details of ICPreManager1** select **Create Valve** from the **Pre Manager Operation** drop-down list.

   b. For the **Valve Hole Edges** select the edge of the hole where the valve should be present and click **Apply**.



   c. Select the faces of the valve seats for **Valve Seats** and click **Apply**.



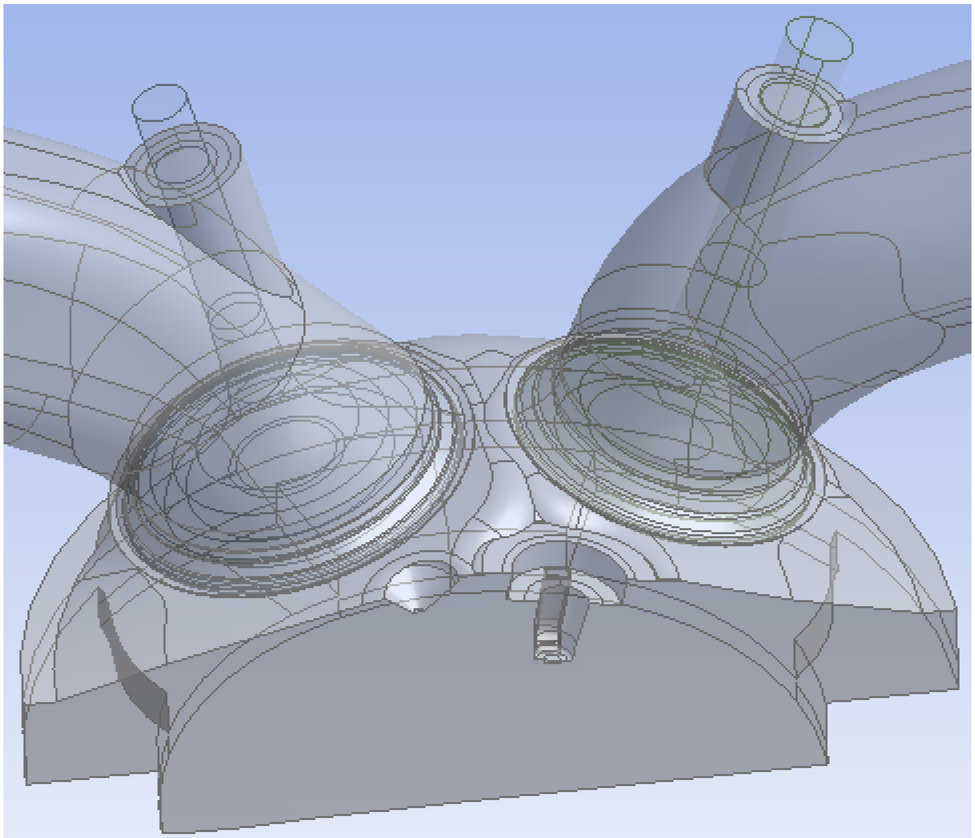
d. The **Valve Stem Extrusion Length** is automatically calculated and you can use the default value or change it if required.

Details View

Details of ICPreManager1	
Name	ICPreManager1
Pre Manager Operation	Create Valve
Valve Hole Edges	2 Edges
Valve Seats	2 Faces
Valve Stem Extrusion Lenght	0.0074918 m

2. Click **Generate** (Generate located in the Ansys DesignModeler toolbar).

3. Click **Pre Process** (Pre Process located in the Ansys DesignModeler toolbar) to complete the procedure.

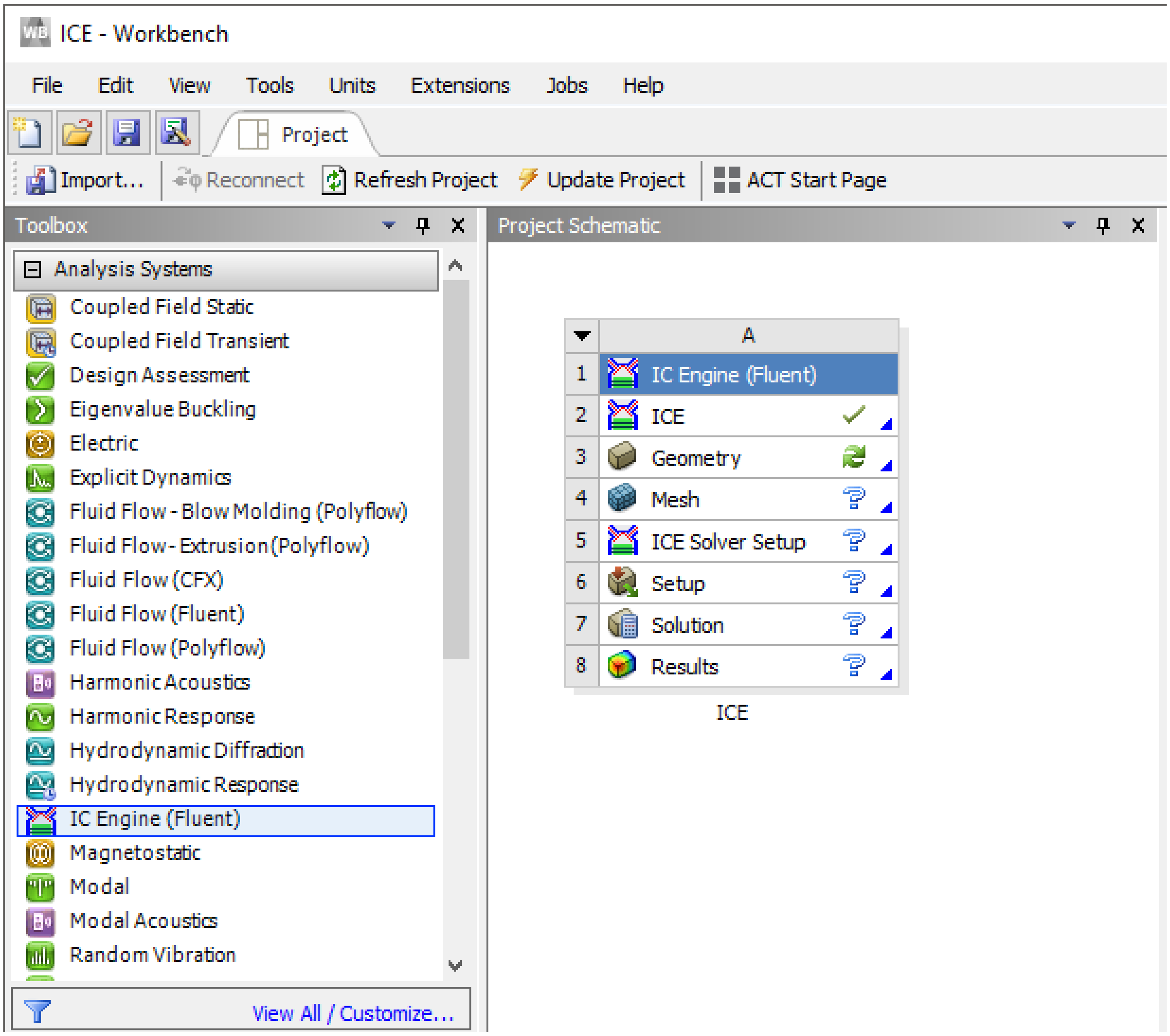


4.2. Geometry Decomposition for a Cold Flow Simulation for IC Engine

Before understanding how the geometry is decomposed into different sub-volumes, let us understand the process of decomposition in DesignModeler.

1. Start ICE DesignModeler.

   a. In Ansys Workbench, double-click or drag **IC Engine (Fluent)** from **Analysis Systems** to the **Project Schematic**.



b. Select **Cold Flow Simulation** from the **Simulation Type** drop-down list.

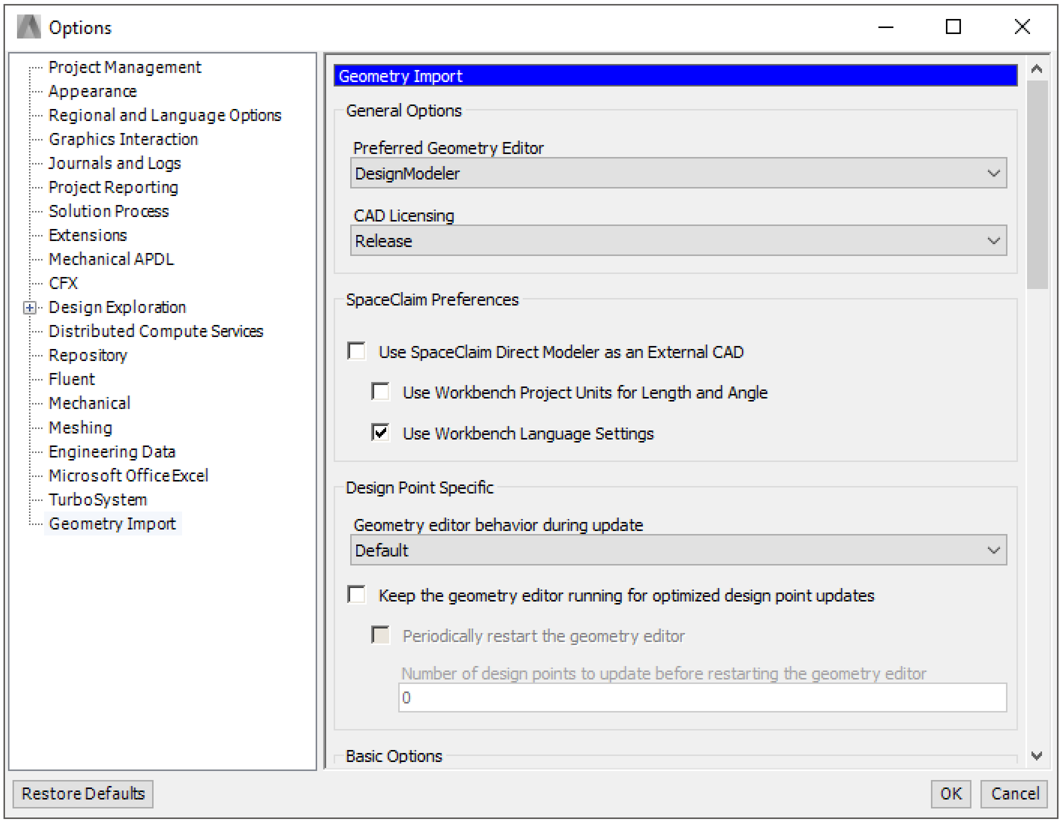
c. Enter the engine details under **Engine Inputs** in the **Properties** view.

d. Provide the path to the profile file for **Valve Lift And Piston Motion Profile**.

e. From **Geometry**, cell 3 in the **IC Engine (Fluent)** analysis system, open **ICE-Design-Modeler** application.

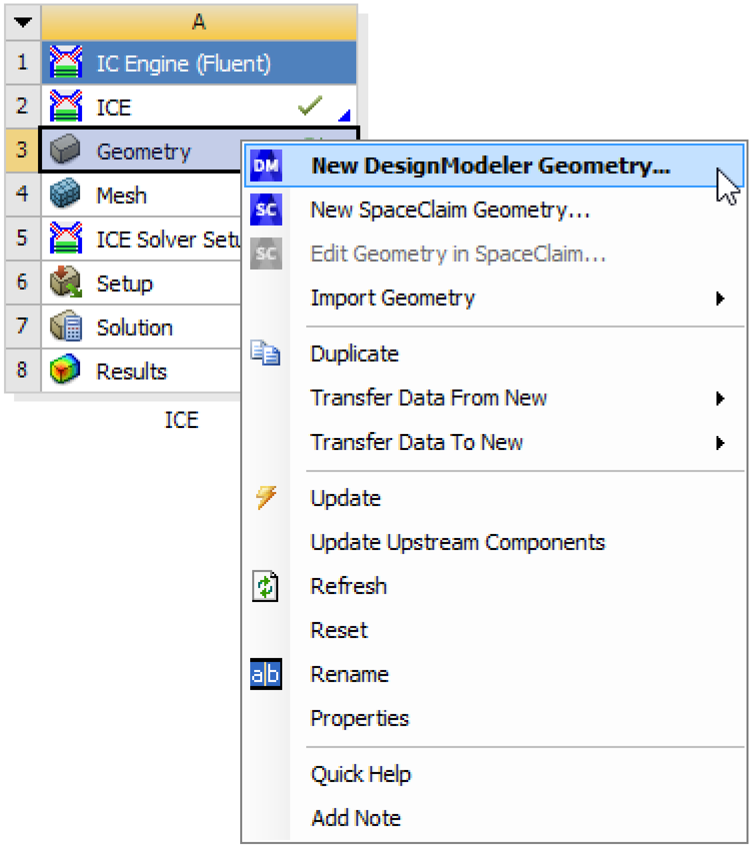
Note:

To open DesignModeler from the **Geometry** cell, you should set the **Preferred Geometry Editor** to **DesignModeler** in the **Geometry Import** section in the **Options** dialog box. The **Options** dialog box can be opened from the **Tools** → **Options...**menu.



Then you can double-click **Geometry**, cell 3, to open DesignModeler.

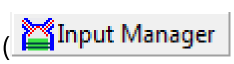
One more way to open DesignModeler is, to right-click on the **Geometry** cell, and select **New DesignModeler Geometry...** from the context menu.



2. Set the units, depending upon the geometry units, in ICE-DesignModeler.

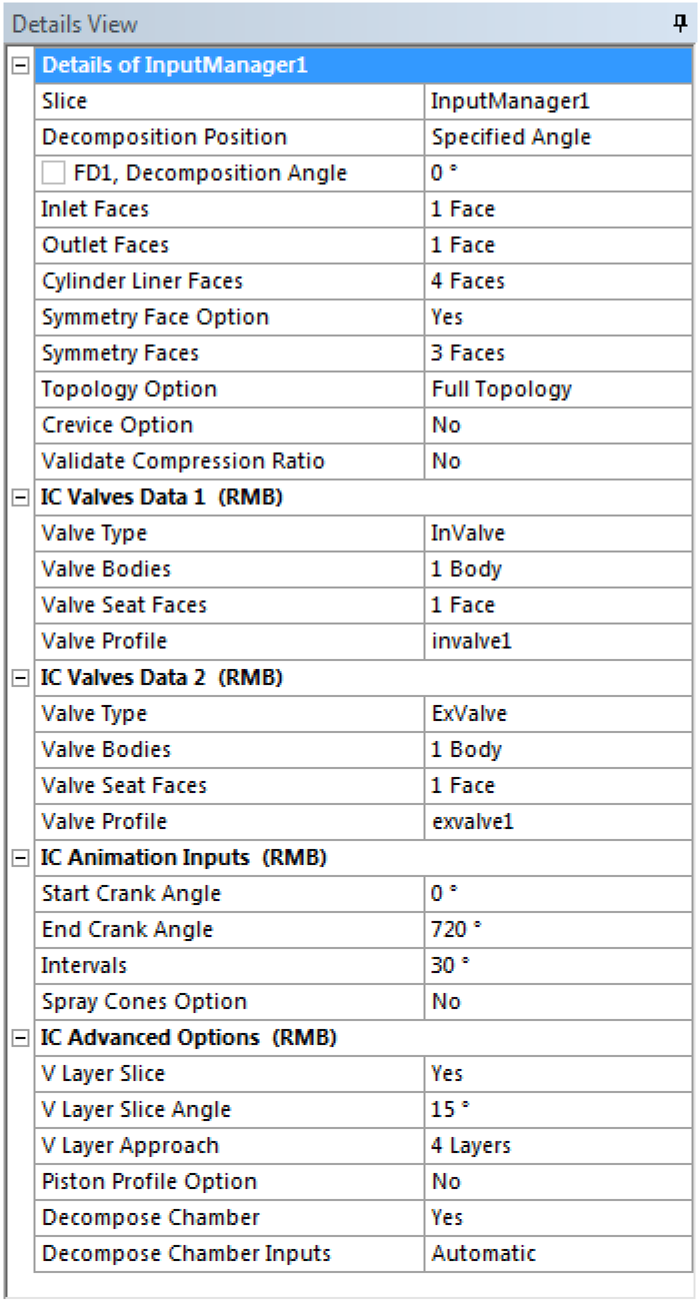
3. Load the geometry file.

   File → Load DesignModeler Database...

4. Set up the **Input Manager** by clicking **Input Manager** (

located in the **IC Engine** toolbar).

> **Note:**
>
> The **IC Engine** toolbar is displayed in the ICE DesignModeler only after installing IC Engine Analysis System.



Details View	
Details of InputManager1	
Slice	InputManager1
Decomposition Position	Specified Angle
FD1, Decomposition Angle	0 °
Inlet Faces	1 Face
Outlet Faces	1 Face
Cylinder Liner Faces	4 Faces
Symmetry Face Option	Yes
Symmetry Faces	3 Faces
Topology Option	Full Topology
Crevice Option	No
Validate Compression Ratio	No
IC Valves Data 1 (RMB)	
Valve Type	InValve
Valve Bodies	1 Body
Valve Seat Faces	1 Face
Valve Profile	invalve1
IC Valves Data 2 (RMB)	
Valve Type	ExValve
Valve Bodies	1 Body
Valve Seat Faces	1 Face
Valve Profile	exvalve1
IC Animation Inputs (RMB)	
Start Crank Angle	0 °
End Crank Angle	720 °
Intervals	30 °
Spray Cones Option	No
IC Advanced Options (RMB)	
V Layer Slice	Yes
V Layer Slice Angle	15 °
V Layer Approach	4 Layers
Piston Profile Option	No
Decompose Chamber	Yes
Decompose Chamber Inputs	Automatic

Details of InputManager

This section in the **Input Manager** dialog box takes the inputs of the engine to set it up for decomposition.

- For **Decomposition Position** you can select the angle at which you want the geometry to be decomposed. You can choose from:
  - **Specified Angle**: If you choose this option then you can specify the particular angle at which you want to decompose the geometry.
  - You can set the angle at which you want the geometry to be decomposed by entering the value in **Decomposition Angle**. It is set to **0** by default. When you want the

solver to start the simulation from a specific crank angle you can do so by changing the **Decomposition Angle**. If you enable **FD1** next to it, a new parameter will be created.

Note:

Setting **Decomposition Angle** to a value will move the piston and valves to the specified crank angle before decomposition. For details, see Moving the Piston to a Specified Crank Angle in **IC Engine** system (p. 183).

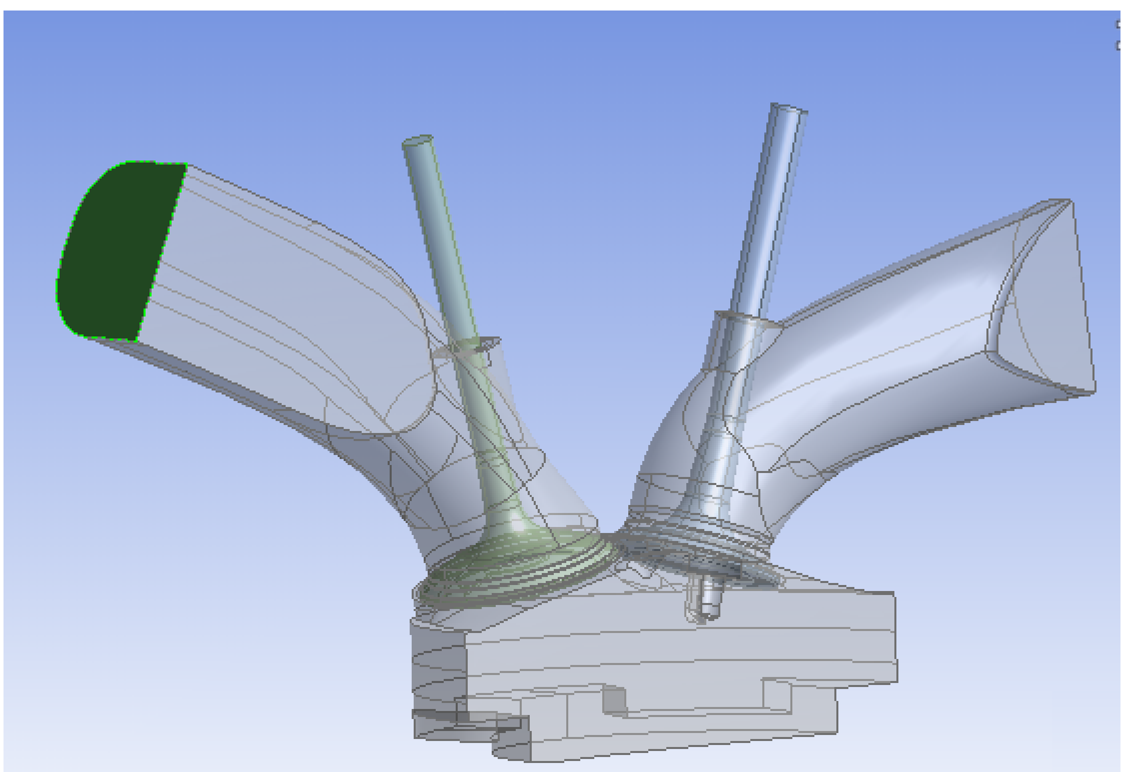
- **FTDC**: This is the firing top dead center angle.
- **IVC**: This is the inlet valve close angle.
- **IVO**: This is the inlet valve open angle.
- **EVC**: This is the exhaust valve close angle.
- **EVO**: This is the exhaust valve open angle.

• Click next to **Inlet Faces** in the **Details View** of **Input Manager**.

Confirm that **Selection Filters** are set to **Model Faces**.

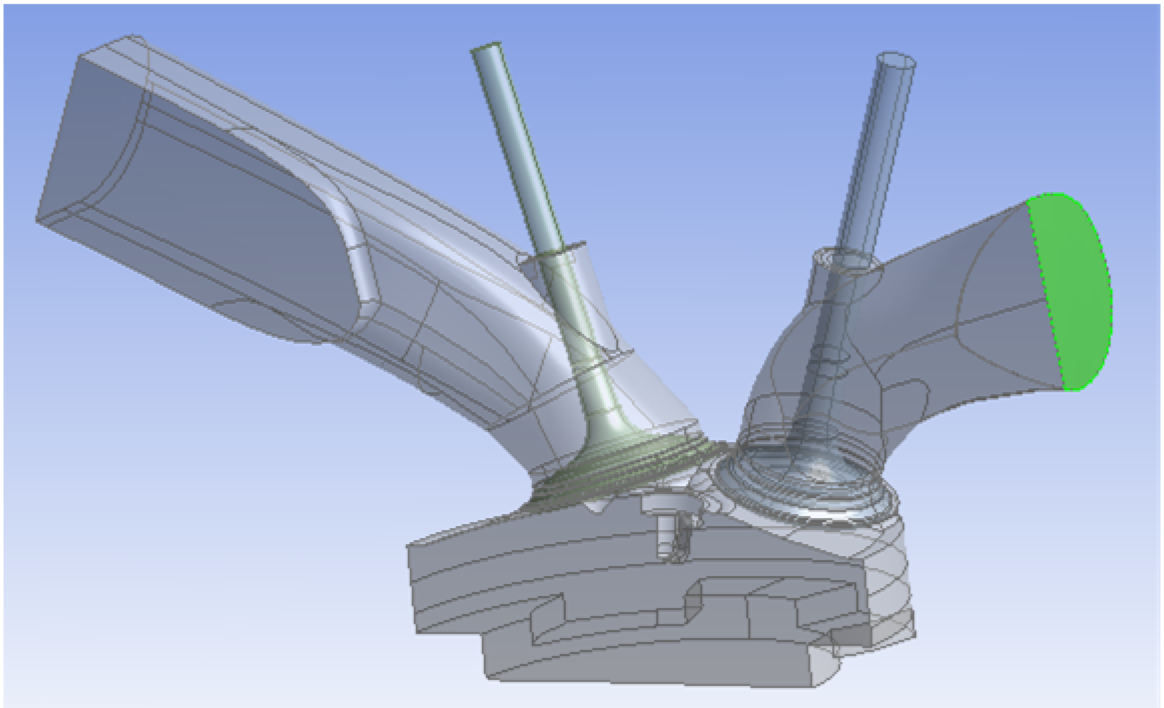
Select the inlet faces and click **Apply**.

Figure 4.1: Inlet Face Selection



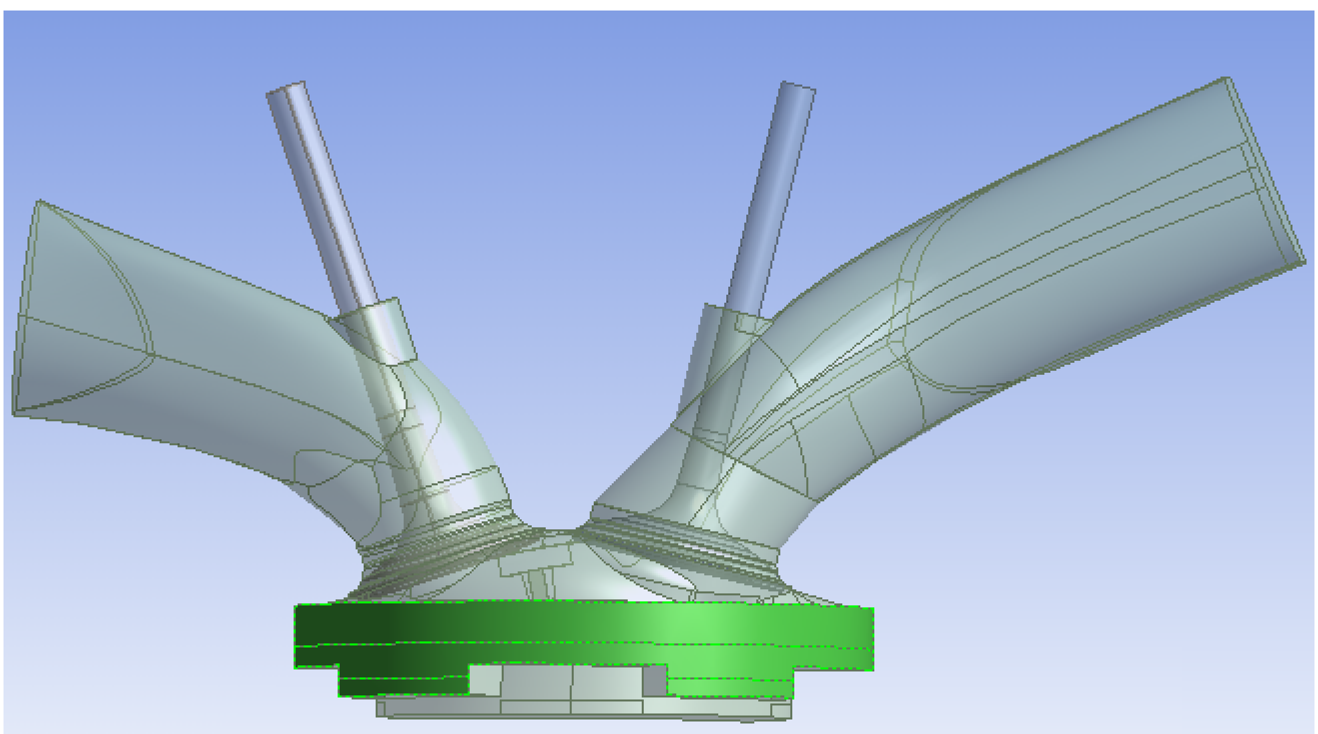
- Similarly, select the outlet faces for **Outlet Faces** and click **Apply**.

Figure 4.2: Outlet Face Selection



- Select all the faces of the engine cylinder for **Cylinder Liner Faces** and click **Apply**.

Figure 4.3: Cylindrical Liner Face Selection



The cylinder radius and the cylinder axis are displayed in the status bar at the bottom of the window.

Note:

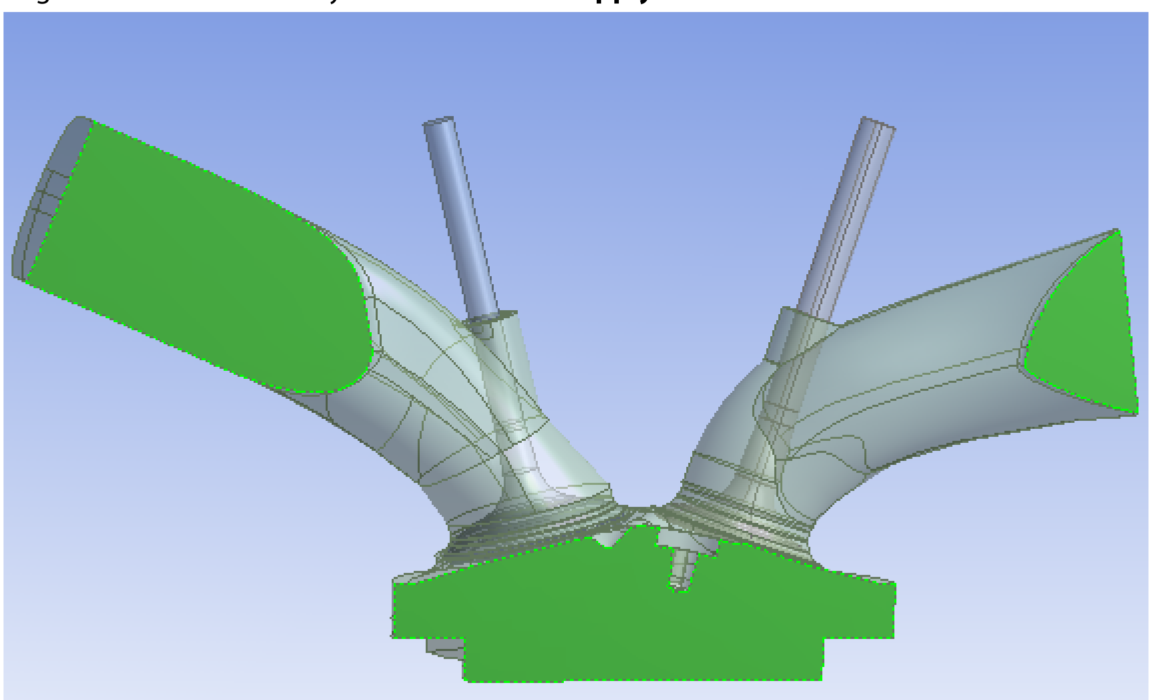
Ensure that all the faces of the cylinder body are selected. If the geometry has a crevice body, ensure that none of the crevice cylinder faces is chosen.

- After selecting the **Cylinder Liner Faces** the program will check for symmetry. It will accordingly set **Yes** or **No** for the **Symmetry Face Option**. If **Yes** is selected you will be required to give the symmetry faces.

Note:

You should select **Yes** for **Symmetry Face Option** if the geometry is symmetrical. Failure to do so will result in the failure of the simulation. Also ensure to select the faces for **Symmetry Faces**.

- Click next to **Symmetry Faces** in the **Details View** of **Input Manager**. Select the faces of the engine about which it is symmetric and click **Apply**.



Note:

If the geometry has a crevice body, select the symmetry faces belonging to the crevice body.

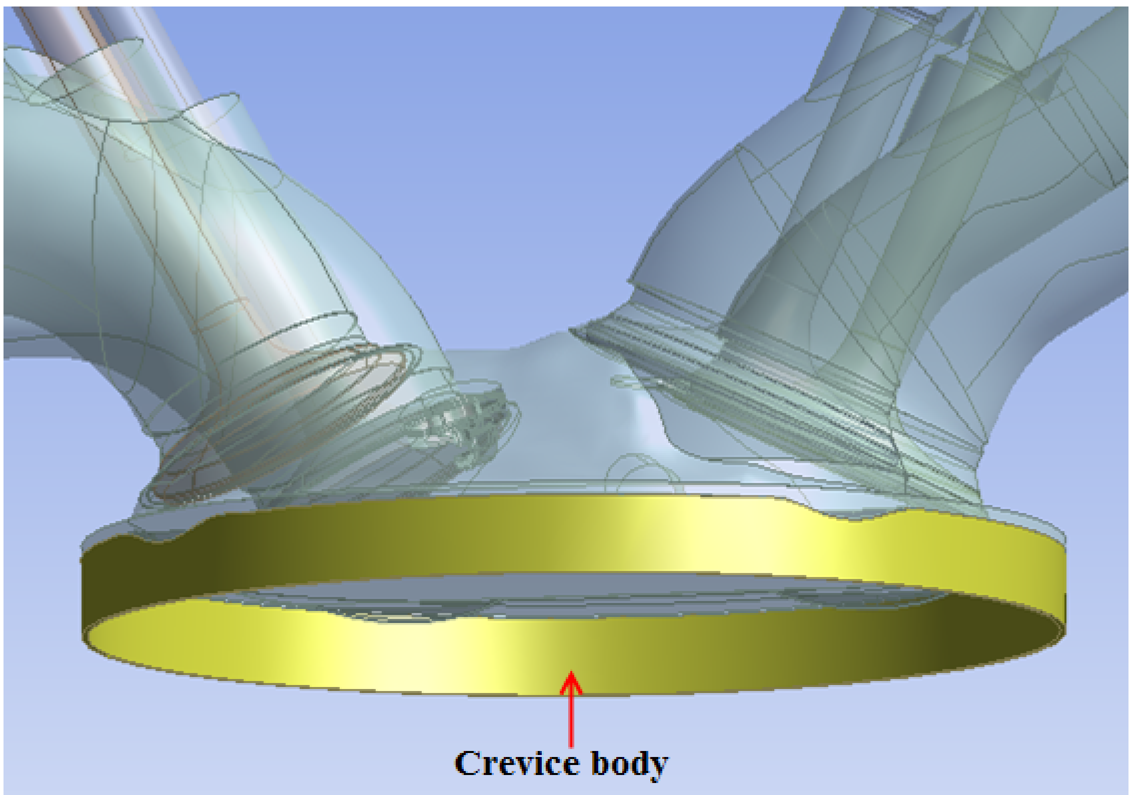
- You can select **Full Topology** or **Single Zone** from the **Topology Option** drop-down list. **Full Topology** will decompose the engine geometry into different parts and bodies according to IC Engine System default decomposition process. If you select **Single Zone** then only a single zone will be generated after decomposition.

- If your geometry has a crevice select **Yes** from the **Crevice Option** drop-down list.

Note:

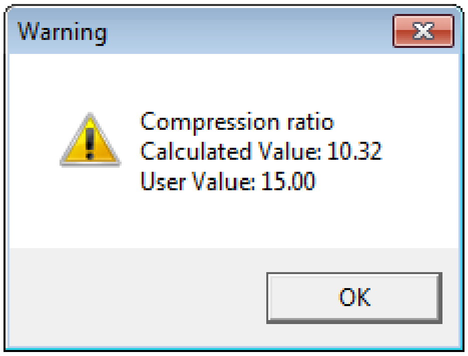
If the geometry has a crevice, ensure that the crevice body is separated before decomposition. For more information, see Separating the Crevice Body (p. 568).

- Select the crevice body for **Crevice Bodies**.



- You can select **Yes** or **No** from the **Validate Compression Ratio** drop-down list. If you select **Yes** then a **Compression Ratio** option appears below it.

- Enter the value for **Compression Ratio**. By default it is set to **15**. The program will internally calculate the compression ratio from the geometry and give a warning if the values do not match.



Note:

For details on how the compression ratio is calculated internally see Calculating Compression Ratio (p. 590).

IC Valves Data

In this section you are required to give details about the valves.

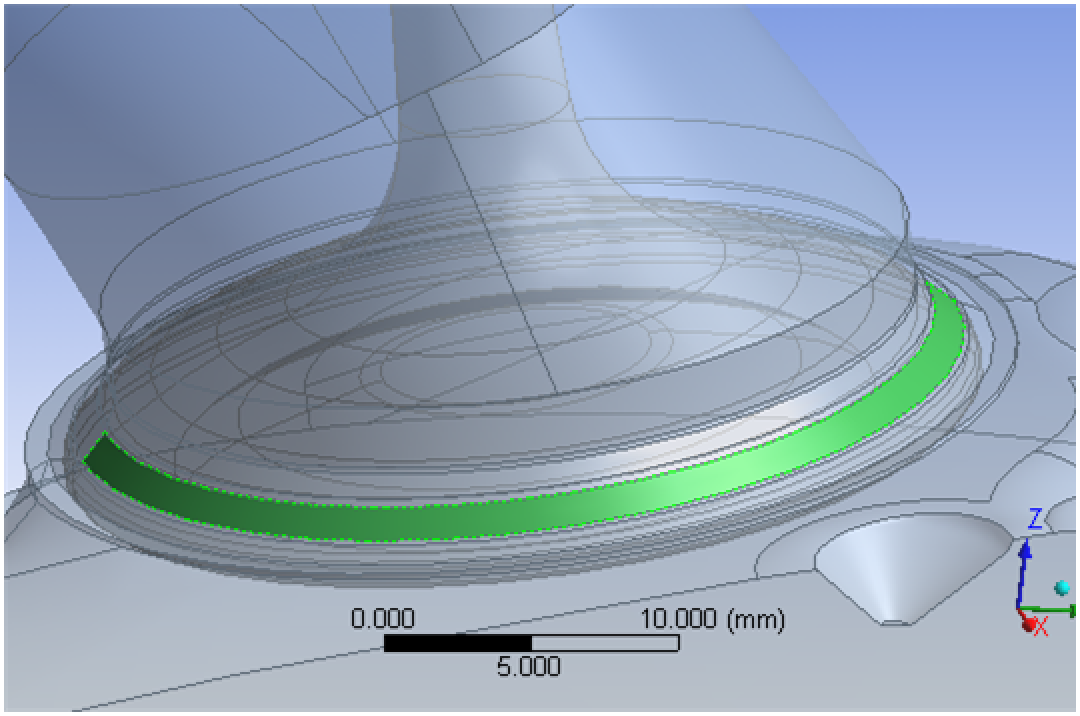
- Select **InValve** or **ExValve** as the **Valve Type** from the drop-down list.
- Select the valves from the figure and click **Apply** next to **Valve Bodies**.

Note:

- You can select multiple valves at a time and set them to the desired type.
- Confirm that **Selection Filters** are set to **Bodies**.
- Remember the order of selection of the valve bodies. This is important especially while using meshes from another setup for the KeyGrid option.

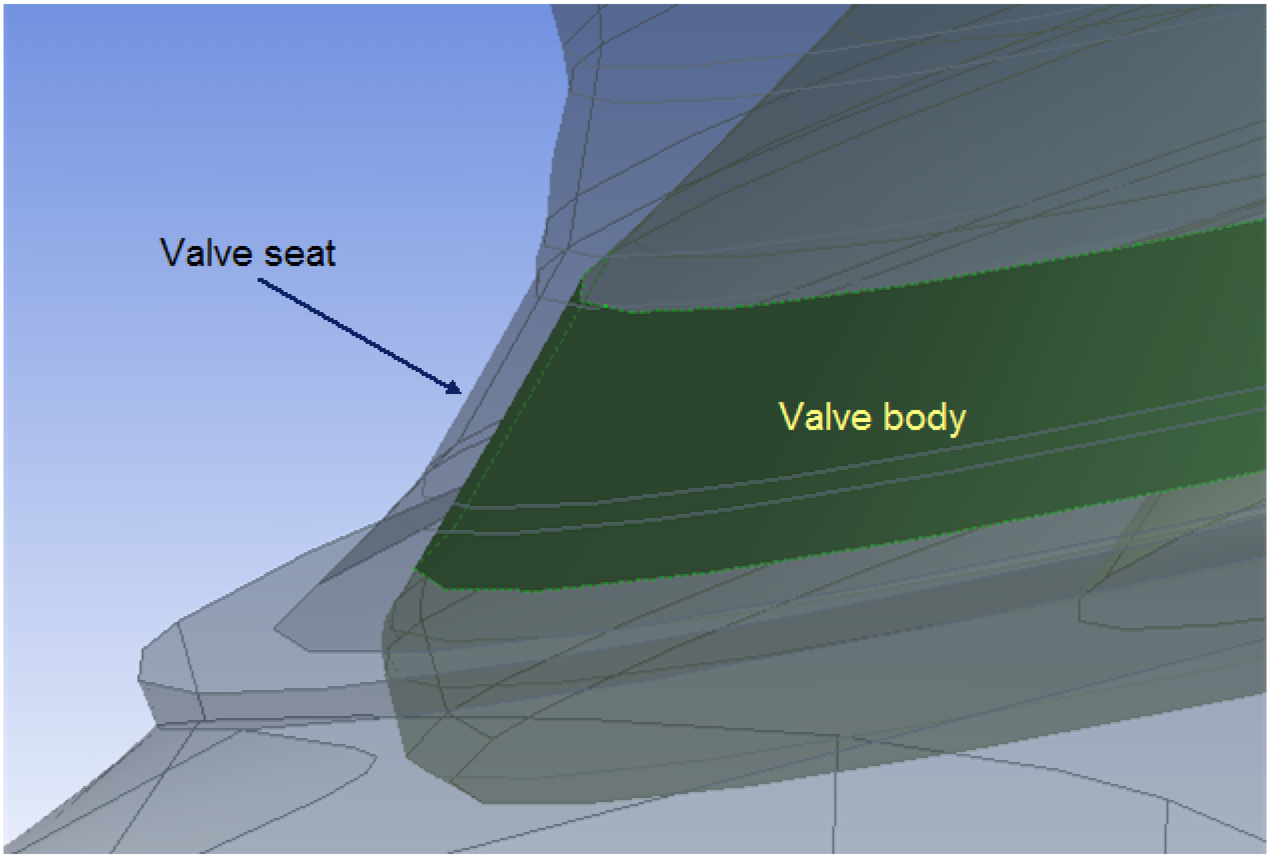
- Select the faces on which the valve rests for **Valve Seat Faces** and click **Apply**.

Figure 4.4: Valve Seats Faces



The valve seat is that face of the port body which comes in contact with the valve body. The Figure 4.5: Valve Seat Selection (p. 163) shows that the highlighted face of the valve body will make contact with the valve seat face of the port body.

Figure 4.5: Valve Seat Selection



Note:

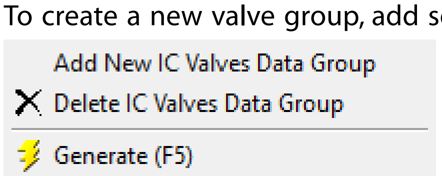
Confirm that **Selection Filters** are set to **Model Faces**. Some engines might have line contact between the valve and the valve seat. In such cases you can select the face which is near the valve and includes the line of contact.

- Select the valve from the **Valve Profile** drop-down list. This will set that particular valve to the selected valve profile.

Note:

The profile file (**Valve Lift and Piston Motion Profile**) (p. 146)contains a table of crank angle against the valve lift for different valves. Each table type has a name. The names are loaded in the **Valve Profile** drop-down list.

- To create a new valve group, add section **IC Valves Data** by right-clicking,



and selecting **Add New IC Valves Data Group**. Set the details for the new group.

IC Animation Inputs

In this section, you can give the inputs required for animating the valve motion. For details, refer to Animating the Valve and Piston (p. 182).

IC Advanced Options

- For **V Layer Slice**, you can select **Yes** or **No** from the drop-down list. By default it is set to **Yes**. This will create extra slices in vlayer. For details, refer to the section on creating extra slices (p. 175).
- **V Layer Slice Angle** is set to **15°** by default. If the angle between two faces in any vlayer body is less than (180–**V Layer Slice Angle**), then the body is split into two bodies using the common edge of these two faces.

  Note:

  This option will not be displayed if the **V Layer Slice** is set to **No**.

- **V Layer Approach**: You can select **4 Layers** or **1 Layer** from the drop-down list. By default it is set to **4 Layers**, which signifies the number of layers created in the V layer.

  Note:

  If **V Layer Approach** is set to **4 Layers**, while meshing inflation is created in port region. For details, see Inflation Port (p. 207).

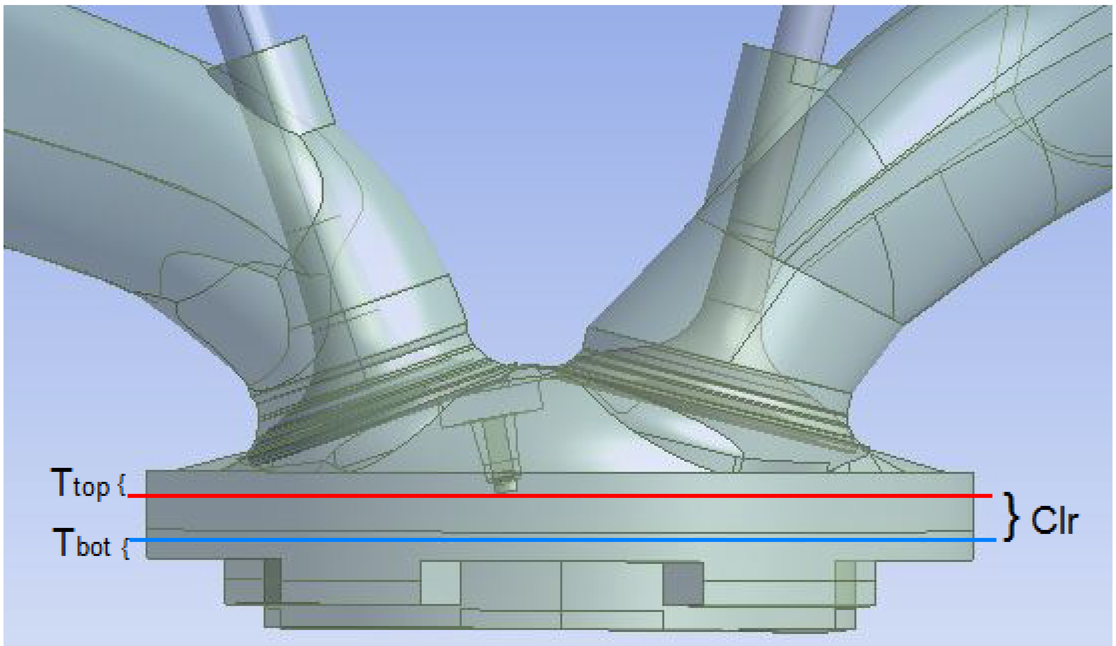
- You can select **Yes** or **No** from the **Piston Profile Option** drop-down list. If you select **Yes** then a **Piston Profile** option appears below it.
- For **Piston Profile**, you can select the piston profile from the drop-down list next to it. The profile is in the file you provided in the properties dialog box. For details, see **Valve Lift and Piston Motion Profile**. (p. 146)
- **Decompose Chamber** can be set to **Yes**, **Program Controlled** or **No**.
  - If you have enough squish volume to separate the chamber and piston by a layer body, select **Yes**.
  - If you have a straight-valve engine, select **Yes** to get a maximum layer mesh.
  - If there is not enough squish volume to insert a layer between chamber and piston, select **No**; this automatically inserts a layer mesh during solver meshing.
  - If you select **Program Controlled** then the program automatically checks if chamber decomposition is possible or not and proceeds accordingly.

  Based on the option selected, decomposition nomenclature will vary. For more information, refer to Nomenclature of Decomposed Geometry (p. 167).

- When **Decompose Chamber** is set to **Yes** then **Decompose Chamber Inputs** is set to **Automatic** by default. This option will not be displayed when **Program Controlled** or **No** is selected from **Decompose Chamber**.
  - When **Automatic** is selected, the **Top Plane/Face** and the **Bottom Plane/Face** are automatically selected by the application.

Note:

The top plane and bottom plane are chosen for the engine automatically as the lines shown in the figure below.



Ttop is the tolerance for the top plane. It is gap between the dome faces and the top plane.

Tbot is the tolerance for the bottom plane. It is gap between the piston faces and the bottom plane.

Clr is the clearance between the top and bottom planes.

Following are the values for the tolerances for a straight valve:

→ Ttop = 0

→ Tbot = 0

→ Clr = S/3 (where S is the mesh size calculated from valve margin radius)

Following are the values for the tolerances for a canted valve:

→ Ttop = 2 * S

→ Tbot = S

→ Clr = S/3 (where S is the mesh size calculated from valve margin radius)

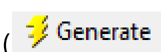
- If you select **Manual**, then you have to provide the **Top Plane/Face** and the **Bottom Plane/Face**.

- **Top Plane/Face** and **Bottom Plane/Face** take the input as a plane or surface; these inputs are used to slice the engine at those positions to form different parts. **Bottom Plane/Face** slices the piston bowl and separates it from the remaining volume. **Top Plane/Face** slices the chamber from the remaining volume. The body remaining between these two planes is filled with layered mesh. For canted valve engines (when the **Decompose Chamber** option is set to **Yes**), this body will be named **layer-cylinder** and for straight valve engines this body will be named **fluid-ch-lower**.

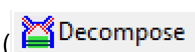
Note:

If you create the planes ensure that the plane created for **Top Plane/Face** does not cut the valve body.

---

5. Click **Generate** (

located in the Ansys DesignModeler toolbar).

6. Click **Decompose** (

located in the **IC Engine** toolbar).

Note:

After decomposition you can see the different parts and bodies created, the different methods used, planes, etc. in the tree. Depending upon the number of valves, cut-plane is automatically created in the tree. In Meshing, you can observe the mesh sliced with the cut-plane.

---

7. Close Ansys DesignModeler.

Note:

Even after decomposition you can change parameters like **Decomposition Crank Angle**, or **Engine Inputs** such as **Connecting Rod Length** and **Minimum Lift** in the properties table. After changing the parameters, you will have to update the cell to regenerate the geometry with the changes. If you change any other parameters you will have to decompose the geometry again.

Important:

If the geometry is of a straight valve engine with valve pockets, then you need to do some manual changes. For more information, refer to Decomposing a Straight Valve Pocket Engine (p. 552).

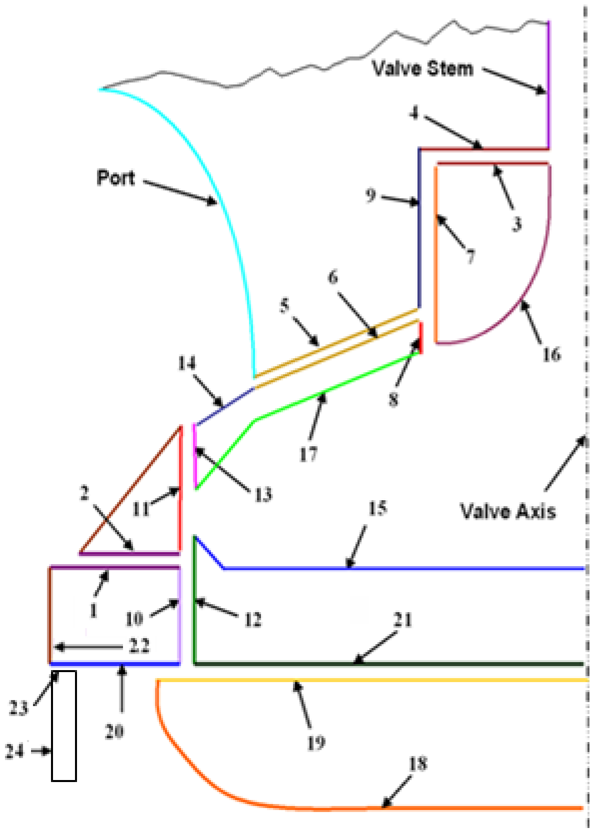
4.3. Nomenclature of Decomposed Geometry

The geometry will be decomposed into different **Parts** and **Bodies**. Each will have a specific name. This naming is important. Depending on the naming, each part will be meshed with different specifications. Refer to Viewing the Bodies and Parts in **IC Engine** system (p. 175) for more information. Types of valves and the naming conventions of their parts are presented in the following sections:

4.3.1. Straight Valve Geometry With Chamber Decomposition for IC Engine

The following figures show the geometry decomposition and the corresponding zone and boundary names specific to a straight valve engine.

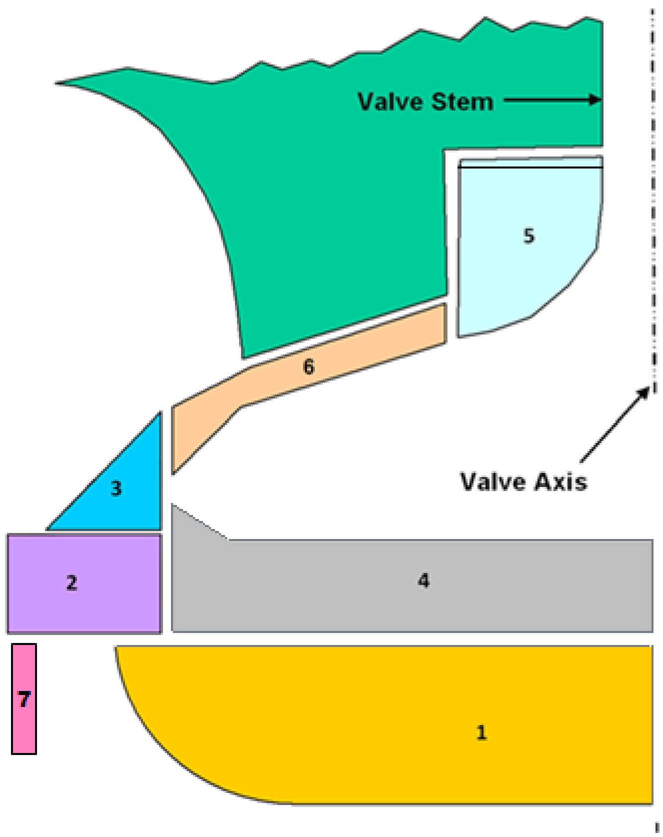
Figure 4.6: Boundary Zone Names and Mesh Requirements for Straight Valve



No.	Boundary Zone
1	intf-deck-fluid-ch-lower
2	intf-deck-fluid-ch-upper
3	intf-int-valveID-ib-fluid-ib
4	intf-int-valveID-ib-fluid-port
5	intf-int-valveID-ob-fluid-port
6	intf-int-valveID-ob-fluid-vlayer
7	intf-valveID-ib-fluid-ib
8	intf-valveID-ib-fluid-ob-quad
9	intf-valveID-ib-fluid-ob-tri
10	intf-valveID-ob-fluid-ch-lower
11	intf-valveID-ob-fluid-ch-upper
12	intf-valveID-ob-fluid-ch-valve
13	intf-valveID-ob-fluid-vlayer
14	valveID-seat
15	valveID-ch
16	valveID-ib
17	valveID-ob
18	piston

19	intf-piston-bowl
20	intf-piston-ch
21	intf-piston-valveID
22	cyl-quad
23	intf-piston-on-crevice
24	cyl-crevice

Figure 4.7: Fluid Zone Names and Mesh Requirements for Straight Valve



No.	Fluid Zone Name	Mesh Requirement
1	fluid-piston	any mesh
2	fluid-ch-lower	layered mesh
3	fluid-ch-upper	any mesh
4	fluid-ch-valveID	layered mesh
5	fluid-valveID-ib	mesh with at least one layer at the top
6	fluid-valveID-vlayer	layered mesh

7	fluid-crevice	any mesh

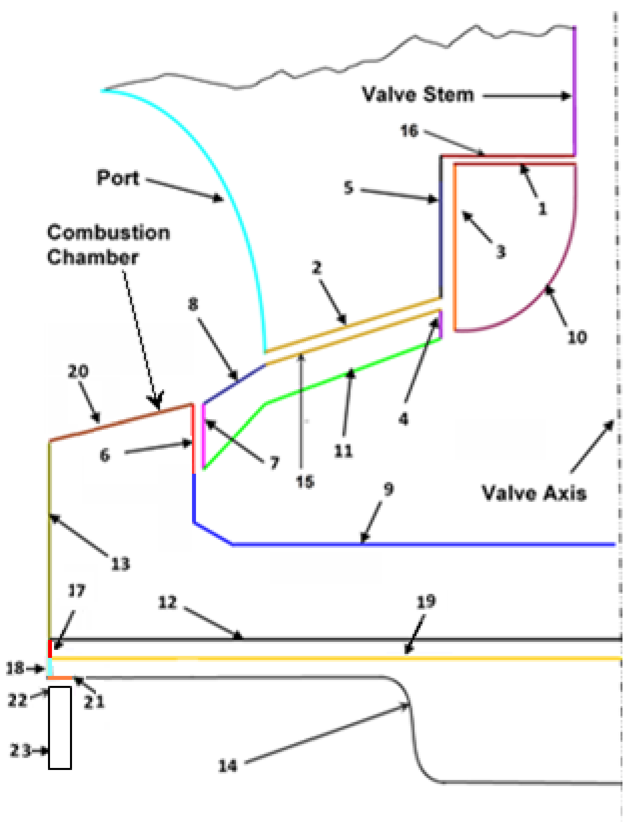
Note:

- `valveID` is a variable. It represents the valve type and number. For example, `valveID` can be exvalve1, which is one of the exhaust valves, or invalve2, which can be the second intake valve in an engine with more than one intake valve.
- Crevice zone and boundary zones are shown for your reference only. They might not be included in every geometry.

4.3.2. Canted Valve Geometry With Chamber Decomposition for IC Engine

The following figures show the geometry decomposition and the corresponding zone and boundary names specific to canted valve engines with the chamber decomposed.

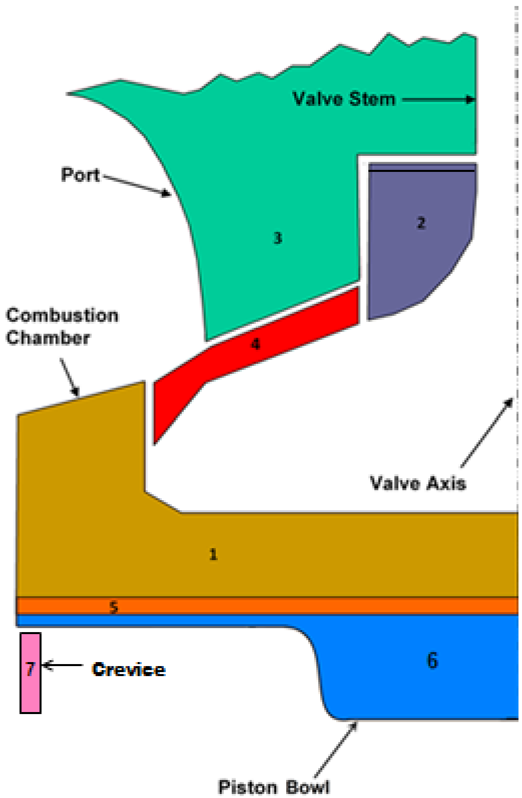
Figure 4.8: Boundary Zone Names and Mesh Requirements for Canted Valve with Chamber Decomposition



No.	Boundary Zone
1	intf-int-valveID-ib-fluid-ib
2	intf-int-valveID-ob-fluid-port
3	intf-valveID-ib-fluid-ib
4	intf-valveID-ib-fluid-ob-quad

5	intf-valveID-ib-fluid-ob-tri
6	intf-valveID-ob-fluid-ch
7	intf-valveID-ob-fluid-vlayer
8	valveID-seat
9	valveID-ch
10	valveID-ib
11	valveID-ob
12	int-piston
13	cyl-tri
14	piston
15	intf-int-valveID-ob-fluid-vlayer
16	intf-int-valveID-ib-fluid-port
17	cyl-quad
18	cyl-piston
19	interior-fluid-layer-cylinder-fluid-piston
20	cyl-head
21	intf-crevice-on-piston
22	intf-piston-on-crevice
23	cyl-crevice

Figure 4.9: Fluid Zone Names and Mesh Requirements for Canted Valve with Chamber Decomposition



No.	Fluid Zone Name	Mesh Requirement
1	fluid-ch	tetrahedron mesh
2	fluid-valveID-ib	mesh with at least one layer at the top
3	fluid-valveID-port	any mesh
4	fluid-valveID-vlayer	layered mesh
5	fluid-layer-cylinder	layered mesh
6	fluid-piston	any mesh
7	fluid-crevice	any mesh

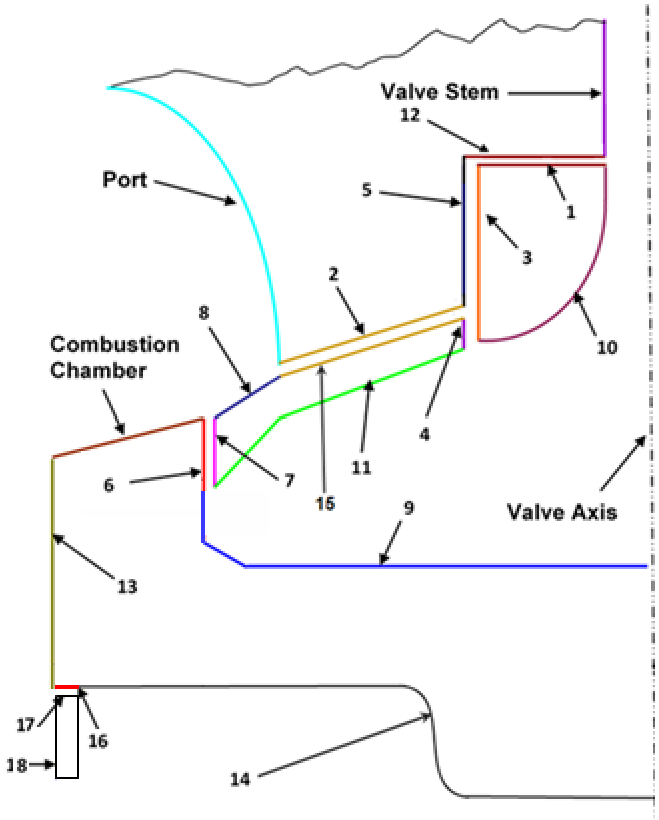
Note:

- `valveID` is a variable. It represents the valve type and number. For example, `valveID` can be exvalve1, which is one of the exhaust valves, or invalve2, which can be the second intake valve in an engine with more than one intake valve.
- Crevice zone and boundary zones are shown for your reference only. They might not be included in every geometry.

4.3.3. Any Engine Geometry Without Chamber Decomposition

The following figures show the geometry decomposition and the corresponding zone and boundary names for canted or straight valve engines without decomposing the chamber.

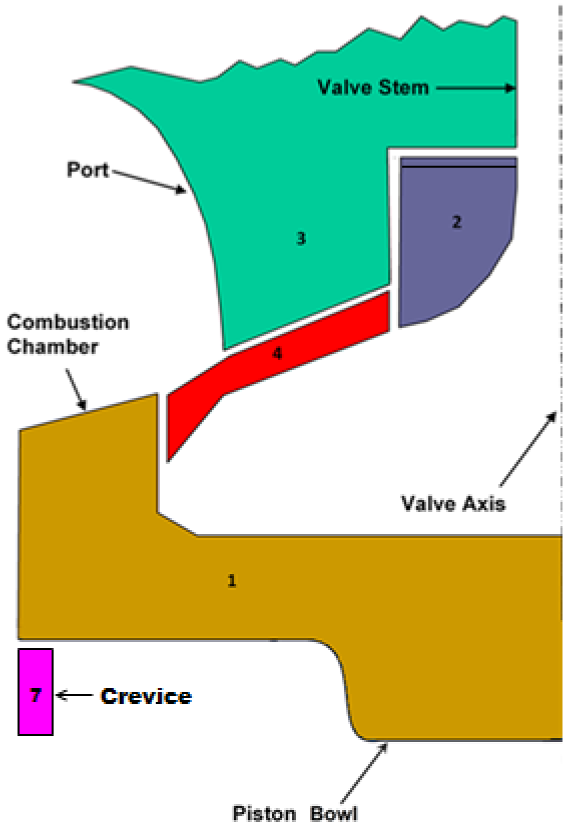
Figure 4.10: Boundary Zone Names and Mesh Requirements Without Chamber Decomposition



No.	Boundary Zone
1	intf-int-valveID-ib-fluid-ib
2	intf-int-valveID-ob-fluid-port
3	intf-valveID-ib-fluid-ib
4	intf-valveID-ib-fluid-ob-quad
5	intf-valveID-ib-fluid-ob-tri
6	intf-valveID-ob-fluid-ch
7	intf-valveID-ob-fluid-vlayer
8	valveID-seat
9	valveID-ch
10	valveID-ib
11	valveID-ob
12	intf-int-valveID-ib-fluid-port
13	cyl-tri
14	piston

15	intf-int-valveID-ob-fluid-vlayer
16	intf-crevice-on-piston
17	intf-piston-on-crevice
18	cyl-crevice

Figure 4.11: Fluid Zone Names and Mesh Requirements Without Chamber Decomposition



No.	Fluid Zone Name	Mesh Requirement
1	fluid-ch	any mesh
2	fluid-valveID-ib	mesh with at least one layer at the top
3	fluid-valveID-port	any mesh
4	fluid-valveID-vlayer	layered mesh
5	fluid-crevice	any mesh

Note:

- `valveID` is a variable. It represents the valve type and number. For example, `valveID` can be exvalve1, which is one of the exhaust valves, or invalve2, which can be the second intake valve in an engine with more than one intake valve.
- Crevice zone and boundary zones are shown for your reference only. They might not be included in every geometry.

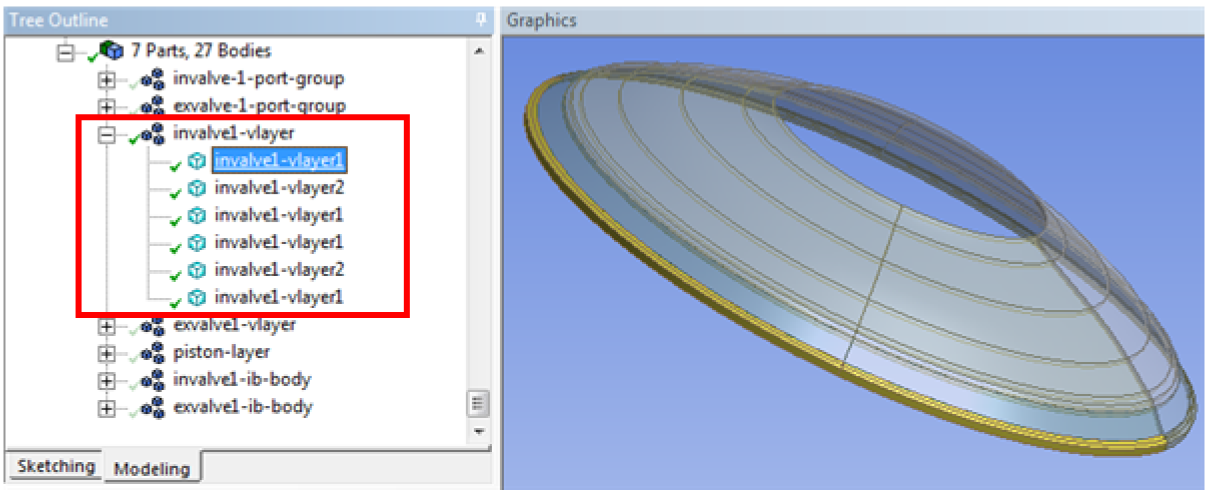
4.4. Viewing the Bodies and Parts in IC Engine system

This section describes in detail the different bodies and parts created after decomposition. For convenience, the engine is divided into three regions.

4.4.1. Valve Region

Following are the parts and bodies created in the valve region.

valveID-vlayer

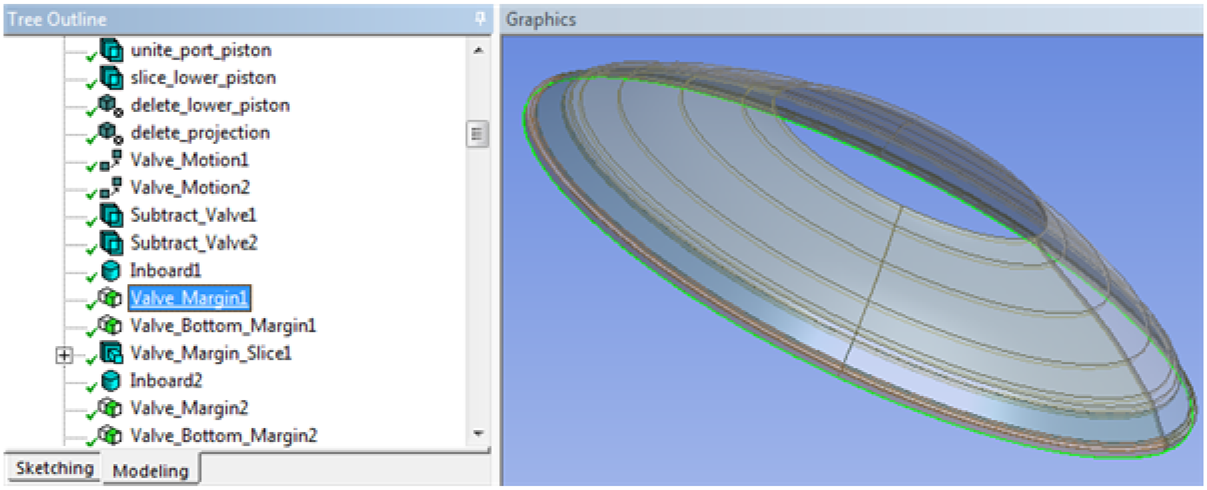


valveID-vlayer is one of the important regions for simulation. Because there will be very high velocities/jets of fluid during the initial phase of valve opening, it is essential to have a good mesh in this region. To achieve this, the part has to be made sweepable so as to create a uniform hex mesh in this region. You can achieve this sweepable part by separating the circular portion between the valve and the valve seat. **Valve_Margin**, **Valve_Margin_Slice**, **Valve_Mid_Radius**, and **Valve_Mid_Radius_Slice** separates the part into different rings.

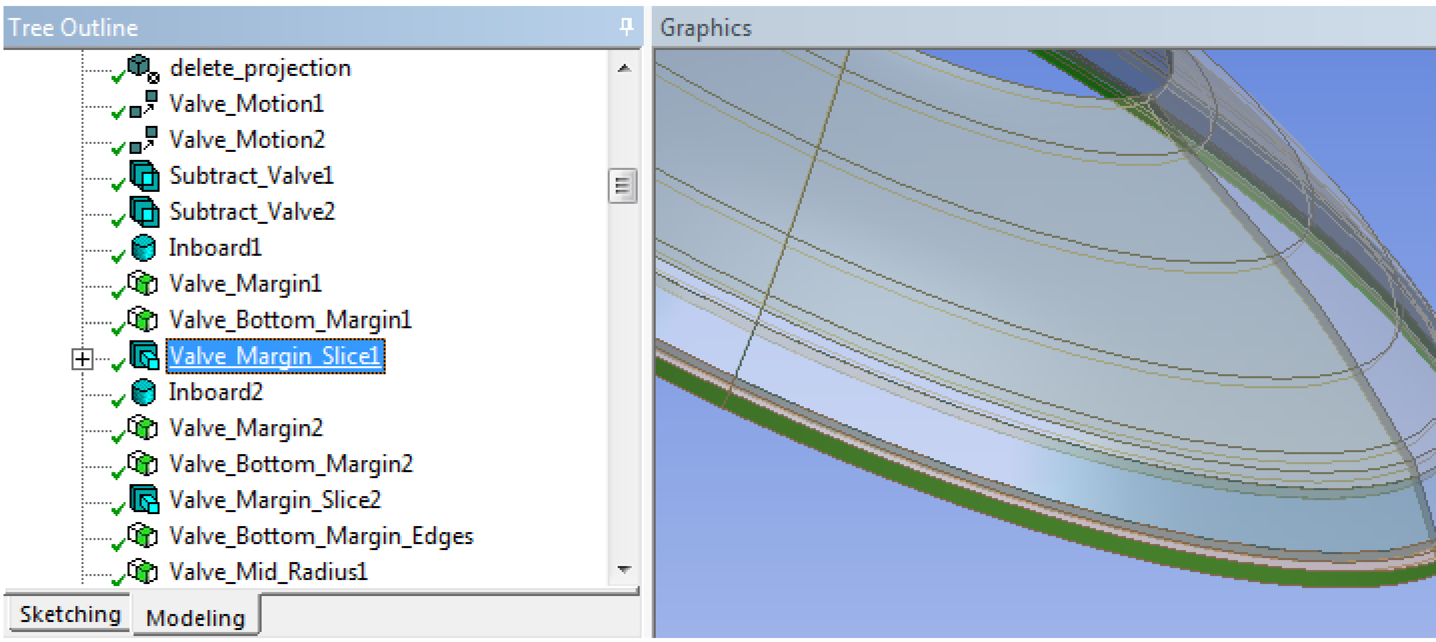
Note:

If you have selected **Yes** for **V Layer Slice** in the **IC Advanced Options** (p. 164) in the **Input Manager**, additional slicing operations are performed. This will happen if, after separation of vlayer, there is a large angular discontinuity within this body. The bodies are separated where there is a change in the angle/inclination of the ring. For details on setting the angle, see the part on **V Layer Slice Angle** (p. 164) in the **Input Manager** section.

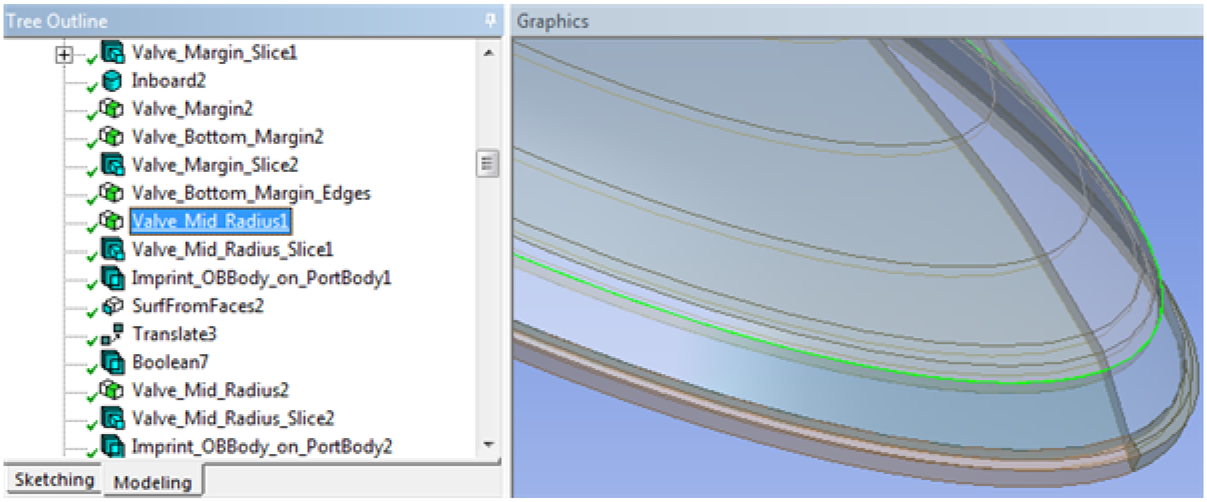
- **Valve_Margin**: This is a named selection corresponding to the maximum diameter closest to the valve seat (with a tolerance of +/- 0.1) of valve body part.



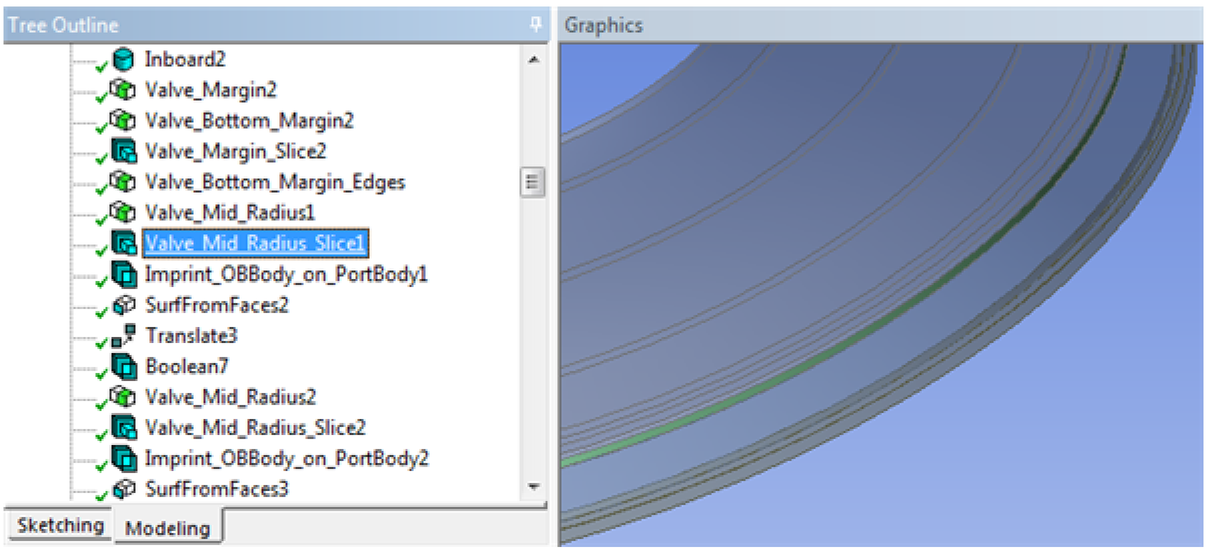
- **Valve_Margin_Slice**: It separates the port body from the chamber region using the extrude operation, along the valve axis of that valve.



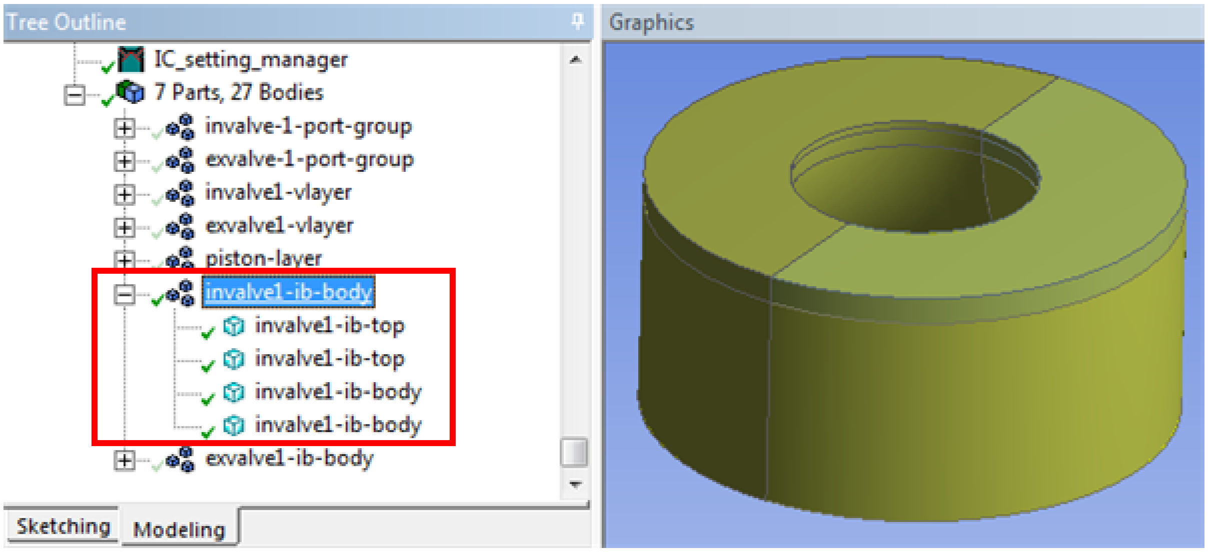
- **Valve_Mid_Radius**: This is the top edge of the valve seat. (It corresponds to the minimum edge diameter of the valve seat for that valve.)



- **Valve_Mid_Radius_Slice**: This is an extrude operation where slicing is carried out on the port body using the **Valve_Mid_Radius**, thus creating the **valveID-vlayer** body.

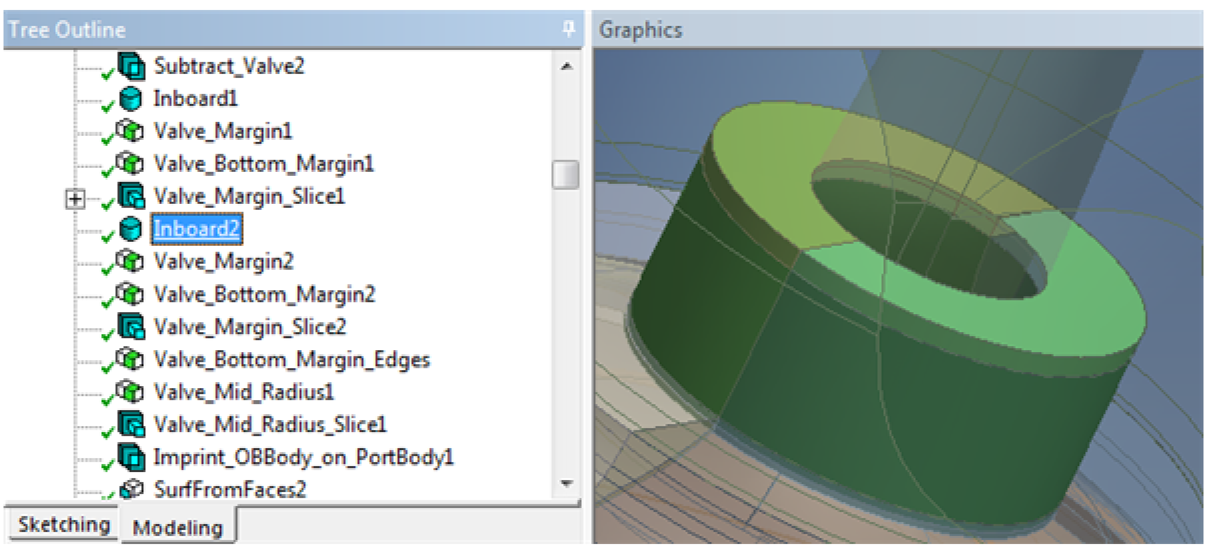


valveID-ib-body



This part consists of four bodies. It is created by the **Cylinder (Inboard)** feature. This cylinder is then sliced into two halves so as to get a layered hex mesh in this region. A top slice is created which also has layered mesh. These bodies move with the valve and thus generate hex layers during the valve motion.

- **Cylinder**: This feature creates a new part called **invalve-ib-body** or **exvalve-ib-body** under **Parts and Bodies**.

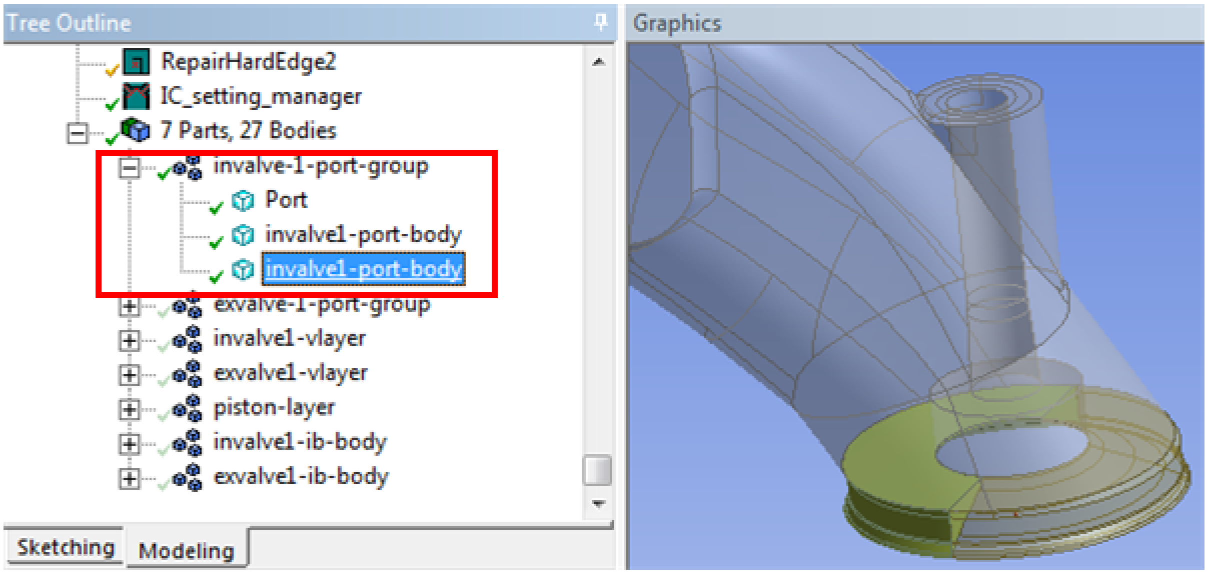


Note:

ib (in board) volume should consist at least of two parts so that sweeping can be achieved.

4.4.2. Port

valveID-port-group



This is the part from where the fluid will enter/exit the engine. Two circular bodies are created at the bottom of the port so as to get good sweepable mesh and mesh transition in this region.

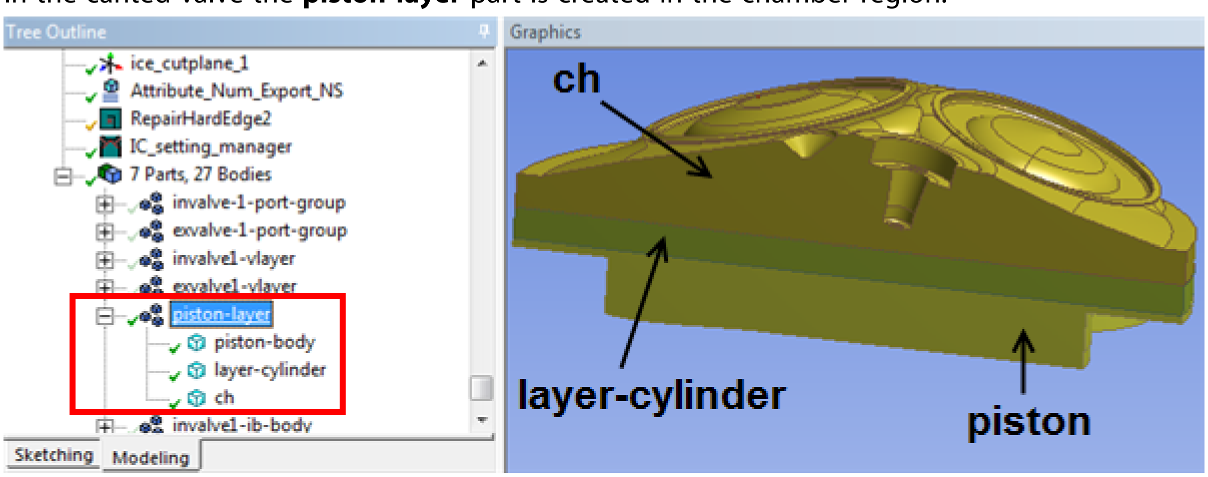
4.4.3. Chamber

The chamber region is decomposed differently for straight and canted valves.

4.4.3.1. Canted Valve

piston-layer

In the canted valve the **piston-layer** part is created in the chamber region.



This part consists of three bodies: **ch**, **layer-cylinder**, and **piston-body**.

- **ch** corresponds to the chamber region. The bottom surface is formed by the **Top Plane/Face**.

- **layer-cylinder** is the body between the **Bottom Plane/Face** and **Top Plane/Face**. Layered mesh is created in this region.
  - **Bottom Plane/Face** is a plane which separates the piston body from the chamber.
  - **Top Plane/Face** is a plane which separates the chamber from the rest of the volume.
- **piston-body** is the body below the **Bottom Plane/Face**. This body does not have any specific mesh requirements.

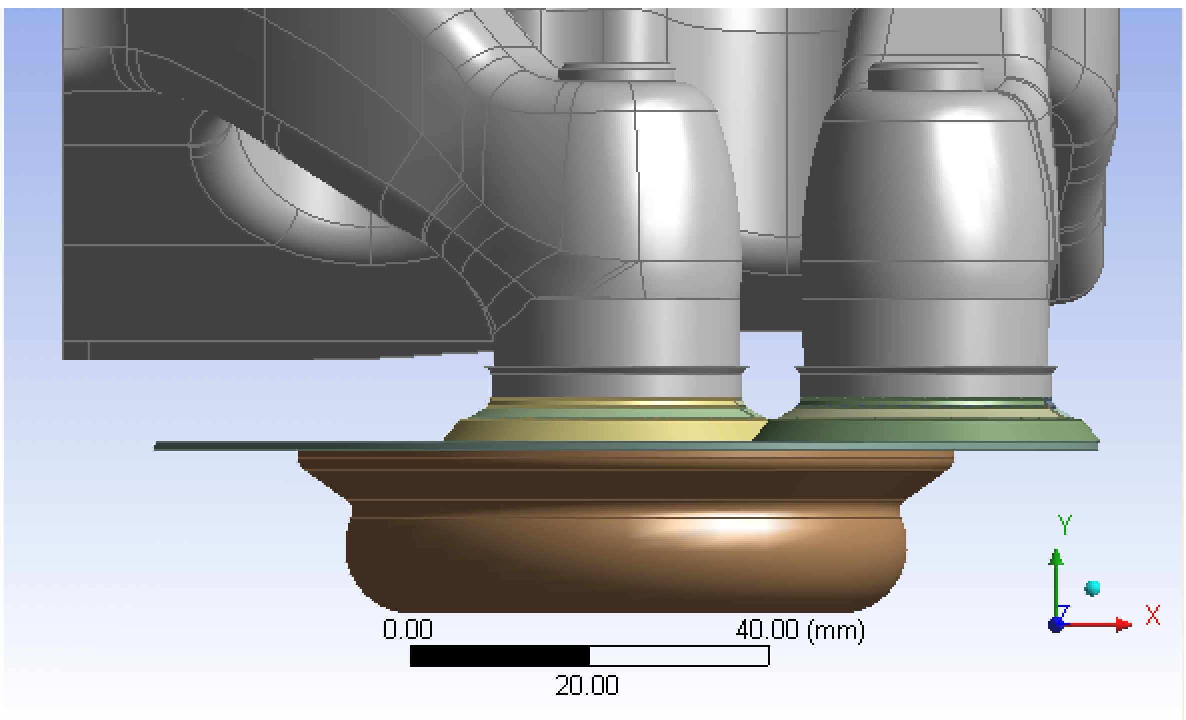
Note:

For canted valves when the **Decompose Chamber** option is set to **No**, instead of these three bodies only one body is created, namely **fluid-ch**.

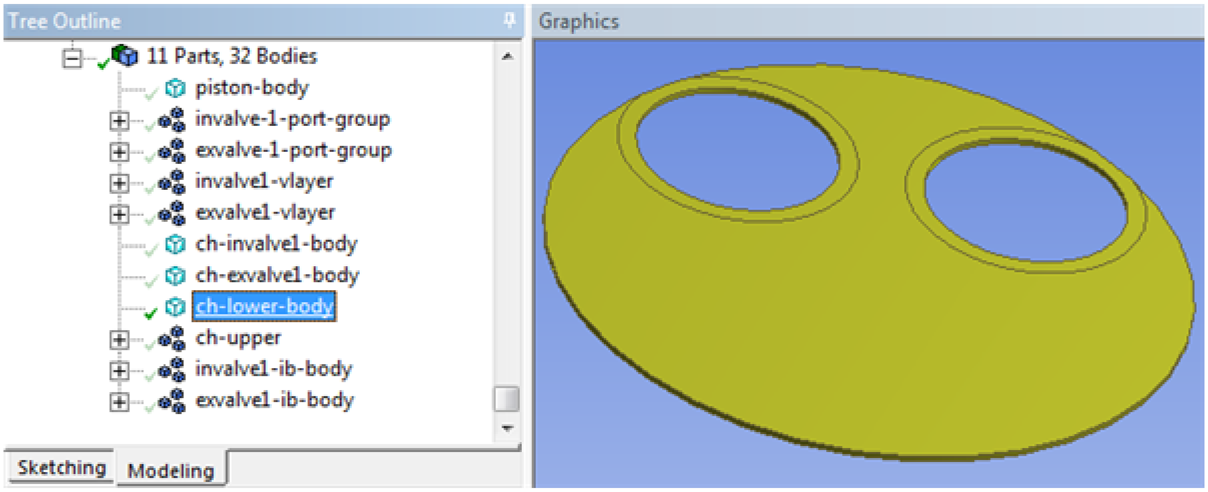
4.4.3.2. Straight Valve

Following are some parts and bodies of a straight valve that have a different named selection than that of a canted valve.

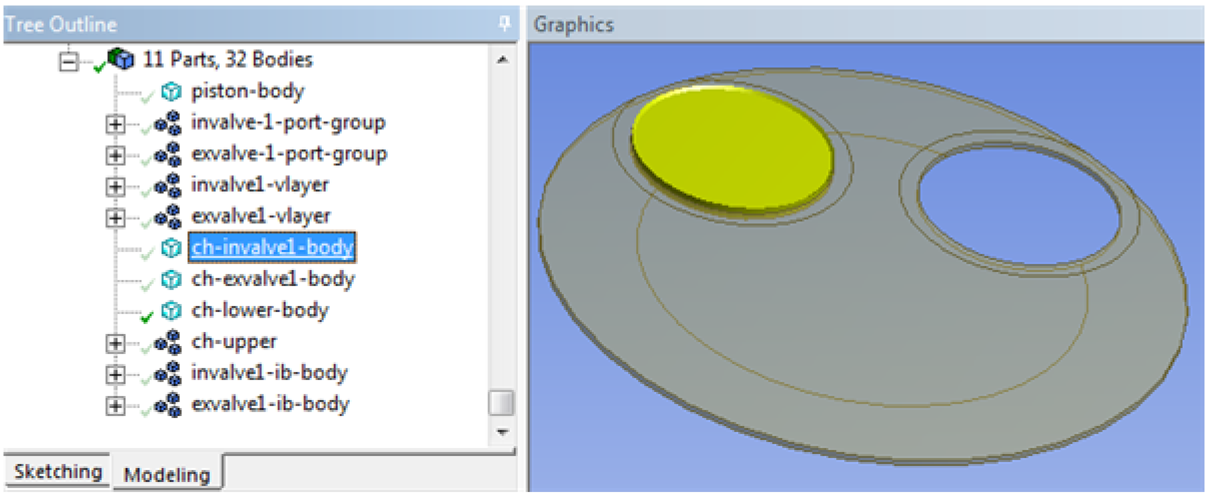
Figure 4.12: Straight Valve



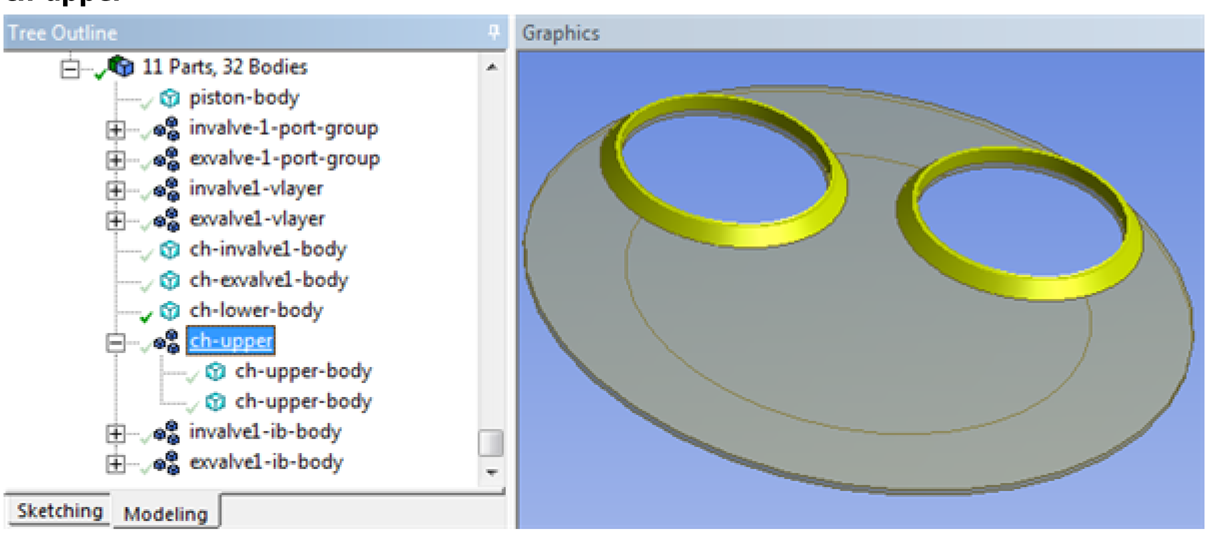
- **ch-lower-body**



- **ch-valveID-body**



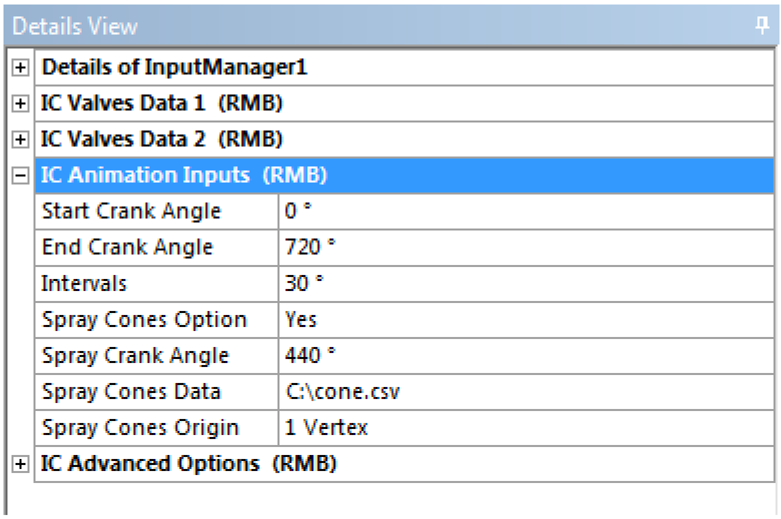
- **ch-upper**



4.5. Animating the Valve and Piston

The **Animate** button () in the **IC Engine** toolbar, will display the valve and piston movement for the required crank angle. It will also show the spray cone in the animation.

Before animating, you need to enter all the required data in the **Input Manager** under **IC Animation Inputs** in addition to the inputs required for decomposition as shown in Geometry Decomposition for a Cold Flow Simulation for **IC Engine** (p. 152).



In the **Details View** under **IC Animation Inputs,** enter the following:

- **Start Crank Angle**: By default, the automatic decomposition is done at zero crank angle, where the piston is assumed to be at TDC position. For animation purposes you can enter the crank angle value of your interest in this field so that the animation starts from that position.

- **End Crank Angle**: This is the crank angle till which you require the animation to continue. It is set to **720°** by default.

- **Intervals**: This term signifies the crank angle intervals at which you want the display to update. It is set to **30°** by default.

- **Spray Cones Option**: You can select **Yes** or **No** from the drop-down list depending on whether your engine has spray cones or not. It is set to **No** by default. If you select **Yes**, you need to give details about the spray cones.

  – You can enter the angle at which the spray starts from the cones for the **Spray Crank Angle**. It is set to **440°** by default.

  – For the **Spray Cones Data** you need to enter the name and the location of the file which contains the details about the spray cones.

    The file about the spray cone contains the following details:

    → `numberofspraycones`: The number of spray cones present.

    → `sprayconeaxis`: The direction of each cone.

→ `sprayconestime-height`: A table that gives information about the spray cone height in millimeters with respect to time in milliseconds.

The format of the file is as follows:

```
numberofspraycones,n
sprayconeaxis,
x1,y1,z1
x2,y2,z2
.
.
.
xn,yn,zn
sprayconestime-height,
t1,h1
t2,h2
.
.
.
tl,yl
```

- For the **Spray Cones Origin** you need to select a vertex on the geometry and click **Apply**. This will be the location at which all the cones are present.

Click **Animate** (

) to display the valve and piston animation.

4.6. Moving the Piston to a Specified Crank Angle in IC Engine system

When the crank angle is specified, the piston should be moved to the desired crank angle. This is based on the assumption that the initial position of the piston is at TDC.

Figure 4.13: Piston at TDC

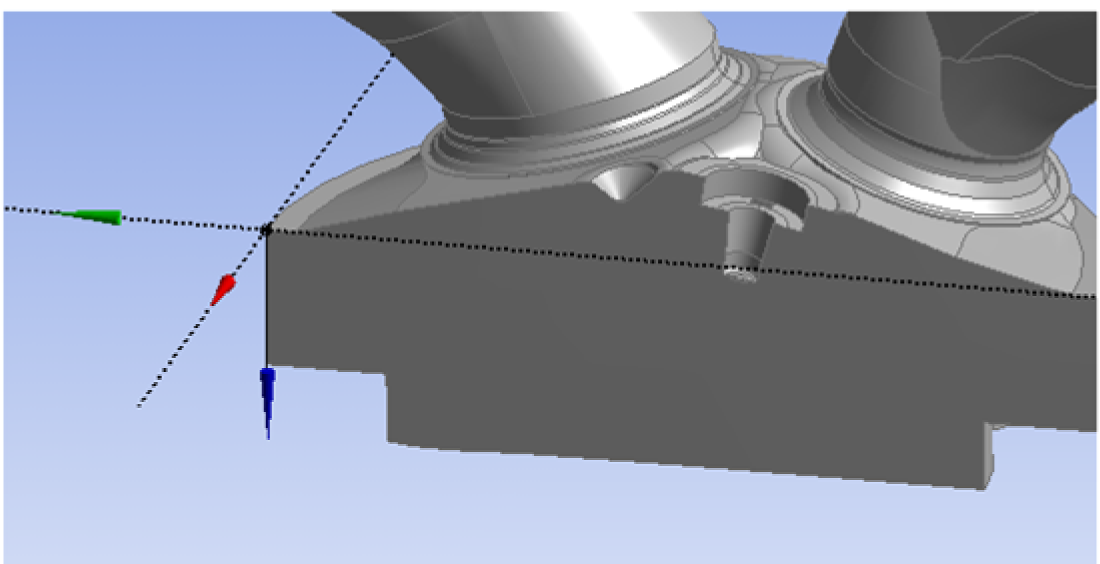
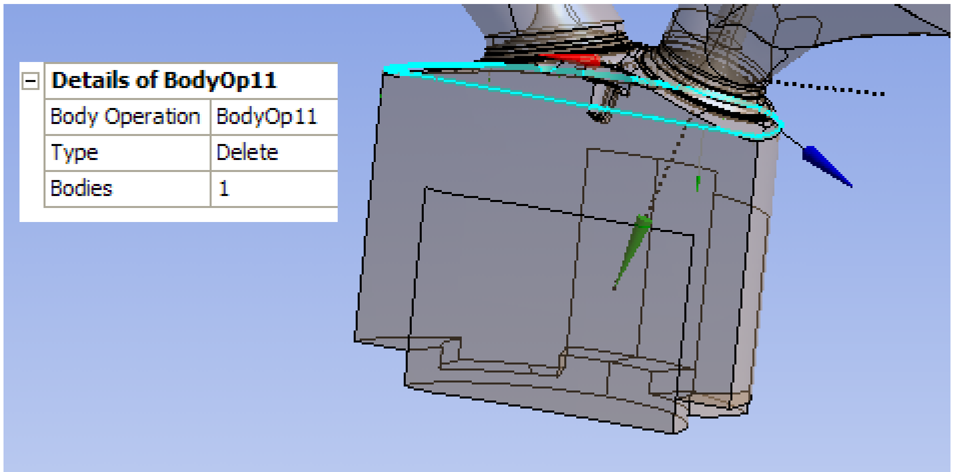


Figure 4.14: Piston Moved to Desired Crank Angle



For more details, refer to How IC Engine System Moves the Piston to the Specified Crank Angle (p. 547)

Chapter 5: Cold Flow Simulation: Meshing

The decomposed geometry is used to generate the mesh. The goal of the IC Engine meshing tool is to minimize the effort required to generate a mesh for the IC Engine specific solver. It uses the named selection created in the decomposition to identify different zones and creates the required mesh controls. The information in this chapter is divided into the following sections:

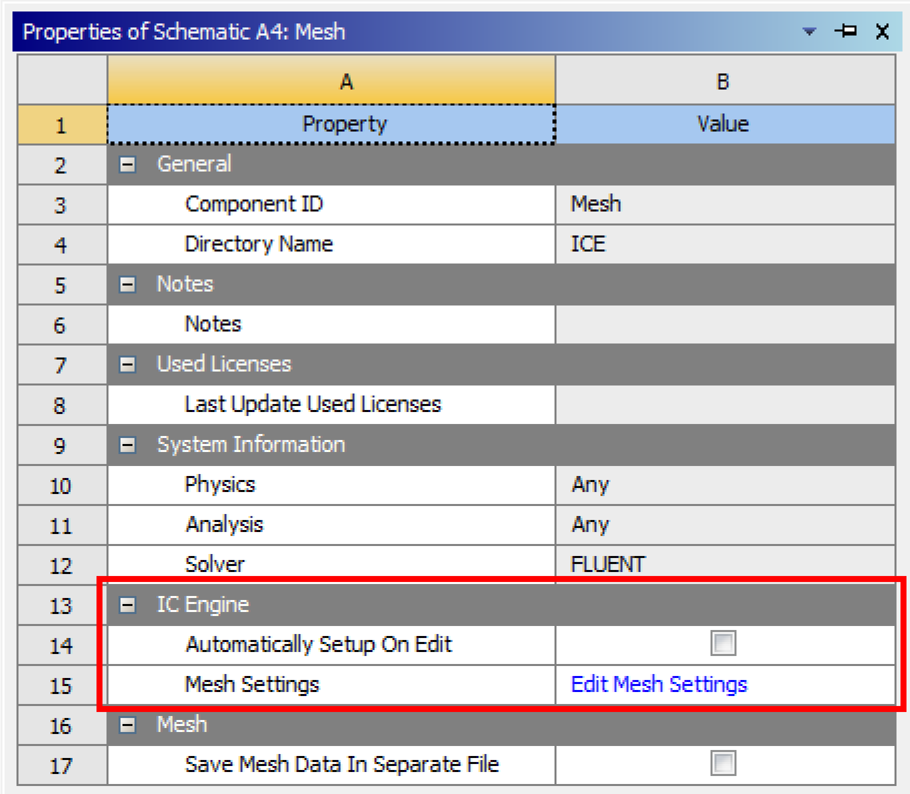
5.1. Meshing Procedure for Cold Flow Simulation in IC Engine

There are two ways to generate the mesh.

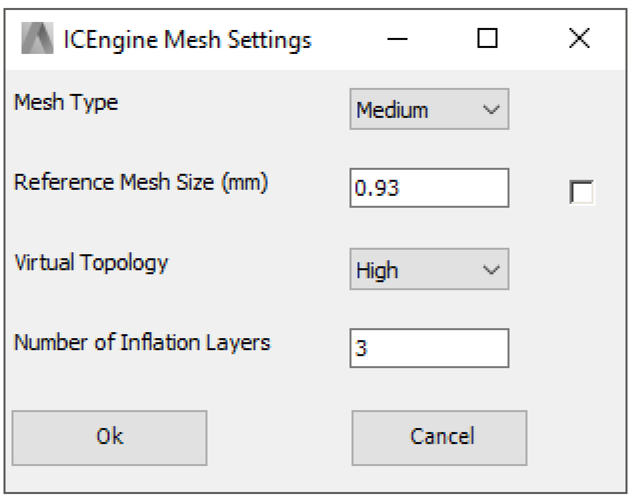
Meshing directly from the Workbench Window

After decomposition is done you can then directly generate the mesh from the Workbench window without opening the Ansys Meshing application. If you want to use the default mesh settings then right-click on the **Mesh** cell and select **Update** from the context menu. This will first create the mesh controls and then generate the mesh.

You can also change the mesh settings from the Workbench window.



In the **Properties** box which is displayed after selecting the **Mesh** cell you can click on **Edit Mesh Settings** next to the **Mesh Settings** property, under **IC Engine**. This will open a **ICEngine Mesh Settings** dialog box.



You can change the settings for the parameters:

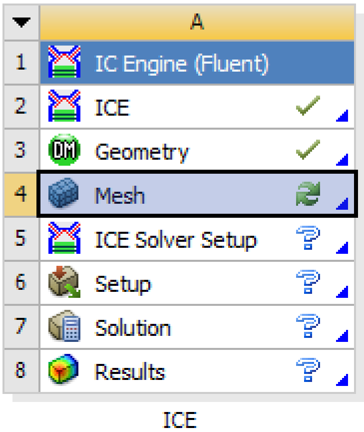
- **Mesh Type**
- **Reference Mesh Size (mm)** (This can be parametrized by enabling the check box next to it.)
- **Virtual Topology**
- **Number of Inflation Layers**

For more information on these check the section on **IC Mesh Parameters** (p. 187).

Meshing using Ansys Meshing Application

In this method of meshing you can check the mesh controls and settings in details.

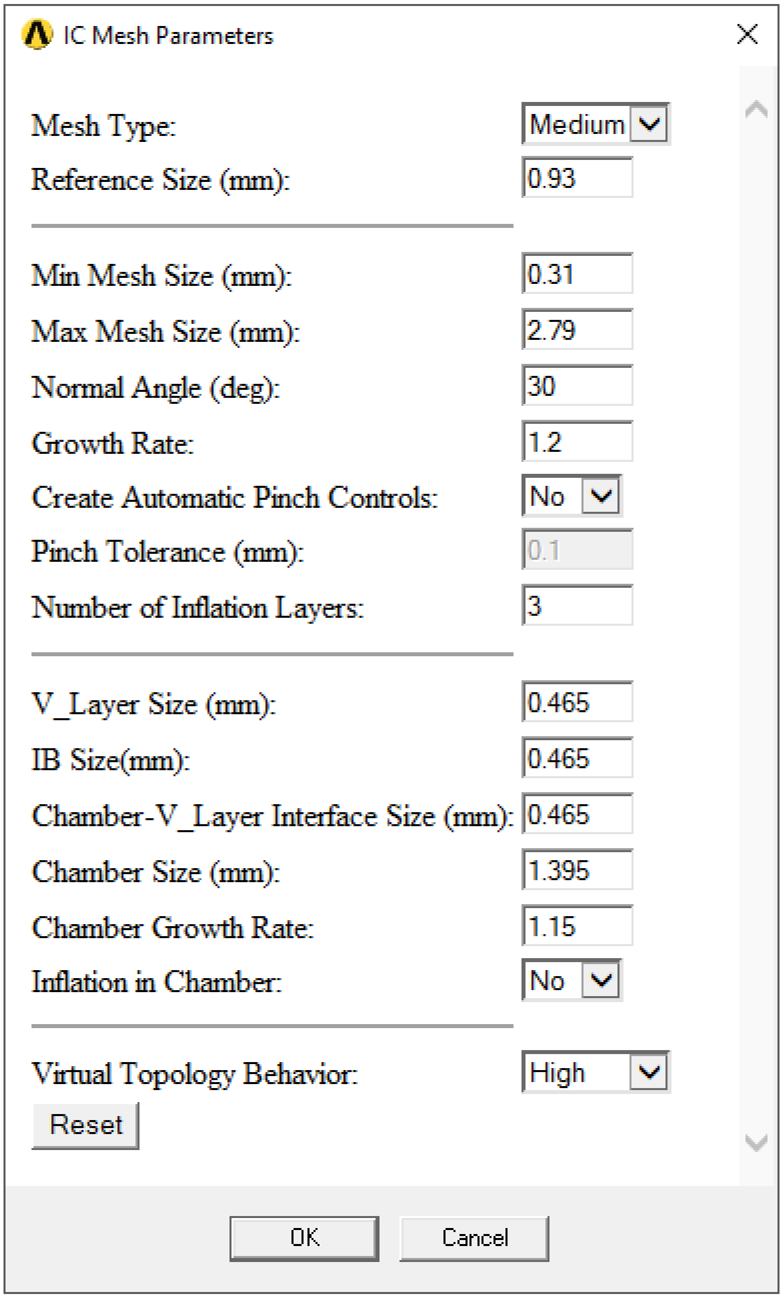
1. Double-click the **Mesh** cell in the **IC Engine** analysis system to open the Ansys Meshing application.



2. Click **IC Setup Mesh** (

located in the **IC Engine** toolbar). This opens the **IC Mesh Parameters** dialog box.

    a. Here you can define different mesh settings for the different parts and virtual topologies.



In the **IC Mesh Parameters** dialog box you can see the default mesh settings. You can change the settings or use the default ones.

- **Mesh Type**: You can select **Fine**, **Medium**, or **Coarse** from the drop-down list. The default setting is **Medium**.
- **Reference Size**: This is a reference value. Some global mesh settings and local mesh setting values are dependent on this term.

Reference Size = (Valve margin perimeter) /100

Note:

Choosing **Fine**, **Medium**, or **Coarse** changes the **Reference Size** and thus all other dependent parameters are also affected.

Global Mesh Settings

- **Min Mesh Size**: This value is set to **Reference Size**/3.
- **Max Mesh Size**: This value is set to **Reference Size** × 3. This can be parametrized by enabling the check box next to it.
- **Normal Angle**: This value changes depending upon the chosen **Mesh Type**.
- **Growth Rate**: This value is set to **1.2**.
- **Create Automatic Pinch Controls**: This is set to **No** by default.
- **Pinch Tolerance**: Automatic pinch controls are not created by default. If you would like to create pinch control select **Yes** from the **Create Automatic Pinch Controls** drop-down list. You can change the default value of **0.1**.
- **Number of Inflation Layers**: This value changes depending upon the chosen **Mesh Type**. It is set to **3** for a **Medium** or **Coarse Mesh Type** and **5** for a **Fine Mesh Type**.

Local Mesh Settings

- **V_Layer Size**: This value is set to **Reference Size** × (1/2).
- **IB Size**: This value is set to **Reference Size**/2.
- **Chamber-V_Layer Interface Size**: This value is set to **Reference Size**/2.
- **Chamber Size**: This value is set to 1.5 X **Reference Size**.
- **Chamber Growth Rate**: This value is set to **1.15** for a **Fine** and **Medium Mesh Type**. It is set to **1.2** for a **Coarse Mesh Type**.
- **Inflation in Chamber**: This is set to **No** by default. If you select **Yes** —
  - For canted valve case, inflation will be created on liner, dome, valve bottom, and vlayer-ch interface.
  - For straight valve case inflation will be created on liner.

  A new inflation control for ch faces is created.
- **Virtual Topology Behavior**: You can select **Low**, **Medium**, **High**, or **None** from the drop-down list. If you select **None** virtual topologies are not created. Once you have selected the behavior type you will not be allowed to change the virtual topology.

- **Crevice-Piston Interface Size**: This option is visible in the dialog box when the geometry has a crevice and you select **Yes** for **Crevice Option** in the **Input Manager** dialog box. The value set depends on the thickness of the crevice and is calculated internally.
- **Number of Layers in Crevice**: This option is visible in the dialog box when the geometry has a crevice and you select **Yes** for **Crevice Option** in the **Input Manager** dialog box. The value set changes upon the selection of the **Mesh Type**. It is set to **2** for **Coarse**, **3** for **Medium**, and **4** for **Fine Mesh Type**.

Note:

If **Reference Size** is changed, all the other parameters will change, depending on their relation with it.

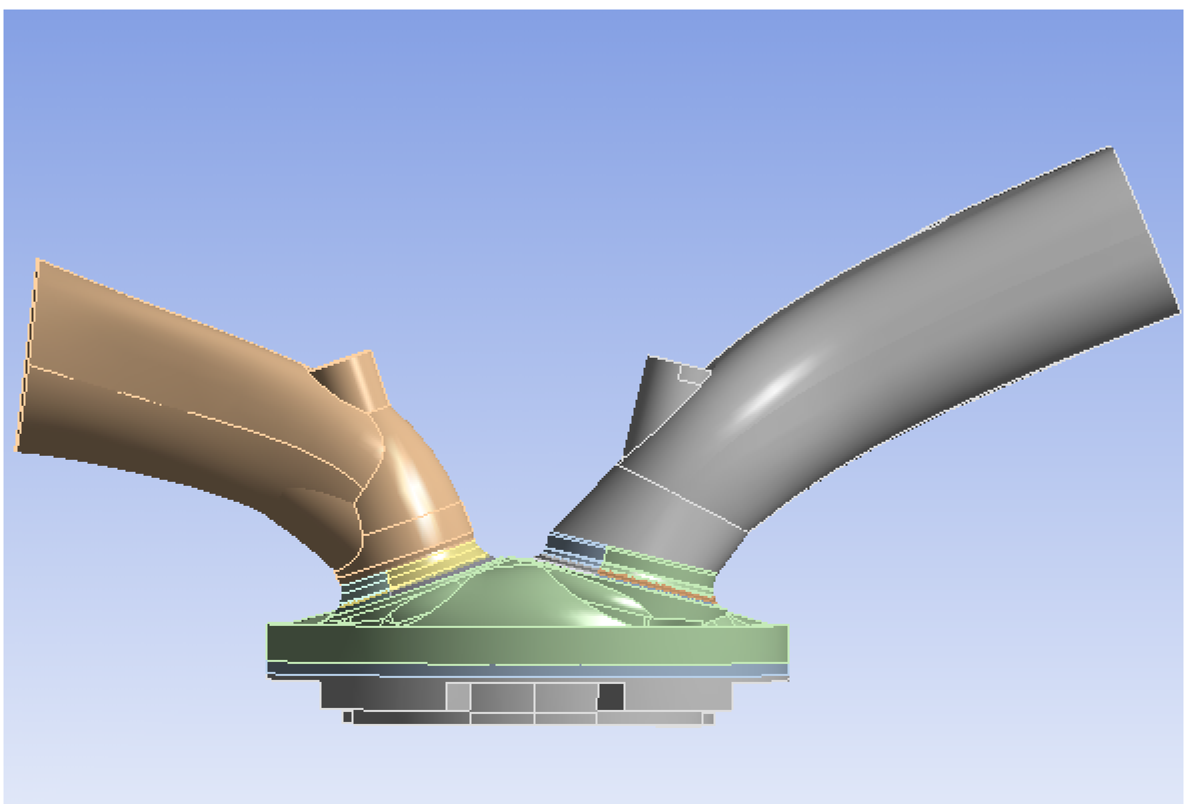
---

b. If after changing the values of the parameters you would like to go back to the default values, click **Reset**.

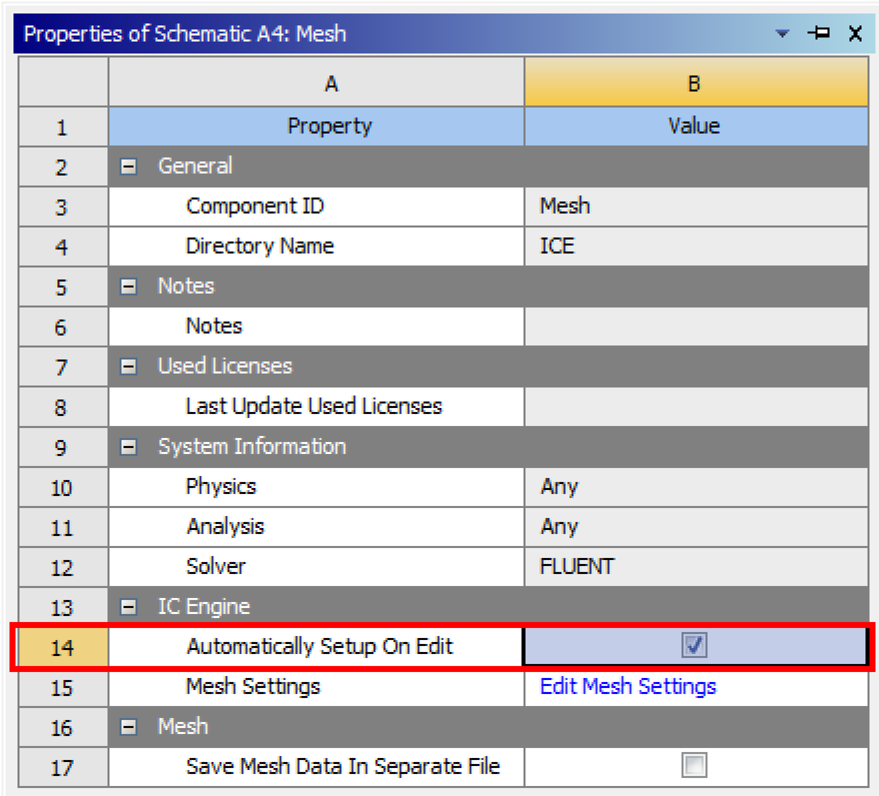
c. Click **OK** to set and close the **IC Mesh Parameters** dialog box.

Note:

Initially when you click **OK**, it creates all the mesh controls and virtual topologies. If you click **OK** again anytime later, it will just update the values and will not re-create any mesh controls or virtual topologies.



You can generate the mesh controls before opening Ansys Meshing application. To do this enable **Automatically Setup On Edit** under **IC Engine** in the **Properties** box which is displayed after selecting the **Mesh** cell in the Workbench window.



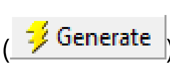
So after you double-click the **Mesh** cell to open Ansys Meshing application the mesh controls are already set.

3. Click **IC Generate Mesh** (

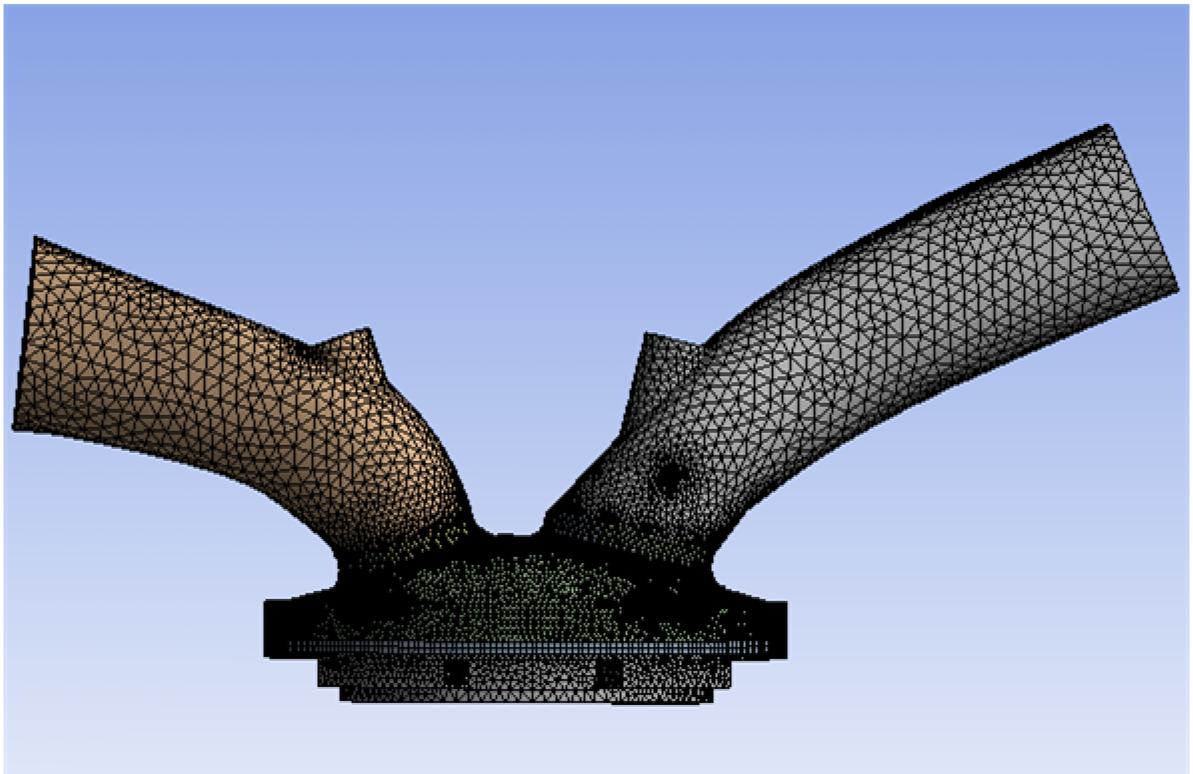
located in the **IC Engine** toolbar) to generate the mesh.

Note:

The order in which you create the mesh is important. So click **IC Generate Mesh** (

) instead of **Generate** (

). Clicking **IC Generate Mesh** creates meshes of parts in a specific order. For example, it creates a mesh of chambers first. This order is important, so that no pyramid cells are generated.



Note:

If you want to modify any of the mesh control parameters, you can change them even after the mesh is generated, by clicking **IC Setup Mesh**(

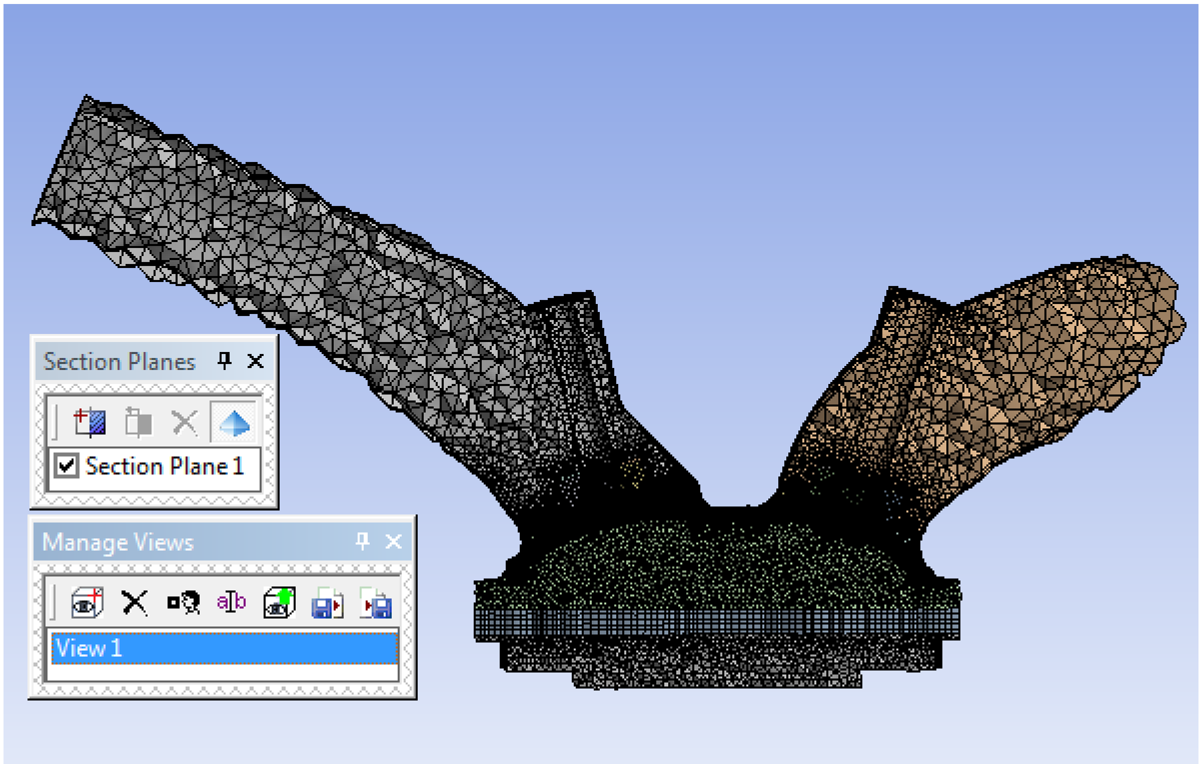
) icon. You will have to create the mesh again with the changed settings.

When you complete the setup through **IC Setup Mesh**, it does the following:

- It creates virtual topologies.
  - It calls the `Generate Virtual Cells` command, for ports (by suppressing all other parts).
  - It creates virtual cells for vlayer, ch bodies. This will help in creating a sweep mesh.
- It makes the required changes in **Global Mesh Settings**. (For details, refer toGlobal Mesh Settings for Cold Flow Simulation in **IC Engine** (p. 193)).
- It creates **Local Mesh Settings**. (For details, refer to Local Mesh Settings for Cold Flow Simulation in **IC Engine** (p. 201)).
- It creates pinch controls.
- Depending upon the number of valves, cut-planes and respective views are created by default. The **Section Planes** dialog box opens after meshing is completed. The number of slice planes

created are visible in the list. By enabling a **Slice Plane** you can observe the mesh at that cut-plane. Double-clicking on the respective **View** in the **Manage Views** dialog box will set the display.

Figure 5.1: Mesh at the Cut Plane



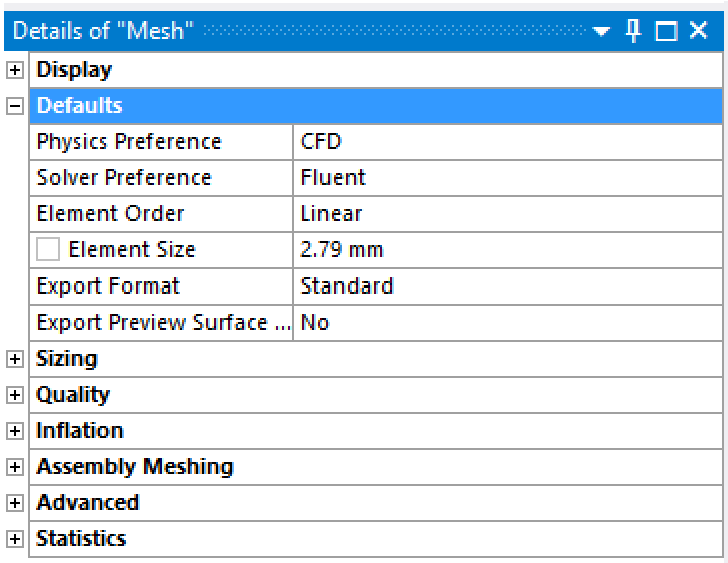
5.2. Global Mesh Settings for Cold Flow Simulation in IC Engine

IC Engine meshing tool creates **Global Mesh Settings** based on the information provided in the **IC Mesh Parameters** (p. 187) dialog box. Following sections describe these settings:

For more information on **Global Mesh Settings**, see Global Mesh Controls in the Meshing User's Guide.

5.2.1. Defaults Group

Under **Details of Mesh**, the following are the global mesh settings defined under **Defaults**.



- **Physics Preference**: This is set to **CFD**.

  This option allows you to establish how Workbench will perform meshing based on the physics of the analysis type that you specify.

- **Solver Preference**: Ansys Fluent is set as the solver for the mesh.

  Since **CFD** is chosen as your **Physics Preference**, it causes a **Solver Preference** option to appear in the **Details View** of the **Mesh** folder. The chosen value sets certain defaults that will result in a mesh that is more favorable to the respective solver.

- **Element Order**: This is set to **Linear**. This option allows you to control whether meshes are to be created with midside nodes (quadratic elements) or without midside nodes (linear elements). Reducing the number of midside nodes reduces the number of degrees of freedom. Choices include **Program Controlled**, **Linear**, and **Quadratic**.

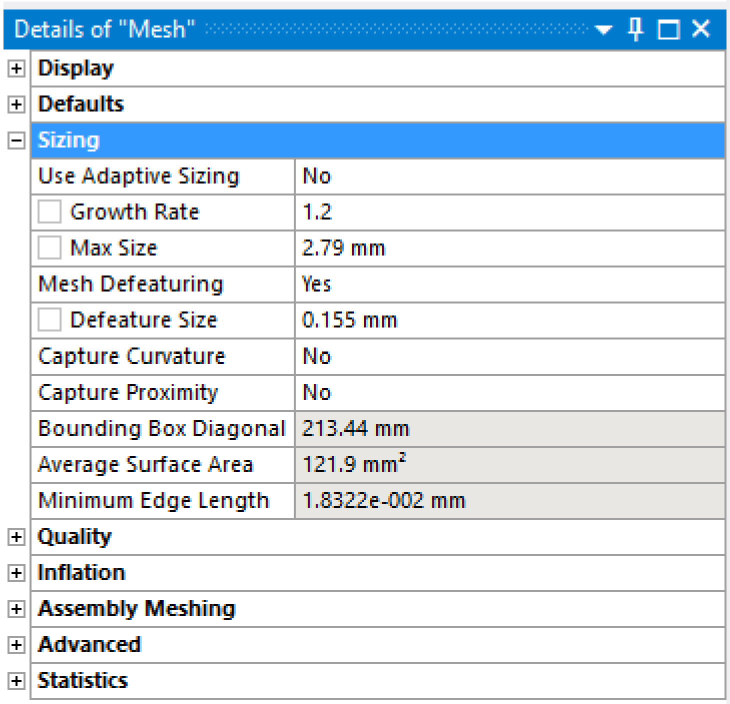
- **Element Size**: This allows you to specify the element size used for the entire model. This size will be used for all edge, face, and body meshing.

- **Export Format**: This option defines the format for the mesh when exported to Ansys Fluent. The default is **Standard**. You can change this to **Large Model Support** to export the mesh as a cell-based Fluent mesh.

- **Export Preview Surface Mesh** : This option controls the export of the preview surface mesh elements. This option can be used when the bodies have been meshed only partially, that is, not all volumes have been filled with elements and only previewing of surface meshes was done. The default is **No**, which results in export of only volume mesh elements to the Fluent mesh file. You can change this to Yes to export both the volume mesh and the preview surface meshes to the Fluent mesh file.

5.2.2. Sizing Group

Under the **Details of Mesh**, the following are the global mesh settings defined under **Sizing**.



- **Use Adaptive Sizing**: This option refers to a 2D curvature and proximity-based refinement approach which refines edges based on curvature and/or proximity but does not propagate the refined mesh along the face. When set to **Yes**, the mesher uses the value of the element size property to determine a starting point for the mesh size. The value of the element size property can be set by the user or automatically computed using defaults. When meshing begins, edges are meshed with this size initially, and then they are refined for curvature and 2D proximity. Next, mesh based defeaturing and pinch control execution occurs. The final edge mesh is then passed into a least-squares fit size function, which guides face and volume meshing.

- **Growth Rate**: It is equal to the value of **Growth Rate**, which is set in the **IC Mesh Parameters** (p. 187) dialog box.

- **Max Size**: It is equal to the value of **Max Mesh Size** set in the **IC Mesh Parameters** (p. 187) dialog box.

- **Mesh Defeaturing**: This option automatically defeatures small features and dirty geometry according to the **Defeature Size** you specify here.

- **Transition**: When **Use Adaptive Sizing** is set to **Yes**, this option affects the rate at which adjacent elements will grow. **Slow** produces smooth transitions while **Fast** produces more abrupt transitions.

- **Span Angle Center**: It is set to **Fine**.

  When **Use Adaptive Sizing** is set to **Yes**, this option sets the goal for curvature-based refinement. The mesh will subdivide in curved regions until the individual elements span this angle. The following choices are available:

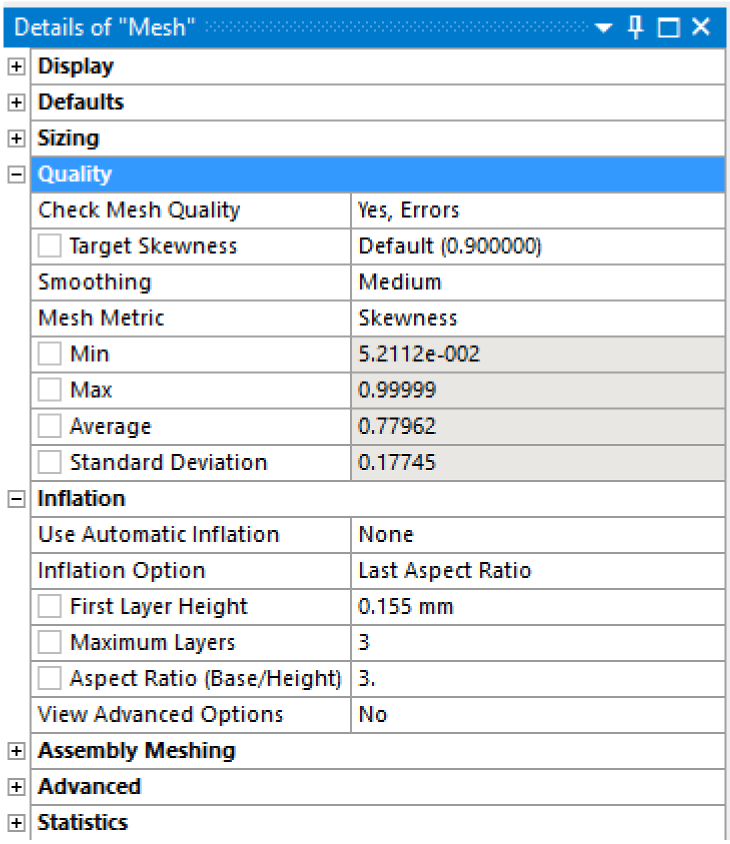
  - **Coarse** — 91^{o} to 60^{o}

- **Medium** — 75^{o} to 24^{o}
- **Fine** — 36^{o} to 12^{o}

- **Initial Size Seed**: When **Use Adaptive Sizing** is set to **Yes**, this option allows you to control the initial seeding of the mesh size for each part.
- **Bounding Box Diagonal**: This option provides a read-only indication of the length of the assembly diagonal.
- **Average Surface Area**: This option provides a read-only indication of the average surface area of the model.
- **Minimum Edge Length**: This option provides a read-only indication of the smallest edge length in the model.
- **Capture Curvature**: This option allows you to take into account curvature effects.
- **Curvature Min Size**: When **Capture Curvature** is set to **Yes**, this option is equal to the value of **Min Mesh Size**, which is set in the **IC Mesh Parameters** (p. 187) dialog box.
- **Curvature Normal Angle**: When **Capture Curvature** is set to **Yes**, this option is set to the value of the **Normal Angle** in the case of **Cold Flow Simulation**. This value is set in the **IC Mesh Parameters** (p. 187) dialog box.
- **Capture Proximity**: This option allows you to account for proximity effects.
- **Proximity Min Size**: When **Capture Proximity** is set to **Yes**, this option allows you to specify a global minimum size to be used in proximity sizing calculations.
- **Num Cells Across Gap**: When **Capture Proximity** is set to **Yes**, this option is the minimum number of layers of elements to be generated in the gaps. You can specify a value from 1 to 100, or accept the default (3).
- **Proximity Size Function Sources**: When **Capture Proximity** is set to **Yes**, this option determines whether regions of proximity between faces and/or edges are considered when proximity size function calculations are performed. You can specify **Edges**, **Faces**, or **Faces and Edges**.

5.2.3. Quality Group

Quality is useful for configuring mesh quality.



- **Check Mesh Quality**: This option determines how the software behaves with respect to error and warning limits

- **Target Skewness**: This option allows you to set a target skewness that you would like the mesh to satisfy.

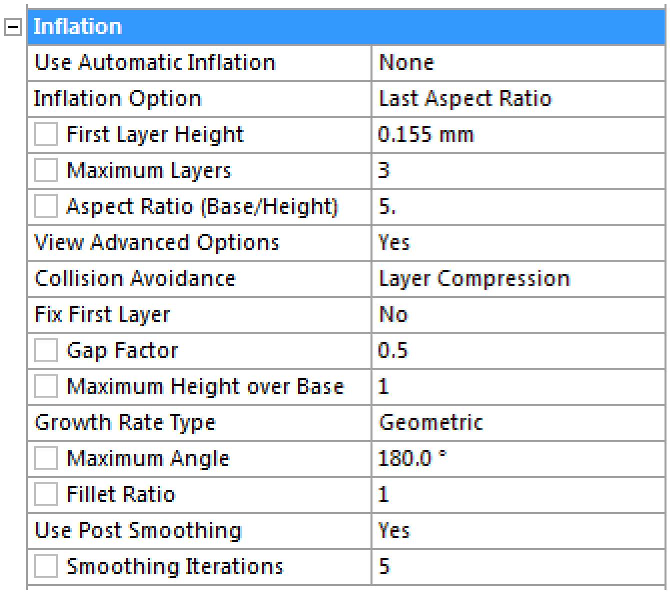
- **Smoothing**: This option attempts to improve element quality by moving locations of nodes with respect to surrounding nodes and elements. The **Low**, **Medium**, or **High** option controls the number of smoothing iterations along with the threshold metric where the mesher will start smoothing.

- **Mesh Metric**: This option allows you to view mesh metric information and thereby evaluate the mesh quality.

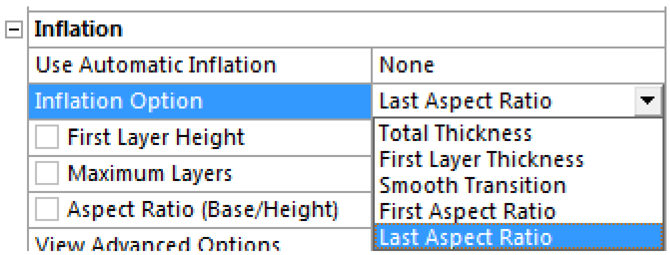
  When you select a mesh metric, its **Min**, **Max**, **Average**, and **Standard Deviation** values are reported in the Details View, and a bar graph is displayed under the Geometry window. The graph is labeled with color-coded bars for each element shape represented in the model's mesh, and can be manipulated to view specific mesh statistics of interest.

5.2.4. Inflation Group

Inflation is useful for CFD boundary layer resolution.



- **Inflation Option**: It is set to **Last Aspect Ratio** for **Cold Flow Simulation**. This setting determines the heights of the inflation layers.



  – The **Last Aspect Ratio** option creates inflation layers using the values of the **First Layer Height**, **Maximum Layers**, and **Aspect Ratio (Base/Height)** controls to generate the inflation mesh.

    → **First Layer Height**: It is equal to (**Reference Size**/**Number of Inflation Layers**), both of which are set in the **IC Mesh Parameters** (p. 187) dialog box.

    This control determines the height of the first inflation layer. This first inflation layer consists of a single layer of prism elements that is formed against the faces of the inflation boundary. You must enter a value for this control, and it must be greater than 0.

    → **Maximum Layers**: It is equal to the value of **Number of Inflation Layers**, which is set in the **IC Mesh Parameters** (p. 187) dialog box.

    This control determines the maximum number of inflation layers to be created in the mesh. Valid values are from 1 to 1000. The default is 5 for solid bodies, and 2 for surface bodies.

    → **Aspect Ratio (Base/Height)**: This value is set to `10`.

    This is defined as the ratio of the local inflation base size to the inflation layer height. The value should be between `0.5` and `20`.

- **View Advanced Options**: This control determines whether advanced inflation options appear in the **Details View**. Choices are **No** and **Yes**. It is set to **Yes** for **Cold Flow Simulation**. When this control is set to **Yes**, the following options are available:

  Collision Avoidance

  Maximum Height over Base

  Growth Rate Type

  Maximum Angle

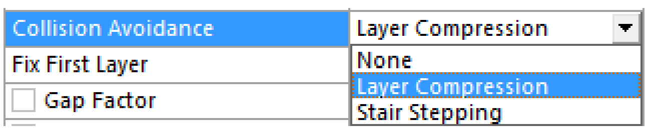
  Fillet Ratio

  Use Post Smoothing

  Smoothing Iterations

  – **Collision Avoidance**: It is set to **Layer Compression**.

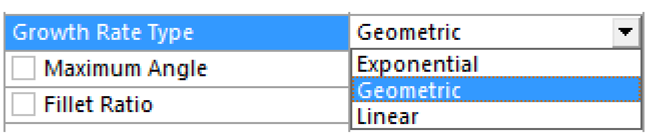
    This control determines the approach that is to be taken in areas of proximity to avoid collisions that may occur from marching inflated surface meshes from opposite sides into each other.

    The **Layer Compression** option compresses inflation layers in areas of collision. In these areas, the defined heights and ratios are reduced to ensure the same number of layers throughout the entire inflation region. Generally, this option is best for avoiding the creation of pyramids in the mesh.

  – **Growth Rate Type**: It is set to **Geometric**.

    This control determines the height of the inflation layers, given the initial height and height ratio.

    With this option, the prism height of a particular layer is defined by

    $h \times r^{(n-1)}$

    where

    h is the initial height

    r is the height ratio

    n is the layer number

    The total height at layer n is:

    $h(1-r^n)/(1-r)$

  – **Maximum Angle**: It is set to 180.

The **Maximum Angle** control determines prism layer growth around angles, and when prisms will adhere (project) to adjacent surfaces/walls. If the inflated mesh involves extruding from one surface and not its neighbor, and the angle between the two surfaces is less than the specified value, the prisms (sides) will adhere (project) to the adjacent wall.

5.2.5. Advanced Group

The **Advanced** group provides some advanced meshing features.

Advanced	
Number of CPUs for Parallel ...	Program Controlled
Straight Sided Elements	
Rigid Body Behavior	Dimensionally Reduced
Triangle Surface Mesher	Program Controlled
Topology Checking	Yes
Pinch Tolerance	0.1 mm
Generate Pinch on Refresh	No

- **Number of CPUs for Parallel Part Meshing**: This option sets the number of processors to be used for parallel part meshing. Using the default for specifying multiple processors will enhance meshing performance on geometries with multiple parts. For parallel part meshing, the default is set to Program Controlled or 0. This instructs the mesher to use all available CPU cores. The default setting inherently limits 2 GB memory per CPU core. An explicit value can be specified between 0 and 256, where 0 is the default.

- **Straight Sided Elements**: This option specifies meshing to straight edge elements when set to **Yes**. This option may affect the placement of midside nodes if the **Element Order** option is set to **Quadratic**.

- **Rigid Body Behavior**: This option determines whether a full mesh is generated for a rigid body, rather than a surface contact mesh. **Rigid Body Behavior** is applicable to all body types. Valid values for **Rigid Body Behavior** are **Dimensionally Reduced** (generate surface contact mesh only) and **Full Mesh** (generate full mesh).

- **Triangle Surface Mesher**: This option determines which triangle surface meshing strategy will be used by patch conforming meshers. In general, the advancing front algorithm provides a smoother size variation and better results for skewness and orthogonal quality. This option is inaccessible when an assembly meshing algorithm is selected.

- **Topology Checking**: This option controls what happens when a user scopes an object (such as loads, boundary conditions, named selections and so on) to geometry (bodies, faces, edges, and vertices) after the mesh has been generated. If **Topology Checking** is set to **Yes** (default), the software will check to see if the scoped geometry has mesh properly associated to it. If the associations are incorrect, the scoping of the object will force the mesh to be out of date. The mesh would need to be re-generated to get proper associations. If the associations are correct, the scoping is performed without any change to the mesh and the mesh stays up to date. Set **Topology Checking** to **No** to avoid the checks and always keep the mesh up to date.

- **Pinch Tolerance**: This is equal to the value of **Pinch Tolerance**, which is set in the **IC Mesh Parameters** (p. 187) dialog box.

This control allows you to specify a tolerance for the Meshing application to use when it generates automatic pinch controls.

- **Generate Pinch on Refresh**: This option determines whether pinch controls will be regenerated following a change made to the geometry (such as a change made via a DesignModeler application operation such as a merge, connect, etc.). If **Generate Pinch on Refresh** is set to **Yes** and you change the geometry, all pinch controls that were created automatically will be deleted and recreated based on the new geometry. If **Generate Pinch on Refresh** is set to **No** and you update the geometry, all pinch controls related to the changed part will appear in the Tree Outline but will be flagged as undefined.

5.3. Local Mesh Settings for Cold Flow Simulation in IC Engine

Based on information provided in the **IC Mesh Parameters** dialog box, IC Engine meshing tool creates some mesh settings at local level. Following section describes these settings for cold flow simulation:

For more information on **Local Mesh Settings**, see Local Mesh Controls in the Meshing User's Guide.

5.3.1. Valve Region Meshing

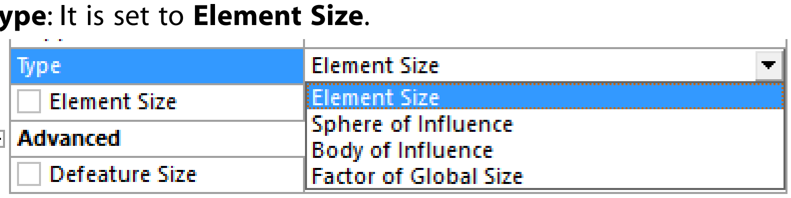
5.3.1.1. Valve Inboard

This corresponds to the part **ib**. **Body Sizing**, **Edge Sizing**, **Sweep Method**, and **Face Sizing** are the sizing methods used for these body parts.

Body Sizing

When you click **Body Sizing (ib)** under **Mesh** in the **Outline**, you can see the details.

- **Geometry**: It shows the number of selected parts for that body.
- **Type**: It is set to **Element Size**.



- **Element Size**: It is equal to the value of **IB Size**, which is set in the **IC Mesh Parameters** (p. 187) dialog box.
- **Behavior**: For this body part it is set to **Soft**.

Edge Sizing

When you click **Edge Sizing (ib)** under **Mesh** in the **Outline**, you can see the details.

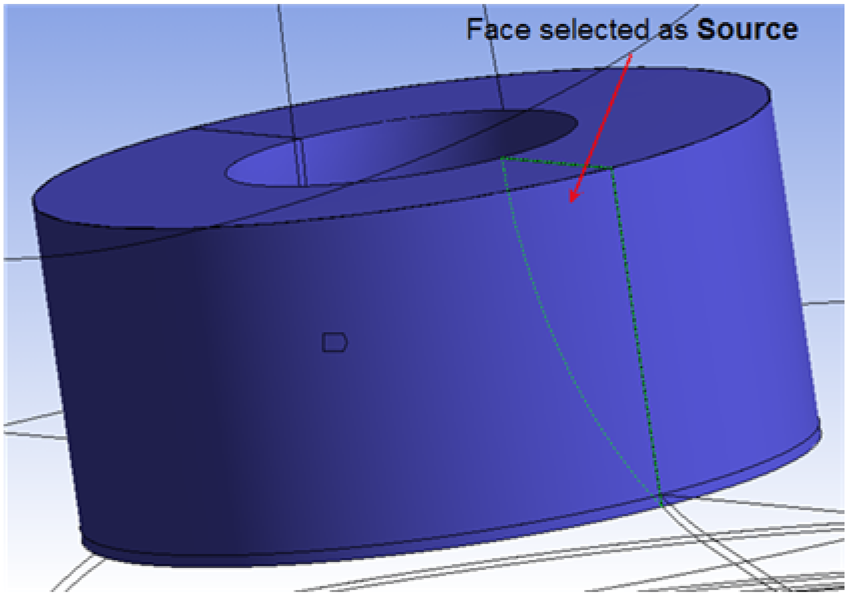
- **Geometry**: It shows the number of selected edges. (Two edges per **vlayer** on **ib**).
- **Type**: It is set to **Number of Divisions**.
- **Number of Divisions**: It is set to 2.
- **Behavior**: For this body part it is set to **Hard**.

Sweep Method

When you click **Sweep Method (valve-ib)** under **Mesh** in the **Outline**, you can see the details. In this method, a swept mesh is forced on the "sweepable" bodies.

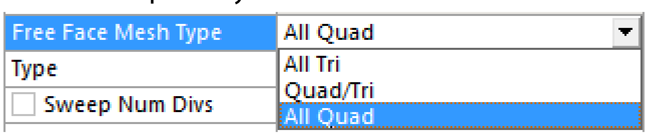
Details of "Sweep Method (invalve1-ib)" - Method	
Scope	
Scoping Method	Geometry Selection
Geometry	2 Bodies
Definition	
Suppressed	No
Method	Sweep
Algorithm	Program Controlled
Element Order	Use Global Setting
Src/Trg Selection	Manual Source
Source	1 Face
Target	Program Controlled
Free Face Mesh Type	All Quad
Type	Number of Divisions
Sweep Num Divs	Default
Element Option	Solid
Constrain Boundary	No
Advanced	
Sweep Bias Type	_____ ___ _ ___ _____
Sweep Bias	1.0

- **Geometry**: It shows the number of selected parts for that body.
- **Source**: It is the face selected. Here one vertical face of **ib** is selected as shown in the figure.

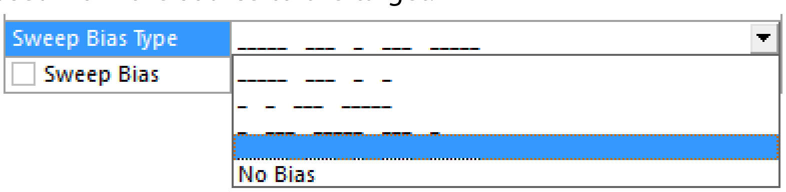


- **Target**: It is **Program Controlled**. This means that the program determines the target for the body.

- **Free Face Mesh Type**: It is set to **All Quad**. This determines the shape of the elements used to fill the swept body.



- **Sweep Bias Type**: It adjusts the spacing ratio of nodes on the edge. Biasing direction is based from the source to the target.



One of the four patterns available from the **Sweep Bias Type** drop-down list is chosen.

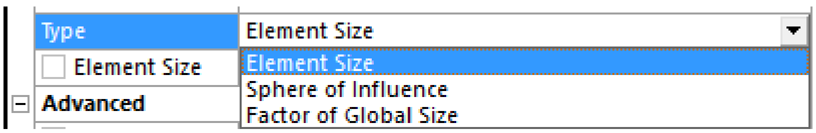
- **Sweep Bias**: This value is set to 1. This will override the mesh gradation in the swept region.

Face Sizing

Details of "Face Sizing (invalve1-ib-sweep-src)" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Suppressed	No
Type	Element Size
Element Size	0.465 mm
Advanced	
Defeature Size	Default (0.155 mm)
Influence Volume	No
Behavior	Hard
Capture Curvature	No
Capture Proximity	No

When you click **Face Sizing (in/ex valve-ib-sweep-src)** under **Mesh** in the **Outline**, you can see the details.

- **Geometry**: The split face of **ib** is selected as the input.
- **Type**: It is set to **Element Size**.



- **Element Size**: It is equal to the value of **IB Size**, which is set in the **IC Mesh Parameters** (p. 187) dialog box.
- **Behavior**: For this body part it is set to **Hard**.

5.3.1.2. Valve Vlayer Meshing

Edge Sizing, **Body Sizing**, and **Sweep Method** are the sizing methods used for mesh sizing control of this body part.

Edge Sizing

This sizing is used for the vertical edge of **vlayer**. When you click **Edge Sizing (in/exvalve-valyer-nbLayers)** under **Mesh** in the **Outline**, you can see the details.

- **Geometry**: This shows the number of selected edges. The circular circumferential edges are selected for this method.
- **Type**: It is set to **Number of Divisions**.
- **Number of Divisions**: This will depend upon the valve lift and layer approach set in the **Input Manager**. For crank angle 0, it is based on the **V Layer Approach** you select in the **Input Manager**. For other crank angles the value is equal to (**Valve Lift**/**Reference Size** X 0.4). It controls the number of layers in the V layer at different valve lifts. For one layer ap-

proach, only one layer is created at the lifts less than or equal to (**Reference Size** X 0.4). For four layer approach, four layers of equal height are created if the lift is less than or equal to (**Reference Size** X 1.6).

- **Behavior**: For this body part it is set to **Hard**.

Body Sizing

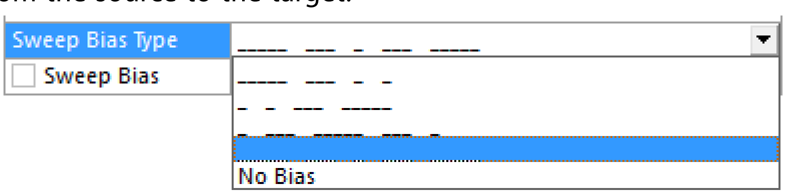
When you click **Body Sizing (vlayer)** under **Mesh** in the **Outline**, you can see the details.

- **Geometry**: This shows the number of **vlayer** bodies selected.
- **Type**: It is set to **Element Size**.
- **Element Size**: This is equal to the value of **V_Layer Size**, which is set in the **IC Mesh Parameters** (p. 187) dialog box.
- **Behavior**: For this body part it is set to **Soft**.

Sweep Method

In this method, a swept mesh is forced on the "sweepable" bodies. When you click **Sweep Method (in/ex valve-vlayer)** under **Mesh** in the **Outline**, you can see the details.

- **Geometry**: All the bodies for the part are selected.
- **Source**: The bottom faces of **vlayer** are selected. (One face per body.)
- **Target**: It is **Program Controlled**. This means that the program determines the target for the body.
- **Free Face Mesh Type**: It is set to **All Quad**. This determines the shape of the elements used to fill the swept body.
- **Sweep Bias Type**: This adjusts the spacing ratio of nodes the edge. Biasing direction is based from the source to the target.



One of the four patterns available from the **Bias Type** drop-down list is chosen.

- **Sweep Bias**: This value is set to 1. This will override the mesh gradation in the swept region.

Edge Sizing

This sizing is used for the peripheral edge of **vlayer**. When you click **Edge Sizing (in/ex valve-vlayer-nbdiv)** under **Mesh** in the **Outline**, you can see the details.

- **Geometry**: This shows the number of selected edges. The top and bottom edges of vertical faces are selected as the input.

- **Type**: It is set to **Number of Divisions**.
- **Number of Divisions**: It is calculated by dividing the larger edge length by (**V_Layer Size**×2/3). **V_Layer Size** is set in the **IC Mesh Parameters** (p. 187) dialog box.
- **Behavior**: For this body part it is set to **Hard**.

Edge Sizing

This sizing is used for the peripheral edge of **vlayer**. When you click **Edge Sizing (in/ex valve-vlayer-sizing)** under **Mesh** in the **Outline**, you can see the details.

- **Geometry**: This shows the number of selected edges. The top and bottom edges of vertical faces are selected as the input.
- **Type**: It is set to **Element Size**.
- **Element Size**: It is calculated as (**V_Layer Size** × 2/3). **V_Layer Size** is set in the **IC Mesh Parameters** (p. 187) dialog box.
- **Behavior**: For this body part it is set to **Hard**.

Edge Sizing

This sizing is used for the circular edge of **vlayer**. When you click **Edge Sizing (in/ex valve-vlayer-cir)** under **Mesh** in the **Outline**, you can see the details.

- **Geometry**: This shows the number of selected edges. The circular circumferential edges are selected for this method.
- **Type**: It is set to **Number of Divisions**.
- **Number of Divisions**: It is calculated as((100 × **MeshSize**/**V_Layer Size**)/2). **V_Layer Size** is set in the **IC Mesh Parameters** (p. 187) dialog box.

  Note:

  MeshSize is obtained from **CAD Attributes** in the **Details** dialog box of **piston-layer**.

- **Behavior**: For this body part it is set to **Hard**.

5.3.2. Port Region Meshing

5.3.2.1. Port

When you click **Inflation (port)** under **Mesh** in the **Outline**, you can see the details.

- **Boundary Scooping Method**: It is set to **Geometry Selection.**

- **Inflation Option**: It is set to **Last Aspect Ratio**. This setting determines the heights of the inflation layers.
  - **First Layer Height**: It is equal to (**Reference Size**/2 * **Number of Inflation Layers**) both of which are set in the **IC Mesh Parameters** (p. 187) dialog box. This control determines the height of the first inflation layer. This first inflation layer consists of a single layer of prism elements that is formed against the faces of the inflation boundary. You must enter a value for this control, and it must be greater than 0.
  - **Maximum Layers**: It is equal to **Number of Inflation Layers** set in the **IC Mesh Parameters** (p. 187) dialog box. This control determines the maximum number of inflation layers to be created in the mesh.
  - **Aspect Ratio**: Here this value is set to 5.
- **Inflation Algorithm**: It is set to **Pre**.

5.3.2.2. Inflation Port

When you click **Edge Sizing (inflation-port)** under **Mesh** in the **Outline**, you can see the details.

- **Geometry**: This shows the number of the selected edges. The edges from port bottom sweep bodies are selected.
- **Type**: It is set to **Number of Divisions**.
- **Number of Divisions**: It is equal to **Number of Inflation Layers** set in the **IC Mesh Parameters** (p. 187) dialog box.
- **Behavior**: For this body part it is set to **Hard**.

Note:

This part will only be created if **V Layer Approach** (p. 164) is set to **4 Layers** in the **Input Manager**.

5.3.2.3. Valve Port

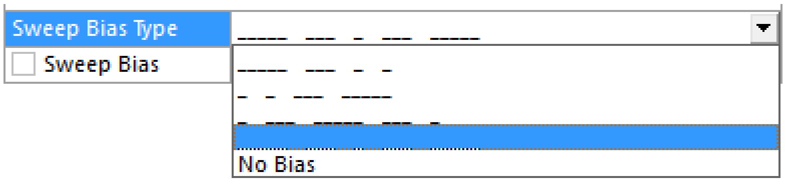
Sweep Method and **Face Sizing** are the sizing methods used for mesh sizing control of the body part.

Sweep Method

When you click **Sweep Method (in/ex valve-port)** under **Mesh** in the **Outline**, you can see the details. In this method, a swept mesh is forced on the “sweepable” bodies.

- **Source**: One vertical face is selected as the source.
- **Free Face Mesh Type**: It is set to **Quad/Tri**. This determines the shape of the elements used to fill the swept body.

- **Sweep Element Size**: It is equal to **V_Layer Size** as set in **IC Mesh Parameters** (p. 187) dialog box).
- **Sweep Bias Type**: It adjusts the spacing ratio of nodes on the edge. Biasing direction is based from the source to the target.



  One of the four patterns available from the **Bias Type** drop-down list is chosen.

- **Sweep Bias**: This value is set to `1`. This will override the mesh gradation in the swept region.

Note:

If during mesh generation the sweep mesh fails, then a tet mesh is grown for this region. In such a case the sweep method is suppressed.

Face Sizing

When you click **Face Sizing (in/ex valve-port-sweep-src)** under **Mesh** in the **Outline**, you can see the details.

- **Geometry**: The face from the port bottom is selected as the input.
- **Type**: It is set to **Element Size**.
- **Element Size**: It is set to **Reference Size**/4. (**Reference Size** is set in the **IC Mesh Parameters** (p. 187) dialog box).
- **Behavior**: For this part it is set to **Hard**.

5.3.2.4. Interface Between Port and Inboard

Face Sizing method is used for this part. When you click **Face Sizing (intf-port-ibVlayer)** under **Mesh** in the **Outline**, you can see the details.

- **Geometry**: The inside lower faces of **Vlayer** and **ib** are selected as the input.
- **Type**: It is set to **Element Size**.
- **Element Size**: It is equal to the value of **V_Layer Size**, set in the **IC Mesh Parameters** (p. 187) dialog box.
- **Behavior**: For this part it is set to **Soft**.

5.3.3. Chamber Meshing

When **Inflation in Chamber** is set to **Yes** in the **IC Mesh Parameters** (p. 187) dialog box, the inflation is created in the chamber.

- For canted valve case, inflation will be created on liner, dome, valve bottom, and vlayer-ch interface.
- For straight valve case inflation will be created on liner.

A new inflation control for ch faces is created.

Inflation (ch)

When you click **Inflation (ch)** under **Mesh** in the **Outline**, you can see the details.

- **Boundary Scooping Method**: It is set to **Geometry Selection.**
- **Inflation Option**: It is set to **Last Aspect Ratio**. This setting determines the heights of the inflation layers.
  - **First Layer Height**: It is equal to (**Reference Size**/4 * **Number of Inflation Layers**) both of which are set in the **IC Mesh Parameters** (p. 187) dialog box. This control determines the height of the first inflation layer. This first inflation layer consists of a single layer of prism elements that is formed against the faces of the inflation boundary. You must enter a value for this control, and it must be greater than 0.
  - **Maximum Layers**: It is equal to **Number of Inflation Layers** set in the **IC Mesh Parameters** (p. 187) dialog box. This control determines the maximum number of inflation layers to be created in the mesh.
  - **Aspect Ratio**: Here this value is set to 5.
- **Inflation Algorithm**: It is set to **Pre**.

Other than inflation, the approach for meshing the chamber region for straight and canted valves is different.

5.3.3.1. Straight Valve

A straight valve has additional parts and bodies. This section shows the automatic settings performed for them.

5.3.3.1.1. Chamber Upper Meshing

Interface Between Chamber Upper

Face Sizing is the sizing method used for this part. When you click **Face Sizing (intf-ch-upper)** under **Mesh** in the **Outline**, you can see the details.

- **Geometry**: The **ch-upper** faces are selected as the input.
- **Type**: It is set to **Element Size**.

- **Element Size**: It is equal to **Reference Size**/2. **Reference Size** is set in the **IC Mesh Parameters** (p. 187) dialog box.
- **Behavior**: For this body part it is set to **Soft**.

ch lower,upper,valve

When you click **Body Sizing (ch lower,upper,valve)** under **Mesh** in the **Outline**, you can see the details.

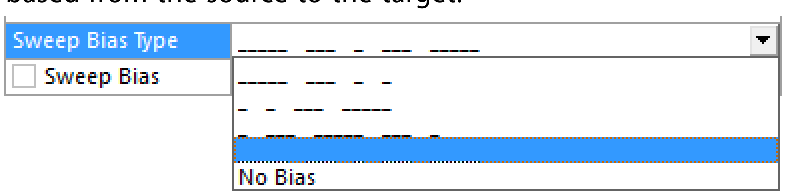
- **Geometry**: This shows the number of bodies selected (**ch-upper, ch-lower, ch-in-valve1–body, ch-exvalve1–body**).
- **Type**: It is set to **Element Size**.
- **Element Size**: It is equal to **Reference Size**, which is set in the **IC Mesh Parameters** (p. 187) dialog box.
- **Behavior**: For this body part it is set to **Soft**.

5.3.3.1.2. Chamber Lower Meshing

ch-lower

For this body part **Sweep Method** is used. Click **Sweep Method (ch-lower)** under **Mesh** in the **Outline**, to see the details.

- **Geometry**: **ch-lower** body part is used as the input.
- **Source**: The upper face of **ch-lower** is used.
- **Target**: It is **Program Controlled**. This means that the program determines the target for the body.
- **Free Face Mesh Type**: It is set to **All Quad**. This determines the shape of the elements used to fill the swept body.
- **Sweep Bias Type**: This adjusts the spacing ratio of nodes on the edge. Biasing direction is based from the source to the target.

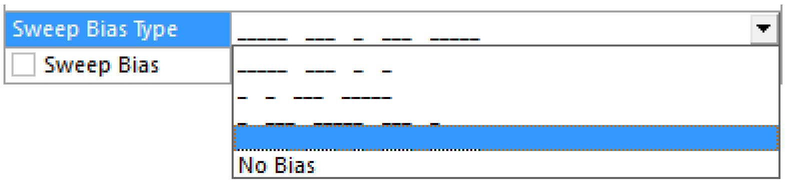
  

  One of the four patterns available from the **Bias Type** drop-down list is chosen.
- **Sweep Bias**: This value is set to 1. This will override the mesh gradation in the swept region.

ch-valve

For this body part **Sweep Method** is used. Click **Sweep Method (ch-valve)** under **Mesh** in the **Outline**, to see the details.

- **Geometry**: **ch-valve** body part is used as the input.
- **Source**: The upper face of **ch-valve** is used.
- **Target**: It is **Program Controlled**. This means that the program determines the target for the body.
- **Free Face Mesh Type**: It is set to **All Quad**. This determines the shape of the elements used to fill the swept body.
- **Sweep Bias Type**: It adjusts the spacing ratio of nodes on the edge. Biasing direction is based from the source to the target.

  One of the four patterns available from the **Bias Type** drop-down list is chosen.

- **Sweep Bias**: This value is set to `1`. This will override the mesh gradation in the swept region.

Interface Between Piston And Chamber

Face Sizing is the sizing method used for this part. When you click **Face Sizing (intf-piston-ch)** under **Mesh** in the **Outline**, you can see the details.

- **Geometry**: The upper face of the piston body is selected as the input.
- **Type**: It is set to **Element Size**.
- **Element Size**: It is equal to **Reference Size**, which is set in the **IC Mesh Parameters** (p. 187) dialog box.
- **Behavior**: For this body part it is set to **Hard**.

5.3.3.1.3. Chamber Valves

Face Sizing is the sizing method used for this part. When you click **Face Sizing (ch-sweep-source)** under **Mesh** in the **Outline**, you can see the details.

- **Geometry**: The upper faces of **ch-invalve**, **ch-exvalve**, and **ch-upper** are selected as the input.
- **Type**: It is set to **Element Size**.
- **Element Size**: It is equal to **Reference Size**, which is set in the **IC Mesh Parameters** (p. 187) dialog box.
- **Behavior**: For this body part it is set to **Hard**.

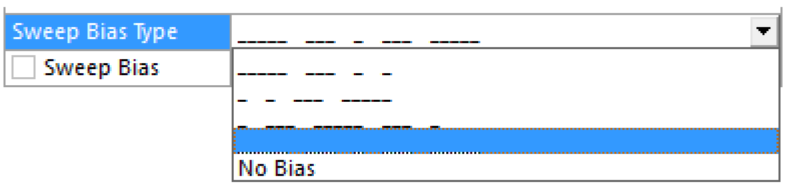
5.3.3.2. Canted Valve

This section shows the chamber region meshing performed for canted valves.

layer-cylinder

Click **Sweep Method (layer-cylinder)** under **Mesh** in the **Outline**, to see the details.

- **Geometry**: **layer-cylinder** body part is used as the input.
- **Source**: It is **Program Controlled**.
- **Target**: It is **Program Controlled**. This means that the program determines the target for the body.
- **Free Face Mesh Type**: It is set to **All Quad**. This determines the shape of the elements used to fill the swept body.
- **Sweep Bias Type**: It adjusts the spacing ratio of nodes on the edge. Biasing direction is based from the source to the target.

  One of the four patterns available from the **Bias Type** drop-down list is chosen.

- **Sweep Bias**: This value is set to 1. This will override the mesh gradation in the swept region.

valves-bottom

Click **Face Sizing(valves-bottom)** under **Mesh** in the **Outline**, to see the details.

- **Type**: It is set to **Element Size**.
- **Element Size**: It is equal to **V_Layer Size**, which is set in the **IC Mesh Parameters** (p. 187) dialog box.
- **Behavior**: For this body part it is set to **Soft**.

5.3.3.2.1. Interface Between Chamber and Vlayer

For engines without chamber decomposition only **intf-vlayer-ch** part is present.

intf-vlayer-ch

Face Sizing is the sizing method used for this part. When you click **Face Sizing (intf-vlayer-ch)** under **Mesh** in the **Outline**, you can see the details.

- **Geometry**: The faces created by **Valve_Margin_Slice** are selected as the input.
- **Type**: It is set to **Element Size**.
- **Element Size**: It is equal to **Chamber-V_Layer Interface Size**, which is set in the **IC Mesh Parameters** (p. 187) dialog box.
- **Behavior**: For this body part it is set to **Soft**.

intf-ch-vlayer-top-edges

Edge Sizing is the sizing method used for this part. When you click **Edge Sizing (intf-ch-vlayer-top-edges)** under **Mesh** in the **Outline**, you can see the details.

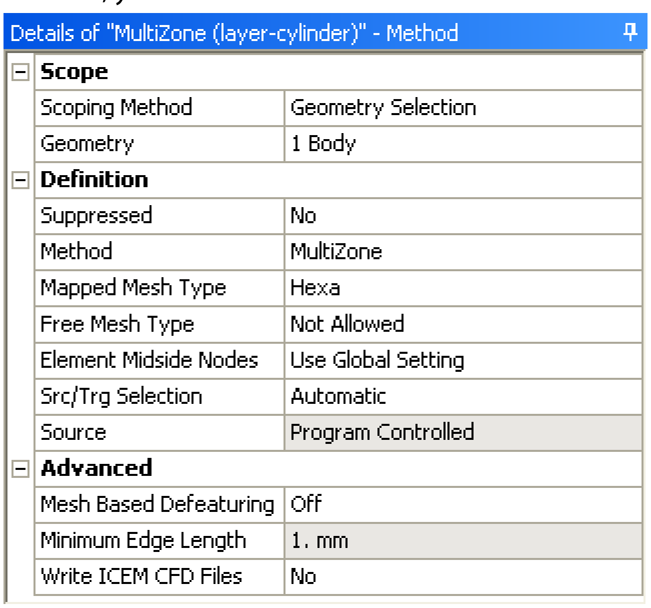
- **Geometry**: Top edges of **Valve_Margin_Slice** are selected as the input for **Geometry**.
- **Type**: It is set to **Element Size**.
- **Element Size**: It is equal to (0.6 X **Minimum Lift**) as defined in the **Properties of Schematic** (p. 141) window of IC Engine System.
- **Behavior**: For this body part it is set to **Hard**.

5.3.3.2.2. layer-cylinder

For this body part either one of **Sweep Method** or **MultiZone**, and **Body Sizing** methods are used. If the number of faces in the int-piston named selection is greater than one, then the method set is **MultiZone**, else it is **Sweep Method**.

Sweep Method / MultiZone

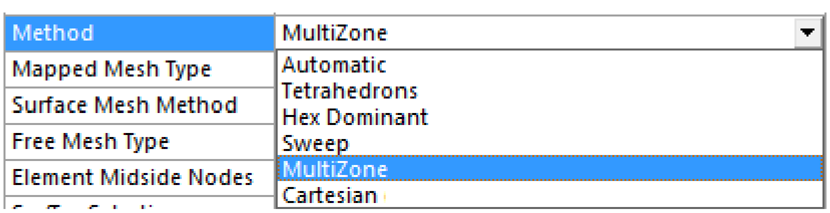
If **MultiZone** method is used, then selecting **MultiZone (layer-cylinder)** under **Mesh** in the **Outline**, you can see the details.



Details of "MultiZone (layer-cylinder)" - Method

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Method	MultiZone
Mapped Mesh Type	Hexa
Free Mesh Type	Not Allowed
Element Midside Nodes	Use Global Setting
Src/Trg Selection	Automatic
Source	Program Controlled
Advanced	
Mesh Based Defeaturing	Off
Minimum Edge Length	1. mm
Write ICEM CFD Files	No

- **Geometry**: This shows the body selected.
- **Method**: **MultiZone** is the method selected.



If **Sweep Method** is used, then selecting **Sweep Method (layer-cylinder)** under **Mesh** in the **Outline**, you can see the details.

Details of "Sweep Method (layer-cylinder)" - Method

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Method	Sweep
Element Midside Nodes	Use Global Setting
Src/Trg Selection	Automatic
Source	Program Controlled
Target	Program Controlled
Free Face Mesh Type	All Tri
Type	Element Size
Sweep Element Size	0.947 mm
Sweep Bias Type	_____ ___ _ ___ _____
Sweep Bias	1.
Element Option	Solid
Constrain Boundary	No

- **Geometry**: This shows the body selected.
- **Method**: **Sweep** is the method selected.

Body Sizing

When you click **Body Sizing (layer-cylinder)** under **Mesh** in the **Outline**, you can see the details.

- **Geometry**: This shows the body selected.
- **Type**: It is set to **Element Size**.
- **Element Size**: It is equal to (1.5 X **Reference Size**), which is set in the **IC Mesh Parameters** (p. 187) dialog box.
- **Behavior**: For this body part is it set to **Soft**.

5.3.3.2.3. Piston

When you click **Body Sizing (piston)** under **Mesh** in the **Outline**, you can see the details.

- **Geometry**: This shows the body selected.
- **Type**: It is set to **Element Size**.
- **Element Size**: It is equal to **Chamber Size**, which is set in the **IC Mesh Parameters** (p. 187) dialog box.
- **Behavior**: For this body part is it set to **Soft**.

5.3.4. Crevice

When the geometry has a crevice and you select **Yes** for **Crevice Option** in the **Input Manager** dialog box then some mesh controls are created for the crevice.

CylCrevice

When you click **Face Sizing(CylCrevice)** under **Mesh** in the **Outline**, you can see the details.

- **Geometry**: This shows the face selected of the crevice.
- **Type**: It is set to **Element Size**.
- **Behavior**: For this body part is it set to **Hard**.

crevice

Click **Sweep Method (crevice)** under **Mesh** in the **Outline**, to see the details.

- **Geometry**: **crevice** body part is used as the input.
- **Source**: It is **Program Controlled**.
- **Free Face Mesh Type**: It is set to **Quad/Tri**. This determines the shape of the elements used to fill the swept body.
- **Sweep Num Divs**: It is equal to the **Number of Layers Crevice** which is set in the **IC Mesh Parameters** (p. 187) dialog box.

intf-crevcie-piston

When you click **Face Sizing(intf-crevcie-piston)** under **Mesh** in the **Outline**, you can see the details.

- **Geometry**: This shows the face selected.
- **Type**: It is set to **Element Size**.
- **Element Size**: It is equal to **Crevice-Piston Interface Size**, which is set in the **IC Mesh Parameters** (p. 187) dialog box.
- **Behavior**: For this body part is it set to **Hard**.

Chapter 6: Cold Flow Simulation: Setting Up the Analysis

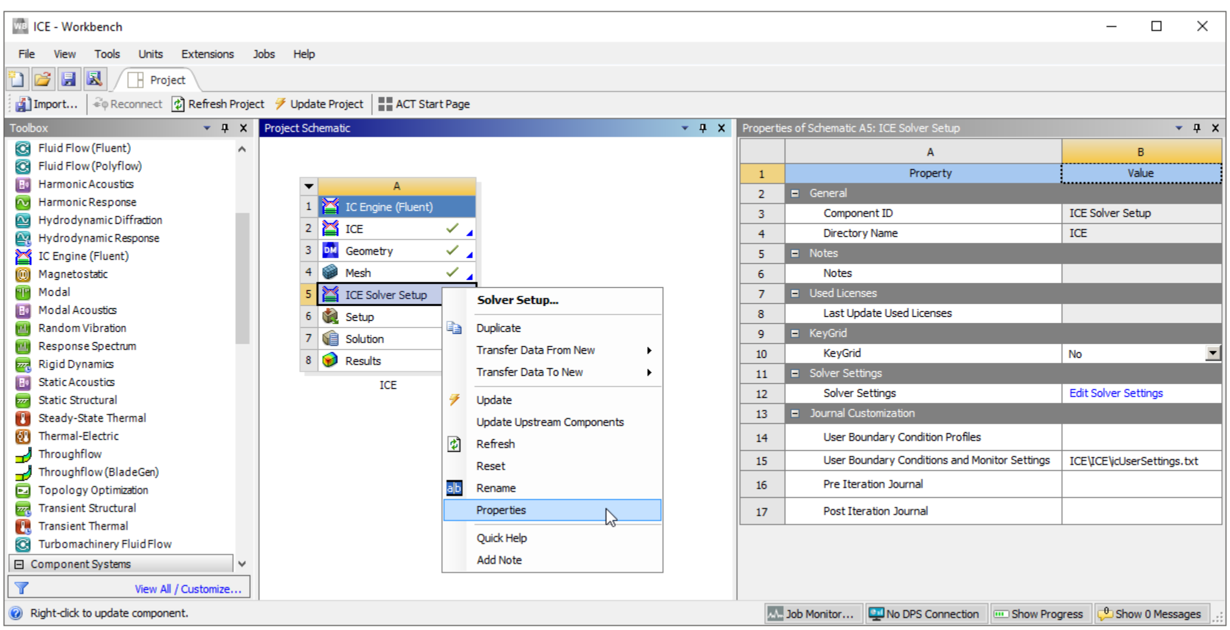
This chapter describes how to configure the analysis for a cold flow simulation.

In the **IC Engine (Fluent)** analysis system, the boundary conditions and solver parameters are set as per the **Decomposition Crank Angle** set in the **Input Manager**. All the settings described in this chapter are done automatically, but you can change them by accessing the **Solver Settings** dialog box or the task pages and the dialog boxes in Ansys Fluent.

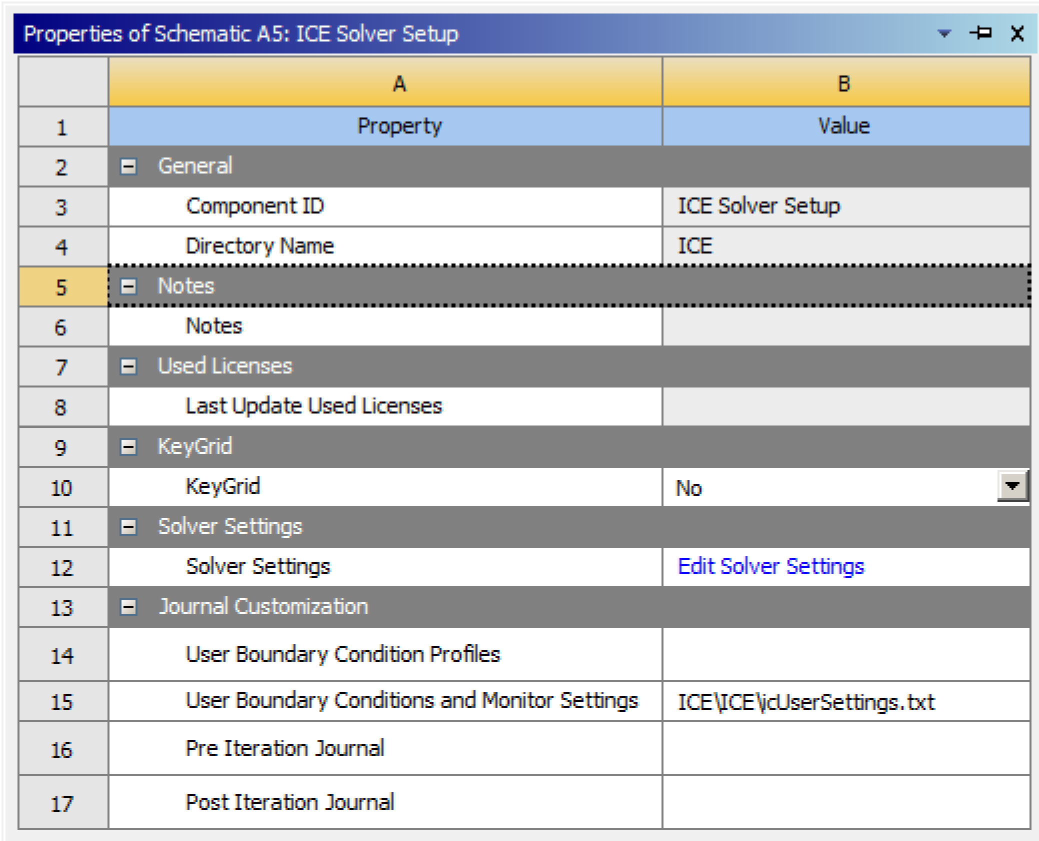
> **Note:**
>
> The piston should be at TDC while decomposing even though you want to start the simulation from an angle other than zero.

After meshing you can set up the solver. In the **Properties** pane of the **ICE Solver Setup** cell you can make changes to the solver settings or set the keygrid crank angles.

6.1. ICE Solver Settings in IC Engine



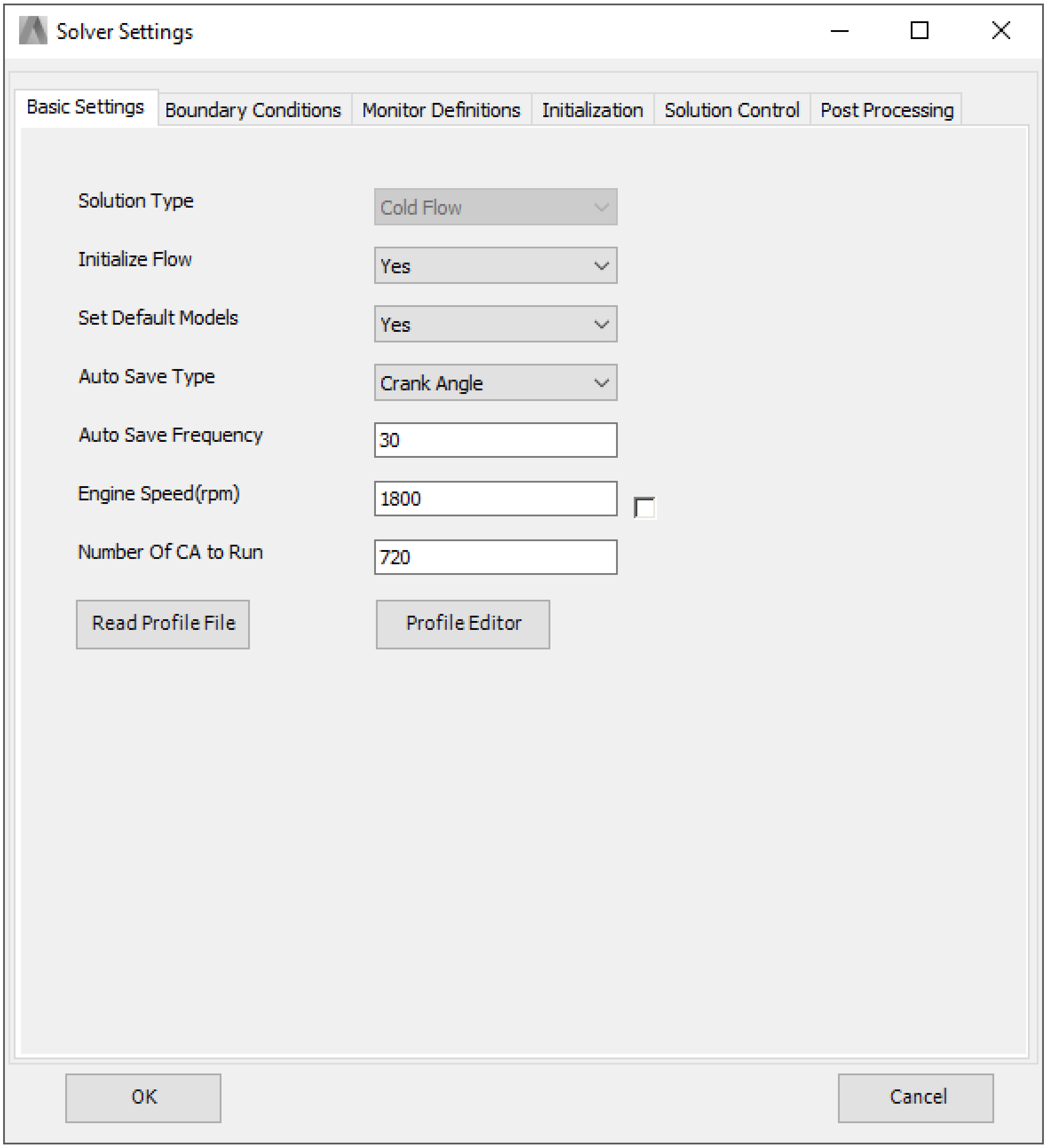
Click **Edit Solver Settings** to open the **Solver Settings** dialog box.



Properties of Schematic A5: ICE Solver Setup

	A	B
1	Property	Value
2	General	
3	Component ID	ICE Solver Setup
4	Directory Name	ICE
5	Notes	
6	Notes	
7	Used Licenses	
8	Last Update Used Licenses	
9	KeyGrid	
10	KeyGrid	No
11	Solver Settings	
12	Solver Settings	Edit Solver Settings
13	Journal Customization	
14	User Boundary Condition Profiles	
15	User Boundary Conditions and Monitor Settings	ICE\ICE\icUserSettings.txt
16	Pre Iteration Journal	
17	Post Iteration Journal	

6.1.1. Basic Settings



In the **Basic Settings** tab you have the following settings:

Solution Type

shows the **Solution Type** you have selected in the **Properties** pane of the **ICE** cell. You cannot make any changes here.

Initialize Flow

is set to **Yes** by default. This will initialize the flow. To check the initialization settings see the **Solution Initialization** task page in Ansys Fluent.

Set Default Models

is set to **Yes** by default. For cold flow simulation some models of Ansys Fluent have been chosen as default for better results. These are the **Energy** model, and the **Standard k-epsilon** model from the list of **Viscous** models with **Standard Wall Functions**. You can check them at the **Models** (p. 245) task page of Ansys Fluent.

Auto Save Type

is set to **Crank Angle** by default. This means that the intermediate case and data files will be saved at the entered frequency crank angles. You can also select **Time** from the drop-down menu if you want to save the case and data files at a specific frequency of time steps.

Auto Save Frequency

shows the number of crank angle or time steps, after which the case and data file will be saved in Ansys Fluent. This will depend upon your selection of **Auto Save Type**. Here the default value is **30**.

Engine Speed(rpm)

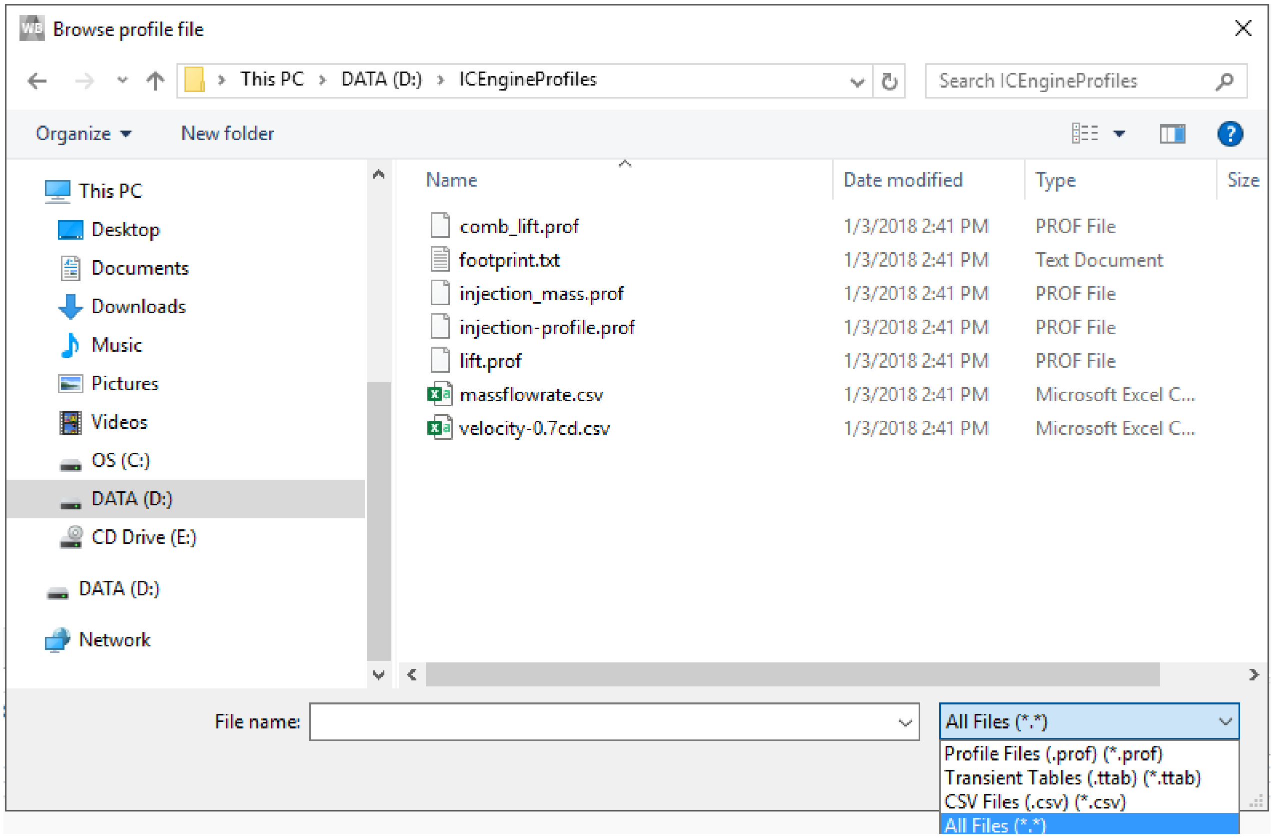
It is set to a default value of **1800**. The engine speed is used along with the crank angle step size to calculate the time step size. You can enter the speed of your engine here. You can enable the check box next to it to parameterize it.

Number of CA to run

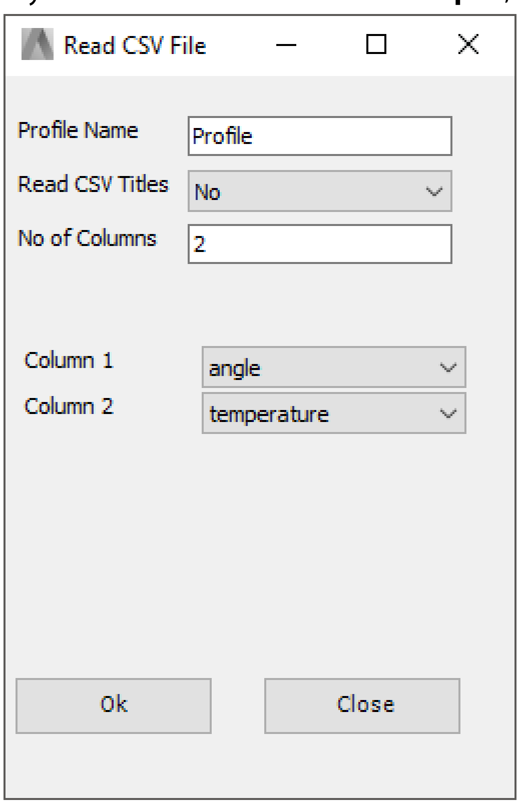
is set to **720** by default. This corresponds to a full cycle. You can enter the number of crank angles you want to run the simulation

Read Profile File

allows you to read multiple profile files. Clicking on **Read Profile File** opens **Browse profile file** dialog box. In the browsing window three types of file extensions will be supported `.prof` (profile file), `.ttab` (transient table), and `.csv`. **All Files** option is also present if you have a file extension which does not match with either of the given extensions. In this case the file type will be automatically identified, and an error will be thrown if the format is not supported.



If you select a `.csv` file and click **Open**, then a **Read CSV File** dialog box opens.



- You need to enter a name for **Profile Name**. This name will appear in the drop-down list of **Profiles** in the **Profile Editor** dialog box.

- If you retain the default setting of **Yes** for **Read CSV Titles** then the quantity or variable names will be as per the names in the CSV file.
- If you choose **No** for **Read CSV Titles** then you have to specify the **No. of Columns** of the CSV file you want to read. For each column you have to select a different variable name from the drop-down list. In this case the titles of the CSV columns will not be read. Your selections for the columns will be the titles.

Important:

- All the columns in CSV should have same number of values. Variable number of values and interpolation is not supported in the current version.
- Ensure that there are no empty spaces in the titles.

The format of the standard profile file is

```
((profile-name transient n periodic?)
(field_name-1 a1 a2 a3 .... an)
(field_name-2 b1 b2 b3 .... bn)
.
.
.
.
(field_name-r r1 r2 r3 .... rn))
```

The profile name as well as the field names have to be shorter than 64 characters. One of the `field_name` should be used for the `time` field, and the `time` field section *must* be in ascending order. `n` is the number of entries per field. The `periodic?` entry indicates whether or not the profile is time-periodic. Set it to `1` for a time-periodic profile, or `0` if the profile is not time-periodic.

An example is shown below:

```
((sampleprofile transient 8 0 0)
(angle
4.400000e+02 4.412000e+02
4.644900e+02 4.656900e+02
6.400000e+02
6.412000e+02 6.480800e+02 6.492800e+02)
(mass-flow
0.000000e+00 2.450040e-03 2.450040e-03 0.000000e+00 0.000000e+00
2.475870e-03 2.475870e-03 0.000000e+00)
(velocity
0.000000e+00 1.941520e+02 1.941520e+02 0.000000e+00 0.000000e+00
1.922430e+02 1.922430e+02 0.000000e+00)
)
```

Important:

All quantities, including coordinate values, must be specified in SI units because Ansys Fluent does not perform unit conversion when reading profile files. Also, profile names must have all lowercase letters (for example, `name`). Uppercase letters in profile names are not acceptable.

The format of the transient table file is

```
profile-name n_field n_data periodic?
field-name-1 field-name-2 field-name-3 .... field-name-n_field
v-1-1  v-2-1... ... ... ... v-n_field-1
v-1-2  v-2-2... ... ... ... v-n_field-2
.
.
.
.
.
v-1-n_data v-2-n_data ... ... ... ... v-n_field-n_data
```

The first field name (for example `field-name-1`) should be used for the `time` field, and the `time` field section, which represents the flow time, *must* be in ascending order. The `periodic?` entry indicates whether or not the profile is time-periodic. Set it to `1` for a time-periodic profile, or `0` if the profile is not time-periodic.

An example is shown below:

```
sampletprofile 3 8 0
angle        mass-flow    velocity
440          0.0          0.0
441.2        0.00245004   194.152
464.49       0.00245004   194.152
465.69       0.0          0.0
640          0.0          0.0
641.2        0.00247587   192.243
648.08       0.00247587   192.243
649.28       0.0           0.0
```

This file defines the same transient profile as the standard profile example above.

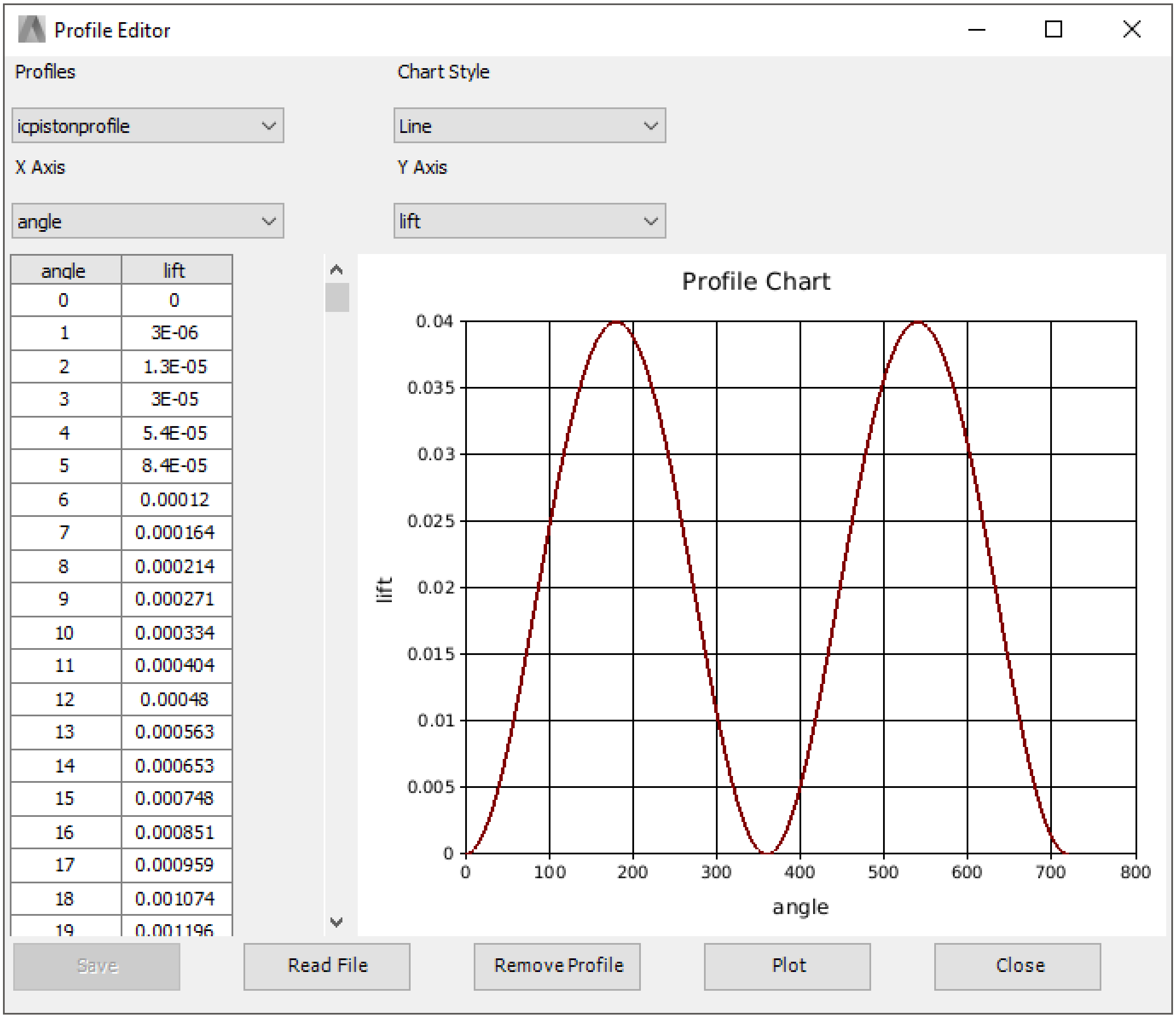
Important:

All quantities, including coordinate values, must be specified in SI units because Ansys Fluent does not perform unit conversion when reading profile files. Also, profile names must have all lowercase letters (for example, `name`). Uppercase letters in profile names are not acceptable. When choosing the field names, spaces or parentheses should not be included.

After reading the files the profiles will be available in the boundary condition drop-down lists.

Profile Editor

allows you to view the plot of the profiles.



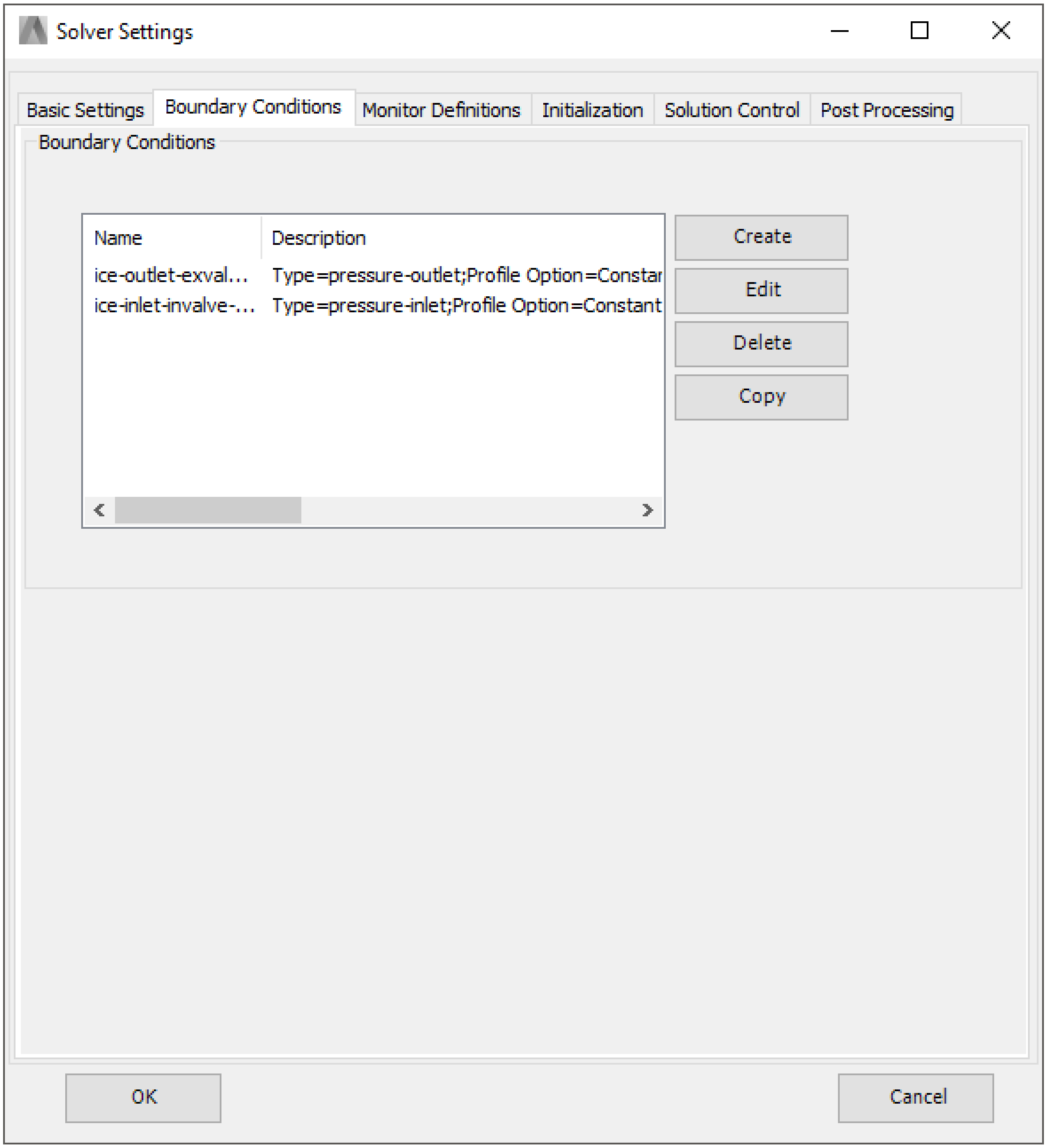
- Select a profile from the drop-down list of **Profiles**. The **TIMESTEP** profile is added to the list by default.
- When a particular profile is selected all corresponding variables will be displayed in the **X Axis** and **Y Axis** drop-down lists.
- Select the variable for **X Axis** and **Y Axis** and click **Plot**. The plot will be displayed in the area of **Profile Chart**.
- You can select the type of chart to be displayed from the **Chart Style** drop-down list. Options of **Spline**, **Step** and **Line** are provided.
- You can also read a profile by clicking on **Read File**.
- The read profile which is presently selected can be deleted from the list by clicking on **Remove Profile**.

- You can make changes to the values in the profile table displayed. Click **Plot** to view the **Profile Chart** with the changed values. You can save the profile with the edited values by clicking on **Save**.
- To manipulate the chart:

Table 6.1: Chart Manipulation

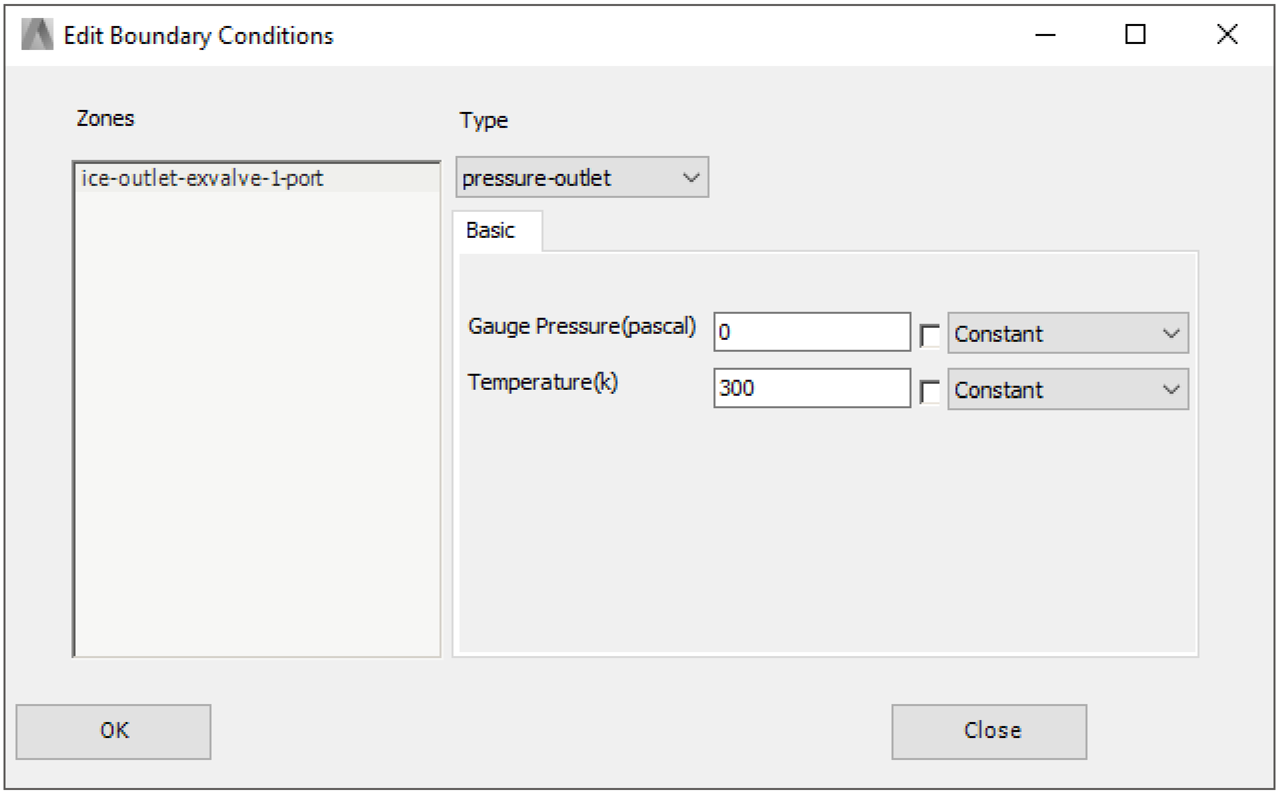
	Operation
Rotate Middle Mouse	Zoom
Shift + Middle Mouse	Zoom
Ctrl + Middle Mouse	Pan
Drag Right Mouse	Box Zoom
F key	Fit to Window

6.1.2. Boundary Conditions



In the **Boundary Conditions** tab you can see the boundary conditions set for two zones. Select each zone and click **Edit** to check the details.

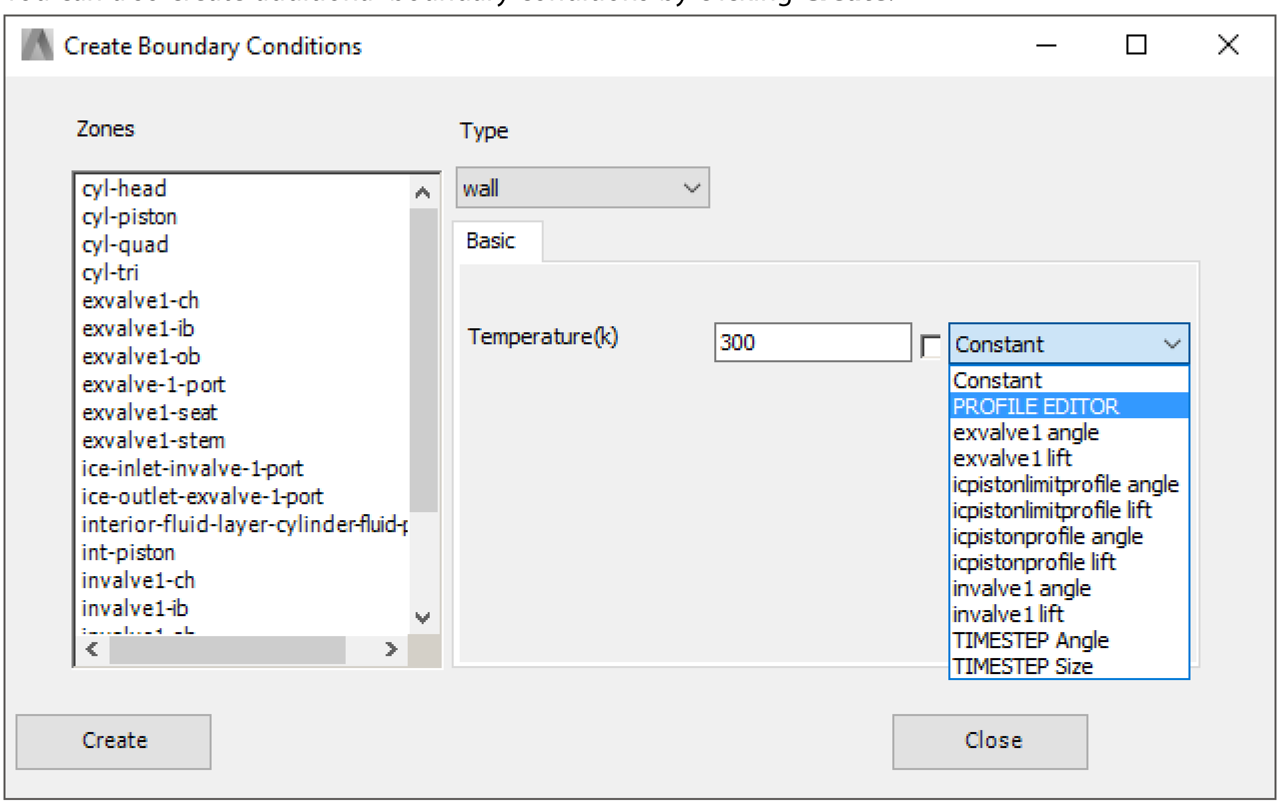
- **ice-outlet-exvalve-1–port** is set to **Type pressure-outlet**.



The **Gauge Pressure** is set to **0**, and the **Temperature** is set to **300**.

- Similarly the zone **ice-inlet-invalve-1–port** is set to **Type pressure-inlet**.

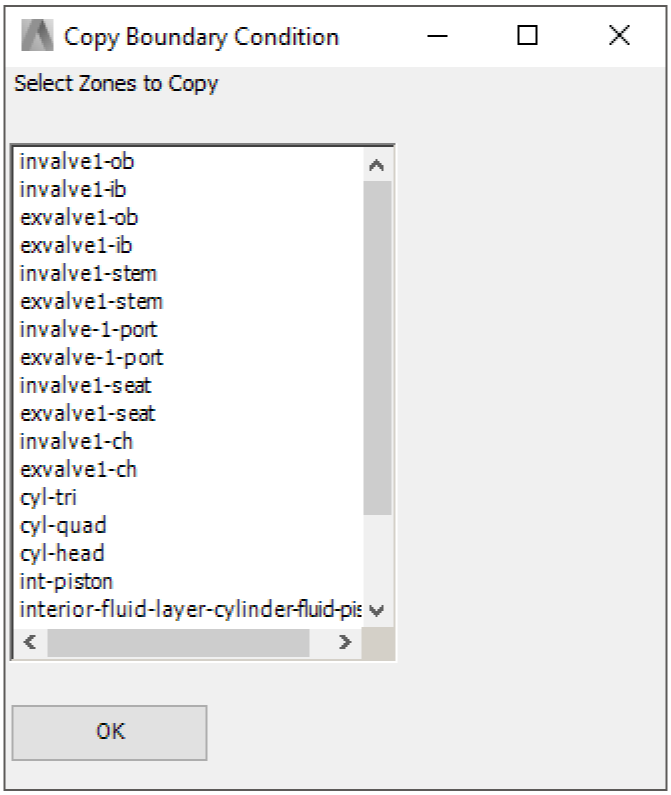
You can check the detailed settings in the **Boundary Conditions** (p. 247) task page in Ansys Fluent. You can also create additional boundary conditions by clicking **Create**.



In the **Create Boundary Conditions** dialog box select the zone to which you like to apply the boundary conditions to from the list under **Zones**. Then select **Type** and enter the required values for the variables, either a constant value or a variable profile from the drop-down list. You can select a profile in case you have read a profile before, or you can select **PROFILE EDITOR** option which will open the **Profile Editor** dialog box where you can check the plot of the profile or read a new profile. After you click **OK** you can see the zone name and the details in the **Boundary Conditions** tab.

You can use the **Copy** button to copy the boundary conditions to other multiple zones.

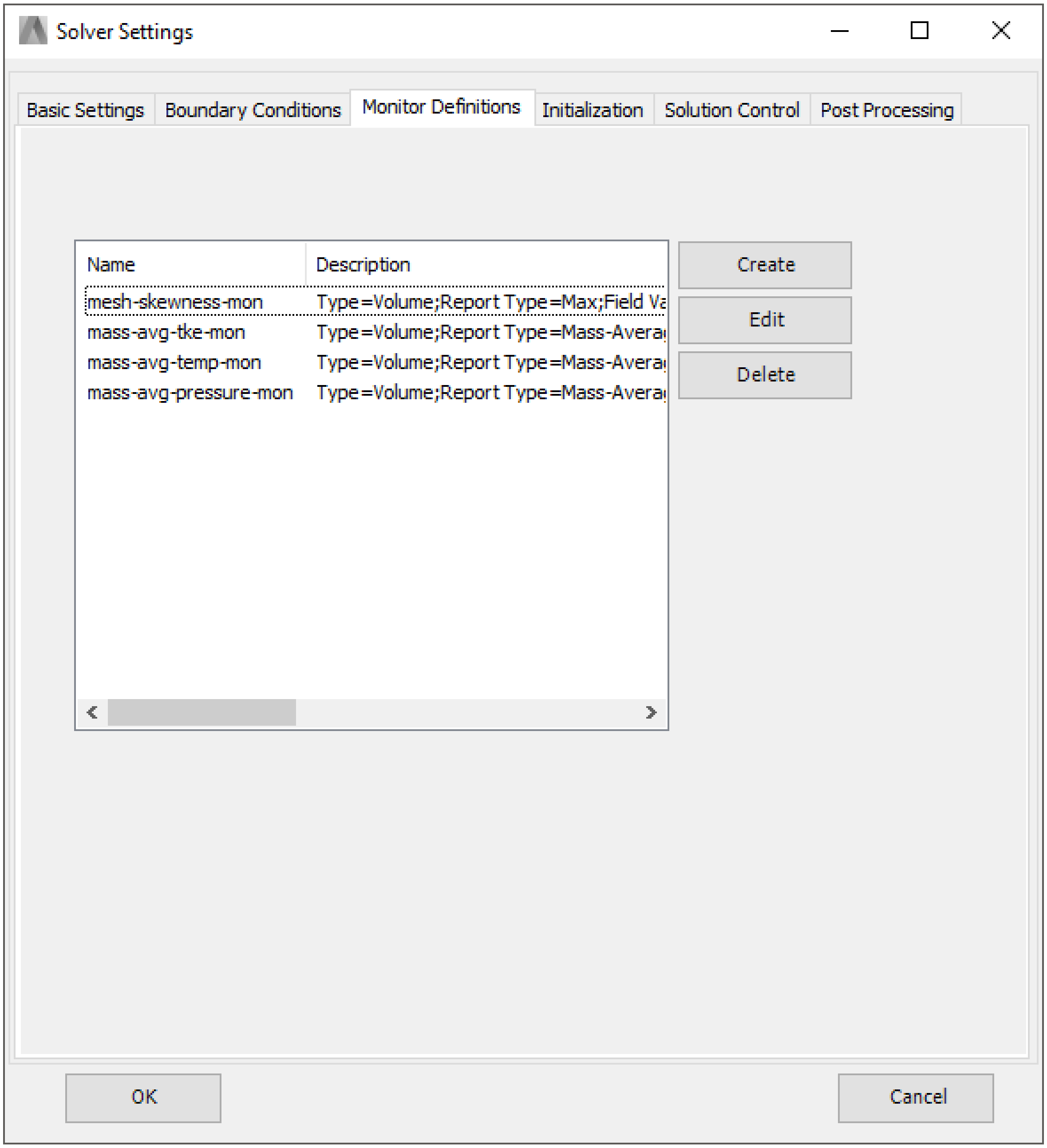
- Select the boundary condition of interest in the **Boundary Conditions** tab.
- Click **Copy**.



- Select the zone(s) to which you want to copy the boundary conditions and click **OK**.
- The zones to which you have copied the boundary conditions appear in the list.

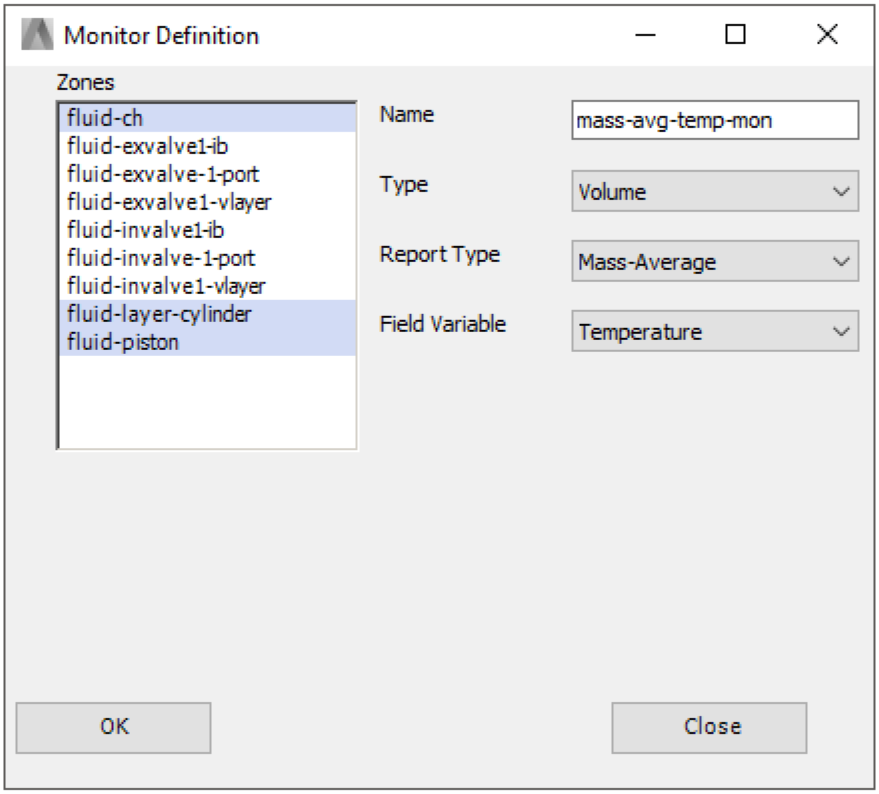
For the boundary conditions, you can parameterize the quantity by enabling the check box next to it.

6.1.3. Monitor Definitions



In the **Monitor Definitions** tab you can see that four volume monitors have been set. Select each monitor and click **Edit** to check the details.

- **mesh-skewness-mon** is a volume monitor which monitors the **Volume-Average** of **Cell Equivolume Skewness** on the zones **fluid-ch**, **fluid-layer-cylinder**, and **fluid-piston**.
- **mass-avg-tke-mon** is a volume monitor which monitors the **Mass-Average** of **Turbulent Kinetic Energy** on the zones **fluid-ch**, **fluid-layer-cylinder**, and **fluid-piston**.
- Similarly **mass-avg-temp-mon** is one more volume monitor.

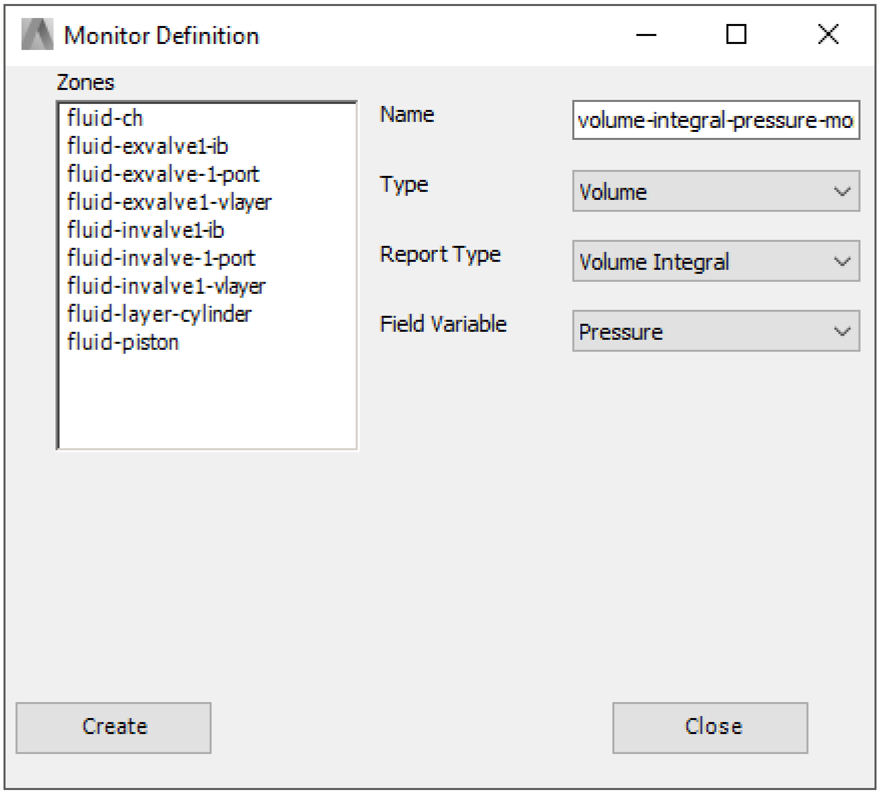


It plots the **Mass-Average** of **Temperature** on the zones **fluid-ch**, **fluid-layer-cylinder**, and **fluid-piston**. This monitor is automatically set for a cold flow simulation.

- The fourth volume monitor **mass-avg-press-mon**, monitors the **Mass-Average** of **Pressure** on the zones **fluid-ch**, **fluid-layer-cylinder**, and **fluid-piston**.

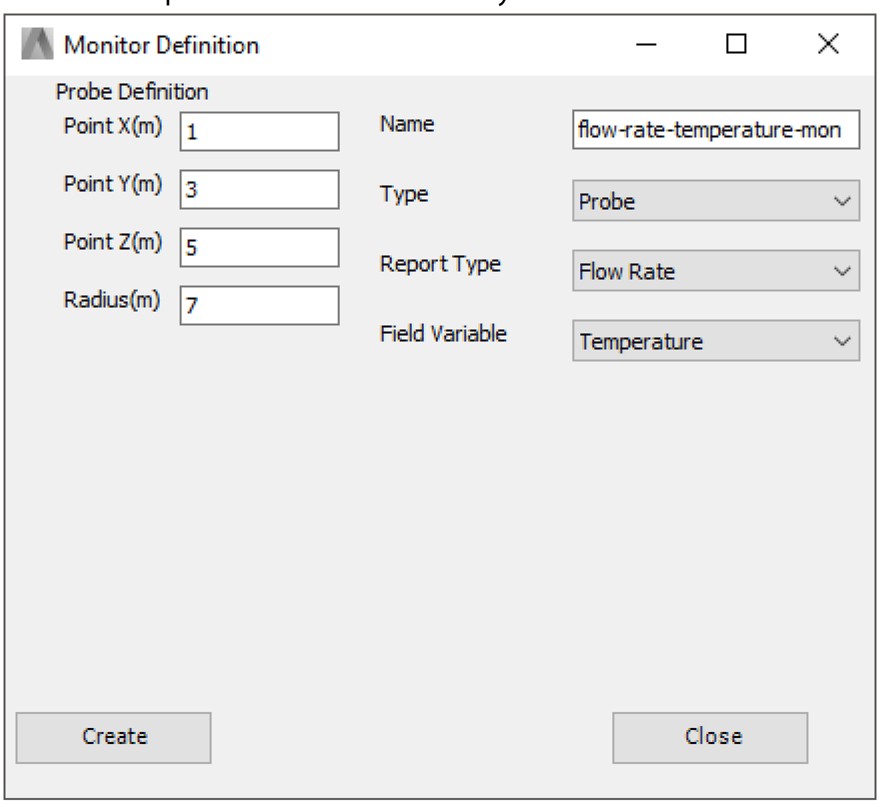
You can change the settings here in the **Monitor Definition** dialog box.

You can see additional details in the **Monitors** (p. 262) task page in Ansys Fluent. You can create additional monitors by clicking **Create**.



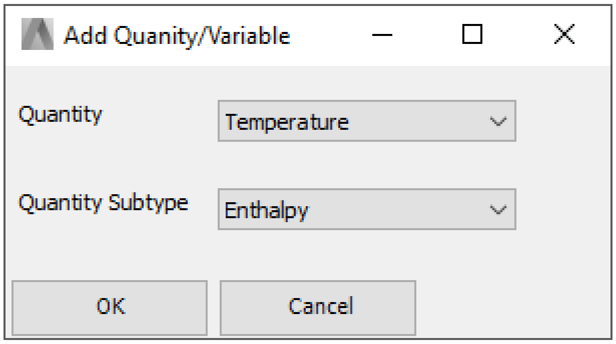
Then select the **Type**, **Report Type**, and the **Field Variable** from the respective drop-down lists.

You can select **Volume**, **Surface**, or **Probe** from the **Type** drop-down list. If you select **Probe** you can create a point or surface on which you can monitor a variable.

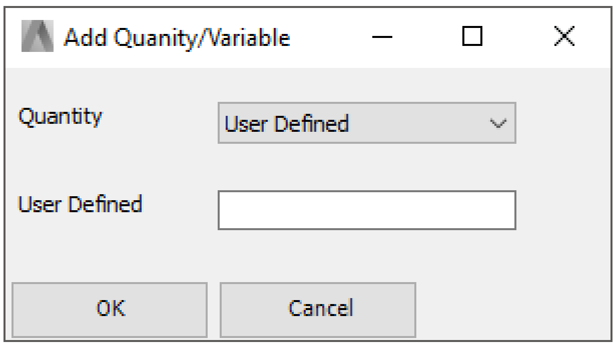


You need to enter the values (in meters) for **Point X**, **Point Y**, and **Point Z** to position the point. If you retain the value of **0** for **Radius** then a point monitor is created. If you enter a value for **Radius** then a circular surface with the given **Radius** and the center as the given **Point** is created. You can monitor the variables of your choice on this surface or point.

You can select **New Variable** from the **Field Variable** drop-down list. This opens the **Add Quantity/Variable** dialog box.



You can select the variable by selecting from the options under the **Quantity** and **Quantity Subtype** drop-down lists. You also have an option of **User Defined** under the **Quantity** drop-down list.

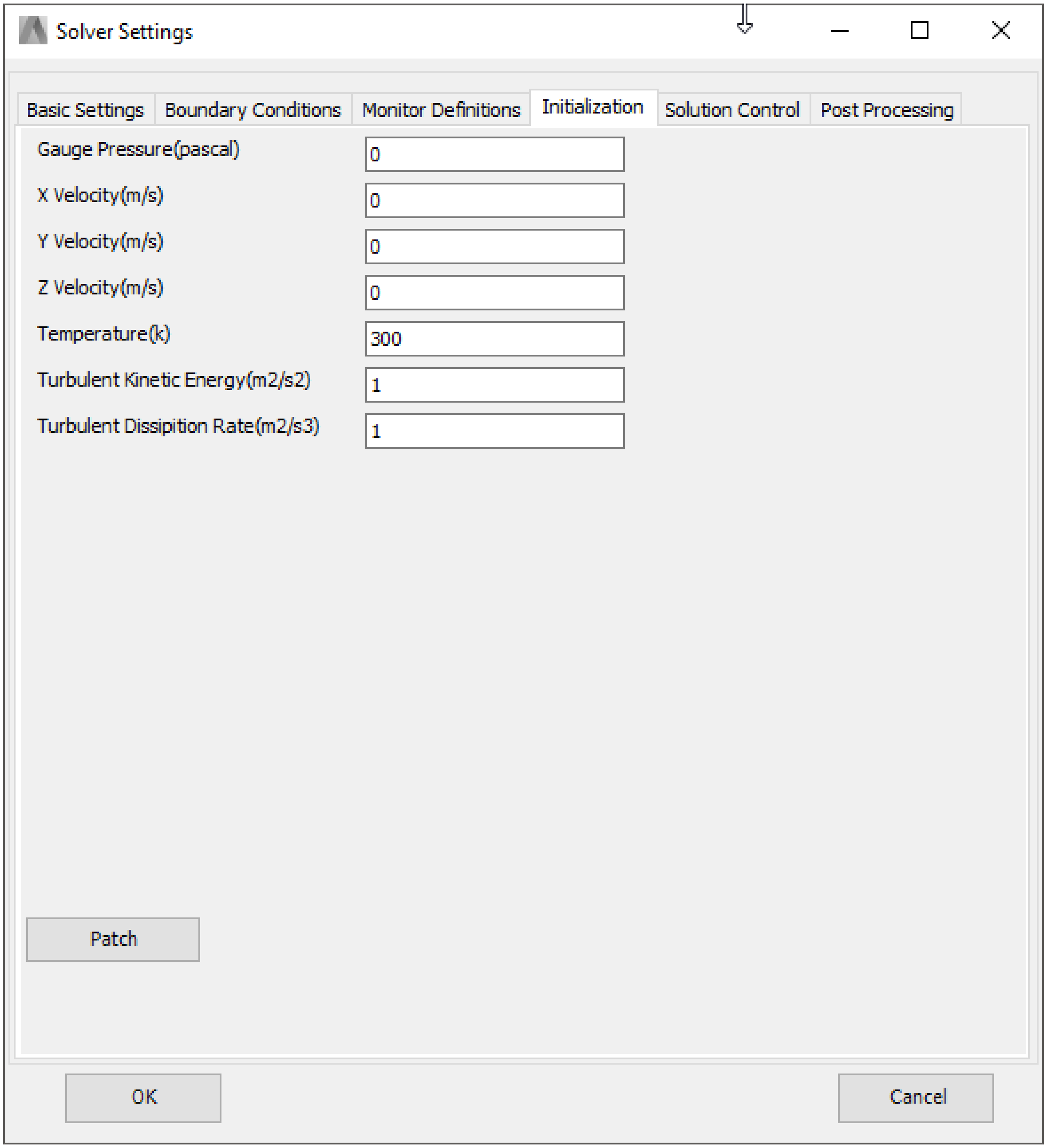


Here you can add the quantity or variable of your choice of which you would like postprocessing images, in the **User Defined** text box. You will have to check Fluent if the term is valid for the simulation. After you click **OK** this quantity will be available in the drop-down list of **Field Variable**.

The **Name** will be set according to your selections. You can enter a name of your choice. Click **Create** to create the monitor. It will now appear in the list in the **Monitor Definitions** tab.

6.1.4. Initialization

In the **Initialization** tab you can see the default set values for the various parameters.

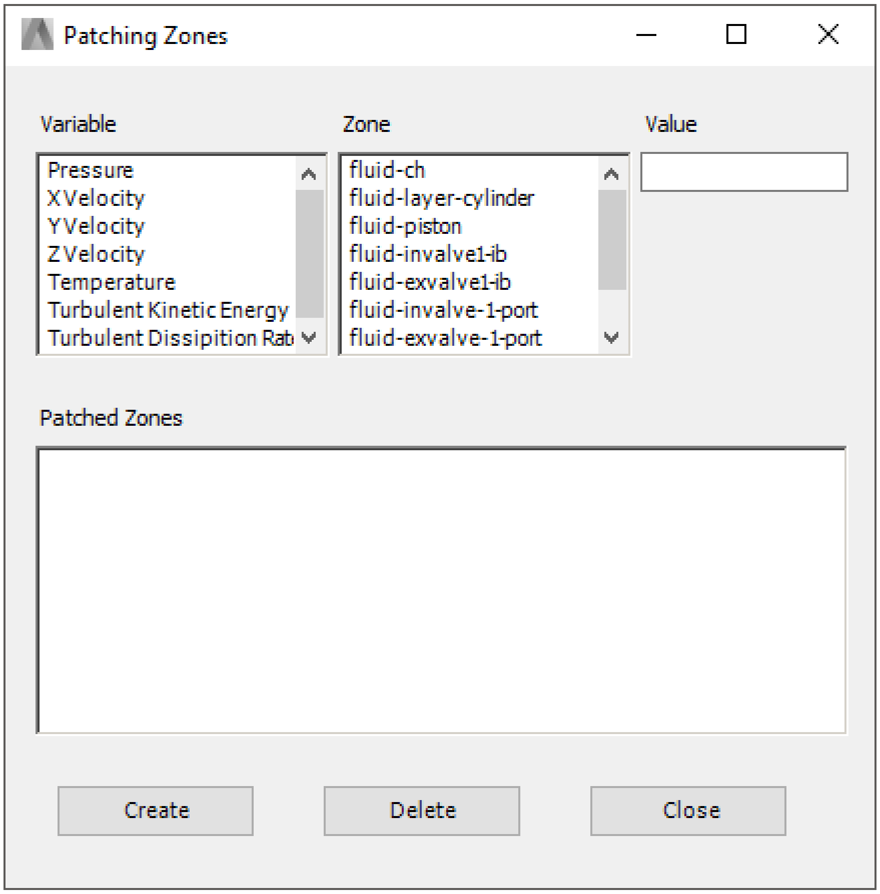


- **Gauge Pressure**, **X Velocity**, **Y Velocity**, and **Z Velocity** are all set to **0** by default.
- **Temperature** is set to **300**.
- **Turbulent Kinetic Energy** and **Turbulent Dissipation Rate** are set to **1** by default.

You can change the values as per your settings.

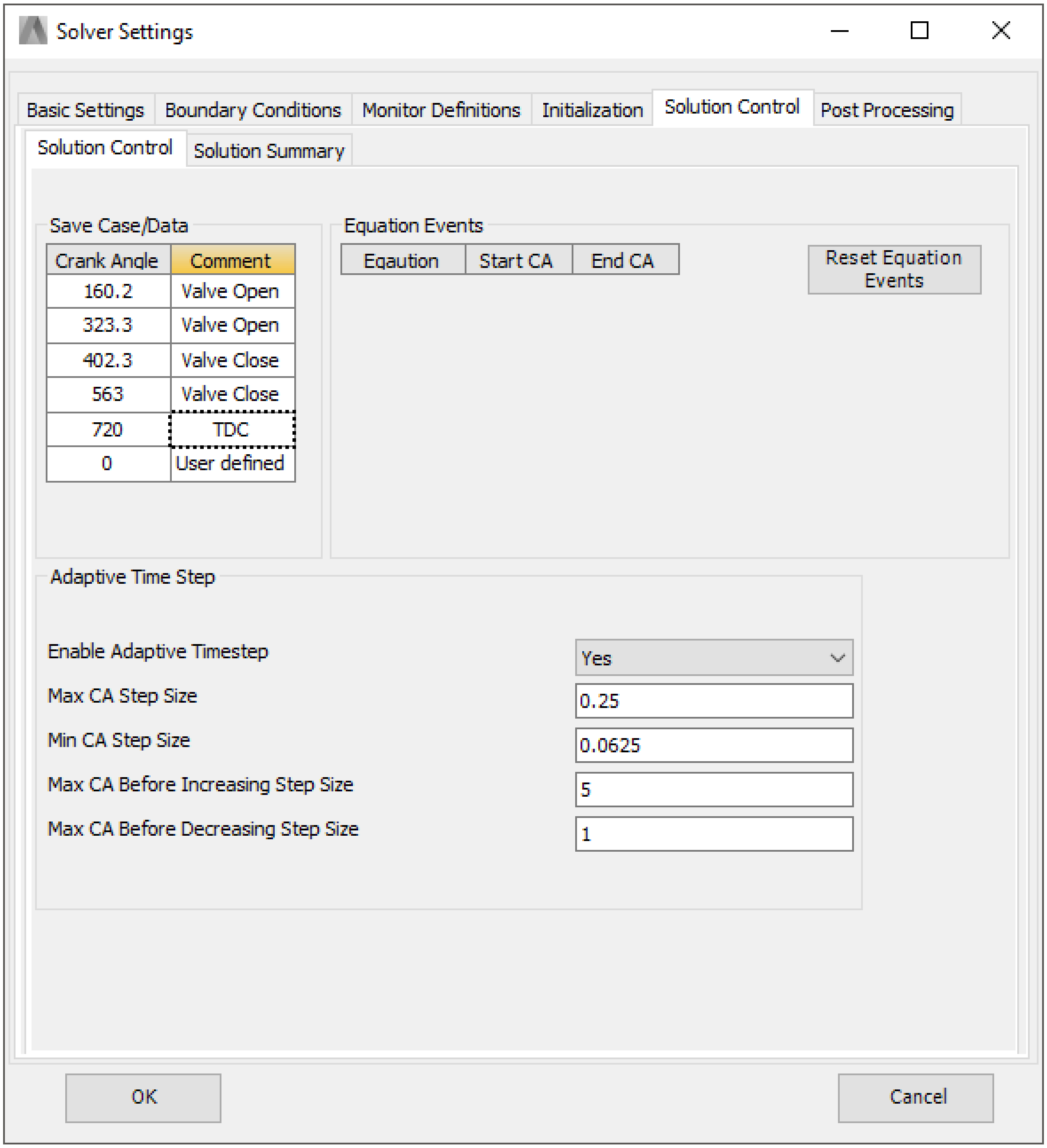
Patching

Click **Patch** to open the **Patching Zones** dialog box.



Here you can patch zones if required. You need to select the zones(s) from the list of **Zone**, a **Variable**, and enter the patching value under **Value**. Then click **Create** and the setting will be displayed in the list of **Patched Zones**.

6.1.5. Solution Control



In the **Solution Control** tab you can set additional specific **Crank Angles** at which the case and data files should be saved and also set the starting and ending crank angles for activating the equations.

6.1.5.1. Solution Control

Save Case/Data

The case and data are saved at a specific frequency which you have entered in the **Basic Settings** (p. 396) tab. In the **Save Case/Data** list you can see the additional crank angles at which the case and data are saved while running the solution. These are at the events of valve opening, valve closing, TDC, etc. You can change these angles. In the **Comment** column you can see the event or definition for the crank angle. You can enter your own definition for the comment column. If you right-click on the list you get options of **Insert Row Below**, **Insert Row Above**, and **Delete Row**. You can use these options to customize your solution.

Equation Events

As per your selection of the combustion models the list of **Equations Events** will change. Here you can enter the starting and ending crank angles for which you want the specific equations activated. The starting crank angle (**Start SA**) is set to the decomposition crank angle by default. If you do not enter anything for the ending crank angle (**End CA**) then the equation will be activated throughout the simulation cycle. To reset the **Equations Events** settings to default values click **Reset Equation Events**.

Adaptive Time Step

By using the options provided under **Adaptive Time Step** you can control how the solver changes the time step size within the specified limits based on convergence criteria. This will help in achieving a better and faster convergence of the solution.

During the solution run, if for one time step the maximum number of iterations are used to converge then it means that the solution is far from converging. In this case the solution will execute the number of time steps according to the crank angle size specified as **Max CA Before Decreasing Step Size** and then reduce the time step by half of the present value. If the solution still uses the maximum number of iterations in the time step then it will again reduce the time step by half. This will go on until **Min CA Step Size** is reached.

On the other hand if the solution is converging in less than half of the maximum number of iterations in the time step, then the time step is doubled after executing the number of time steps according to the crank angle size specified as **Max CA Before Increasing Step Size**. This will go on until **Max CA Step Size** is reached.

In case there is an event of KeyGrid replacement then the time step is reduced such that the event is within the tolerance limit of **Min CA Step Size**. This will take priority over any other criteria which will try to increase the time step. After the KeyGrid replacement the solution will go back to run on the settings of **Adaptive Time Step**.

Also, if there are any direct events which try to change time step size, the program will respect them but from next time step the **Adaptive Time Step** criteria will control the solution.

- For using the option select **Yes** from the **Enable Adaptive Timestep** drop-down list.
- When the solution starts to converge you can increase the time step size. Enter the crank angle step size to which it can increase in **Max CA Step Size**.
- Similarly if the solution reaches the maximum iterations in a time step then the time step size is decreased. Enter the crank angle step size to which the time step can decrease in **Min CA Step Size**.
- The number of angles the solution will wait before increasing the step size is entered for **Max CA Before Increasing Step Size**.
- The number of angles the solution will wait before decreasing the step size is entered for **Max CA Before Decreasing Step Size**.

.

6.1.5.2. Solution Summary

In this group box you can view different profiles overlapping each other. You can enable the profiles you would like to see on the chart from the list on the left hand side. All the profiles are plotted on the base of crank angles. The **Summary Chart** gives a graphic view of the valve profiles, piston profiles, timesteps, etc. Observing the chart gives you an account of the position of the piston and valve at the injection time. This will help if you want to manipulate the default time step settings, or any other events. The chart can be manipulated using the following table.

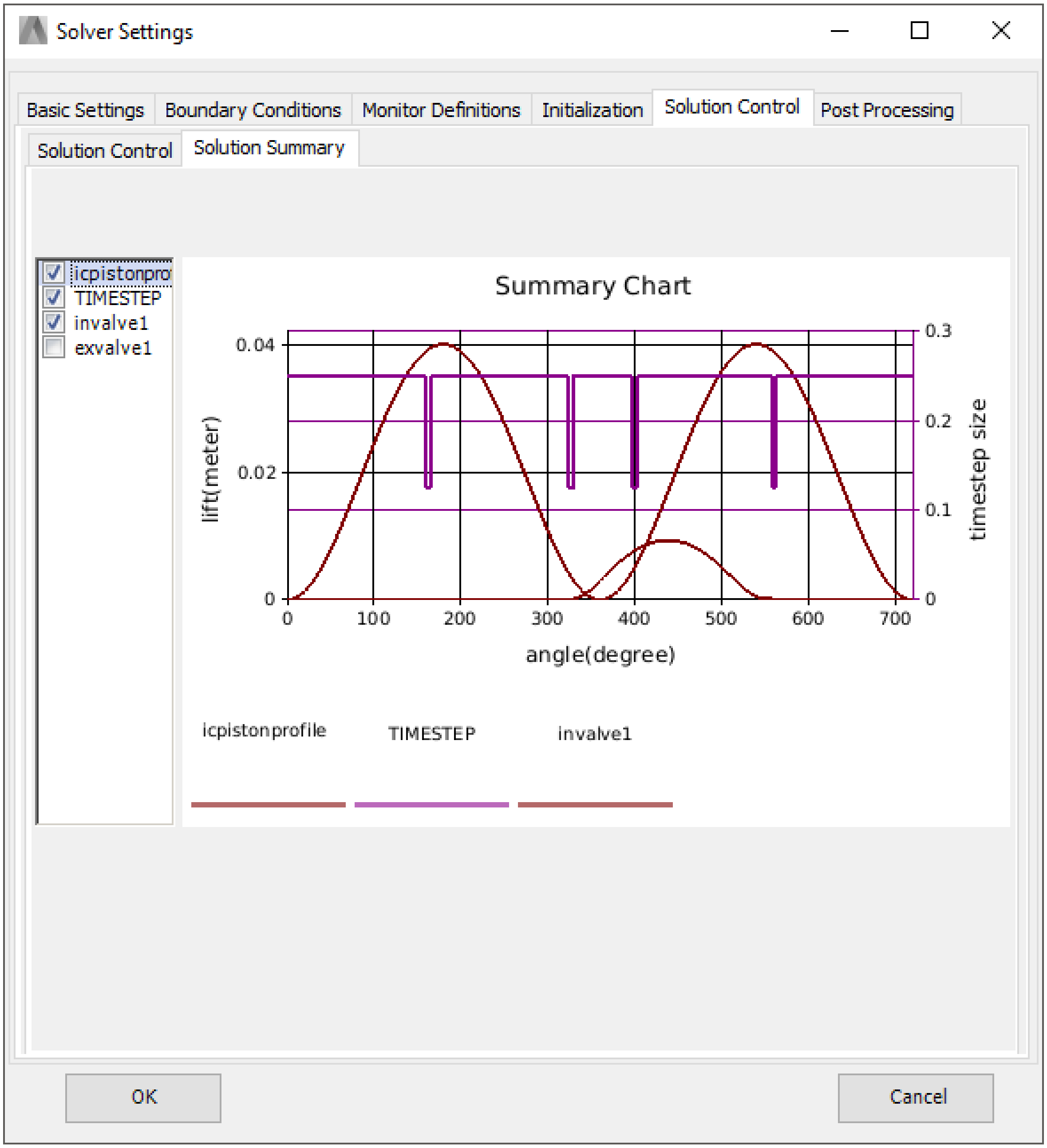
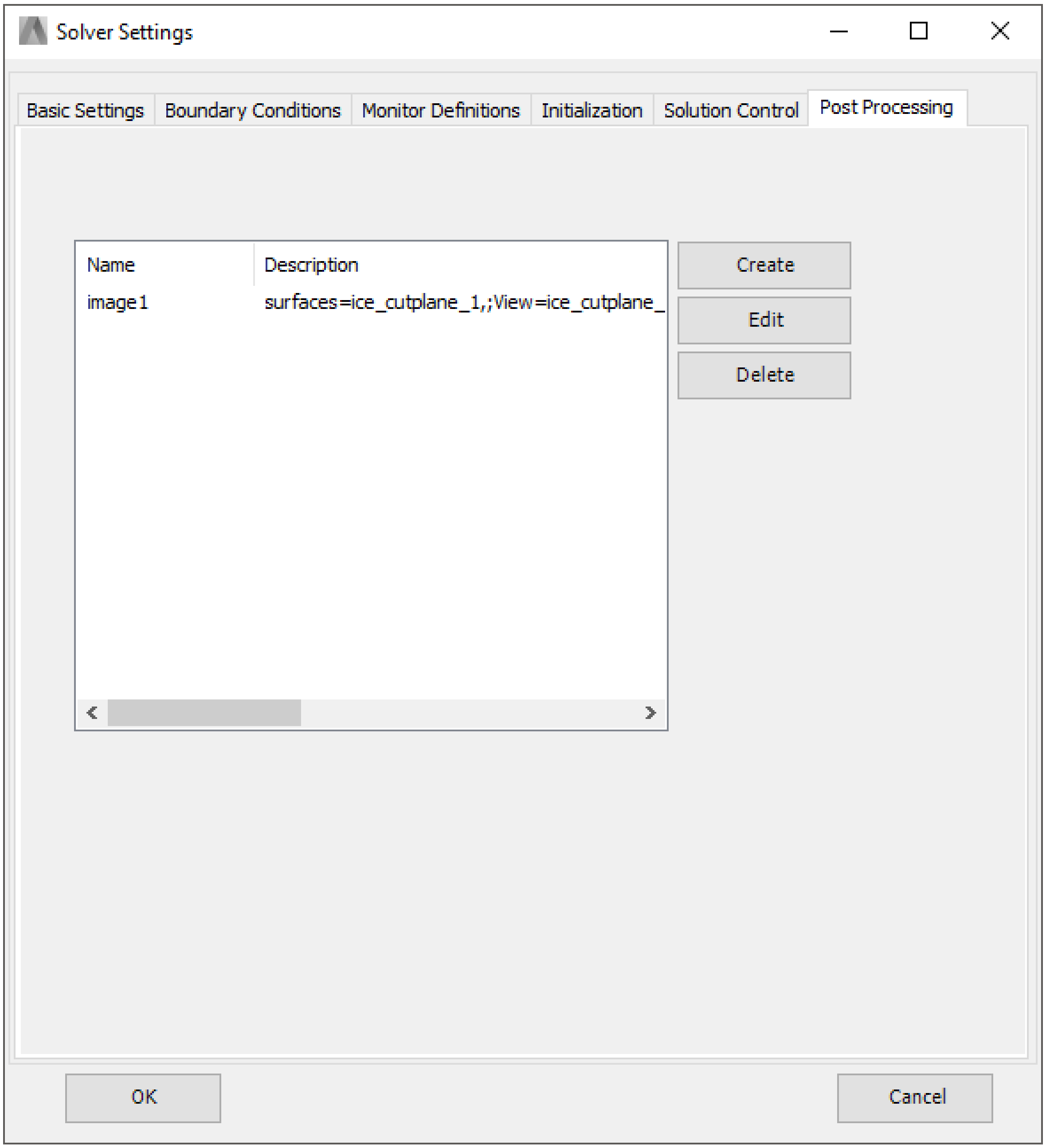


Table 6.2: Chart Manipulation

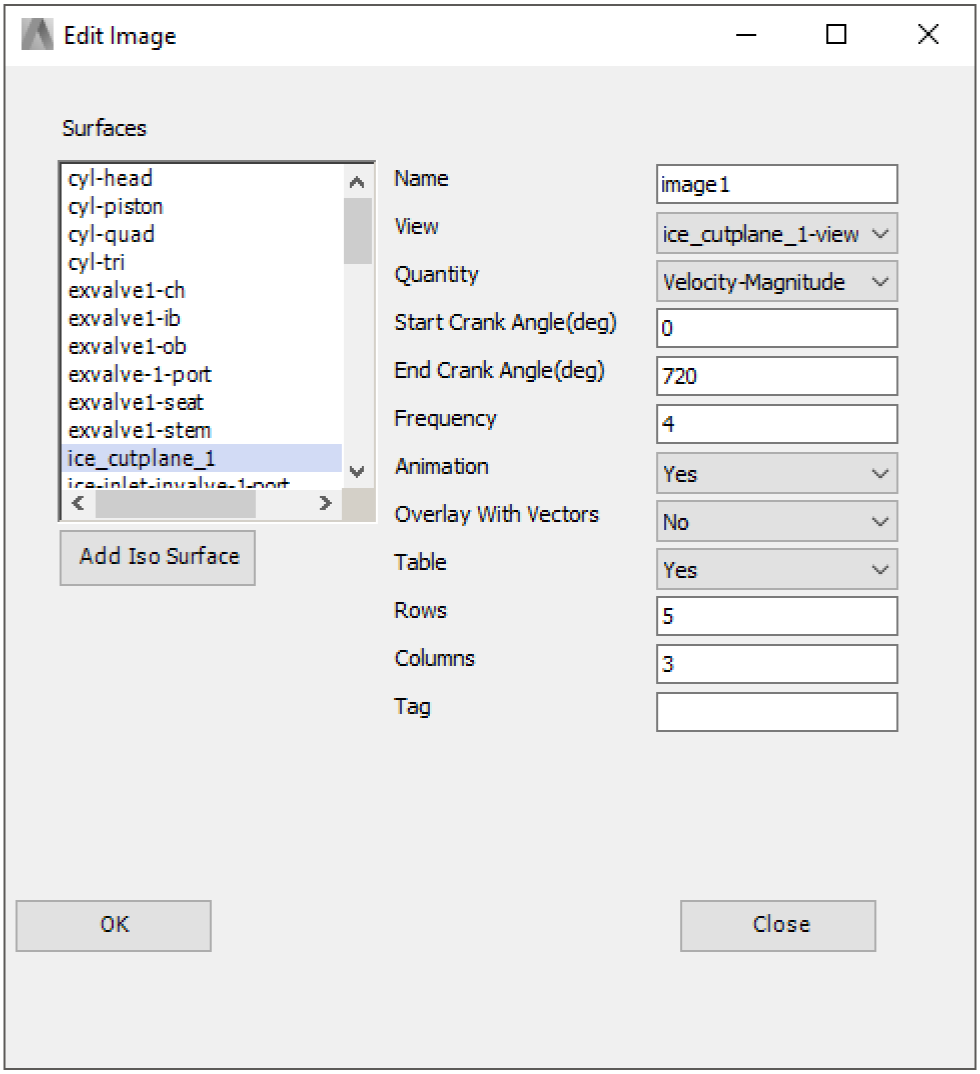
	Operation

Rotate Middle Mouse	Zoom
Shift + Middle Mouse	Zoom
Ctrl + Middle Mouse	Pan
Drag Right Mouse	Box Zoom
F key	Fit to Window

6.1.6. Postprocessing

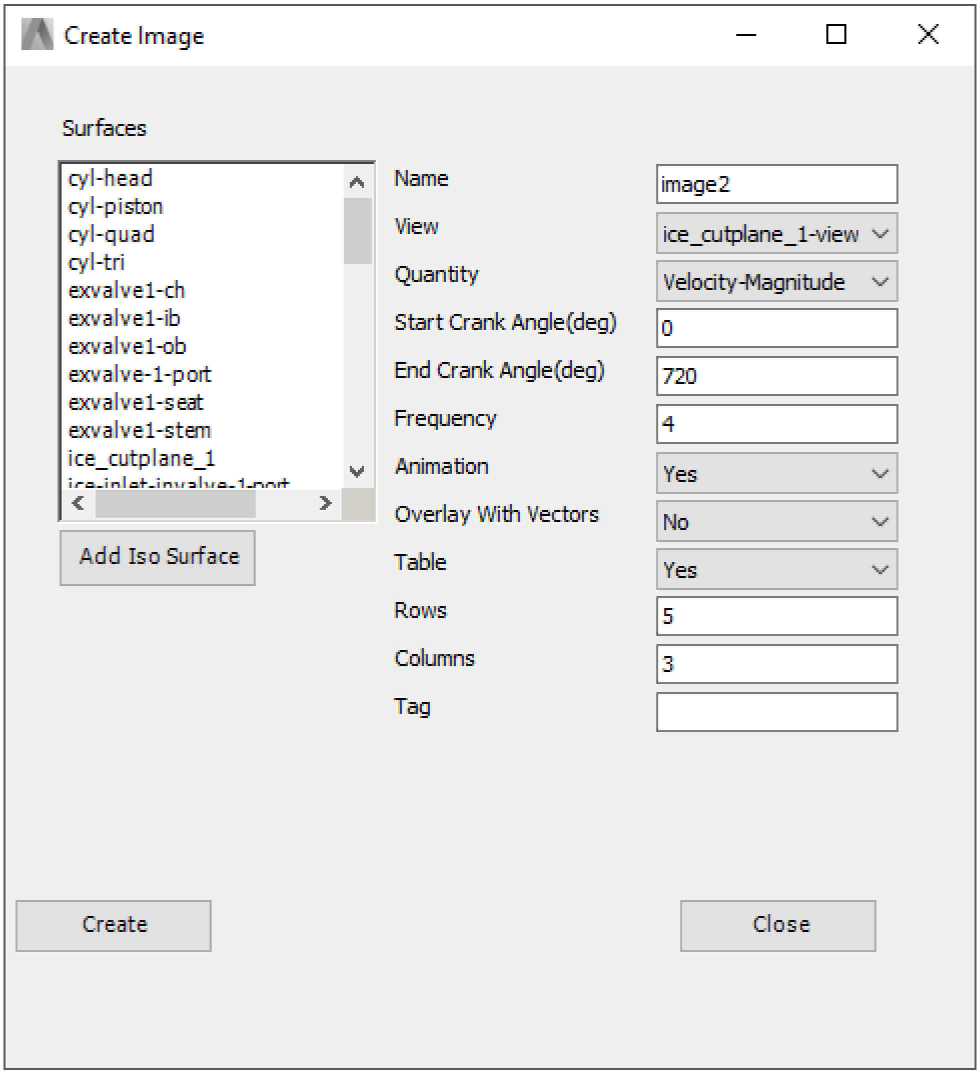


In the **Post Processing** tab you can see that **image1** is being automatically saved. Click **Edit** to check the details.



- **image1** is plotted on surface **ice_cutplane_1**.
- The image is captured at a saved view, **ice_cutplane_1–view**.
- **velocity-magnitude** contours are displayed.
- Images are saved from the **Start Crank Angle** of **0** to the **End Crank Angle** of **720** at a **Frequency** of every **4** time-steps.
- **Animation** is set to **Yes**, signifying that animation of the velocity contour images will be created at the end of the simulation.
- **Overlay With Vectors** is set to **No**.
- **Table** is set to **Yes**, signifying that the saved images will be displayed in the report in a table **5 Rows** and **3 Columns**.

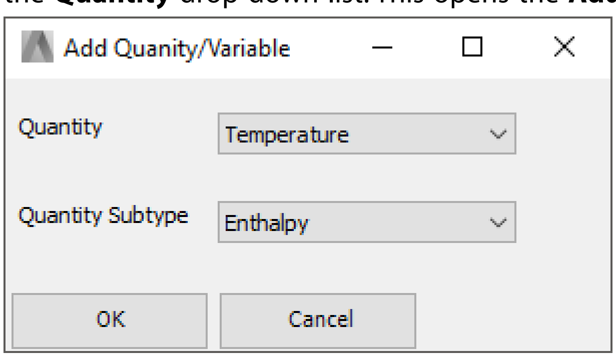
In the **Edit Image** dialog box you can change the settings and values as required. To create additional postprocessing images click **Create**.



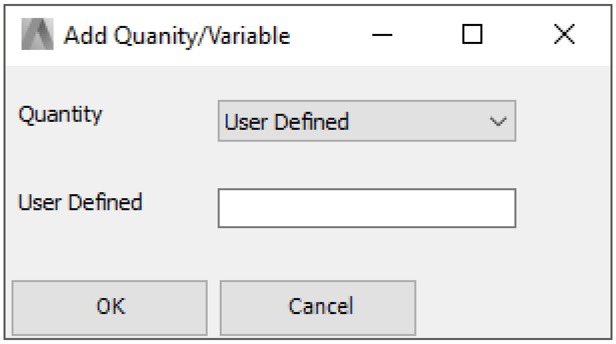
1. Select the **Surfaces** on which you want to plot the contours.

2. Select a view from the **View** drop-down list.

3. Select the parameter from the **Quantity** drop-down list. You can select **New Variable** from the **Quantity** drop-down list. This opens the **Add Quantity/Variable** dialog box.



You also have an option of **User Defined** under the **Quantity** drop-down list.



Here you can add the quantity or variable of your choice of which you would like postprocessing images, in the **User Defined** text box. You will have to check Fluent if the term is valid for the simulation.

4. Enter the **Start Crank Angle** and **End Crank Angle**. The images will be saved only within these two crank angles.

5. Enter a value for **Frequency**. The images will be captured at the entered frequency.

6. If you need to create an animation from the saved images you can select **Yes** from the **Animation** drop-down list.

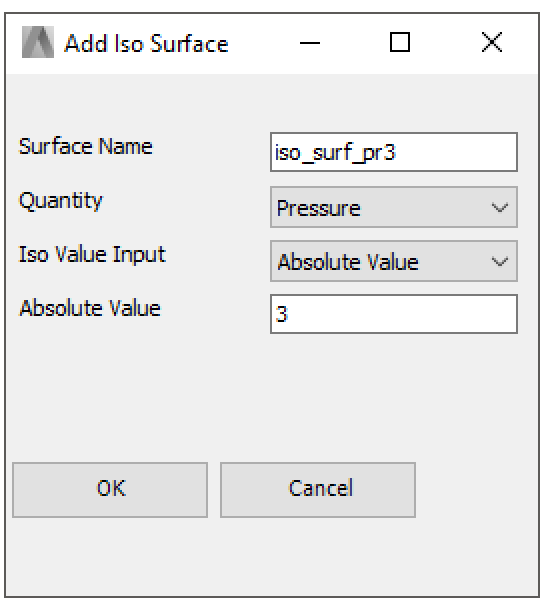
7. The option **Overlay With Vectors** is by default set to **No**. You can set it to **Yes** if you want the vectors to overlap the contour images.

8. If you want the saved images to be displayed in the report in a table format select **Yes** from the **Table** drop-down list.

9. Enter the values for **Rows** and **Columns** depending on how you want to format the table.

10. You can add a tag for the image. You can use this **Tag** if you want only the final images of different surfaces in a single table. In this case you have to provide the same tag to all the images.

11. Click **Add Iso Surface** if you want to create an iso-surfaces. These surfaces are isovalued sections of the entire domain.



a. Enter a name for the isosurface you want to create at **Surface Name**.

b. Select the **Quantity** from the drop-down list. You can select a quantity from the list or create a new variable by selecting **New Variable**.

c. From the **Iso Value Input** drop-down list you can select either of **Absolute Value** or **Percentage of Range**.

- **Absolute Value**: If you select this from the drop-down options, you can enter a value for **Absolute Value**. The iso-surface created will be of this value.

- **Percentage of Range**: If you select this from the drop-down options, then you need to enter a percentage value for **Percentage of Range**. The iso-surface created will be of the percentage value of the minimum and maximum values obtained of the quantity.

An iso-surface of the given name and of the selected quantity will be created and appear in the list of **Surfaces**. This isosurface will be of the **Absolute Value** entered or of the percentage of the minimum and maximum values of the quantity. You can select this isosurface to create images in postprocessing.

12. Click **Create** to create the image.

13. Click **Close** to close the **Create Image** dialog box.

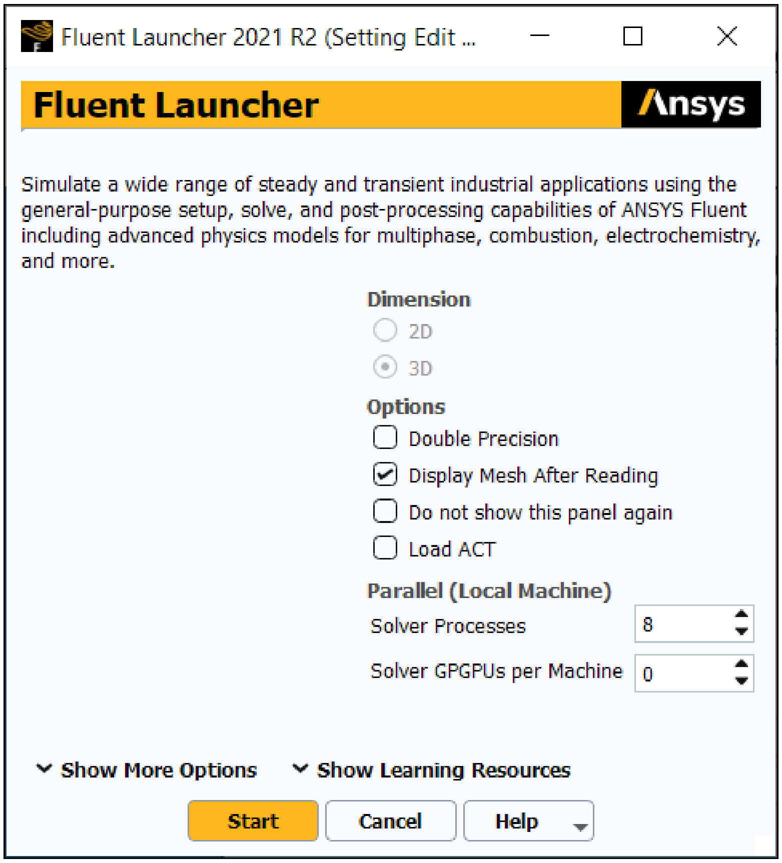
Once the changes are done in the **Solver Settings** dialog box you can close it and then update the **ICE Solver Setup** cell by choosing **Update** from the context menu.

6.2. Solver Default Settings for IC Engine

When you double-click the **Setup** cell, **FLUENT Launcher** opens.

Note:

Depending upon your machine configuration you may be able to increase the **Number of Processes** to decrease the time taken to arrive at a solution. A reasonable value for the **Number of Processes** is the number of cores on your machine.

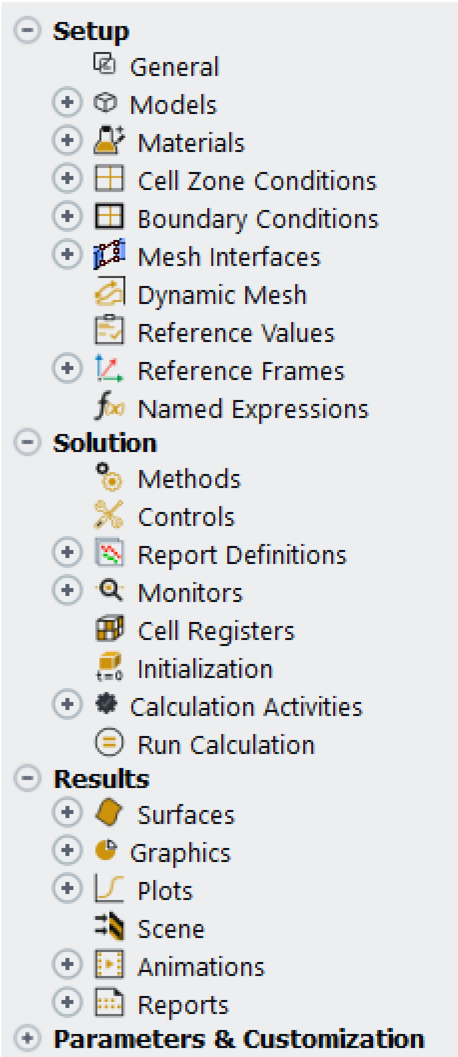


When you click **OK**, Ansys Fluent reads the mesh file and sets up the IC Engine case. It will:

- Read the valve and piston profile.
- Create various dynamic mesh zones.
- Create interfaces required for dynamic mesh setup.
- Set up the dynamic mesh parameters.
- Set up the required models.
- Set up the default boundary conditions and material.
- Create all the required events, to model opening and closing of valves, and corresponding modifications in solver settings and under-relaxations factors.
- Set up the default monitors.
- Initialize and patch the solution.

In the Ansys Fluent application, you can check the default settings by highlighting the items in the navigation pane.

Figure 6.1: The Ansys Fluent Navigation Pane

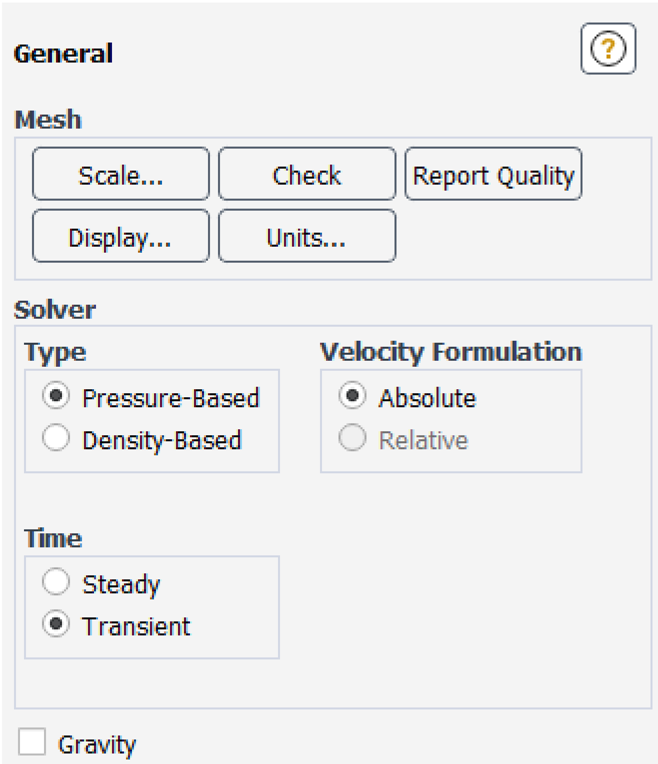


The details about the settings in the different task pages are described in the following sections:

- 6.2.1. Solver General Settings
- 6.2.2. Models Set in Solver
- 6.2.3. Materials Set in Solver
- 6.2.4. Boundary Condition Settings in Solver
- 6.2.5. Dynamic Mesh Settings in Solver
- 6.2.6. Events Set in Solver
- 6.2.7. Solution Methods Set in Solver
- 6.2.8. Solution Controls Set in Solver
- 6.2.9. Monitors Set in Solver
- 6.2.10. Run Calculation

6.2.1. Solver General Settings

In the **General** task page the following settings are done:

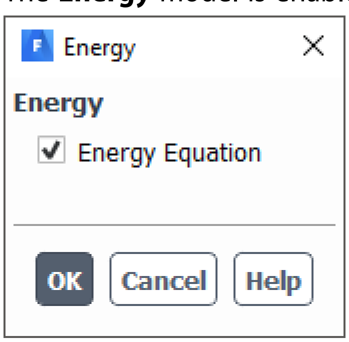


- Solver **Type** is set to **Pressure-Based**.
- Solver **Time** is set to **Transient**.

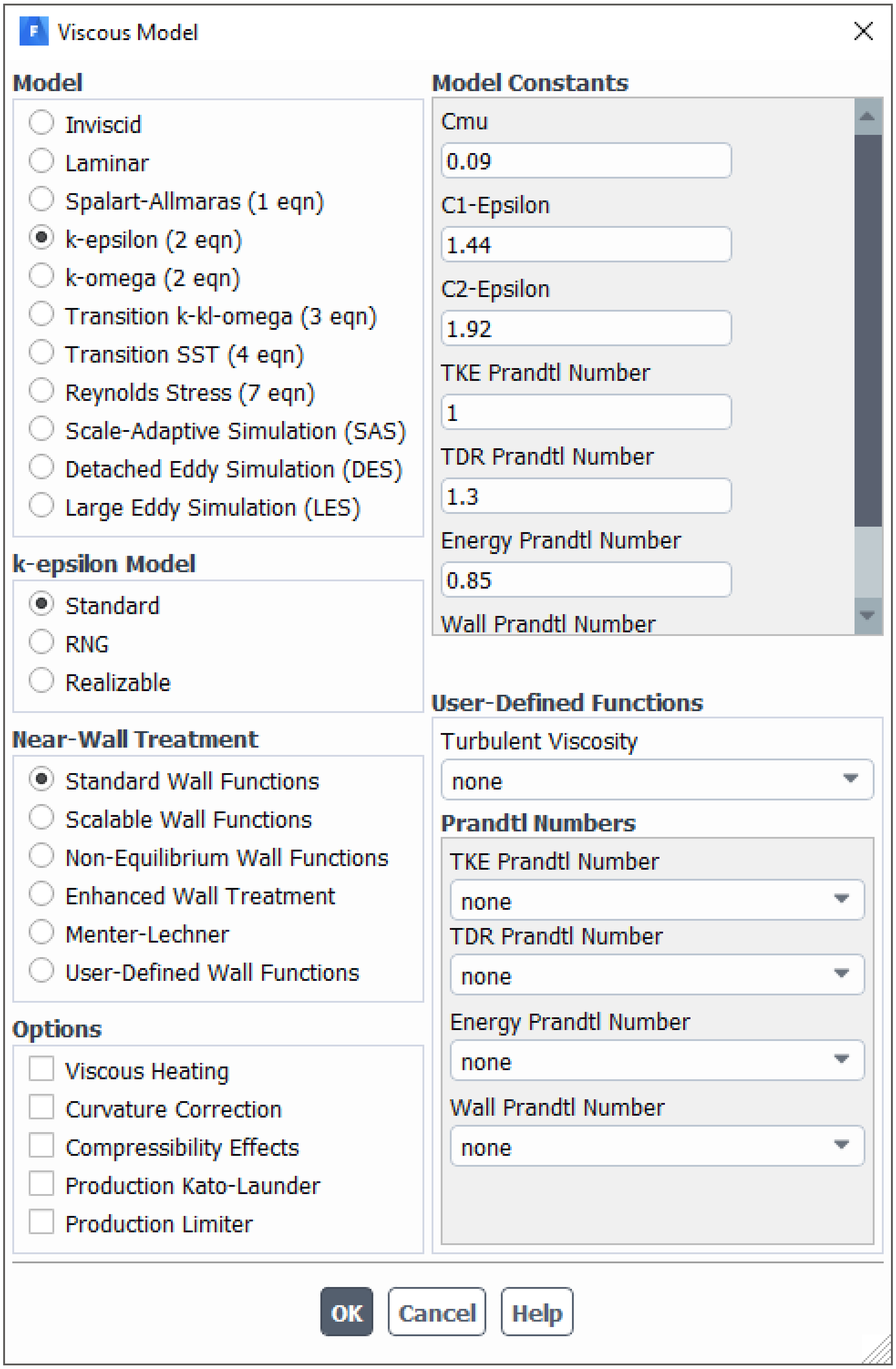
6.2.2. Models Set in Solver

The following models are selected for the analysis:

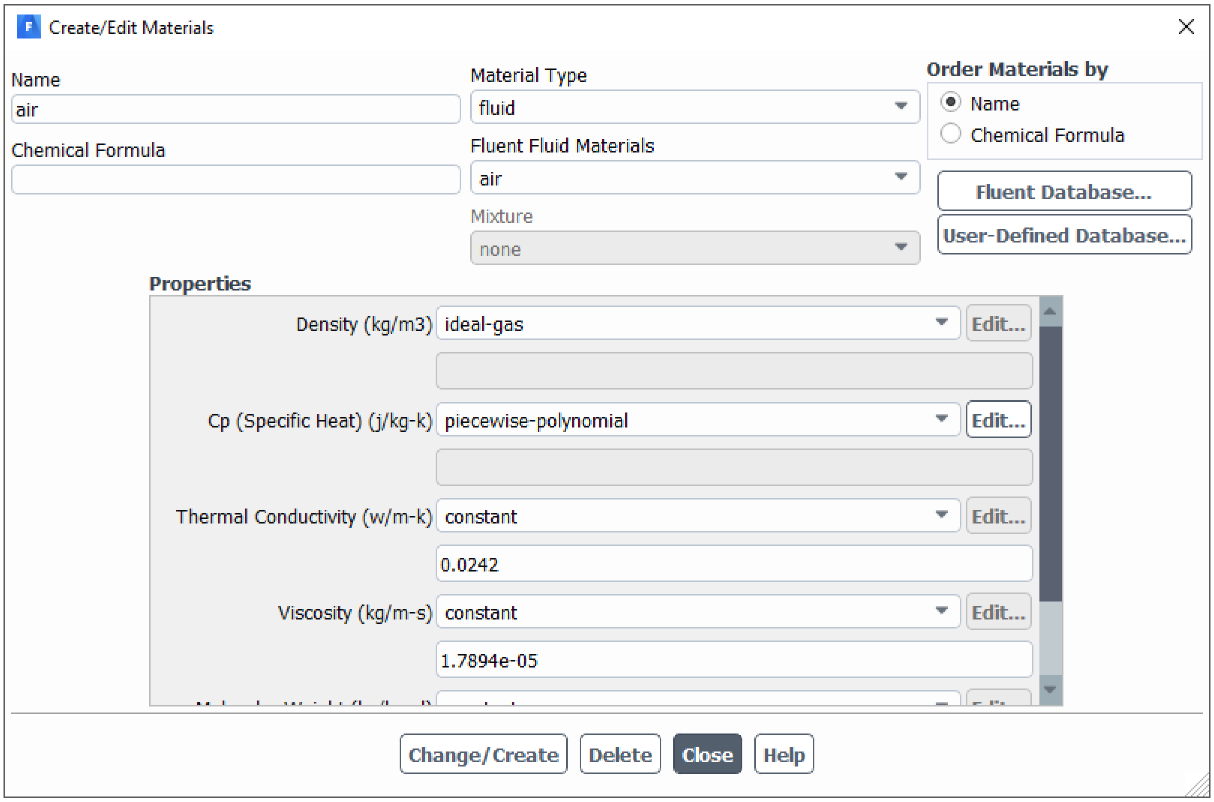
- The **Energy** model is enabled.



- From the **Viscous** models, the **Standard k-epsilon** model is selected, with **Standard Wall Functions** as **Near-Wall Treatment**.


Viscous Model
Model
Inviscid
Laminar
Spalart-Allmaras (1 eqn)
k-epsilon (2 eqn)
k-omega (2 eqn)
Transition k-kl-omega (3 eqn)
Transition SST (4 eqn)
Reynolds Stress (7 eqn)
Scale-Adaptive Simulation (SAS)
Detached Eddy Simulation (DES)
Large Eddy Simulation (LES)
k-epsilon Model
Standard
RNG
Realizable
Near-Wall Treatment
Standard Wall Functions
Scalable Wall Functions
Non-Equilibrium Wall Functions
Enhanced Wall Treatment
Menter-Lechner
User-Defined Wall Functions
Options
Viscous Heating
Curvature Correction
Compressibility Effects
Production Kato-Launder
Production Limiter
Model Constants
Cmu
0.09
C1-Epsilon
1.44
C2-Epsilon
1.92
TKE Prandtl Number
1
TDR Prandtl Number
1.3
Energy Prandtl Number
0.85
Wall Prandtl Number
User-Defined Functions
Turbulent Viscosity
none
Prandtl Numbers
TKE Prandtl Number
none
TDR Prandtl Number
none
Energy Prandtl Number
none
Wall Prandtl Number
none
OK
Cancel
Help

6.2.3. Materials Set in Solver

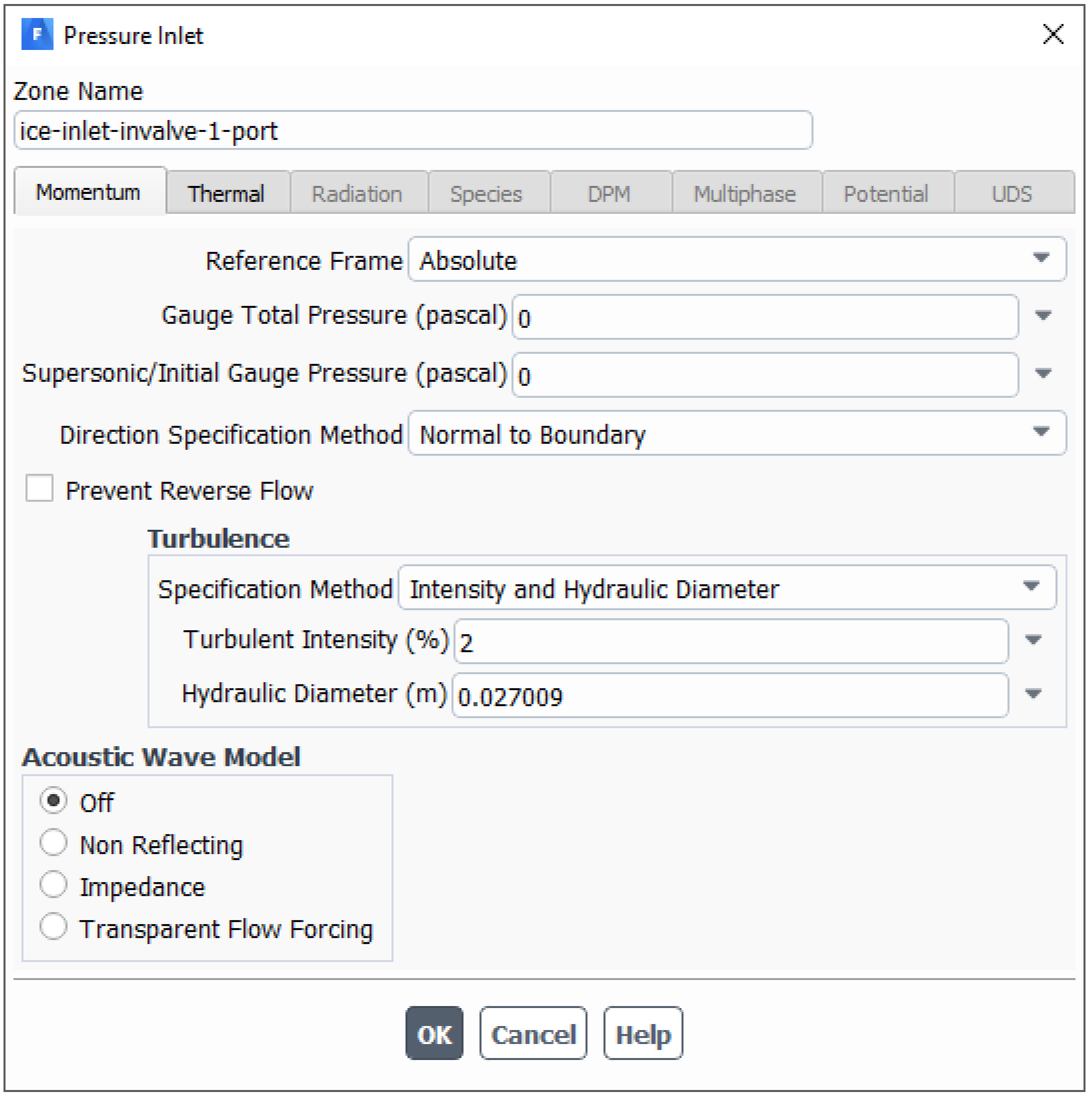


- **Air** is set as the material.
- The **Density** of air is set to **ideal-gas**.
- The **Specific Heat (Cp)** of air is set to temperature dependent.

6.2.4. Boundary Condition Settings in Solver

You can check the boundary conditions and monitors settings for cold flow simulation in the `icBC-Settings` file.

- For the **ice-inlet-invalve-1-port** boundary conditions, the following settings are done in the **Pressure Inlet** dialog box:



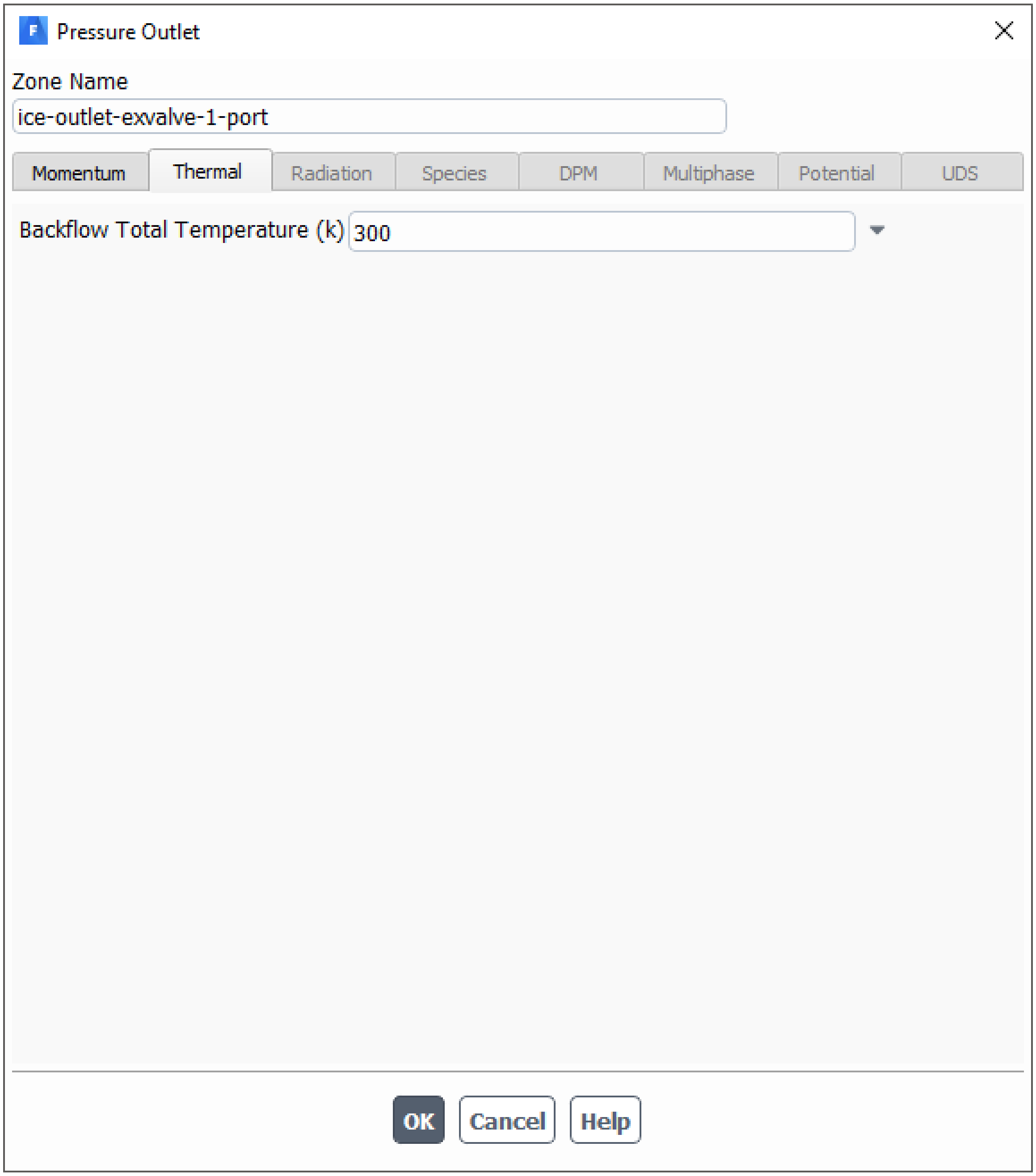
- **Gauge Total Pressure** is set to `0` Pa.
- The **Total Temperature** is set to `300` K, assuming that the engine is naturally aspirated.
- The **Specification Method** in the **Turbulence** group box is set to **Intensity and Hydraulic Diameter**.

Note:

The **Specification Method** can also be set to, **Intensity and Viscosity Ratio**.

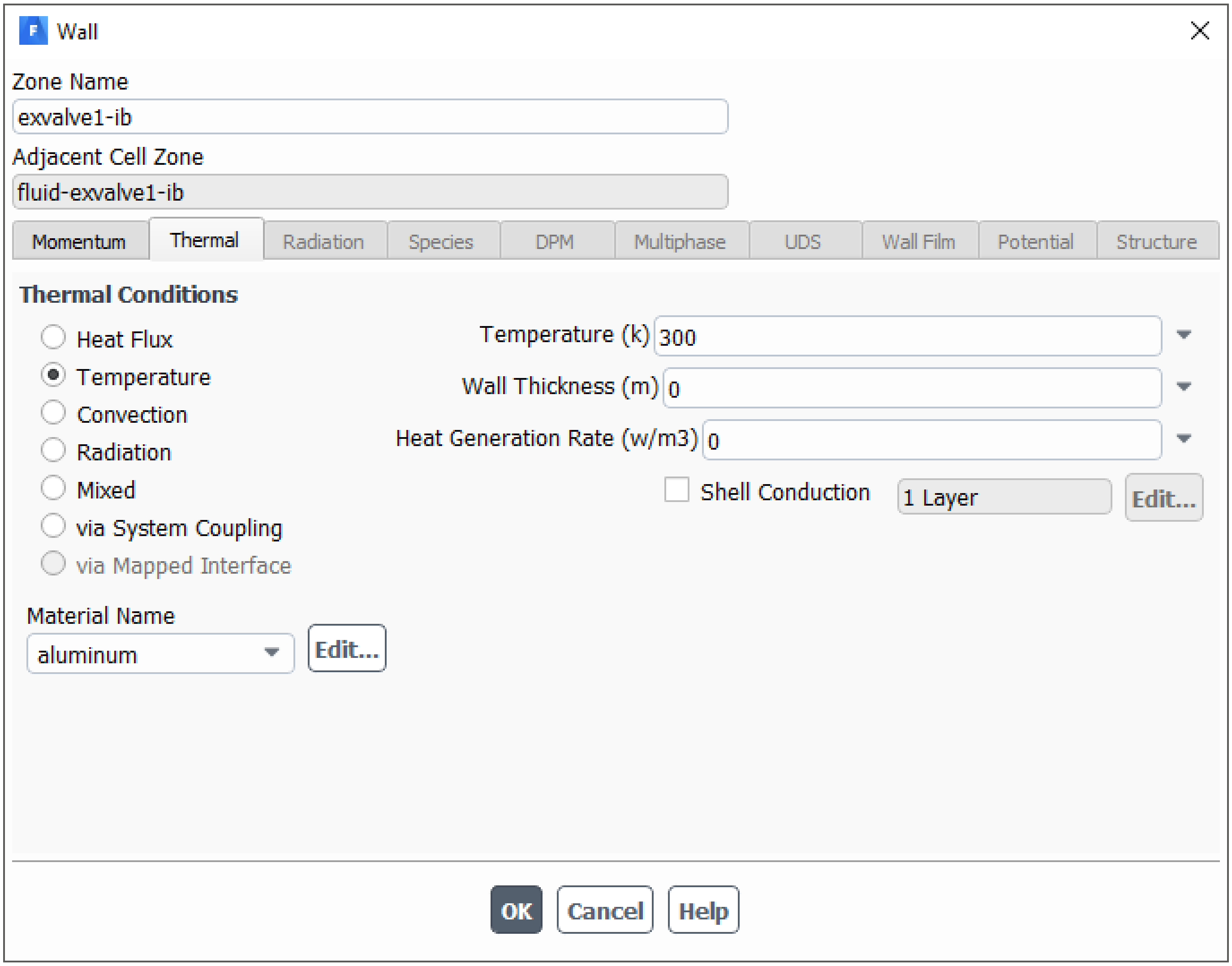
- **Turbulent Intensity** is set to 2%, and **Hydraulic Diameter** is automatically calculated.

• For the **ice-outlet-exvalve-1-port** boundary condition the following settings are done in the **Pressure Outlet** dialog box:

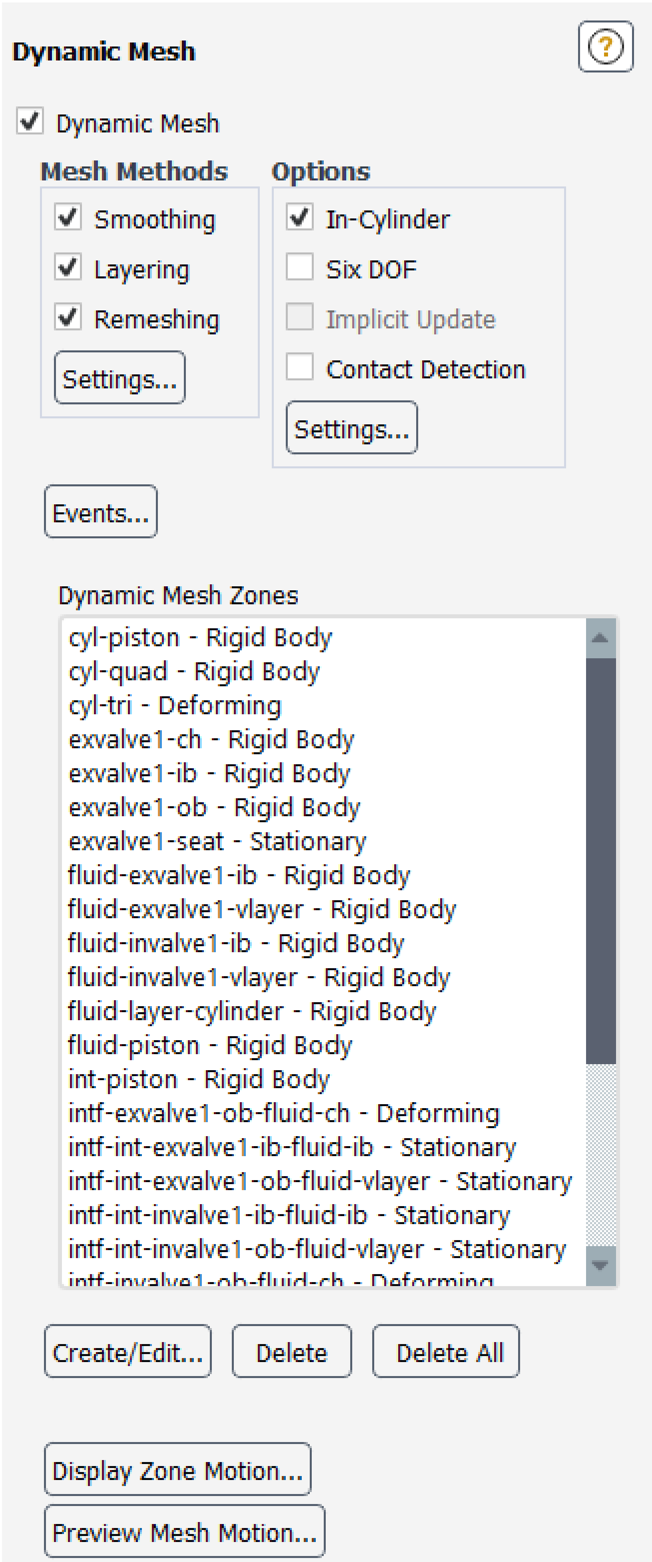


- **Gauge Pressure** is set to `0` Pa.
- **Backflow Total Temperature** is set to `300` K.

• For the wall boundary condition **Temperature** is set to `300` k.

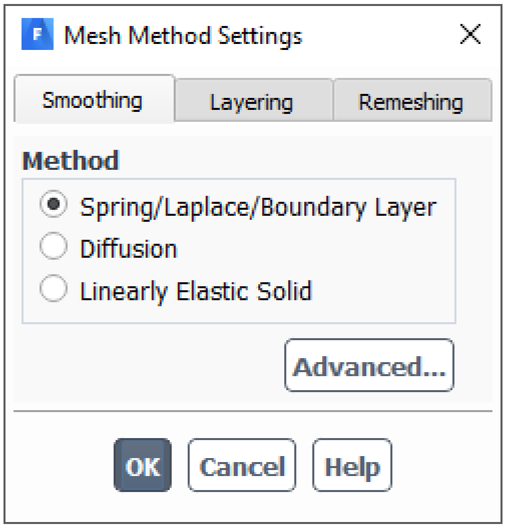

Wall
Zone Name
exvalve1-ib
Adjacent Cell Zone
fluid-exvalve1-ib
Momentum
Thermal
Radiation
Species
DPM
Multiphase
UDS
Wall Film
Potential
Structure
Thermal Conditions
Heat Flux
Temperature
Convection
Radiation
Mixed
via System Coupling
via Mapped Interface
Temperature (k) 300
Wall Thickness (m) 0
Heat Generation Rate (w/m3) 0
Shell Conduction
1 Layer
Edit...
Material Name
aluminum
Edit...
OK
Cancel
Help

6.2.5. Dynamic Mesh Settings in Solver

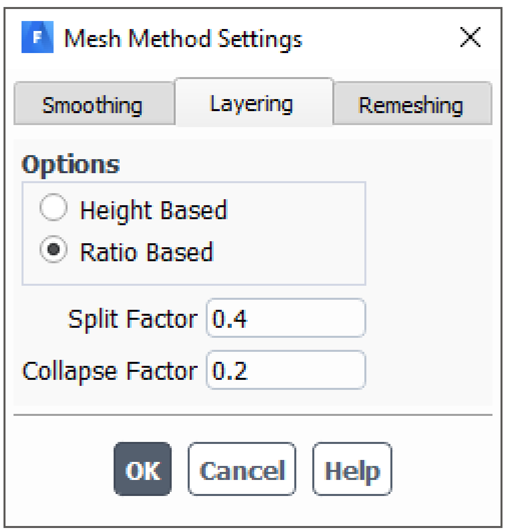


In the **Dynamic Mesh** dialog box you can check the mesh methods and their settings. Click **Settings...** in the **Mesh Methods** group box to open the **Mesh Method Settings** dialog box.

In the **Smoothing** tab the method is set to **Spring/Laplace/Boundary Layer**. The **Parameters** are set by default. For more information, refer to Smoothing Methods in the Fluent User's Guide.



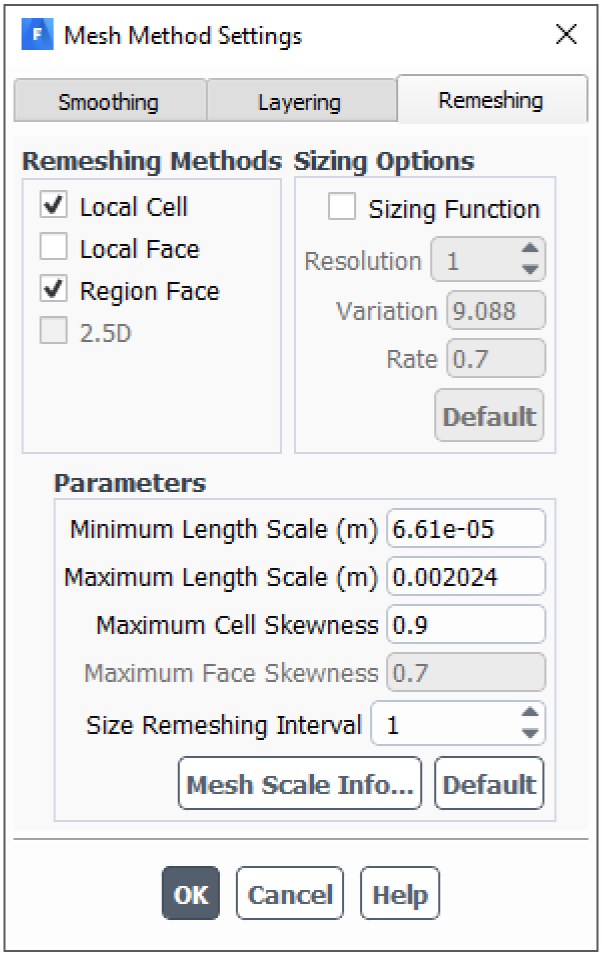
In the **Layering** tab **Ratio Based** is chosen from the **Options** group box. You can control how a cell layer is split by specifying either **Height Based** or **Ratio Based**. The **Split Factor** and **Collapse Factor** are the factors that determine when a layer of cells that is next to a moving boundary is split or merged with the adjacent cell layer, respectively. For more information, refer to Dynamic Layering in the Fluent User's Guide.



Click the **Remeshing** tab to check the methods and parameters. For details, see Remeshing Methods in the Fluent User's Guide. In the **Parameters** group box, the following parameters are set:

- **Maximum Length Scale** = `0.4 X Average length scale of fluid-ch`.
- **Minimum Length Scale** = `1.4 X Average length scale of fluid-ch`.
- By default, the **Maximum Cell Skewness** is set to `0.9` for 3D simulations.
- **Size Remeshing Interval** is set to `1`.

For details, see Local Remeshing Method in the Fluent User's Guide.



You can also see the dynamic mesh zones created and the type to which each zone is set. The zones created differ for a straight and canted valve.

Refer to Straight Valve Geometry With Chamber Decomposition for **IC Engine** (p. 167) for the zones created for a straight valve. The Table 6.3: Dynamic Mesh Zones for a Straight Valve (p. 253) shows the dynamic mesh zones created and the type each zone is set to for a straight valve.

Table 6.3: Dynamic Mesh Zones for a Straight Valve

Dynamic Mesh Zones	Type
`fluid-ch-invalve1`	**Rigid Body**
`fluid-ch-invalve2`	**Rigid Body**
`fluid-ch-lower`	**Rigid Body**
`fluid-invalve1-ib`	**Rigid Body**
`fluid-invalve1-vlayer`	**Rigid Body**
`fluid-invalve2-vlayer`	**Rigid Body**
`fluid-invalve2-vlayer`	**Rigid Body**
`fluid-piston`	**Rigid Body**
`intf-deck-fluid-ch-lower`	**Stationary**
`intf-int-invalve1-ib-fluid-ib`	**Stationary**

`intf-int-invalve1-ob-fluid-vlayer`	**Stationary**
`intf-int-invalve2-ib-fluid-ib`	**Stationary**
`intf-int-invalve2-ob-fluid-vlayer`	**Stationary**
`intf-piston-ch`	**Rigid Body**
`invalve1-ch`	**Rigid Body**
`invalve1-ib`	**Rigid Body**
`invalve1-ob`	**Rigid Body**
`invalve1-seat`	**Stationary**
`invalve2-ch`	**Rigid Body**
`invalve2-ib`	**Rigid Body**
`invalve2-ob`	**Rigid Body**
`invalve2-seat`	**Stationary**
`piston`	**Rigid Body**

For checking zones created for a canted valve with the chamber decomposed, refer to Canted Valve Geometry With Chamber Decomposition for **IC Engine** (p. 170). The Table 6.4: Dynamic Mesh Zones for a Canted Valve With Chamber Decomposition (p. 254) shows the dynamic mesh zones created and the type each zone is set to for a canted valve with the chamber decomposed.

Table 6.4: Dynamic Mesh Zones for a Canted Valve With Chamber Decomposition

Dynamic Mesh Zones	Type
`cyl-piston`	**Rigid Body**
`cyl-quad`	**Rigid Body**
`cyl-tri`	**Deforming**
`exvalve1-ch`	**Rigid Body**
`exvalve1-ib`	**Rigid Body**
`exvalve1-ob`	**Rigid Body**
`exvalve1-seat`	**Stationary**
`fluid-exvalve1-ib`	**Rigid Body**
`fluid-exvalve1-vlayer`	**Rigid Body**
`fluid-invalve1-ib`	**Rigid Body**
`fluid-invalve1-vlayer`	**Rigid Body**
`fluid-layer-cylinder`	**Rigid Body**
`fluid-piston`	**Rigid Body**
`int-piston`	**Rigid Body**
`intf-exvalve1-ob-fluid-ch`	**Deforming**
`intf-int-exvalve1-ib-fluid-ib`	**Stationary**
`intf-int-exvalve1-ob-fluid-vlayer`	**Stationary**
`intf-int-invalve1-ib-fluid-ib`	**Stationary**
`intf-int-invalve1-ob-fluid-vlayer`	**Stationary**

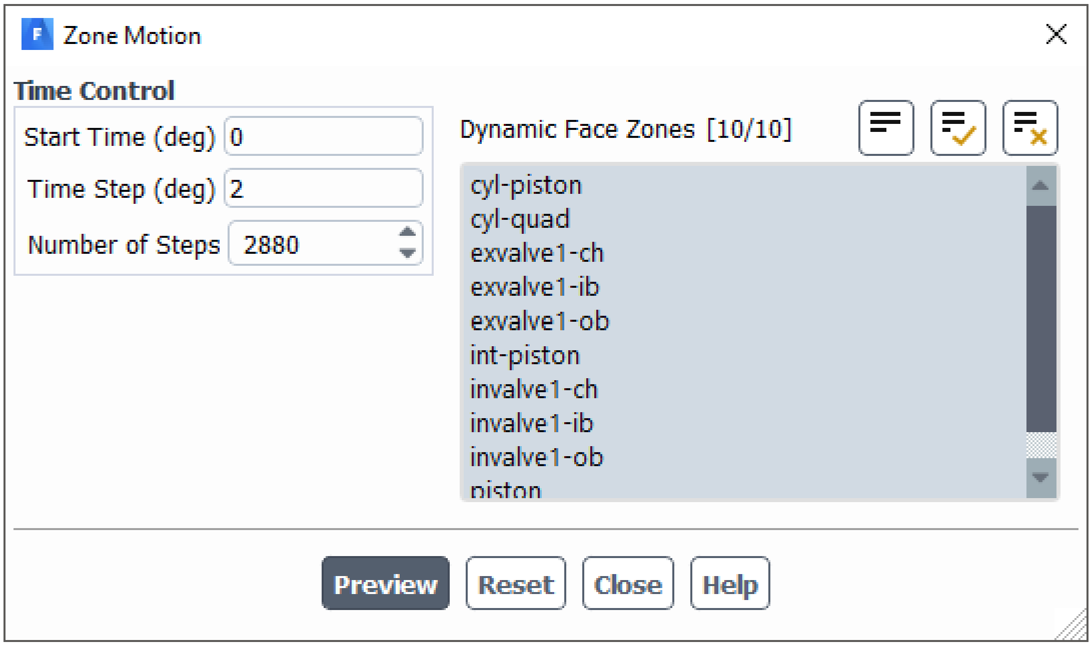
`intf-invalve1-ob-fluid-ch`	**Deforming**
`invalve1-ch`	**Rigid Body**
`invalve1-ib`	**Rigid Body**
`invalve1-ob`	**Rigid Body**
`invalve1-seat`	**Stationary**
`piston`	**Rigid Body**
`symm-cyl-tri`	**Deforming**

For checking zones created for a canted valve without chamber decomposition, refer to Any Engine Geometry Without Chamber Decomposition (p. 173). The Table 6.5: Dynamic Mesh Zones for a Canted Valve Without Chamber Decomposition (p. 255) shows the dynamic mesh zones created and the type each zone is set to for a canted valve without chamber decomposition:

Table 6.5: Dynamic Mesh Zones for a Canted Valve Without Chamber Decomposition

Dynamic Mesh Zones	Type
`cyl-tri`	**Deforming**
`fluid-invalve1-ib`	**Rigid Body**
`fluid-invalve1-vlayer`	**Rigid Body**
`fluid-exvalve1-ib`	**Rigid Body**
`fluid-exvalve1-vlayer`	**Rigid Body**
`intf-int-invalve1-ib-fluid-ib`	**Stationary**
`intf-int-invalve1-ob-fluid-valyer`	**Stationary**
`intf-int-exvalve1-ib-fluid-ib`	**Stationary**
`intf-int-exvalve1-ob-fluid-valyer`	**Stationary**
`intf-invalve1-ob-fluid-ch`	**Deforming**
`intf-exvalve1-ob-fluid-ch`	**Deforming**
`invalve1-ch`	**Rigid Body**
`invalve1-ib`	**Rigid Body**
`invalve1-ob`	**Rigid Body**
`invalve1-seat`	**Stationary**
`exvalve1-ch`	**Rigid Body**
`exvalve1-ib`	**Rigid Body**
`exvalve1-ob`	**Rigid Body**
`exvalve1-seat`	**Stationary**
`piston`	**Rigid Body**
`symm-cyl-tri`	**Deforming**

You can click **Display Zone Motion...** to check the zone motion preview.



Select the **Dynamic Face Zones** and then click **Preview**.

6.2.6. Events Set in Solver

Events are used to model opening and closing of the valves. This is done by making and breaking some non-conformal interfaces. Since you can observe sharp flow gradients around the time when the valves open and close, it is a good practice to reduce the time step size from the default 0.25 crank angle to 0.125 crank angle. Also the URF changes can be synchronized according to the valve opening and closing events. When the dynamic mesh is set up, some events are set to change the time step depending upon the valve opening.

The events are specified for one complete engine cycle. In the subsequent cycles, the events are executed whenever

$$\theta_{event}=\theta_c \pm n\theta_{period} \tag{6.1}$$

where θ_{event} is the event crank angle, θ_c is the current crank angle, θ_{period} is the crank angle period for one cycle, and n is some integer.

As an example, for in-cylinder simulations, events may not execute at the exact event angle. Ansys Fluent will execute an event if the current crank angle is between

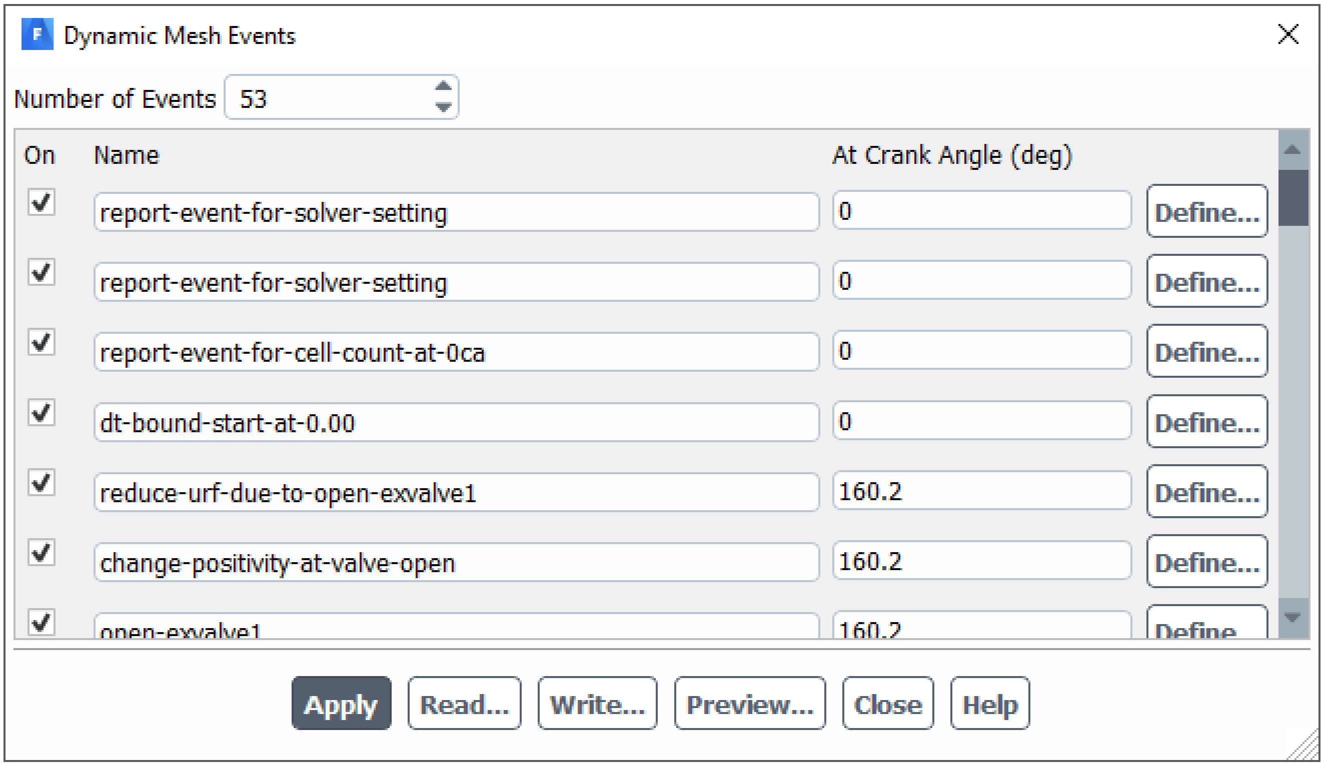
$$\pm 0.5\Delta\theta \tag{6.2}$$

where $\Delta\theta$ is the equivalent change in crank angle for the time step. For example, check the report of Ansys Fluent in the following text window.

```
Execute Event: open-exvalve1 (defined at: 160.3000, current angle: 160.2500)
Execute Event: open-invalve1 (defined at: 324.2000, current angle: 324.2500)
Execute Event: close-exvalve1 (defined at: 402.3000, current angle: 402.2500)
Execute Event: close-invalve1 (defined at: 563.0000, current angle: 563.0000)
```

Notice that event defined at `160.3` is executed at `160.25`. Similarly event defined at `324.2` is executed at `324.25`. This is because they satisfy the condition of Equation 6.2 (p. 256).

If you click **Events** in the **Dynamic Mesh** task page, you can see these events.



Some general events are as follows:

- The time step is set to `0.25` degrees for the simulation.
- When valves open, the time-step changes from:
  - `0.25` to `0.125` at valve opening crank angle
  - `0.125` to `0.25` at least 5 degree after valve opening
- When valves close, the time-step changes from
  - `0.25` to `0.125` at least 5 degrees before the valve closing
  - `0.125` to `0.25` at valve closing crank angle

> **Note:**
>
> The change in time-step size from `0.25` to `0.125` is not made exactly at valve opening or closing, but a few degrees before and after that.

- The number of iterations per time step is set to `30`.

- In case of chamber without decomposition two additional events will be automatically created: one for layer insertion and other for layer deletion.

Note:

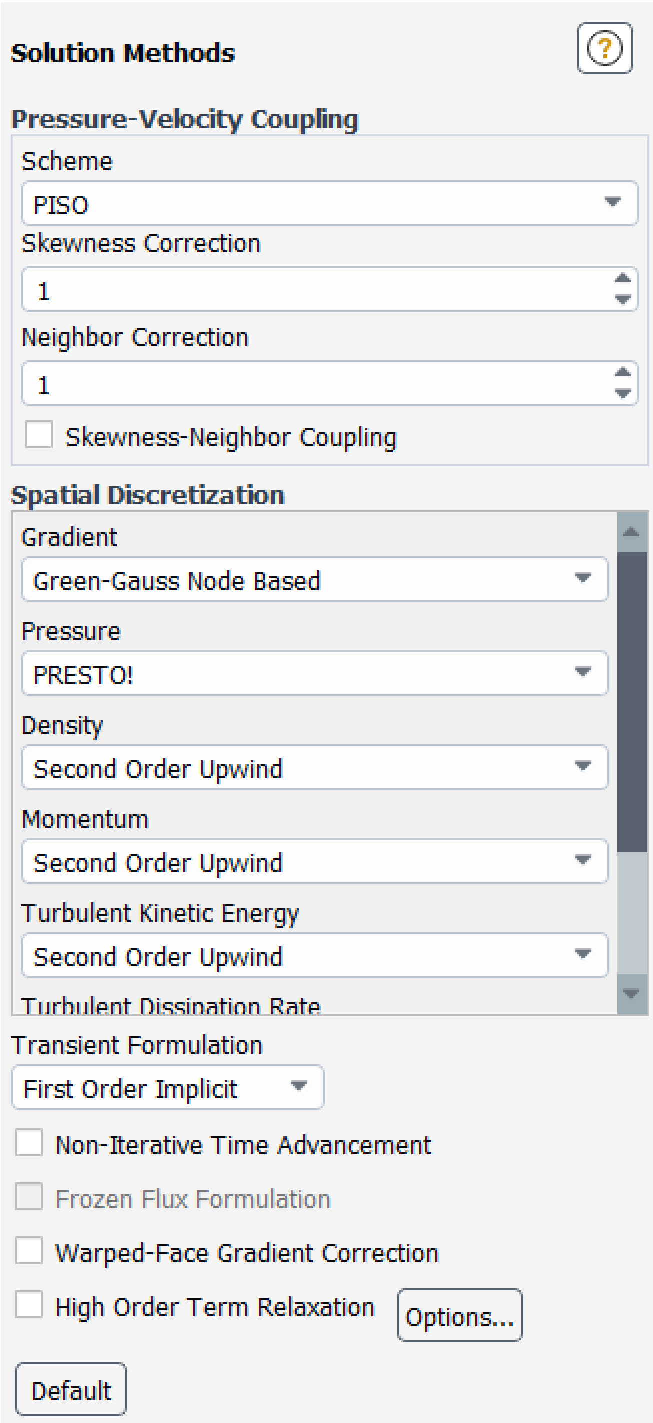
If you are changing the time step size and/or layer height of the cylinder, ensure that at the time of layer deletion there is only one layer. If there are more than one cell layers, delete layer will fail. If all the layers collapse by the time delete layer event is called, negative volume error will occur.

You can check the events by clicking **Define**, next to each event name.

Note:

The calculated valve opening and closing angles are different from the actual angles. For more information, refer to How IC Engine System Calculates Valve Opening and Closing Angles (p. 549).

6.2.7. Solution Methods Set in Solver



- The **Scheme** for the analysis is set to **PISO** in the **Pressure-Velocity Coupling** group box.

Note:

This scheme is recommended for all the transient solutions, especially when larger time steps have to be used. Also, PISO with neighbor correction and skewness correction will be used for highly distorted meshes.

- **Skewness Correction** is set to 1.

Note:

By default this value is 0 but considering that due to MDM the mesh quality deteriorates, it is good practice to set the skewness correction to 1.

- **Neighbor Correction** is set to 1.
- **Skewness-Neighbor Coupling** is disabled.

Note:

For highly distorted meshes, disabling this is recommended.

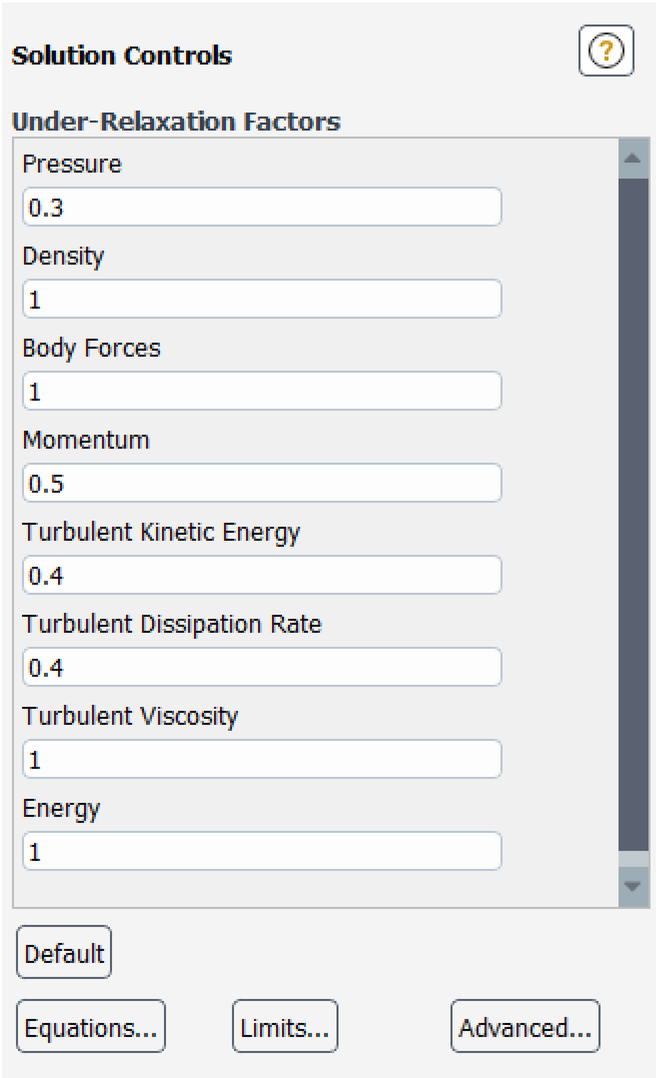
- The **Gradient** in the **Spatial Discretization** group box is set to **Green Gauss Node Based**.

Note:

For tet meshes it is good to use this gradient option. **Least Square Cell Based** can be another good option to use.

- **Pressure** is set to **PRESTO!**.
- **Density**, **Momentum**, **Turbulent Kinetic Energy**, and **Energy**, are all set to **Second Order Upwind**.
- **Turbulent Dissipation Rate** is set to **First Order Upwind**.

6.2.8. Solution Controls Set in Solver



At the beginning of the solution the **Under-Relaxation Factors** are set as following:

- **Pressure**: `0.3`
- **Momentum**: `0.5`
- **Turbulent Kinetic Energy** and **Turbulent Dissipation Rate**: `0.4`
- **Turbulent Viscosity**: `1`
- **Density**, **Body Force**, and **Energy** are all set to `1`

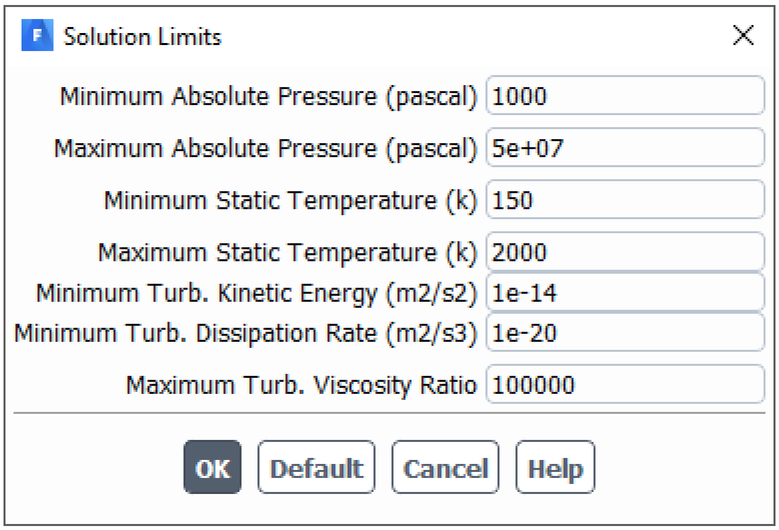
Five degrees before the valve opens/closes and till five degrees after the valve opens/closes, the following will be the **Under-Relaxation Factors**:

- **Pressure**: `0.2`
- **Momentum**: `0.4`

- **Turbulent Kinetic Energy** and **Turbulent Dissipation Rate**: `0.2`
- **Turbulent Viscosity**: `1`
- **Density**, **Body Force**, and **Energy** are all set to `1`

> **Note:**
>
> After the valve opens/closes to more than five degrees, all the **Under-Relaxation Factors** are reset to the settings at the beginning of the solution (p. 261).

- **Solution Limits** for pressure and temperature are set as shown below:

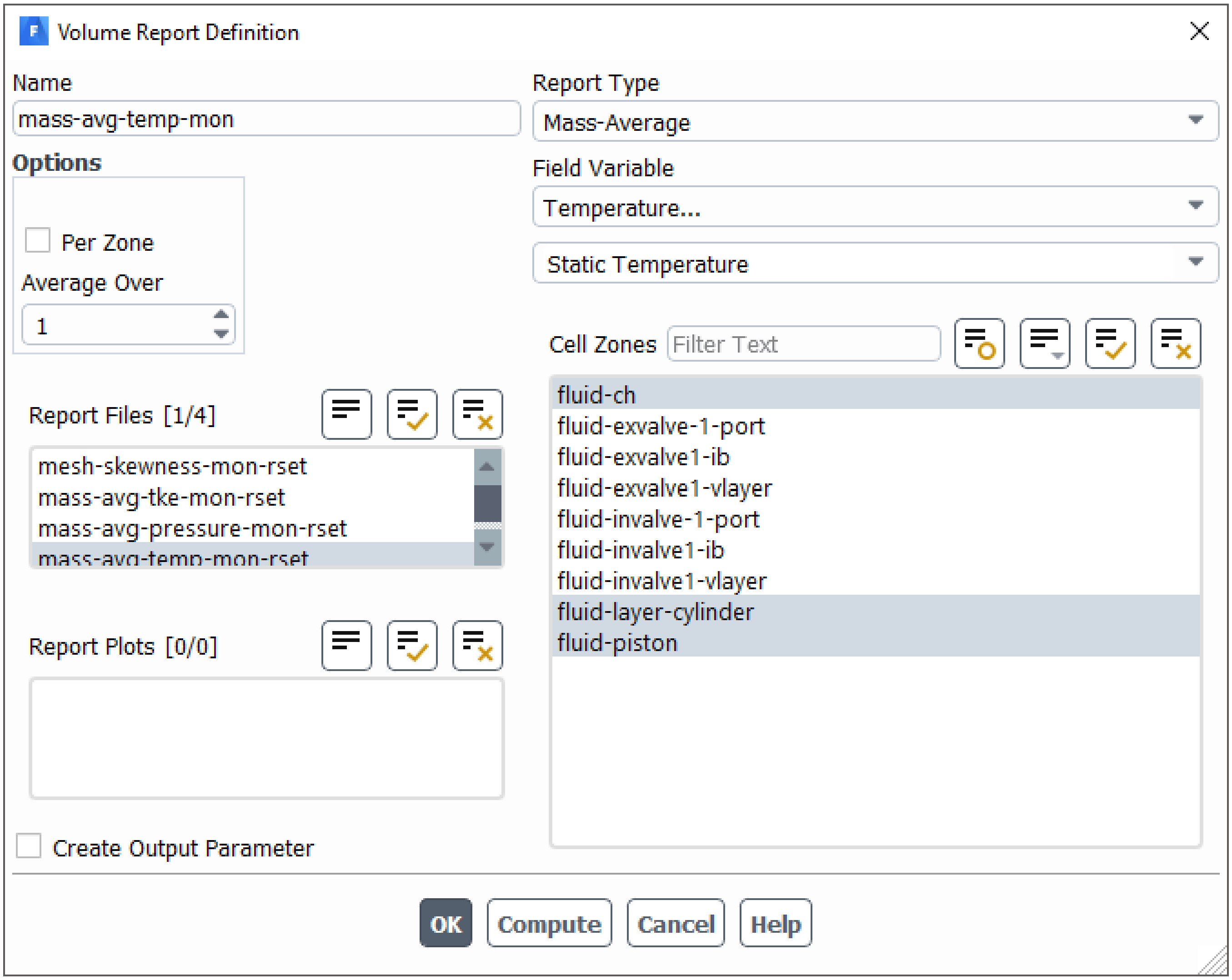


  - **Minimum Absolute Pressure (pascal)** is set to `1000`.
  - **Maximum Absolute Pressure (pascal)** is set to `5e+07`.
  - **Minimum Static Temperature (k)** is set to `150`.
  - **Maximum Static Temperature (k)** is set to `2000`.

> **Note:**
>
> The temperature and pressure limits are restricted as indicated above because the values of these variables should not cross this specified range in some cells, typically in cold-flow simulations. However, if you are modeling combustion or gas exchange, or if you know that these limits are too aggressive for your case, then you can modify them appropriately.

6.2.9. Monitors Set in Solver

To check the output results of the parameters, some monitors have been created.



The monitors defined by default are **Volume Report Definitions**. The variables they plot are

- **Mass Average** of temperature
- **Mass Average** of pressure
- **Volume Average** of cell skewness
- **Mass Average** of turbulent kinetic energy.

These monitors are defined on **fluid-ch**, **fluid-layer-cylinder**, **fluid-piston** cell zones.

You can create your own **Surface Monitors** or **Volume Monitors** from the **Monitors** task page. Enable **Write** so that an output file is written, which can be loaded and viewed later. The monitor settings which are done by default can be seen in the `icBcSettings.txt` file.

The swirl ratio and tumble about x-axis in the chamber region is also plotted. A file `ice-incylinder-output.txt`, is written in the Fluent directory of the project. Details of the quantities written to the file are as follows:

CA = Crank Angle

L = Angular momentum vector of fluid mass contained in selected cell zones with respect to the swirl center

$|\vec{L}|$ = Magnitude of angular momentum of fluid

$\vec{sa}$ = Swirl Axis

$\vec{ta}$ = Tumble Axis

$\vec{cta}$ = Cross Tumble Axis

I $_{sa}$ = Moment of inertia of the fluid mass about Swirl axis

I $_{ta}$ = Moment of inertia of the fluid mass about Tumble Axis

I $_{cta}$ = Moment of inertia of the fluid mass about Cross Tumble Axis

$\cdot$ = Dot product between two vectors

Altogether, the previous quantities are combined to yield eight columns of data in the output file, as shown in the figure that follows:

CA	(L , sa)	(L , ta)	(L ,cta)	|L|	Isa	Ita	Icta
350.00	0.0000e+00	0.0000e+00	0.0000e+00	0.0000e+00	1.1474e-07	7.0881e-08	6.8234e-08
365.00	6.4910e-07	7.6332e-07	-2.3923e-08	1.0023e-06	9.9355e-08	6.1951e-08	5.9120e-08
380.00	-1.6684e-06	-2.3585e-06	-7.8704e-08	2.8900e-06	1.5460e-07	9.2601e-08	9.2951e-08
395.00	-3.0125e-05	8.1780e-06	2.9730e-06	3.1357e-05	2.9557e-07	1.7709e-07	1.8031e-07
410.00	-8.9259e-05	1.7637e-05	-1.3191e-05	9.1936e-05	4.9396e-07	3.1077e-07	3.1660e-07
425.00	-2.1336e-04	2.4657e-05	-4.6908e-05	2.1984e-04	7.3845e-07	5.1056e-07	5.1811e-07
440.00	-3.9555e-04	6.6114e-05	-1.0156e-04	4.1370e-04	1.0084e-06	7.8920e-07	7.9931e-07
455.00	-6.0621e-04	1.2651e-04	-1.7990e-04	6.4487e-04	1.2833e-06	1.1499e-06	1.1623e-06
470.00	-8.1472e-04	2.0109e-04	-2.6251e-04	8.7927e-04	1.5486e-06	1.5784e-06	1.5913e-06
485.00	-9.9456e-04	2.7342e-04	-3.6243e-04	1.0933e-03	1.7921e-06	2.0409e-06	2.0526e-06
500.00	-1.1160e-03	2.9711e-04	-4.5477e-04	1.2412e-03	2.0003e-06	2.4850e-06	2.4945e-06

The swirl ratio can be deduced from the IC output file using the following expression:

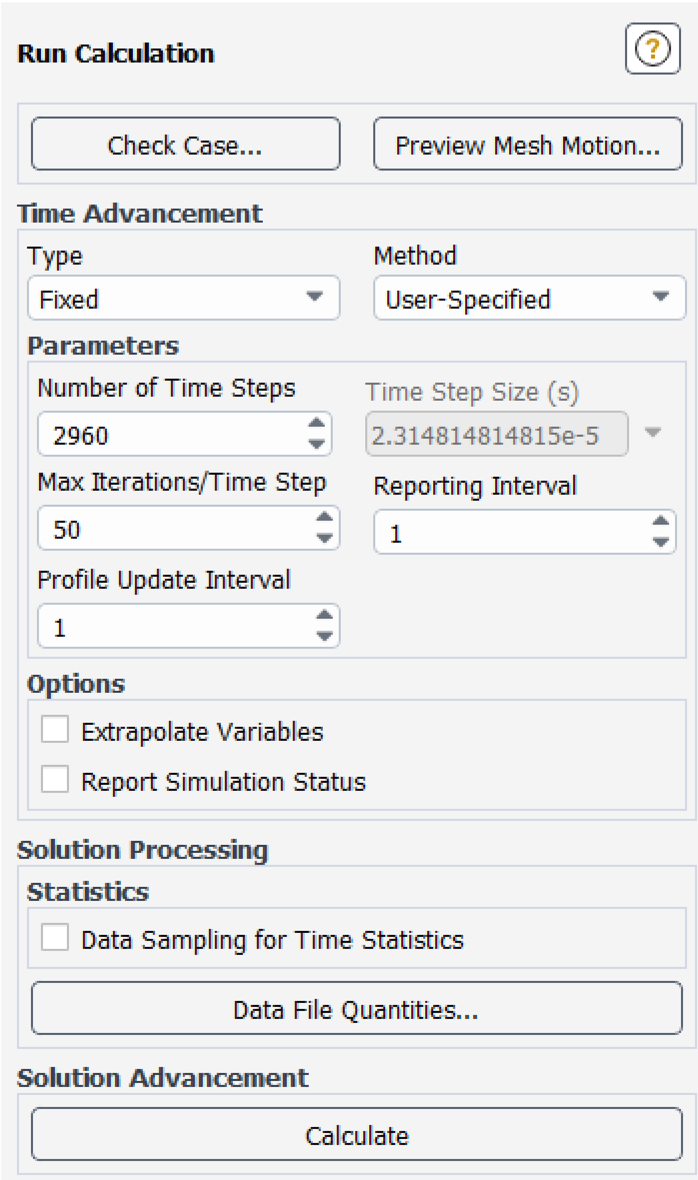
$$\frac{L{\cdot}sa \big/ I_{sa}}{(EngineRPM*2*3.141) \big/ 60} \tag{6.3}$$

Similarly the tumble ratio can be computed as follows:

$$\frac{L{\cdot}ta \big/ I_{ta}}{(EngineRPM*2*3.141) \big/ 60} \tag{6.4}$$

For more information refer to, In-Cylinder Settings in the Fluent User's Guide.

6.2.10. Run Calculation



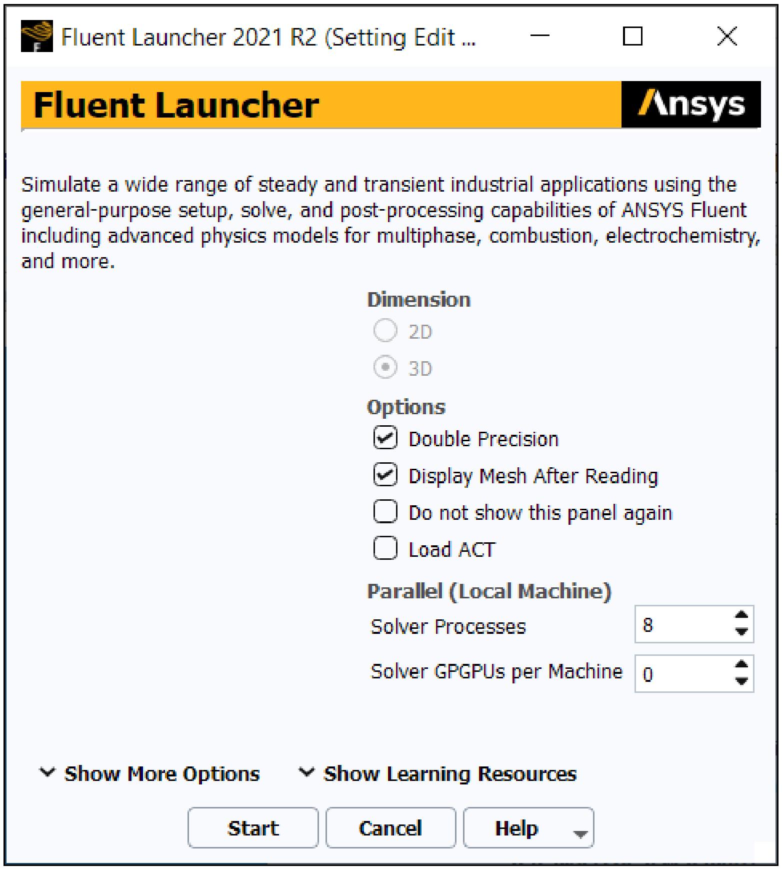
You can enter the **Number of Time Steps** you want. By default it is set to a calculated value from the **Number of CA to Run** which you set in the **Basic Settings** tab of **ICE Solver Settings** dialog box. Click **Calculate** to run the simulation.

Note:

If time-step size is 0.25 CA and there are no changes in it through events, one cycle which is 720 degrees would mean 2880 time-steps. If you are reducing the time-step size during valve closing and opening through events, you will have to account for that also.

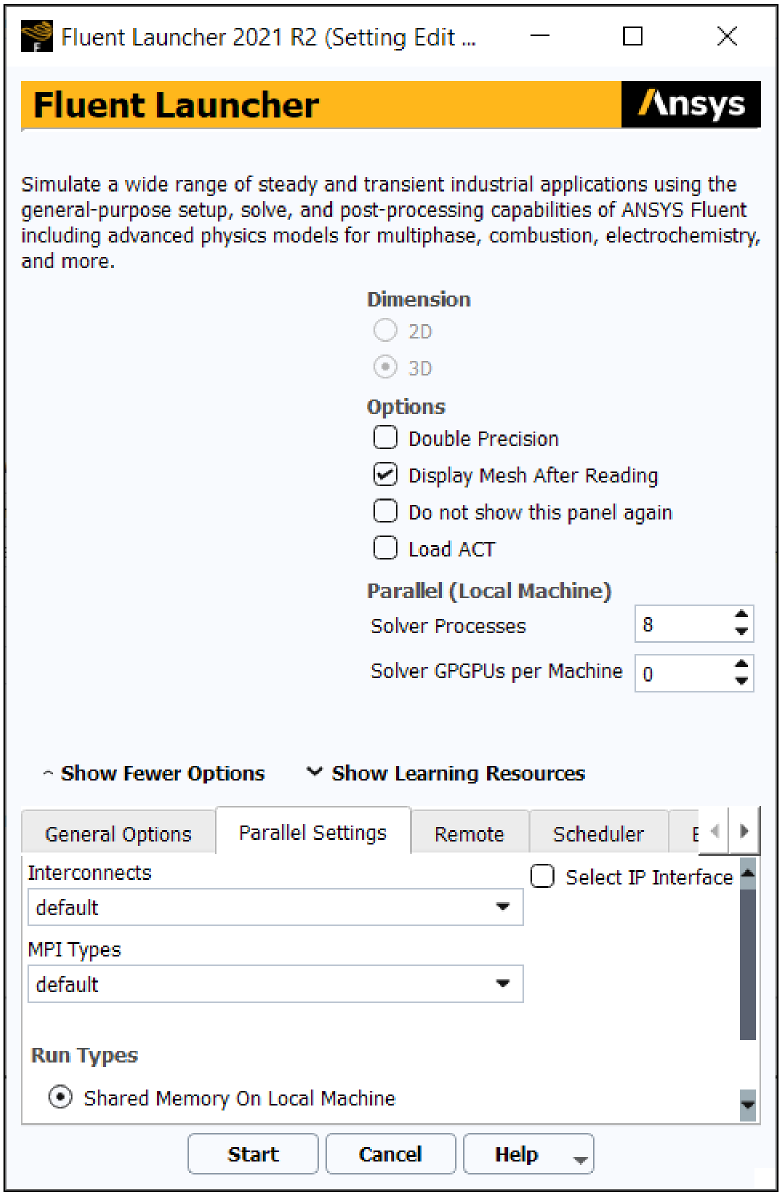
Running the Calculation from Windows on Remote Linux Machines

To run the simulation on Linux, some changes in the setup are required.



1. Click **Show More Options**.

2. Click the **Parallel Settings** tab.



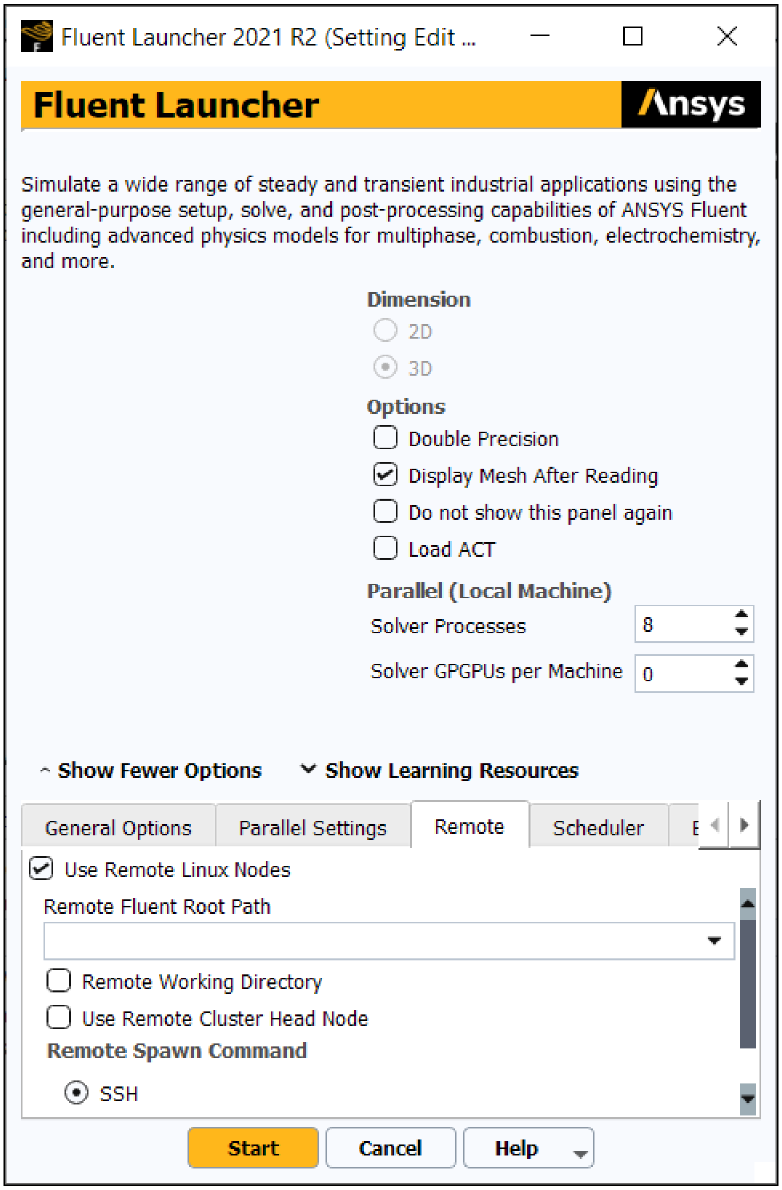
Select the **Run Types** from the list.

Note:

If you selected **Distributed Memory on a Cluster**, then you have to provide the **Machine Names**.

---

3. Click the **Remote** tab.



a. Enable **Use Remote Linux Nodes** option.

b. Enter the path for **Remote FLUENT Root Path**. It is the root directory of Fluent installation.

c. Select your choice from the list of **Remote Spawn Command**.

Note:

For **SSH**, you need to set up a password-less connection before running the solution remotely.

d. Enable **Use Remote Cluster Head Node** and provide the node.

e. Enter the number for **Number of Processes** under **Processing Options**.

Note:

When the node for **Use Remote Cluster Head Node** is specified, then that name will be reflected under **Processing Options**.

When the remote connection is set, it will use the nodes of the remote Linux machine. The details are displayed in the Fluent console.

```
Host spawning Node 0 on machine "pundevrhel64r6" (lnamd64).
/usr/local/fluent/develop/fluent14.0.0/bin/fluent -r14.0.0 3d -pdefault -node -t4 -ssh -mport 10.14.6.198:10.14.2.108:2673:0
Starting /usr/local/fluent/develop/fluent14.0.0/multiport/mpi/lnamd64/pcmpi/bin/mpirun -np 4 /usr/local/fluent/develop/fluent14.0.0/l
Platform-MPI licensed for FLUENT.

-------------------------------------------------------------------------------
ID      Comm.   Hostname         O.S.         PID      Mach ID HW ID    Name
-------------------------------------------------------------------------------
host    net     punptstxp64t1    Windows-x64 177216   1       5        Fluent Host
n3      pcmpi   pundevrhel64r6. Linux-64     17969    0       3        Fluent Node
n2      pcmpi   pundevrhel64r6. Linux-64     17968    0       2        Fluent Node
n1      pcmpi   pundevrhel64r6. Linux-64     17967    0       1        Fluent Node
n0*     pcmpi   pundevrhel64r6. Linux-64     17966    0       0        Fluent Node

Selected system interconnect: shared-memory
```

All the files will be saved in the working directory. Once the solution is completed, the project is updated and you can proceed with your postprocessing.

For details see Setting Additional Options When Running on Remote Linux Machines in the Fluent User's Guide.

Note:

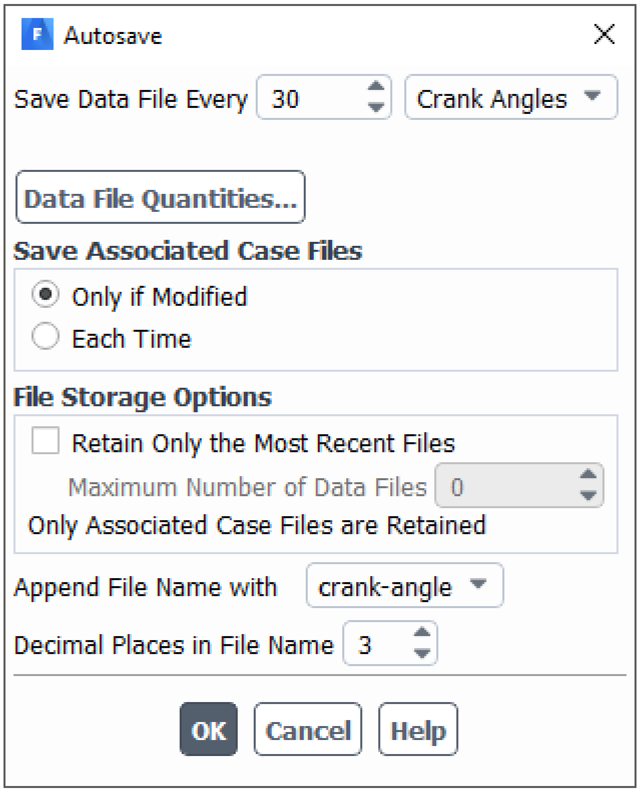
You can also submit the run on different Linux machines though RSM.

Running on Standalone Fluent

1. Open Fluent and read the case and data file from the `~project-name_files\dp0\ICE\Fluent` directory.

   File → **Read** → **Case & Data...**

2. In **Calculation Activities** task page click **Edit....** next to **Autosave Every (Time Steps)** and check if that the name and path under **File Name** is correct.



3. In **Run Calculation** task page enter the required **Number of Time Steps** and click **Calculate**.

   Note:

   If the case is setup in the previously released versions, then you might have to read the scheme file, (`WB-ICE-Solver-Setup.scm`) from `~ANSYS Inc\v150\Addins\ICEngine\CustomizationFiles` folder in the mapped directory before running it in the present version.

Note:

If you want the generate the report from IC Engine System report template, then you need to first make sure that **Solution** cell is updated. This can be done by running the solution for 1 timestep. Then, copy all the files generated by your standalone run into the Fluent directory and update the **Results** cell.

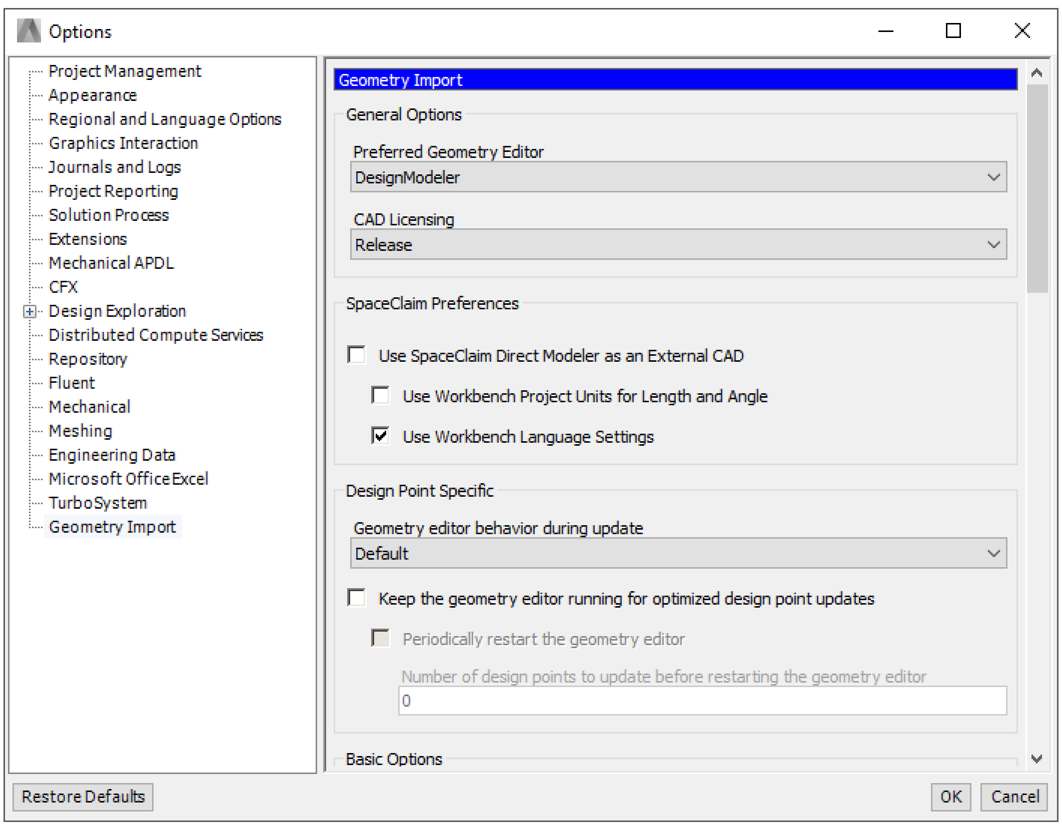
Chapter 7: Port Flow Simulation: Preparing the Geometry in IC Engine

This chapter provides instructions and information about preparing the IC engine geometry for port flow simulation.

Once you have selected **Port Flow Simulation** from the **Simulation Type** drop-down list in the **Properties** view, you can open DesignModeler from the **Geometry** cell in the **IC Engine** analysis system.

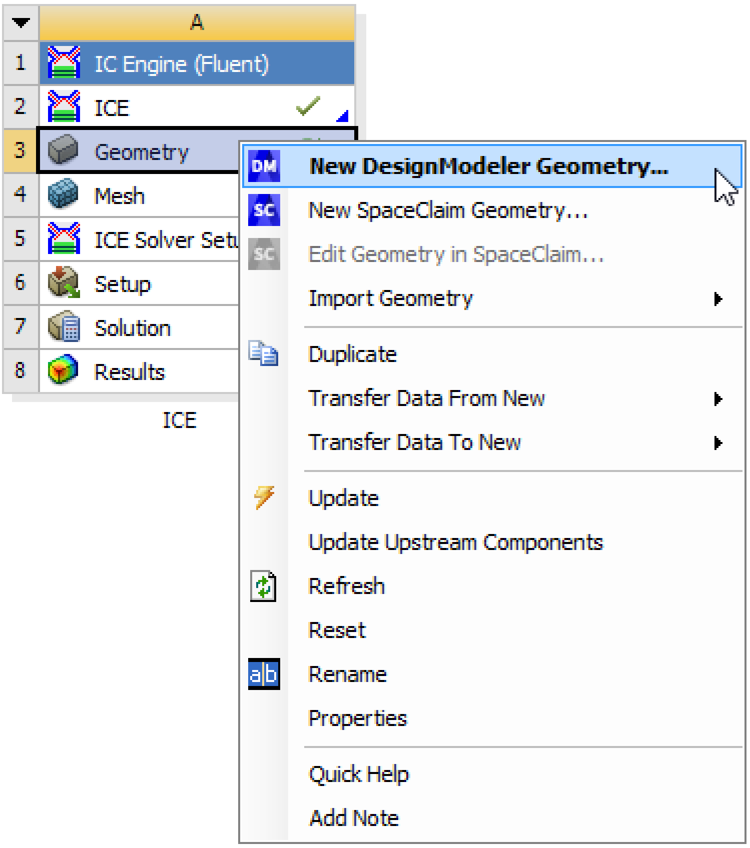
Note:

To open DesignModeler from the **Geometry** cell, you should set the **Preferred Geometry Editor** to **DesignModeler** in the **Geometry Import** section in the **Options** dialog box. The **Options** dialog box can be opened from the **Tools** → **Options...**menu.



Then you can double-click **Geometry**, cell 3, to open DesignModeler.

One more way to open DesignModeler is, to right-click on the **Geometry** cell, and select **New DesignModeler Geometry...** from the context menu.



7.1. Geometry Decomposition for Port Flow Simulation

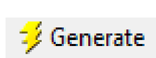
Note:

It is a good practice to check the geometry before decomposing; there are a few things that you should verify so that fewer problems occur later on. For more information, see Geometry Check (p. 517).

1. Set the units, depending upon the geometry units, in ICE-DesignModeler.

2. Load the geometry file.

   File → Import External Geometry File...

3. Click **Generate** 

 to complete the import.

4. Set up the **Input Manager** by clicking **Input Manager** (Input Manager located in the **IC Engine** toolbar).

   Note:

   The **IC Engine** toolbar is displayed in the ICE DesignModeler only after installing IC Engine Analysis System.

Details View

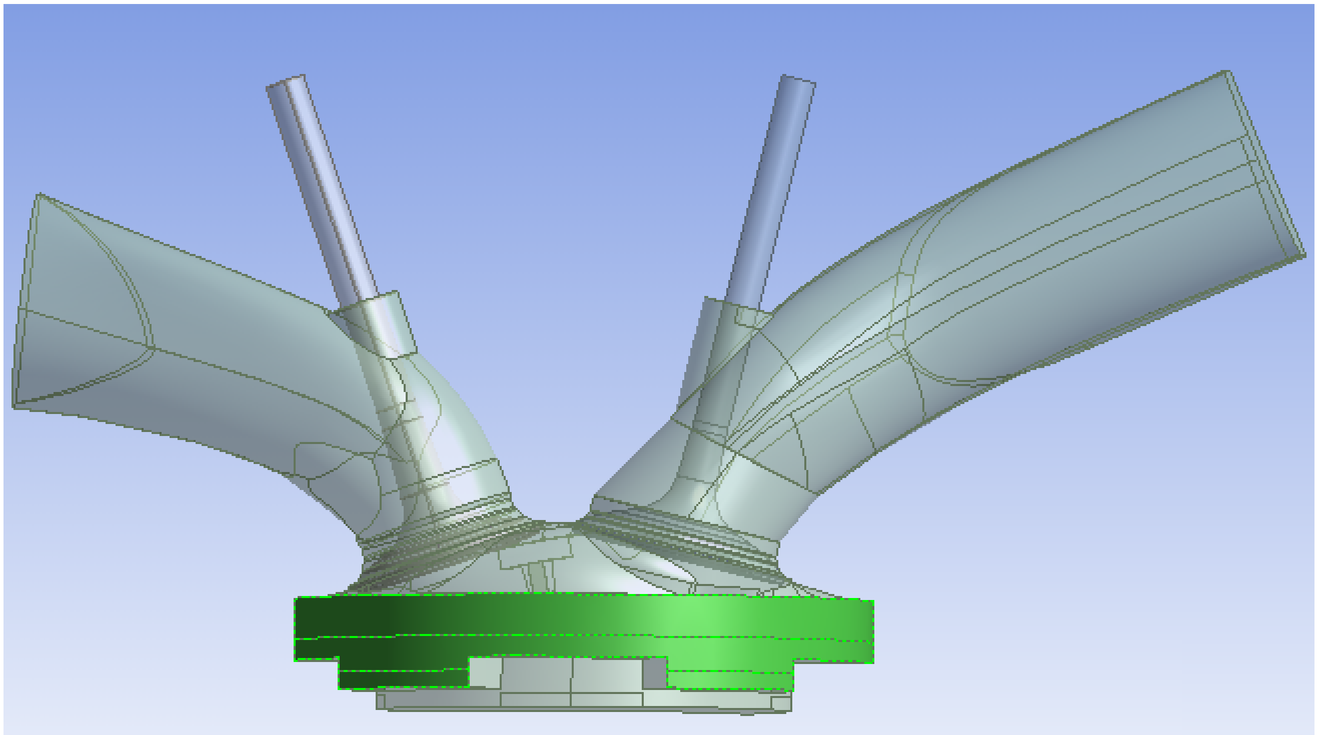
Details of InputManager1	
Slice	InputManager1
Cylinder Liner Faces	1 Face
Symmetry Face Option	Yes
Symmetry Faces	1 Face
Post Planes Dist. From Ref.	12.0; 17.1; 25.0; 33.0 (mm)
IC Valves Data 1 (RMB)	
Valve Type	InValve
Valve Bodies	1 Body
Valve Seat Faces	1 Face
FD1, Valve Lift	7 mm
IC Valves Data 2 (RMB)	
Valve Type	ExValve
Valve Bodies	1 Body
Valve Seat Faces	1 Face
FD2, Valve Lift	12 mm
IC Inlet Plenum 1 (RMB)	
Inlet/(Plenum Inlet) Faces	1 Face
Plenum Type	Hemisphere
Inlet Extension Length	38.1 mm
Plenum Size	139.7 mm
Plenum Blend Rad	25 mm
IC Outlet Plenum (RMB)	
Outlet Plenum Option	Yes
Cylinder Extension Length	267.71 mm
Plenum Type	Cylinder
Plenum Size	341.66 mm

Details of InputManager

This section in the **Input Manager** dialog box takes inputs of the engine to set it up for decomposition.

- Select all the faces of the engine cylinder for **Cylinder Liner Faces** and click **Apply**.

Figure 7.1: Cylindrical Face Selection



The cylinder radius and the cylinder axis are displayed in the status bar at the bottom of the window.

Note:

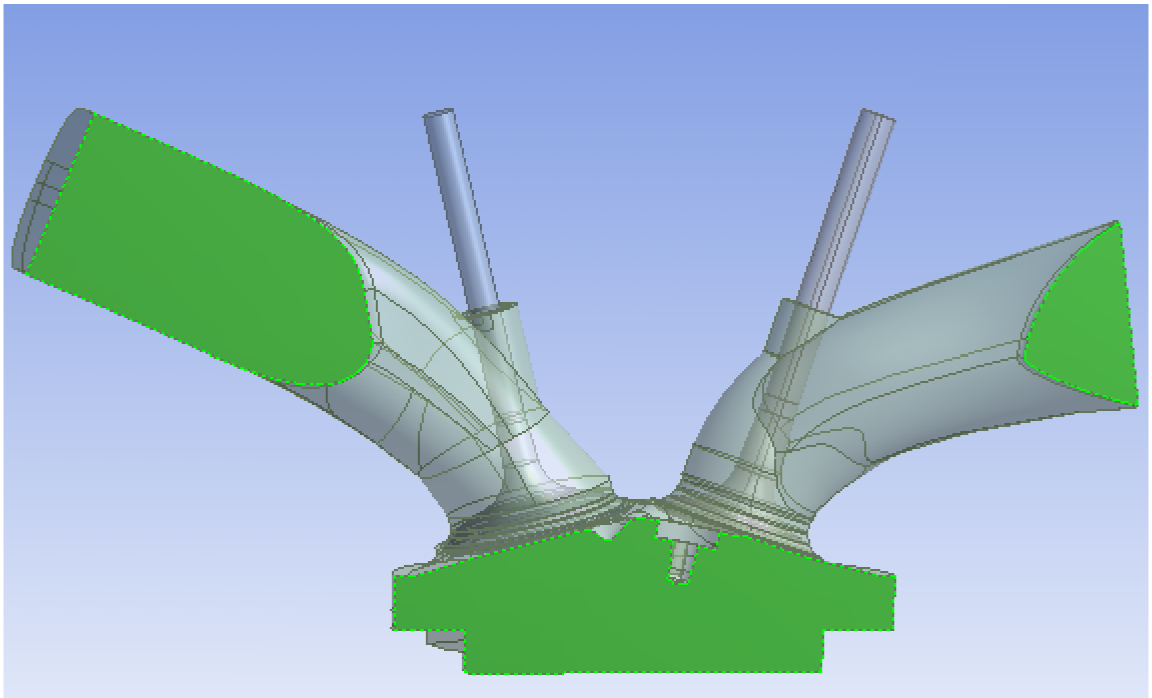
Ensure that all the faces of the cylinder body are selected.

- After selecting the **Cylinder Liner Faces** the program will check for symmetry. It will accordingly set **Yes** or **No** for the **Symmetry Face Option**. If **Yes** is selected you will be required to give the symmetry faces.

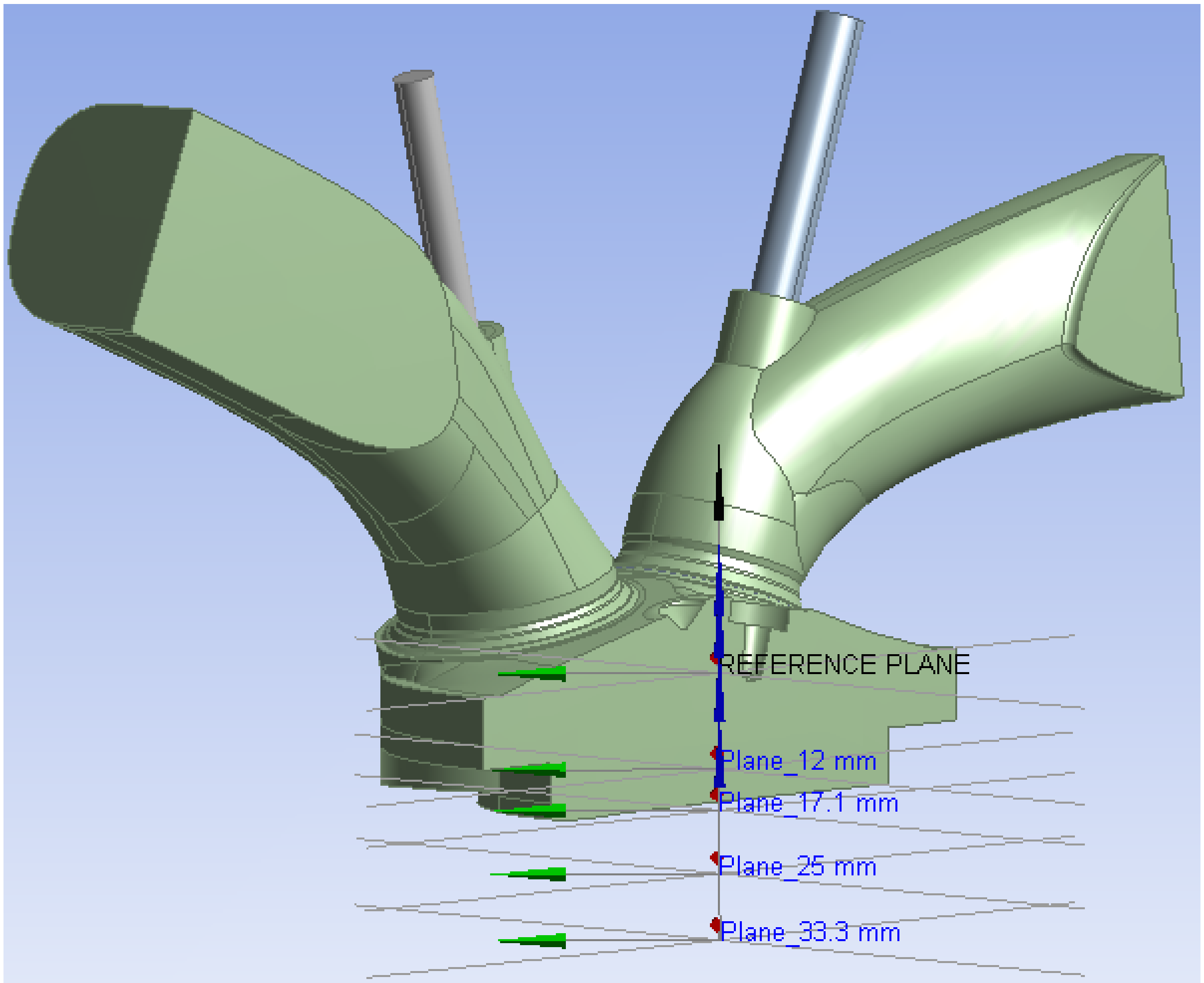
Note:

You should select **Yes** for **Symmetry Face Option** if the geometry is symmetrical. Failure to do so will result in the failure of the simulation. Also ensure to select the faces for **Symmetry Faces**.

- Click next to **Symmetry Faces** in the **Details View** of **Input Manager**. Select the faces of the engine about which it is symmetric and click **Apply**.



- For **Post Planes Dist. From Ref.** you can enter the distance from the reference plane at which you would like to have the postprocessing planes. These planes are required for creating swirl monitors in Fluent. If you do not provide any distance, a default plane is created below the valve at an approximate distance of (4 X **Reference Size** (p. 288)) from the Reference Plane. This helps in creating sweep mesh in the cylinder and plenum.



The Reference Plane is created near the top of the cylinder, where the cylinder meets the dome. For multiple planes separate the distances by a semicolon (**;**). For example, `12.0;17.1;25.0;33.3 (mm)`. The distances will be sorted in the ascending order automatically. The planes are visible in the geometry after you enter the distances.

For information on how the swirl ratio is monitored on the swirl planes see the section on swirl definition (p. 582).

IC Valves Data

In this section you are required to give details about the valves.

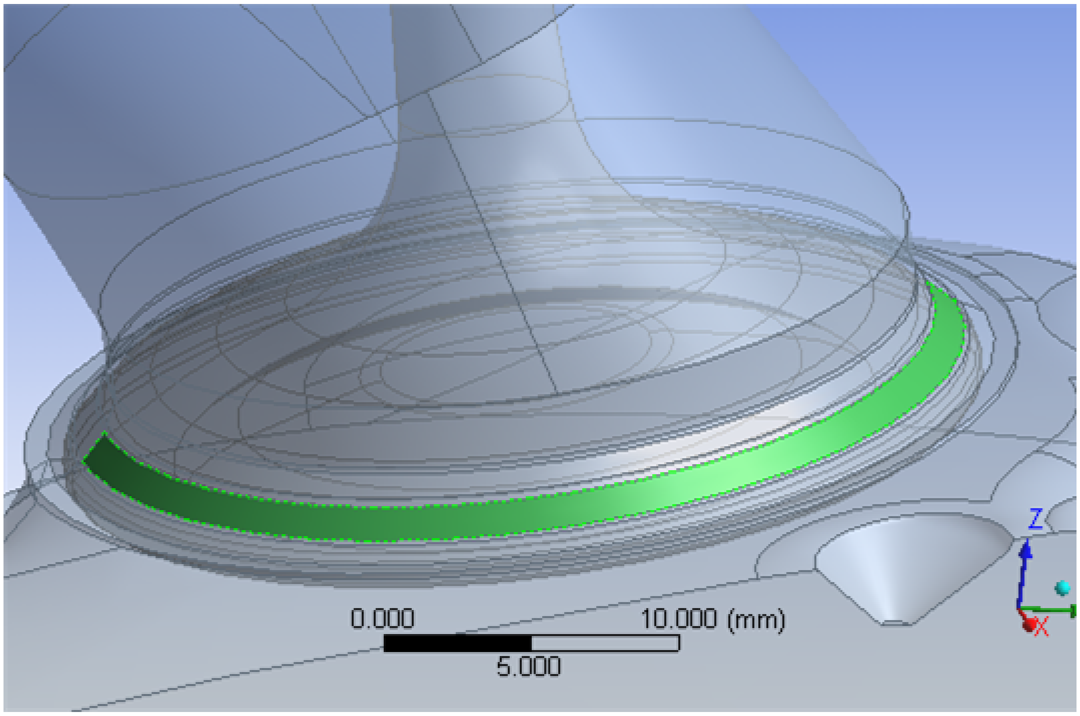
- Select **InValve** or **ExValve** as the **Valve Type** from the drop-down list.
- Select the valve from the figure and click **Apply** next to **Valve Bodies**.

Note:

- You can select multiple valves at a time and set them to the desired type.
- Confirm that **Selection Filters** are set to **Bodies**.

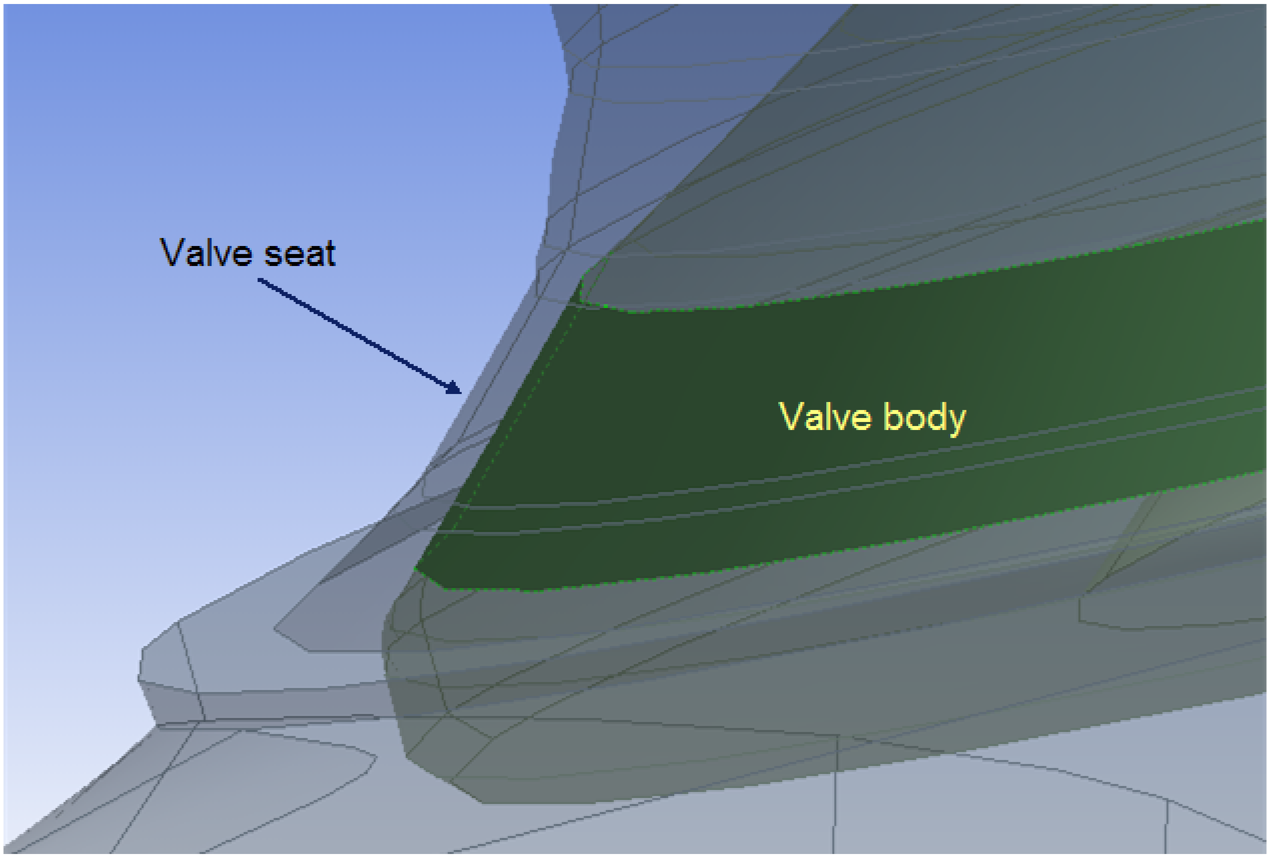
- Select the faces on which the valves rests for **Valve Seat Faces** and click **Apply**.

Figure 7.2: Valve Seat Faces



The valve seat is that face of the port body which comes in contact with the valve body. The Figure 4.5: Valve Seat Selection (p. 163) shows that the highlighted face of the valve body will make contact with the valve seat face of the port body.

Figure 7.3: Valve Seat Selection

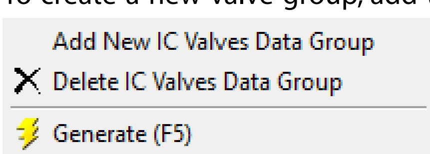


Note:

Confirm that **Selection Filters** are set to **Model Faces**. Some engines might have line contact between the valve and the valve seat. In such cases you can select the face which is near the valve and includes the line of contact.

- Enter a value for **Valve Lift**. This will be the distance to which you want the valves to be moved. If you retain the default value of **0** then the valve is closed and the selected ports will be deactivated.

- To create a new valve group, add the section **IC Valves Data** by right-clicking,

  and selecting **Add New IC Valves Data Group**. Set the details for the new group.

IC Inlet Plenum

In this section, you give the inputs required for constructing the inlet plenum.

- Select the inlet face where the inlet plenum is created for **Inlet/(Plenum Inlet) Faces**. If the plenum is already created, select its inlet face for this option. When the face is selected, the hydraulic diameter is displayed in the status bar.

- You can select the shape of the plenum from the **Plenum Type** drop-down list. You have four options to choose from:

  - **None**

  - **Hemisphere**

  - **Cylinder**

  - **Box**

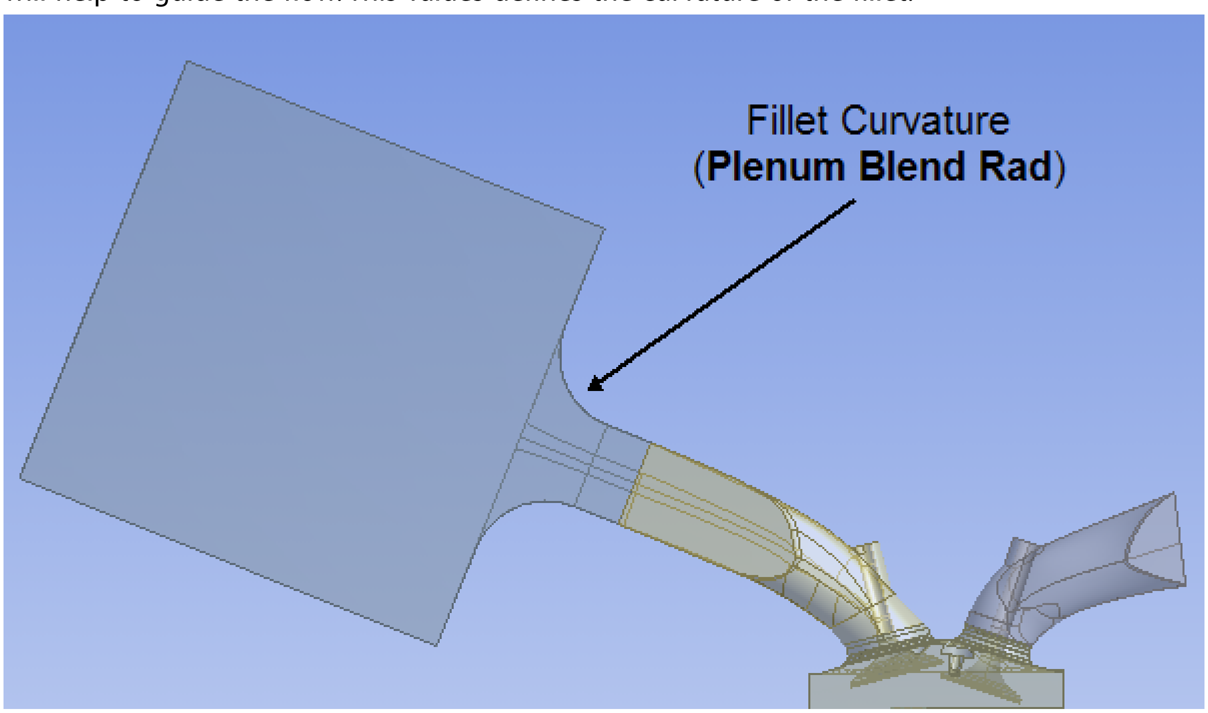
  On selection of either of **Hemisphere**, **Cylinder**, or **Box** you will have to provide details for the location of the plenum and its size. If you select **None** it is assumed that the geometry includes the inlet plenum.

- Enter a value for **Inlet Extension Length**. This value is the distance at which the inlet plenum is created from inlet face. By default it is set to 1.5 times the hydraulic diameter of the inlet.

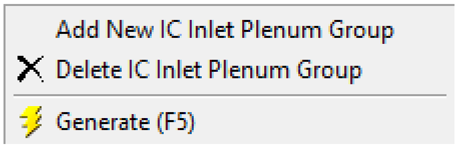
- Enter a value for **Plenum Size**. This value is a set by default to 5.5 times the size of the hydraulic diameter of the inlet. .

  The size of the input plenum will be as per the **Plenum Size** and the shape as per the selection of **Plenum Type**.

- Enter a value for **Plenum Blend Rad**. This will create a curve of the given radius which will help to guide the flow. This values defines the curvature of the fillet.



- To create a new inlet plenum group, add the section **IC Inlet Plenum** by right-clicking



and selecting **Add New IC Inlet Plenum Group**. Set the details for the new group.

Note:

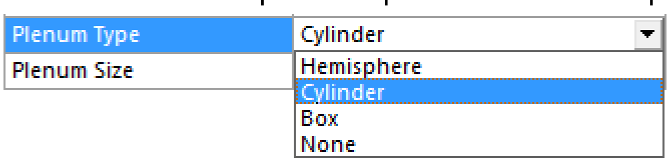
The significance of a group is to create a common plenum for all the faces of the group. For example, If you have two separate inlet faces, and if you select both of them in a single group, then it will create a single plenum joining these faces. On the other hand if for the same input faces, you create two different inlet plenum groups, then disconnected plenums are created.

IC Outlet Plenum

In this section, you give the inputs required for constructing the outlet plenum.

- You can select **Yes** or **No** from the **Outlet Plenum Option** drop-down list.
  - If you select **No** it is assumed that you have provided the outlet pleum in the geometry. In this case you will have to provide the outlet face of the plenum for **Outlet Plenum Faces**.
  - If you select **Yes** you can create the outlet plenum.
- Enter a value for **Cylinder Extension Length**, the length to which the cylinder is extended from the present position. This value is set by default to three times the cylinder diameter.

  You can select the shape of the plenum from the drop-down list of **Plenum Type**.



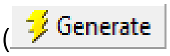
  You have four options to choose from:
  - **Hemisphere**
  - **Cylinder**
  - **Box**
  - **None**

Note:

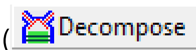
If you select the option **None** from the **Plenum Type** drop-down list, then you can extend the cylinder without creating an outlet plenum.

- Enter a value for **Plenum Size**. It is set by default to 3.5 times the cylinder diameter.

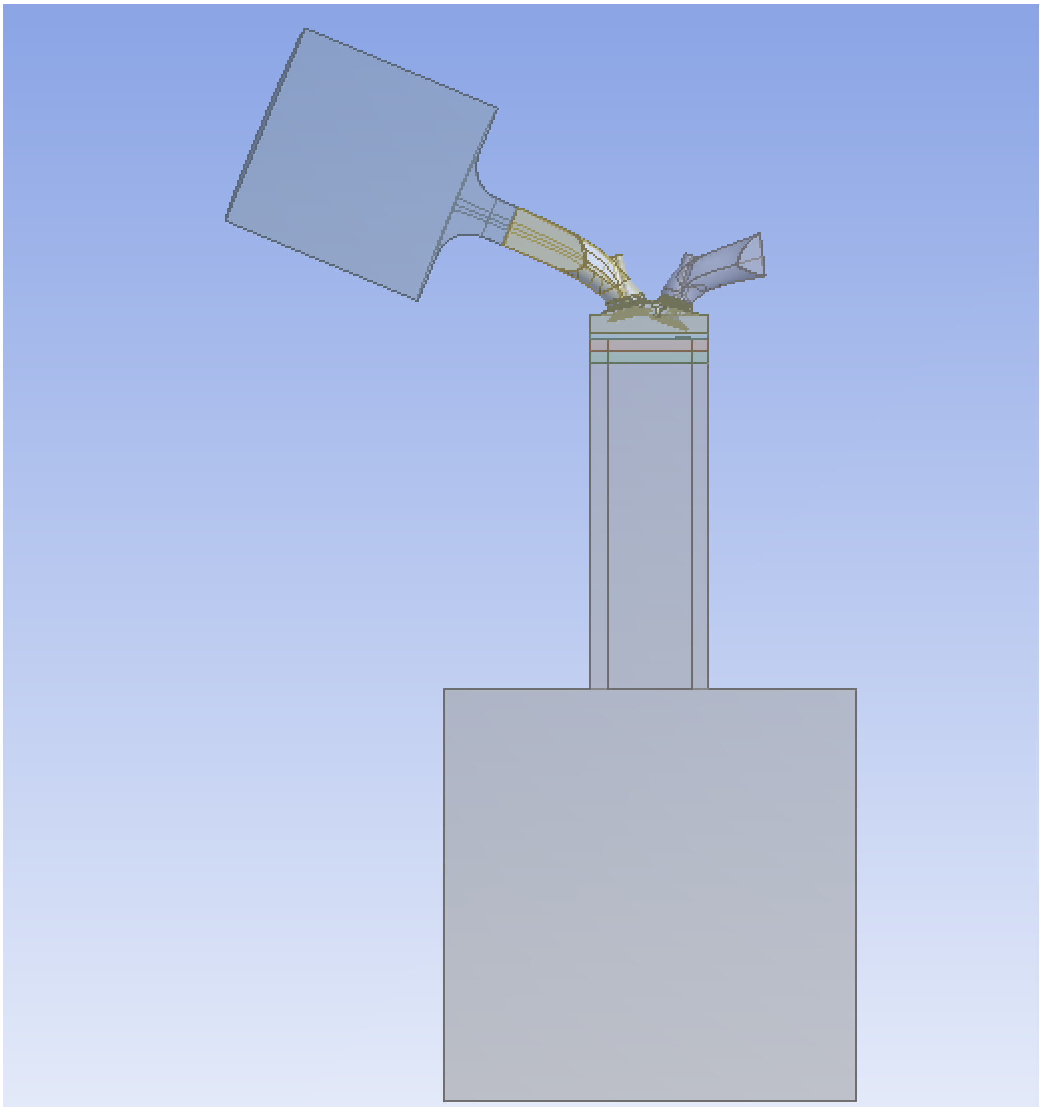
  The size of the output plenum will be as per the **Plenum Size** and the shape as per the selection of **Plenum Type**.

5. Click **Generate** (

located in the Ansys DesignModeler toolbar).

6. Click **Decompose** (

located in the **IC Engine** toolbar).



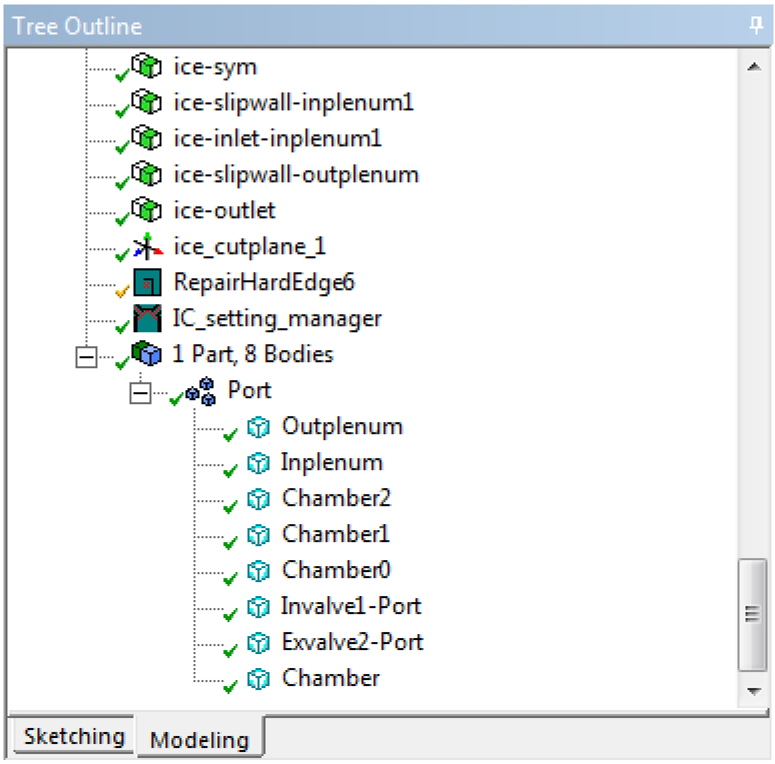
7. Close the Ansys DesignModeler.

Note:

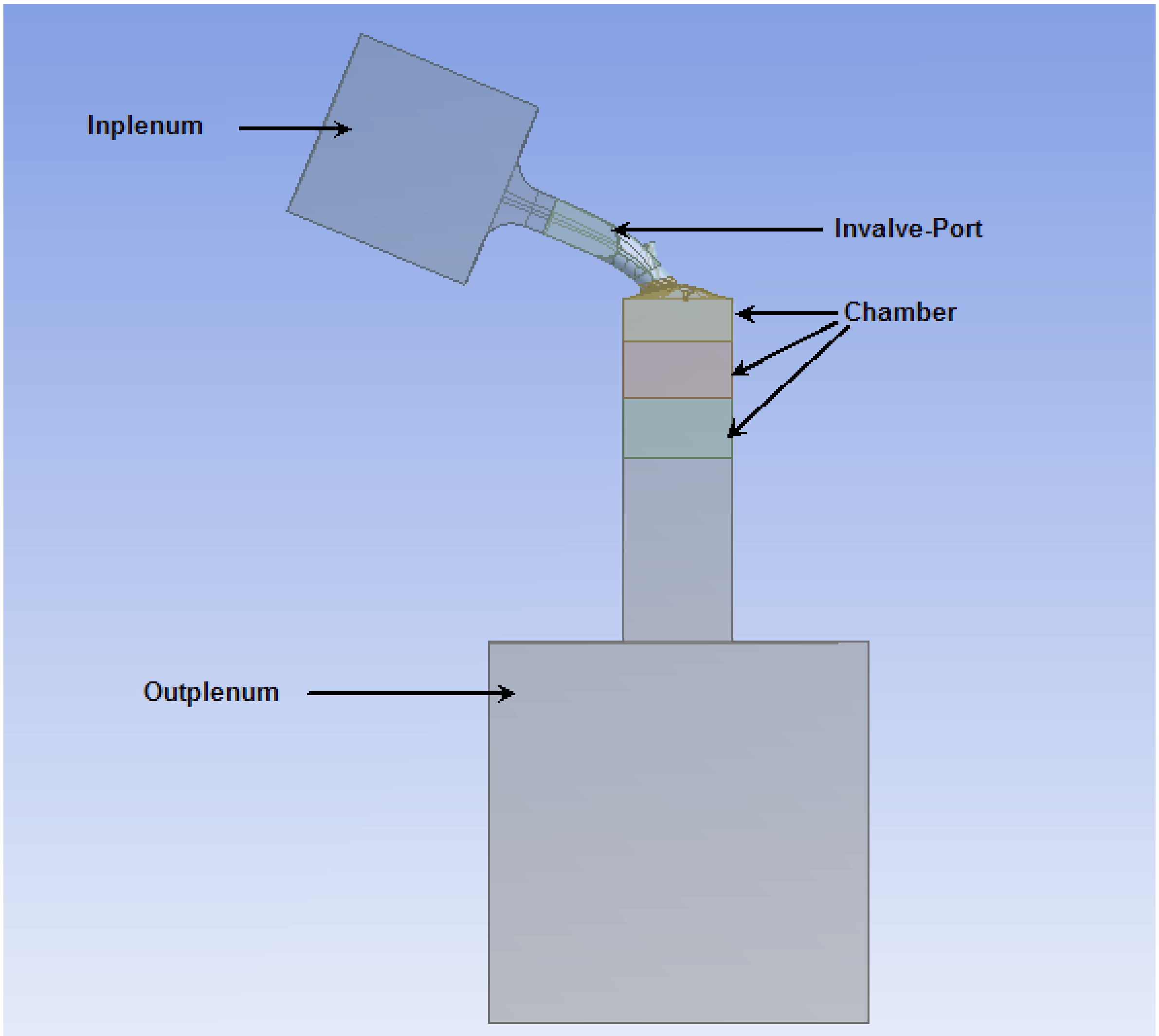
Even after decomposition you can change the parameters like **Valve Lift**, **Inlet Extension Length**, **Plenum Size**, **Plenum Blend Rad**, or **Cylinder Extension Length** in the **Input Manager**. After changing the parameters you will have to update the cell to regenerate the geometry with the changed parameters. For **Post Planes Dist. From Ref.** you can change the plane distances but the number of postprocessing planes added the first time should remain the same. If you change any other parameters you will have to decompose the geometry again.

7.2. Viewing the Bodies and Parts

After decomposition the engine is divided into one part, **Port** and several bodies.



- **Outplenum**: The plenum created at the outlet.
- **Inplenum**: The plenum created at the inlet.
- **Chamber**: The chamber is divided into parts depending upon the number of postprocessing planes.
- **Invalve-Port**: Depending upon the number of inlet valves in the geometry, **Invalve-Port** is created.
- **Exvalve-Port**: Depending upon the number of inlet valves in the geometry, **Exvalve-Port** is created.


Inplenum
Invalve-Port
Chamber
Outplenum

Chapter 8: Port Flow Simulation: Meshing in IC Engine

The decomposed geometry is used to generate the mesh. The goal of the IC Engine meshing tool is to minimize the effort required to generate a mesh for the IC Engine specific solver. It uses the named selection created in the decomposition to identify different zones and creates the required mesh controls. The information in this chapter is divided into the following sections:

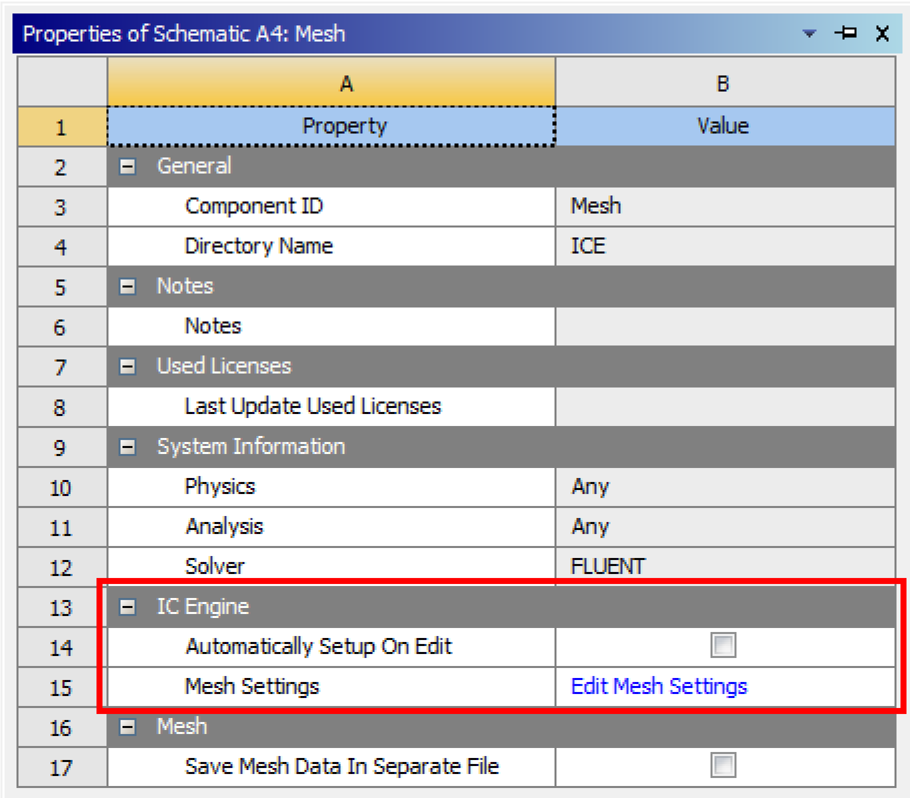
8.1. Meshing Procedure for Port Flow Simulation

There are two ways to generate the mesh.

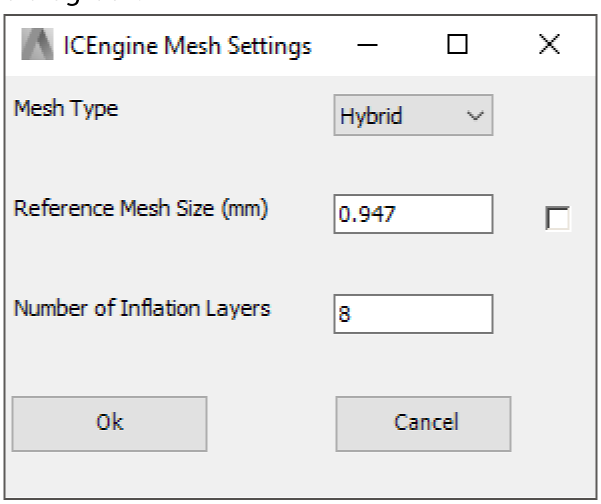
Meshing directly from the Workbench Window

After decomposition is done you can then directly generate the mesh from the Workbench window without opening the Ansys Meshing application. If you want to use the default mesh settings then right-click on the **Mesh** cell and select **Update** from the context menu. This will first create the mesh controls and then generate the mesh.

You can also change the mesh settings from the Workbench window.



In the **Properties** box which is displayed after selecting the **Mesh** cell you can click on **Edit Mesh Settings** next to the **Mesh Settings** property, under **IC Engine**. This will open a **ICEngine Mesh Settings** dialog box.



You can change the settings for the parameters:

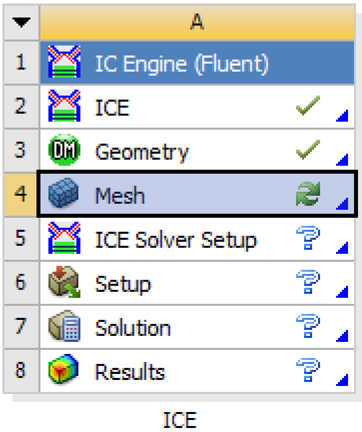
- **Mesh Type**
- **Reference Mesh Size (mm)** (This can be parametrized by enabling the check box next to it.)
- **Number of Inflation Layers**

For more information on these check the section on **IC Mesh Parameters** (p. 187).

Meshing using Ansys Meshing Application

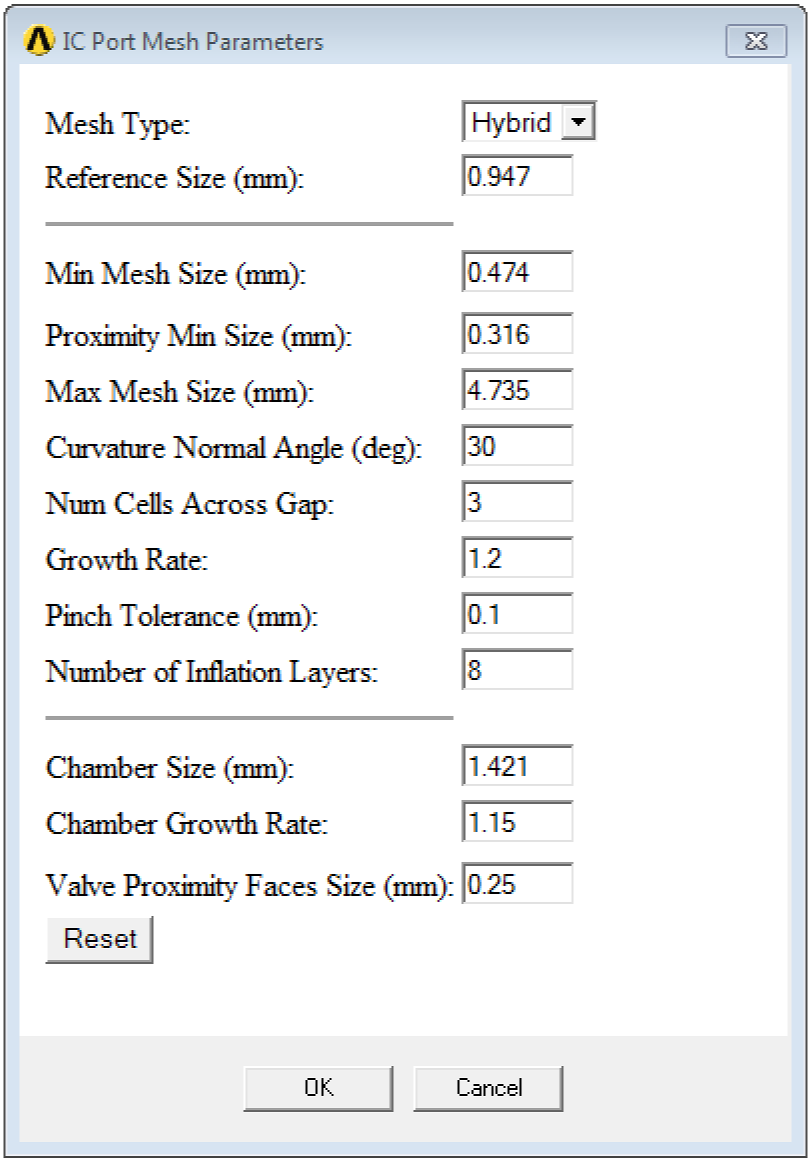
In this method of meshing you can check the mesh controls and settings in details.

1. Double-click the **Mesh** cell in the **IC Engine** analysis system to open the Ansys Meshing application.

2. Click **IC Setup Mesh** ( located in the **IC Engine** toolbar). This opens the **IC Mesh Parameters** dialog box.

   a. Here you can define different mesh settings for the different parts and virtual topologies.



In the **IC Port Mesh Parameters** dialog box you can see the default mesh settings. You can change the settings or use the default ones.

- **Mesh Type**: You can select **Hybrid** or **CutCell** from the drop-down list.
- **Reference Size**: This is a reference value. Some global mesh settings and local mesh setting values are dependent on this term.

  Reference Size = (Valve margin perimeter) /100

Global Mesh Settings

- **Min Mesh Size**: This value is set to **Reference Size**/2 for **Hybrid** mesh type and **Reference Size**/3 for **CutCell** mesh type.
- **Proximity Min Size**: This value is set to **Reference Size**/3.

- **Max Mesh Size**: This value is set to **Reference Size** × 5 for **Hybrid** mesh type. For **CutCell** mesh type it is set to (**Proximity Min Size** × 16) or (**Min Mesh Size** × 16) whichever is lesser.
- **Curvature Normal Angle**: This value changes depending upon the chosen **Mesh Type**. It is **30** for **Hybrid** mesh type and **18** for **CutCell** mesh type.
- **Num Cells Across Gap**: This value is set to **3** for **Hybrid** mesh type and **5** for **CutCell** mesh type.
- **Growth Rate**: This value is set to **1.2** for **Hybrid** mesh type and **1.05** for **CutCell** mesh type.
- **Pinch Tolerance**: This value is set to **0.1** for **Hybrid** mesh type. It is not valid for **CutCell** mesh type.
- **Number of Inflation Layers**: This value is set to **8**.

Local Mesh Settings

- **Chamber Size**: This value is set to **Reference Size** × 1.5.
- **Chamber Growth Rate**: This value is set to **1.15** for **Hybrid** mesh type and **1.05** for **CutCell** mesh type.

Note:

Chamber Growth Rate should be less than **Growth Rate**.

- **Valve Proximity Faces Size**: This value is set to (least value among valve lifts/8). The minimum limit is **Reference Size**/8 and maximum limit is **Reference Size**.

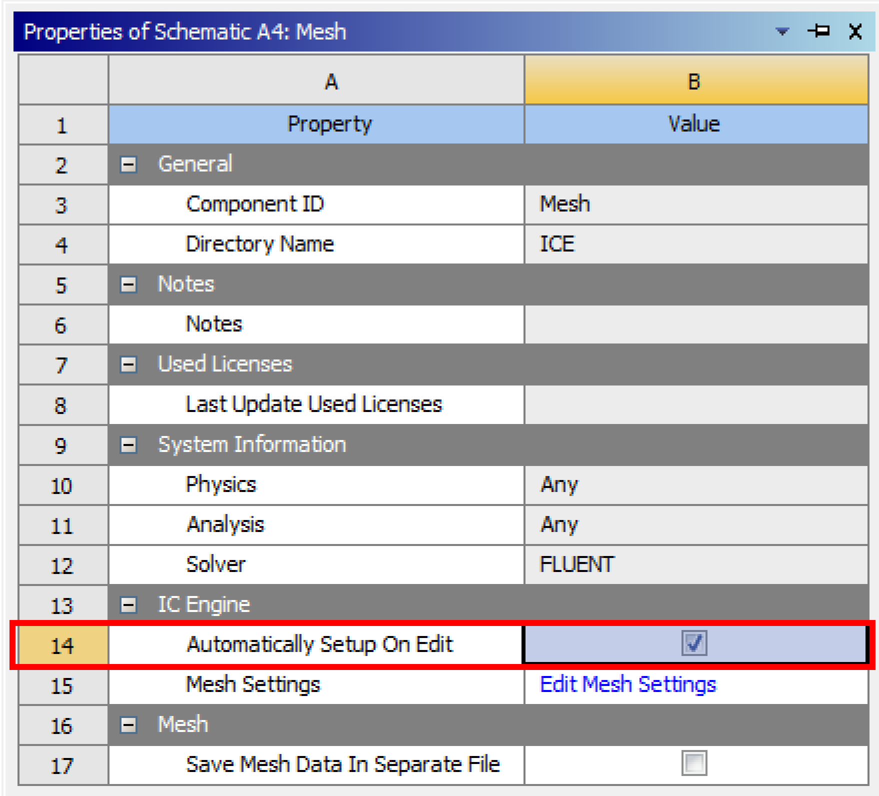
Note:

If **Reference Size** is changed, all the other parameters will change, depending on their relation with it.

b. If after changing the values of the parameters you would like to go back to the default values, click **Reset**.

c. Click **OK** to set and close the **IC Port Mesh Parameters** dialog box.

You can generate the mesh controls before opening Ansys Meshing application. To do this enable **Automatically Setup On Edit** under **IC Engine** in the **Properties** box which is displayed after selecting the **Mesh** cell in the Workbench window.



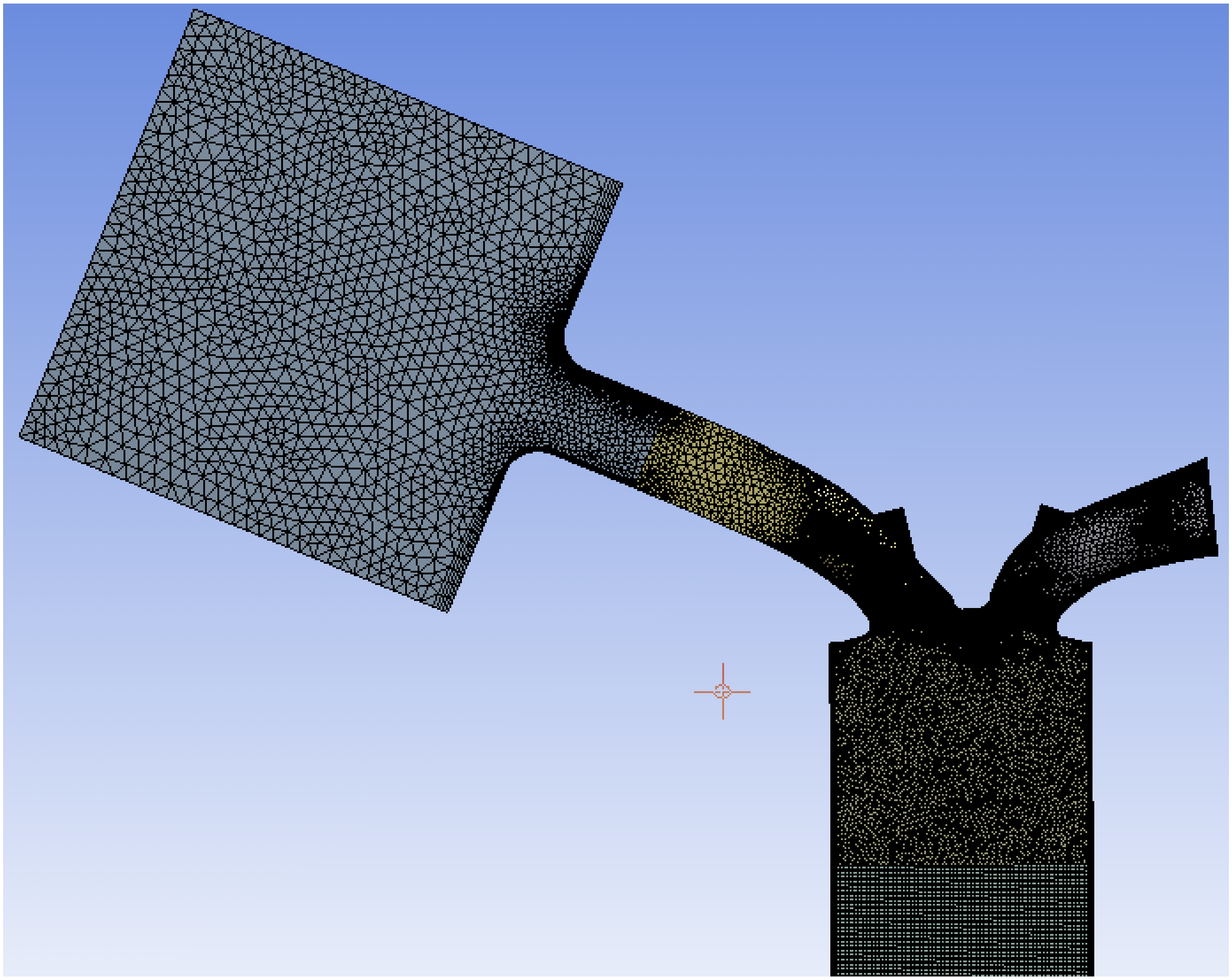
Properties of Schematic A4: Mesh

	A	B
1	Property	Value
2	⊟ General	
3	Component ID	Mesh
4	Directory Name	ICE
5	⊟ Notes	
6	Notes	
7	⊟ Used Licenses	
8	Last Update Used Licenses	
9	⊟ System Information	
10	Physics	Any
11	Analysis	Any
12	Solver	FLUENT
13	⊟ IC Engine	
14	Automatically Setup On Edit	☑
15	Mesh Settings	Edit Mesh Settings
16	⊟ Mesh	
17	Save Mesh Data In Separate File	☐

So after you double-click the **Mesh** cell to open Ansys Meshing application the mesh controls are already set.

3. Click **IC Generate Mesh** (

located in the **IC Engine** toolbar) to generate the mesh.



When you complete the setup through **IC Setup Mesh**, it does the following:

- It makes the required changes in **Global Mesh Settings**. (For details refer to, Global Mesh Settings for Port Flow Simulation (p. 291)).
- It creates **Local Mesh Settings**. (For details refer to Local Mesh Settings for Port Flow Simulation (p. 297)).
- It creates pinch controls (for hybrid mesh).
- It defines the “Worksheet” for controlling the order of meshing bodies (for hybrid mesh).

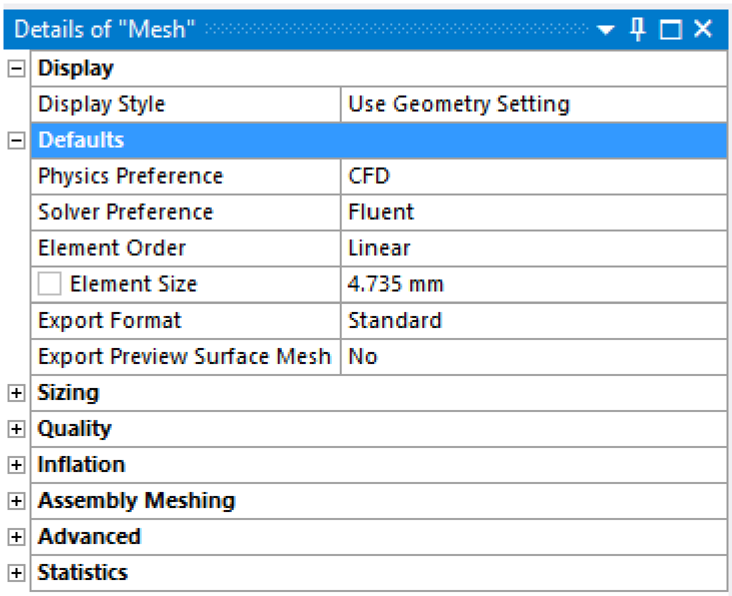
8.2. Global Mesh Settings for Port Flow Simulation

IC Engine meshing tool creates **Global Mesh Settings** based on the information provided in the **IC Port Mesh Parameters** (p. 287) dialog box. Following sections describe these settings:

For more information on **Global Mesh Settings**, see Global Mesh Controls in the Meshing User's Guide.

8.2.1. Defaults Group

Under **Details of Mesh**, the following are the global mesh settings defined under **Defaults**.



- **Physics Preference**: This is set to **CFD**.

  This option allows you to establish how Workbench will perform meshing based on the physics of the analysis type that you specify.

- **Solver Preference**: Ansys Fluent is set as the solver for the mesh.

  Since **CFD** is chosen as your **Physics Preference**, it causes a **Solver Preference** option to appear in the **Details View** of the **Mesh** folder. The chosen value sets certain defaults that will result in a mesh that is more favorable to the respective solver.

- **Element Order**: This is set to **Linear**. This option allows you to control whether meshes are to be created with midside nodes (quadratic elements) or without midside nodes (linear elements). Reducing the number of midside nodes reduces the number of degrees of freedom. Choices include **Program Controlled**, **Linear**, and **Quadratic**.

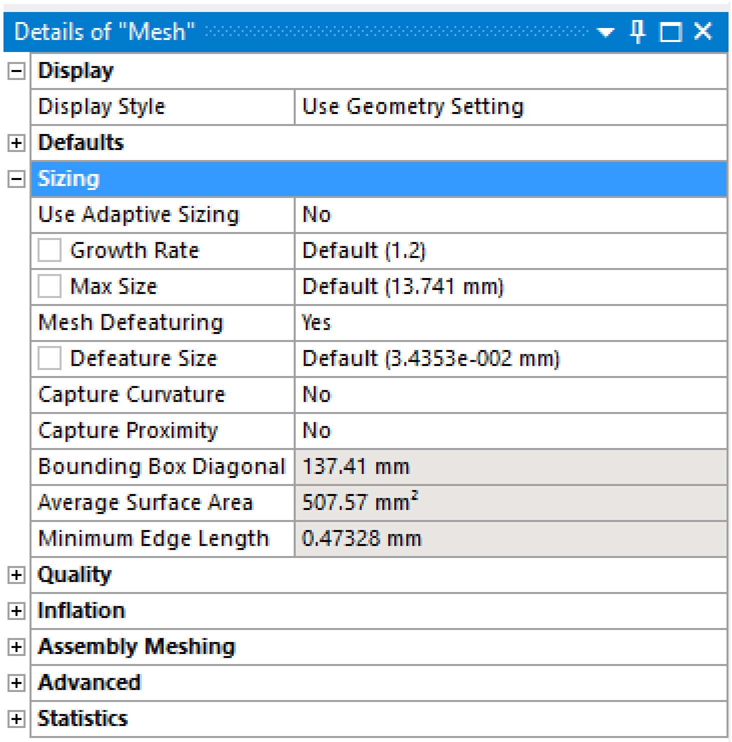
- **Element Size**: This allows you to specify the element size used for the entire model. This size will be used for all edge, face, and body meshing.

- **Export Format**: This option defines the format for the mesh when exported to Ansys Fluent. The default is **Standard**. You can change this to **Large Model Support** to export the mesh as a cell-based Fluent mesh.

- **Export Preview Surface Mesh** : This option controls the export of the preview surface mesh elements. This option can be used when the bodies have been meshed only partially, that is, not all volumes have been filled with elements and only previewing of surface meshes was done. The default is **No**, which results in export of only volume mesh elements to the Fluent mesh file. You can change this to Yes to export both the volume mesh and the preview surface meshes to the Fluent mesh file.

8.2.2. Sizing Group

Under the **Details of Mesh**, the following are the global mesh settings defined under **Sizing**.



- **Use Adaptive Sizing**: This option refers to a 2D curvature and proximity-based refinement approach which refines edges based on curvature and/or proximity but does not propagate the refined mesh along the face. When set to **Yes**, the mesher uses the value of the element size property to determine a starting point for the mesh size. The value of the element size property can be set by the user or automatically computed using defaults. When meshing begins, edges are meshed with this size initially, and then they are refined for curvature and 2D proximity. Next, mesh based defeaturing and pinch control execution occurs. The final edge mesh is then passed into a least-squares fit size function, which guides face and volume meshing.

- **Growth Rate**: It is equal to the value of **Growth Rate**, which is set in the **IC Port Mesh Parameters** (p. 287) dialog box.

- **Max Size**: It is equal to the value of **Max Mesh Size**, which is set in the **IC Port Mesh Parameters** (p. 287) dialog box.

- **Mesh Defeaturing**: This option automatically defeatures small features and dirty geometry according to the **Defeature Size** you specify here.

- **Transition**: When **Use Adaptive Sizing** is set to **Yes**, this option affects the rate at which adjacent elements will grow. **Slow** produces smooth transitions while **Fast** produces more abrupt transitions.
- **Span Angle Center**: It is set to **Fine**.

  When **Use Adaptive Sizing** is set to **Yes**, this option sets the goal for curvature based refinement. The mesh will subdivide in curved regions until the individual elements span this angle. The following choices are available:

  - **Coarse** — 91° to 60°
  - **Medium** — 75° to 24°
  - **Fine** — 36° to 12°
- **Capture Curvature**: This option allows you to take into account curvature effects.
- **Curvature Min Size**: When **Capture Curvature** is set to **Yes**, this option is equal to the value of **Min Mesh Size**, which is set in the **IC Port Mesh Parameters** (p. 287) dialog box.
- **Curvature Normal Angle**: When **Capture Curvature** is set to **Yes**, this option is set to the value of the **Normal Angle** in the case of **Cold Flow Simulation**. This value is set in the **IC Port Mesh Parameters** (p. 287) dialog box.
- **Capture Proximity**: This option allows you to account for proximity effects.
- **Proximity Min Size**: When **Capture Proximity** is set to **Yes**, this option is equal to the value of **Proximity Min Size**, which is set in the **IC Port Mesh Parameters** (p. 287) dialog box.
- **Num Cells Across Gap**: It is set to the value of **Num Cells Across Gap** in the **IC Port Mesh Parameters** (p. 287) dialog box. When **Capture Proximity** is set to **Yes**, this option is the minimum number of layers of elements to be generated in the gaps.
- **Proximity Size Function Sources**: When **Capture Proximity** is set to **Yes**, this parameter is set to **Faces and Edges**.

8.2.3. Quality Group

Quality is useful for configuring mesh quality.

Details of "Mesh"	
Display	
Display Style	Use Geometry Setting
Defaults	
Sizing	
Quality	
Check Mesh Quality	Yes, Errors
Target Skewness	Default (0.900000)
Smoothing	High
Mesh Metric	None
Inflation	
Assembly Meshing	
Advanced	
Statistics	

- **Check Mesh Quality**: This option determines how the software behaves with respect to error and warning limits

- **Target Skewness**: This option allows you to set a target skewness that you would like the mesh to satisfy.

- **Smoothing**: This option attempts to improve element quality by moving locations of nodes with respect to surrounding nodes and elements. The **Low**, **Medium**, or **High** option controls the number of smoothing iterations along with the threshold metric where the mesher will start smoothing.

- **Mesh Metric**: This option allows you to view mesh metric information and thereby evaluate the mesh quality.

  When you select a mesh metric, its **Min**, **Max**, **Average**, and **Standard Deviation** values are reported in the Details View, and a bar graph is displayed under the Geometry window. The graph is labeled with color-coded bars for each element shape represented in the model's mesh, and can be manipulated to view specific mesh statistics of interest.

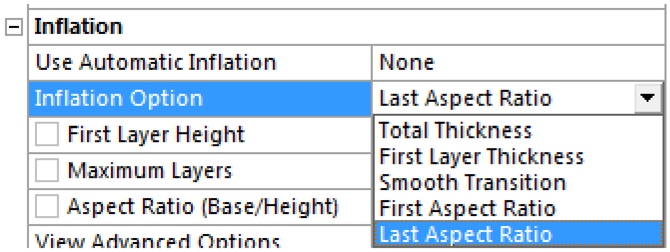
8.2.4. Inflation Group

Inflation is useful for CFD boundary layer resolution.

Inflation	
Use Automatic Inflation	None
Inflation Option	Last Aspect Ratio
First Layer Height	1.4e-002 mm
Maximum Layers	8
Aspect Ratio (Base/Height)	3.
View Advanced Options	No

- **Use Automatic Inflation**: It is set to **None** for **Port Flow Simulation**.

- **Inflation Option**: It is set to **Last Aspect Ratio** for **Port Flow Simulation**. This setting determines the heights of the inflation layers.



– The **Last Aspect Ratio** option creates inflation layers using the values of the **First Layer Height**, **Maximum Layers**, and **Aspect Ratio (Base/Height)** controls to generate the inflation mesh.

→ **First Layer Height**: It is equal to (**Reference Size**/70), for **Hybrid Mesh Type** and (**Reference Size**/20), for **CutCell Mesh Type**.

This control determines the height of the first inflation layer. This first inflation layer consists of a single layer of prism elements that is formed against the faces of the inflation boundary. You must enter a value for this control, and it must be greater than 0.

→ **Maximum Layers**: It is equal to the value of **Number of Inflation Layers**, which is set in the **IC Mesh Parameters** (p. 187) dialog box.

This control determines the maximum number of inflation layers to be created in the mesh. Valid values are from 1 to 1000.

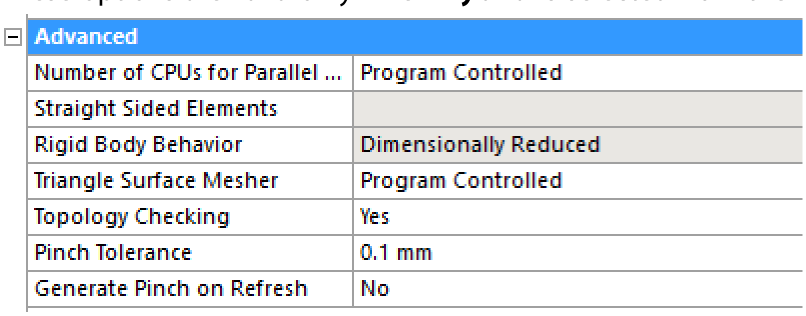
→ **Aspect Ratio (Base/Height)**: This value is set to 3.

This is defined as the ratio of the local inflation base size to the inflation layer height. The value should be between `0.5` and `20`.

- **View Advanced Options**: This control determines whether advanced inflation options appear in the **Details View**. Choices are **No** and **Yes**. It is set to **No** for **Port Flow Simulation**.

8.2.5. Advanced Group

These options are valid only when **Hybrid** is selected from the **Mesh Type** drop-down list.



The **Advanced** group allows you to use the following features and controls:

- **Number of CPUs for Parallel Part Meshing**: This option sets the number of processors to be used for parallel part meshing. Using the default for specifying multiple processors will enhance meshing performance on geometries with multiple parts. For parallel part meshing, the default is set to

Program Controlled or 0. This instructs the mesher to use all available CPU cores. The default setting inherently limits 2 GB memory per CPU core. An explicit value can be specified between 0 and 256, where 0 is the default.

- **Straight Sided Elements**: This option specifies meshing to straight edge elements when set to **Yes**. This option may affect the placement of midside nodes if the **Element Order** option is set to **Quadratic**.
- **Rigid Body Behavior**: This option determines whether a full mesh is generated for a rigid body, rather than a surface contact mesh. **Rigid Body Behavior** is applicable to all body types. Valid values for **Rigid Body Behavior** are **Dimensionally Reduced** (generate surface contact mesh only) and **Full Mesh** (generate full mesh).
- **Triangle Surface Mesher**: This option determines which triangle surface meshing strategy will be used by patch conforming meshers. In general, the advancing front algorithm provides a smoother size variation and better results for skewness and orthogonal quality. This option is inaccessible when an assembly meshing algorithm is selected.
- **Topology Checking**: This option controls what happens when a user scopes an object (such as loads, boundary conditions, named selections and so on) to geometry (bodies, faces, edges, and vertices) after the mesh has been generated. If **Topology Checking** is set to **Yes** (default), the software will check to see if the scoped geometry has mesh properly associated to it. If the associations are incorrect, the scoping of the object will force the mesh to be out of date. The mesh would need to be re-generated to get proper associations. If the associations are correct, the scoping is performed without any change to the mesh and the mesh stays up to date. Set **Topology Checking** to **No** to avoid the checks and always keep the mesh up to date.
- **Pinch Tolerance**: This is equal to the value of **Pinch Tolerance**, which is set in the **IC Port Mesh Parameters** (p. 287) dialog box.

  This control allows you to specify a tolerance for the Meshing application to use when it generates automatic pinch controls.
- **Generate Pinch on Refresh**: This option determines whether pinch controls will be regenerated following a change made to the geometry (such as a change made via a DesignModeler application operation such as a merge, connect, etc.). If **Generate Pinch on Refresh** is set to **Yes** and you change the geometry, all pinch controls that were created automatically will be deleted and recreated based on the new geometry. If **Generate Pinch on Refresh** is set to **No** and you update the geometry, all pinch controls related to the changed part will appear in the Tree Outline but will be flagged as undefined.

8.3. Local Mesh Settings for Port Flow Simulation

Based on information provided in the **IC Port Mesh Parameters** dialog box, IC Engine meshing tool creates some mesh settings at local level. Following section describes these settings for port flow simulation:

Face Sizing on Valve Proximity Faces

When you click **Face Sizing (valve-proximity-faces)** under **Mesh** in the **Outline**, you can see the details.

- **Type**: It is set to **Element Size**.
- **Element Size**: It is equal to **Valve Proximity Faces Size**, which is set in the **IC Port Mesh Parameters** (p. 287) dialog box.
- **Behavior**: For this body part it is set to **Hard**.

> **Note:**
>
> This setting will be present in the tree only for hybrid meshing.

Face Sizing on Cylinder Faces of Outlet Plenum

When you click **Face Sizing (Outplenum-cyl)** under **Mesh** in the **Outline**, you can see the details.

- **Type**: It is set to **Element Size**.
- **Element Size**: It is equal to **Reference Size** × 1.5, which is set in the **IC Port Mesh Parameters** (p. 287) dialog box.
- **Behavior**: For this body part it is set to **Soft**.

Body Sizing for Chamber

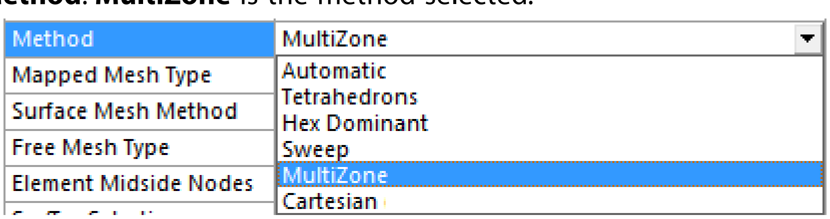
When you click **Body Sizing (Chamber)** under **Mesh** in the **Outline**, you can see the details.

- **Geometry**: This shows the body selected.
- **Type**: It is set to **Element Size**.
- **Element Size**: It is equal to **Chamber Size** which is set in the **IC Port Mesh Parameters** (p. 287) dialog box.
- **Behavior**: For this body part is it set to **Soft**.
- **Growth Rate**: It is equal to the **Growth Rate** which is set in the **IC Port Mesh Parameters** (p. 287) dialog box.

MultiZone

When you click **MultiZone(Outplenum)** under **Mesh** in the **Outline**, you can see the details.

- **Geometry**: This shows the body selected.
- **Method**: **MultiZone** is the method selected.



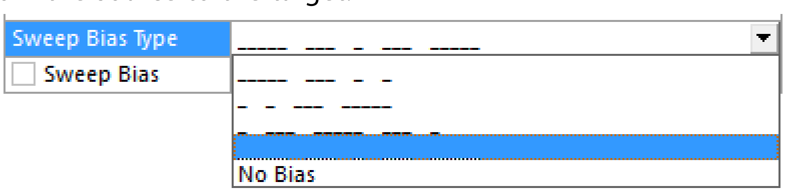
- **Mapped Mesh Type**: **Hexa/Prism** is selected from the drop-down list.
- **Src/Trg Selection**: **Manual Source** is selected from the drop-down list.
- **Source**: This shows the faces selected. The faces, **ice-slipwall-outletplenum** and **int-ch-*** are selected.

> **Note:**
>
> This setting will be present in the tree only for hybrid meshing.

Sweep Method for Chamber

Click **Sweep Method (Chamber)** under **Mesh** in the **Outline**, to see the details.

- **Geometry**: The chamber body part is used as input.
- **Source**: It is **Program Controlled**.
- **Target**: It is **Program Controlled**. This means that the program determines the target for the body.
- **Free Face Mesh Type**: It is set to **All Quad**. This determines the shape of the elements used to fill the swept body.
- **Sweep Num Divs**: It is equal to — (The length of the chamber body/**Reference Size**).
- **Sweep Bias Type**: It adjusts the spacing ratio of nodes on the edge. Biasing direction is based from the source to the target.

  One of the four patterns available from the **Bias Type** drop-down list is chosen.

- **Sweep Bias**: This value is set to 1. This will override the mesh gradation in the swept region.

> **Note:**
>
> This setting will be present in the tree only for hybrid meshing.

Inflation For All Faces

Click **Inflation (all-faces)** under **Mesh** in the **Outline**, to see the details.

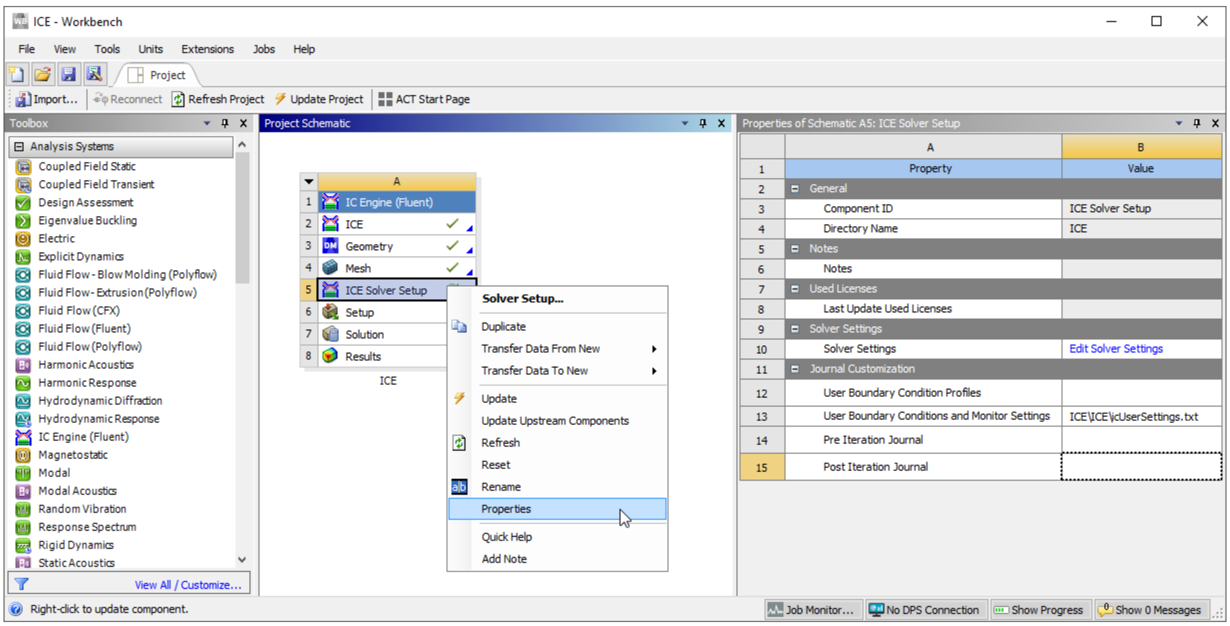
- **Boundary**: For this all the faces of the geometry except the inlet faces, symmetry faces, and the outlet faces are selected.

- **Inflation Option**: It is set to **Smooth Transition**. This setting determines the heights of the inflation layers.
  - The **Smooth Transition** option uses the local tetrahedral element size to compute each local initial height and total height so that the rate of volume change is smooth. Each triangle that is being inflated will have an initial height that is computed with respect to its area, averaged at the nodes. This means that, for a uniform mesh, the initial heights will be roughly the same, while for a varying mesh, the initial heights will vary.
    - → **Transition Ratio**: It is set to `0.272`. This control determines the rate at which adjacent elements grow. This is the default value when **Physics Preference** is set to **CFD** and when **Solver Preference** is set to **Fluent**.
    - → **Number of Layers**: It is equal to the **Number of Inflation Layers** as specified in the **IC Port Mesh Parameters** (p. 287) dialog box. This control determines the maximum number of inflation layers to be created in the mesh. The default is **5** for solid bodies.
    - → **Growth Rate**: It is set to `1.1`. This control determines the relative thickness of adjacent inflation layers. As you move away from the face to which the inflation control is applied, each successive layer is approximately one growth rate factor thicker than the previous one.
    - → **Inflation Algorithm**: It is set to **Pre** for hybrid meshing. For the **CutCell** algorithm, inflation is neither **Pre** nor **Post**. Rather, it may be considered a hybrid of the two, in that the technology used is like that of the **Pre** algorithm, but inflation occurs **Post** mesh generation.

Chapter 9: Port Flow Simulation: Setting Up the Analysis in IC Engine

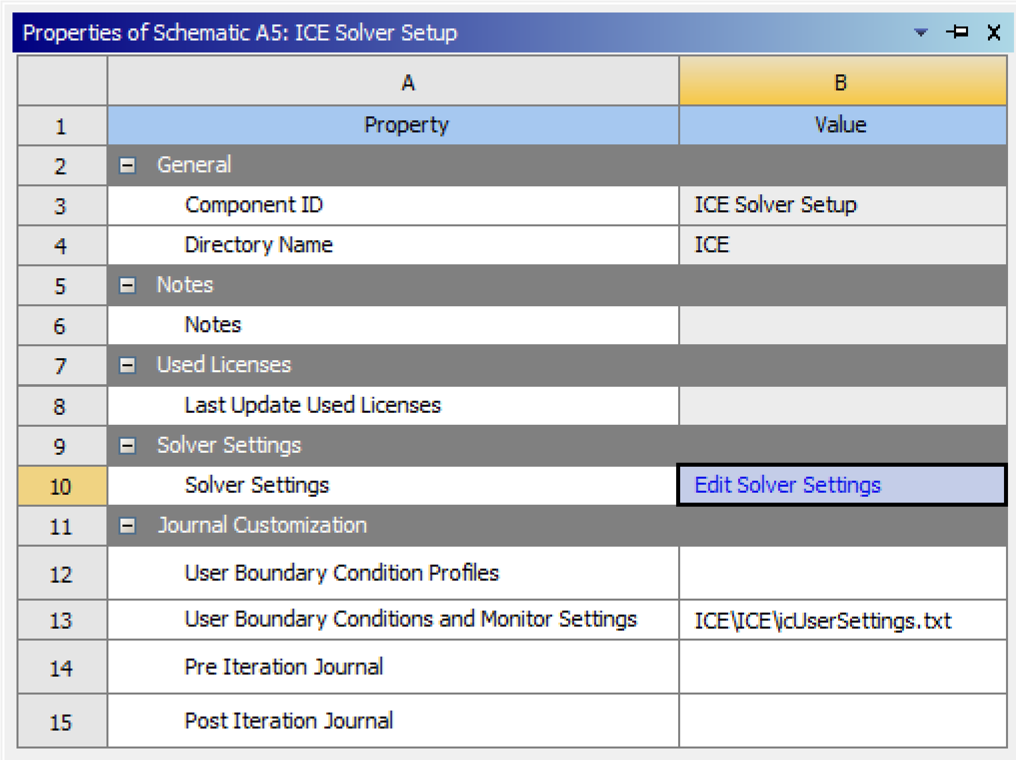
This chapter describes how to configure the analysis for a port flow simulation. All the settings described in this chapter are done automatically, but you can change them by accessing the **Solver Settings** dialog box or the task pages and the dialog boxes in Ansys Fluent.

After meshing you can set up the solver. In the **Properties** pane of the **ICE Solver Setup** cell you can make changes to the solver settings.



9.1. ICE Solver Settings

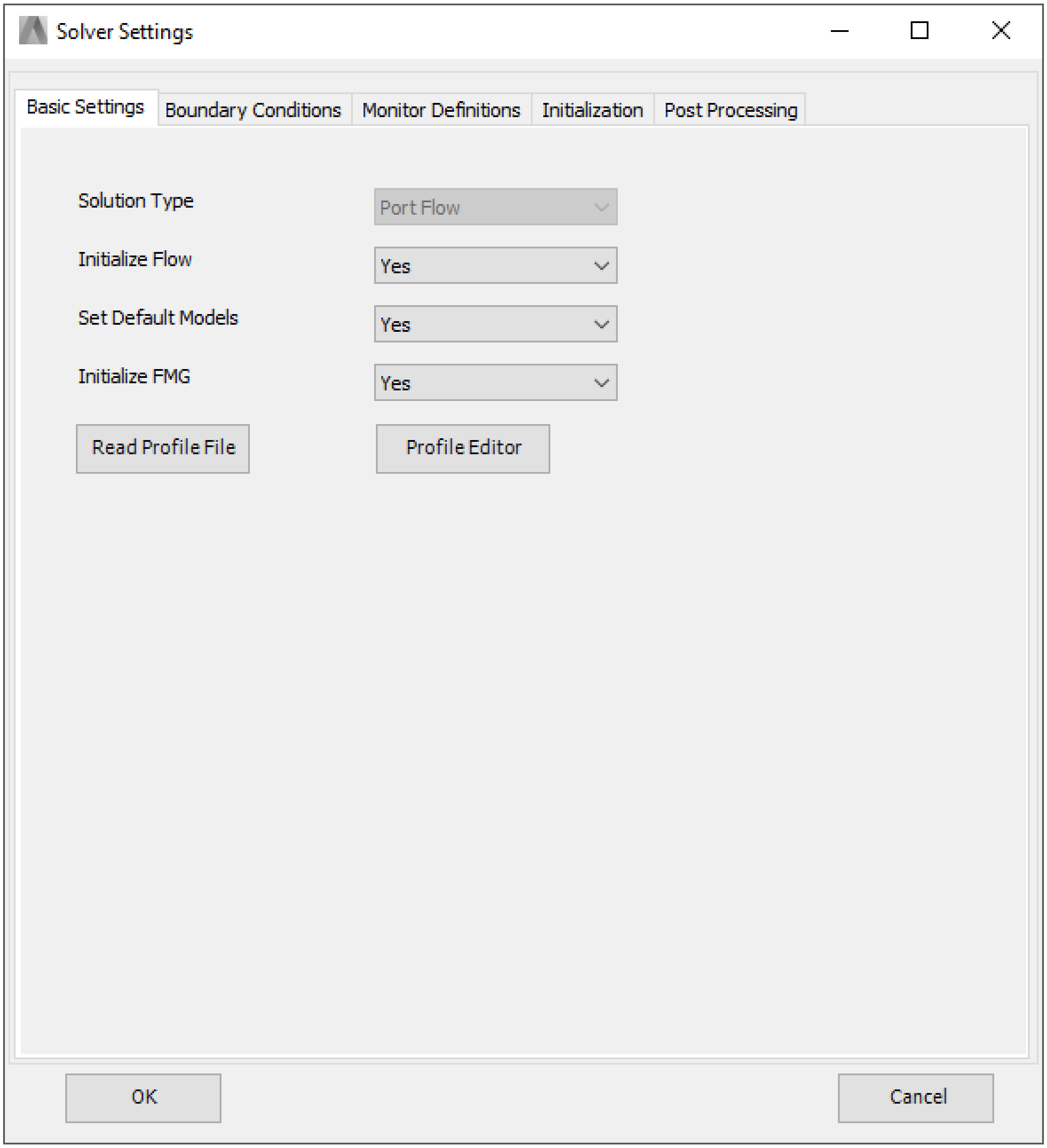
Click **Edit Solver Settings** to open the **Solver Settings** dialog box.



Properties of Schematic A5: ICE Solver Setup

	A	B
1	Property	Value
2	⊟ General	
3	Component ID	ICE Solver Setup
4	Directory Name	ICE
5	⊟ Notes	
6	Notes	
7	⊟ Used Licenses	
8	Last Update Used Licenses	
9	⊟ Solver Settings	
10	Solver Settings	Edit Solver Settings
11	⊟ Journal Customization	
12	User Boundary Condition Profiles	
13	User Boundary Conditions and Monitor Settings	ICE\ICE\icUserSettings.txt
14	Pre Iteration Journal	
15	Post Iteration Journal	

9.1.1. Basic Solver Settings



In the **Basic Settings** tab you have the following settings:

Solution Type

shows the **Solution Type** as **Port Flow** which you have selected in the **Properties** pane of the **ICE** cell. You cannot make any changes here.

Initialize Flow

is set to **Yes** by default. This will initialize the flow. To check the initialization settings see the **Solution Initialization** task page in Ansys Fluent.

Set Default Models

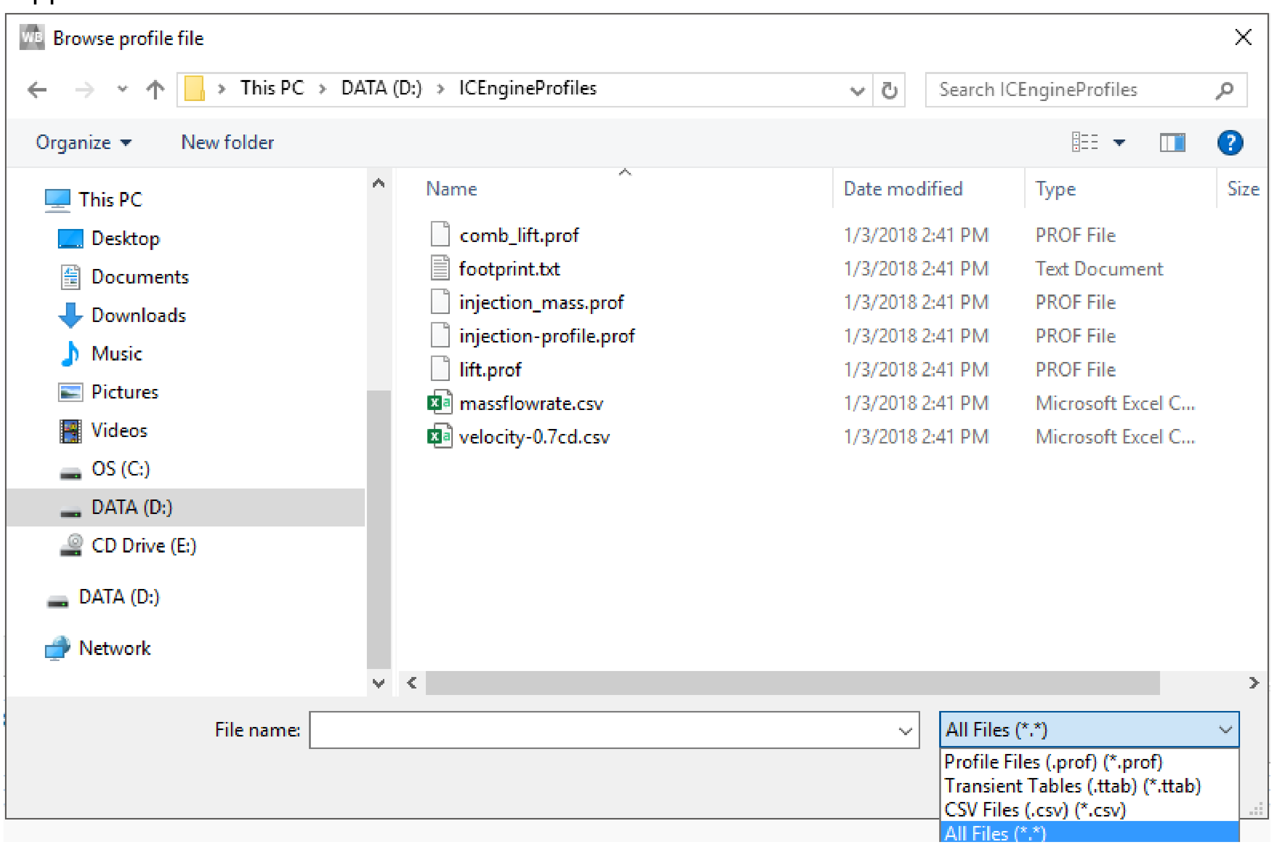
is set to **Yes** by default. For port flow simulation some models of Ansys Fluent have been chosen as default for better results. These are the **Energy** model, and the **Standard k-omega** model from the list of **Viscous** models. You can check them at the **Models** (p. 324) task page of Ansys Fluent.

Initialize FMG

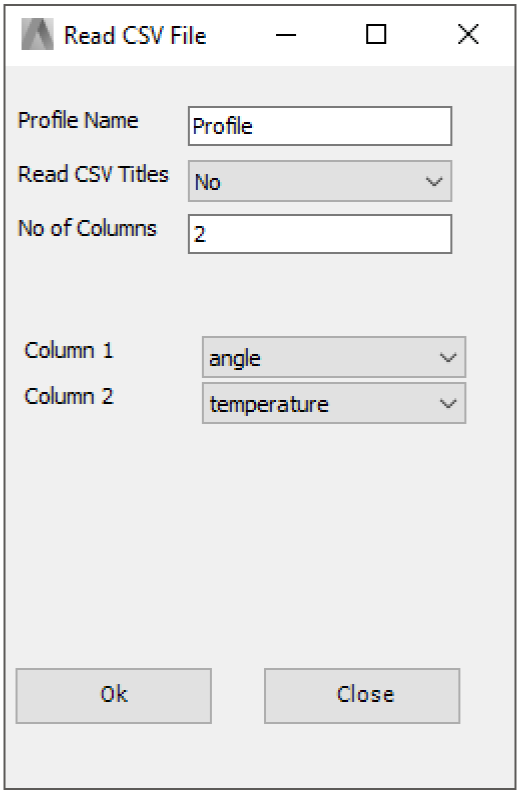
is set to **Yes** by default. This will use FMG initialization. For more information see Full Multigrid (FMG) Initialization in the Fluent Theory Guide.

Read Profile File

allows you to read multiple profile files. Clicking on **Read Profile File** opens **Browse profile file** dialog box. In the browsing window three types of file extensions will be supported `.prof` (profile file), `.ttab` (transient table), and `.csv`. **All Files** option is also present if you have a file extension which does not match with either of the given extensions. In this case the file type will be automatically identified, and an error will be thrown if the format is not supported.



If you select a `.csv` file and click **Open**, then a **Read CSV File** dialog box opens.



- You need to enter a name for **Profile Name**. This name will appear in the drop-down list of **Profiles** in the **Profile Editor** dialog box.
- If you retain the default setting of **Yes** for **Read CSV Titles** then the quantity or variable names will be as per the names in the CSV file.
- If you choose **No** for **Read CSV Titles** then you have to specify the **No. of Columns** of the CSV file you want to read. For each column you have to select a different variable name from the drop-down list. In this case the titles of the CSV columns will not be read. Your selections for the columns will be the titles.

Important:

- All the columns in CSV should have same number of values. Variable number of values and interpolation is not supported in the current version.
- Ensure that there are no empty spaces in the titles.

The format of the standard profile file is

```
((profile-name transient n periodic?)
(field_name-1 a1 a2 a3 .... an)
(field_name-2 b1 b2 b3 .... bn)
.
.
.
.
(field_name-r r1 r2 r3 .... rn))
```

The profile name as well as the field names have to be shorter than 64 characters. One of the `field_name` should be used for the `time` field, and the `time` field section *must* be in ascending order. n is the number of entries per field. The `periodic?` entry indicates whether or not the profile is time-periodic. Set it to `1` for a time-periodic profile, or `0` if the profile is not time-periodic.

An example is shown below:

```
((sampleprofile transient 8 0 0)
(angle
4.400000e+02 4.412000e+02
4.644900e+02 4.656900e+02
6.400000e+02
6.412000e+02 6.480800e+02 6.492800e+02)
(mass-flow
0.000000e+00 2.450040e-03 2.450040e-03 0.000000e+00 0.000000e+00
2.475870e-03 2.475870e-03 0.000000e+00)
(velocity
0.000000e+00 1.941520e+02 1.941520e+02 0.000000e+00 0.000000e+00
1.922430e+02 1.922430e+02 0.000000e+00)
)
```

Important:

All quantities, including coordinate values, must be specified in SI units because Ansys Fluent does not perform unit conversion when reading profile files. Also, profile names must have all lowercase letters (for example, `name`). Uppercase letters in profile names are not acceptable.

The format of the transient table file is

```
profile-name n_field n_data periodic?
field-name-1 field-name-2 field-name-3 .... field-name-n_field
v-1-1  v-2-1... ... ... ... v-n_field-1
v-1-2  v-2-2... ... ... ... v-n_field-2
.
.
.
.
.
v-1-n_data v-2-n_data ... ... ... ... v-n_field-n_data
```

The first field name (for example `field-name-1`) should be used for the `time` field, and the `time` field section, which represents the flow time, *must* be in ascending order. The `periodic?` entry indicates whether or not the profile is time-periodic. Set it to `1` for a time-periodic profile, or `0` if the profile is not time-periodic.

An example is shown below:

```
sampletprofile 3 8 0
angle        mass-flow    velocity
440          0.0          0.0
441.2        0.00245004   194.152
464.49       0.00245004   194.152
465.69       0.0          0.0
640          0.0          0.0
641.2        0.00247587   192.243
648.08       0.00247587   192.243
649.28       0.0           0.0
```

This file defines the same transient profile as the standard profile example above.

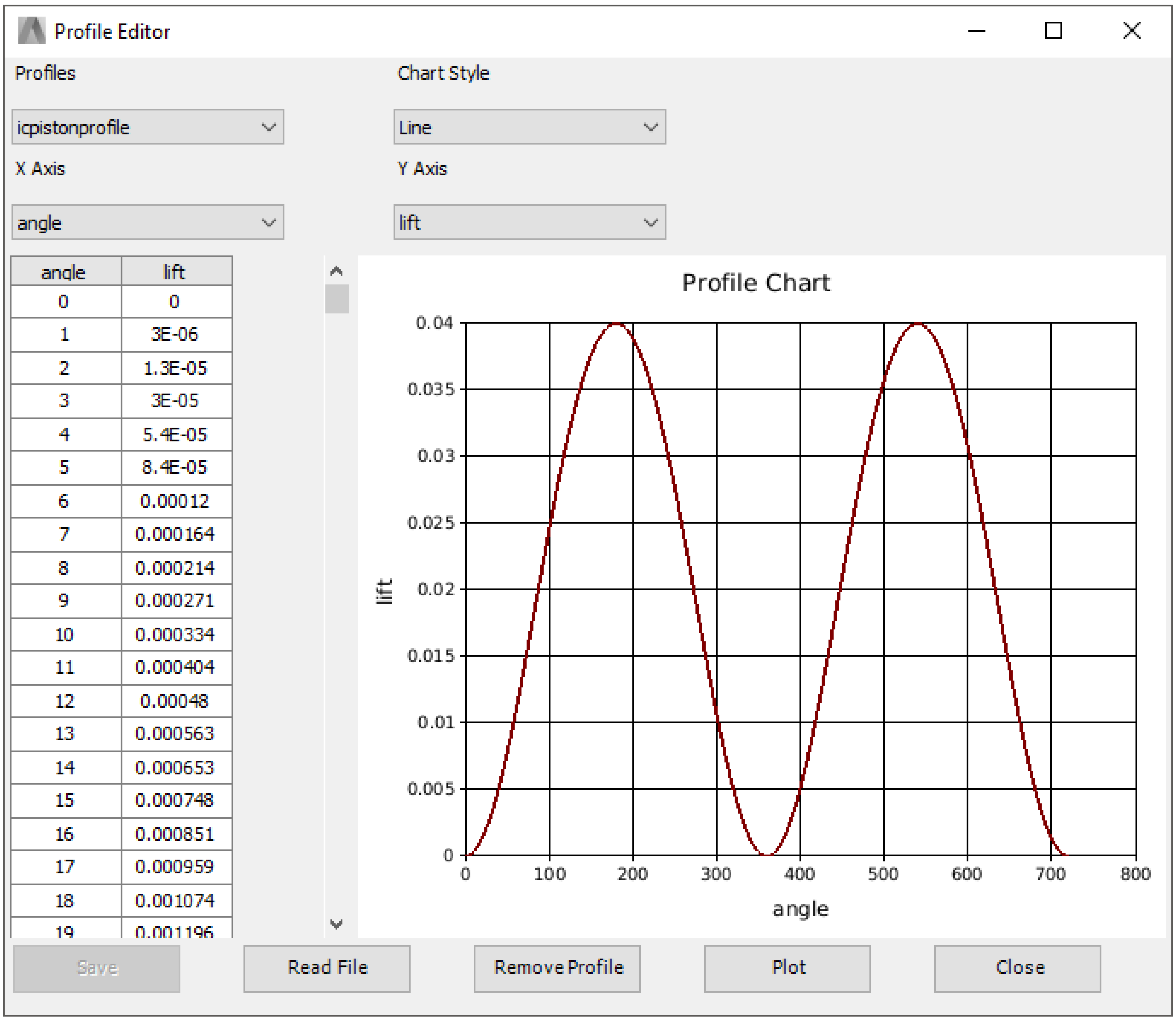
Important:

All quantities, including coordinate values, must be specified in SI units because Ansys Fluent does not perform unit conversion when reading profile files. Also, profile names must have all lowercase letters (for example, `name`). Uppercase letters in profile names are not acceptable. When choosing the field names, spaces or parentheses should not be included.

After reading the files the profiles will be available in the boundary condition drop-down lists.

Profile Editor

allows you to view the plot of the profiles.

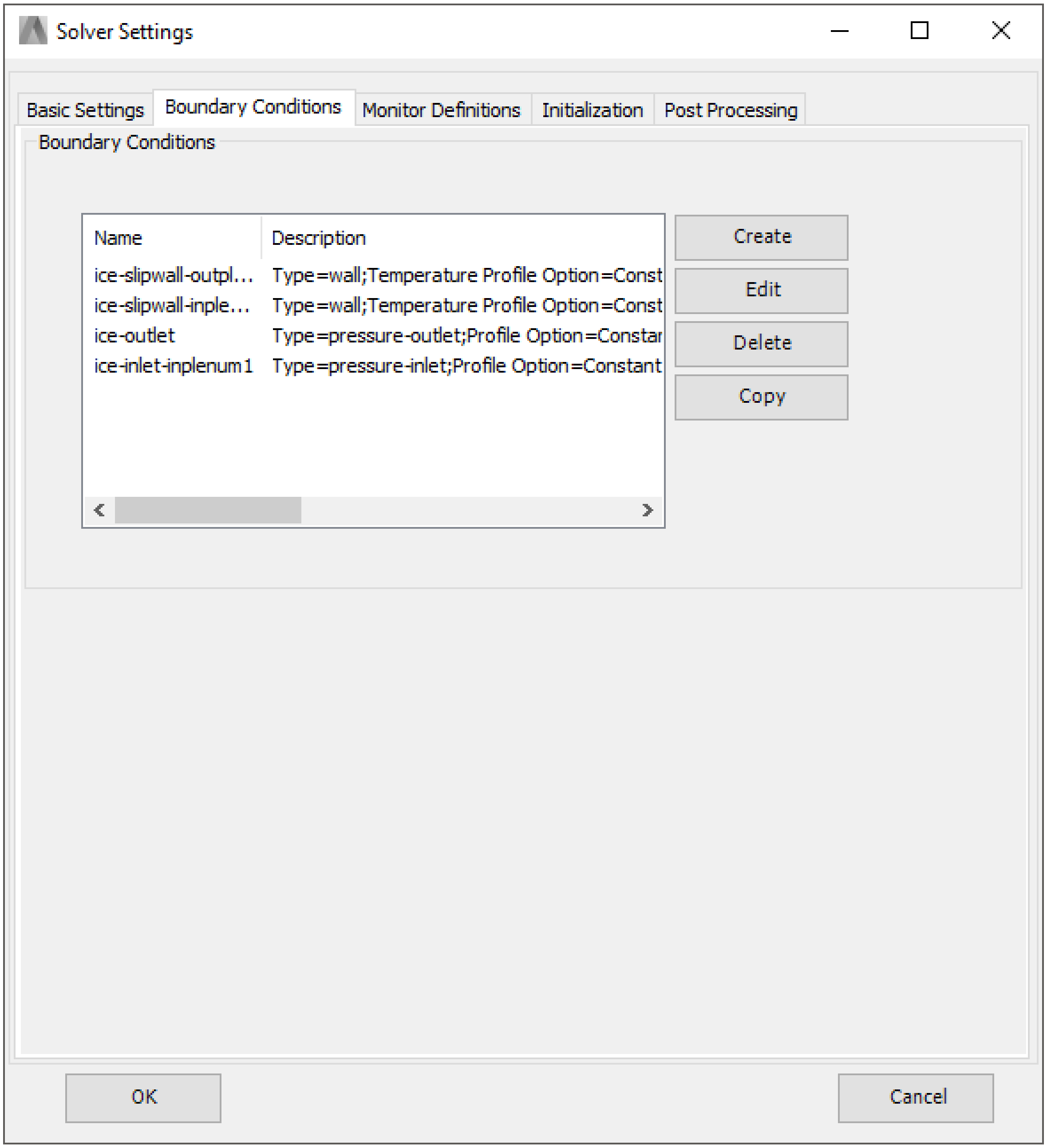


- Select a profile from the drop-down list of **Profiles**.
- When a particular profile is selected all corresponding variables will be displayed in the **X Axis** and **Y Axis** drop-down lists.
- Select the variable for **X Axis** and **Y Axis** and click **Plot**. The plot will be displayed in the area of **Profile Chart**.
- You can also read a profile by clicking on **Read File**.
- The read profile which is presently selected can be deleted from the list by clicking on **Remove Profile**.
- You can make changes to the values in the profile table displayed. Click **Plot** to view the **Profile Chart** with the changed values. You can save the profile with the edited values by clicking on **Save**.
- To manipulate the chart:

Table 9.1: Chart Manipulation

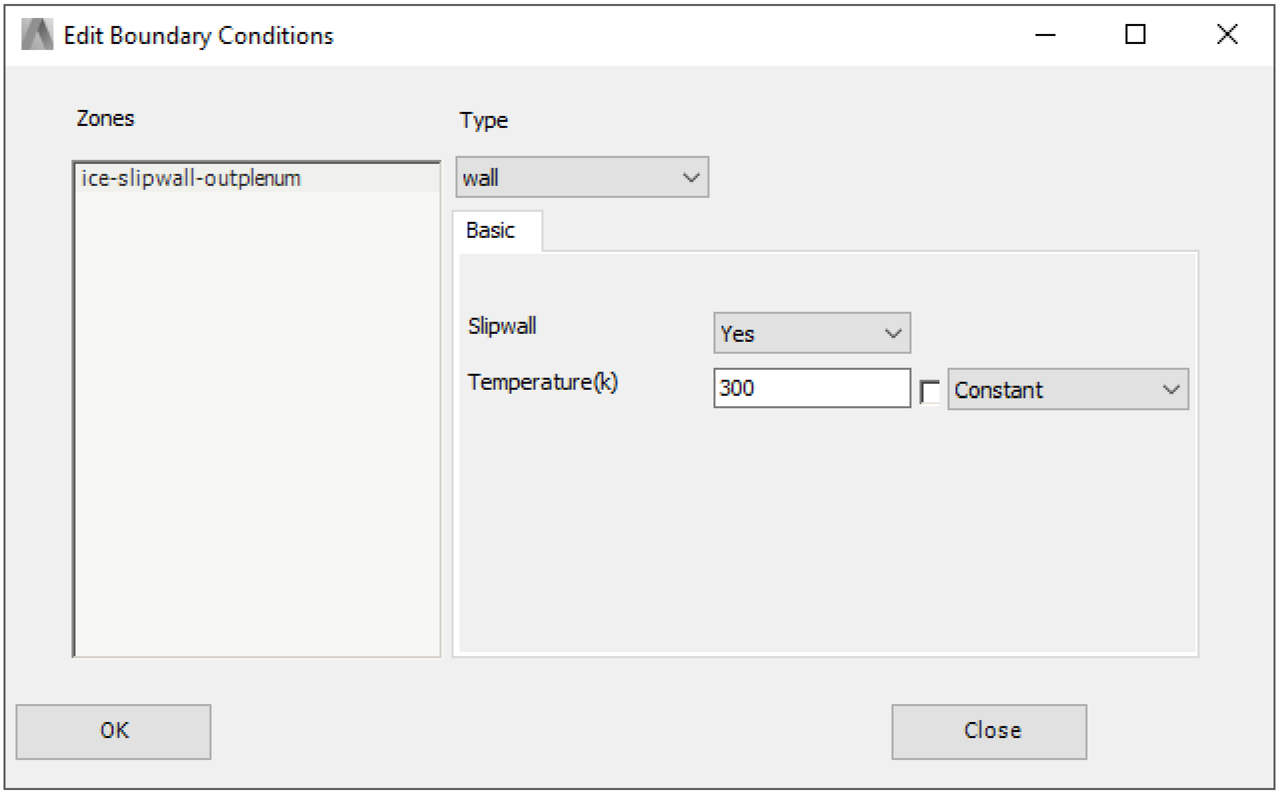
	Operation
Rotate Middle Mouse	Zoom
Shift + Middle Mouse	Zoom
Ctrl + Middle Mouse	Pan
Drag Right Mouse	Box Zoom
F key	Fit to Window

9.1.2. Boundary Conditions



In the **Boundary Conditions** tab you can see boundary conditions set for four zones. Select each zone and click **Edit** to check the details.

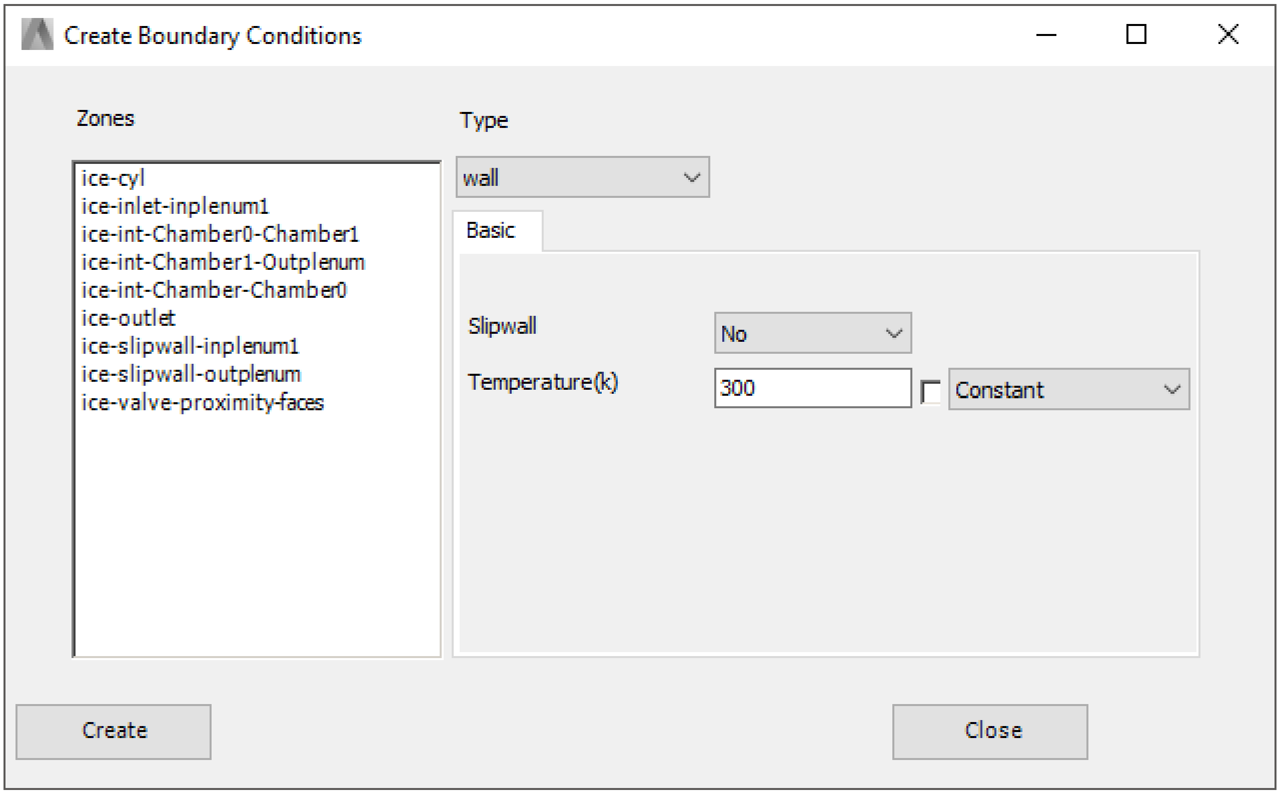
- **ice-slipwall-outplenum** is set to **Type wall**.



The wall is set as slipwall and the **Temperature** is set to **300**.

- Similar conditions are set for **ice-slipwall_inplenum1**.
- **ice-outlet** is set as **Pressure Outlet** with **Gauge Pressure** set to **–5000** and **Temperature** set to **300**.
- **ice-inlet-inplenum1** is set as **Pressure Inlet** with **Gauge Pressure** set to **0** and **Temperature** set to **300**.

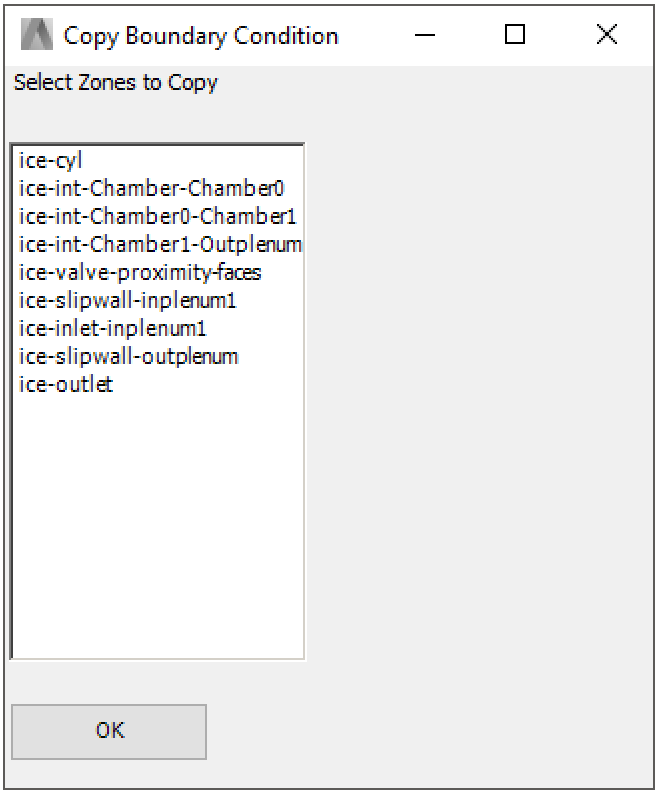
You can check the detailed settings in the **Boundary Conditions** (p. 327) task page in Ansys Fluent. You can also create additional boundary conditions by clicking **Create**.



In the **Create Boundary Conditions** dialog box select the zone to which you like to apply the boundary conditions to from the list under **Zones**. Then select **Type** and enter the required values for the variables, either a constant value or a variable profile from the drop-down list. You can select a profile in case you have read a profile before, or you can select **PROFILE EDITOR** option which will open the **Profile Editor** dialog box where you can check the plot of the profile or read a new profile. After you click **OK** you can see the zone name and the details in the **Boundary Conditions** tab.

You can use the **Copy** button to copy the boundary conditions to other multiple zones.

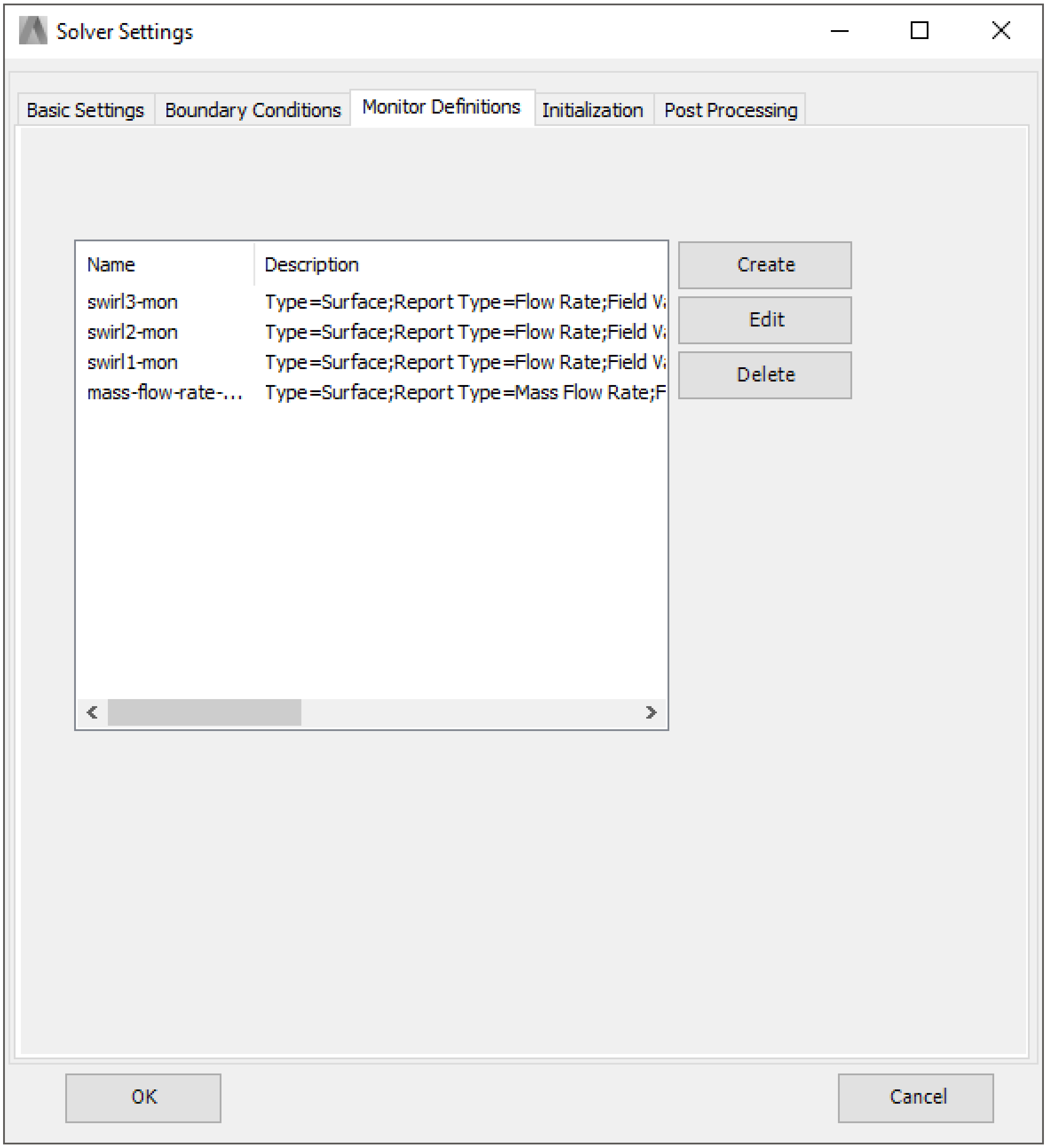
- Select the boundary condition of interest in the **Boundary Conditions** tab.
- Click **Copy**.



- Select the zone(s) to which you want to copy the boundary conditions and click **OK**.
- The zones to which you have copied the boundary conditions appear in the list.

For the boundary conditions, you can parametrize the quantity by enabling the check box next to it.

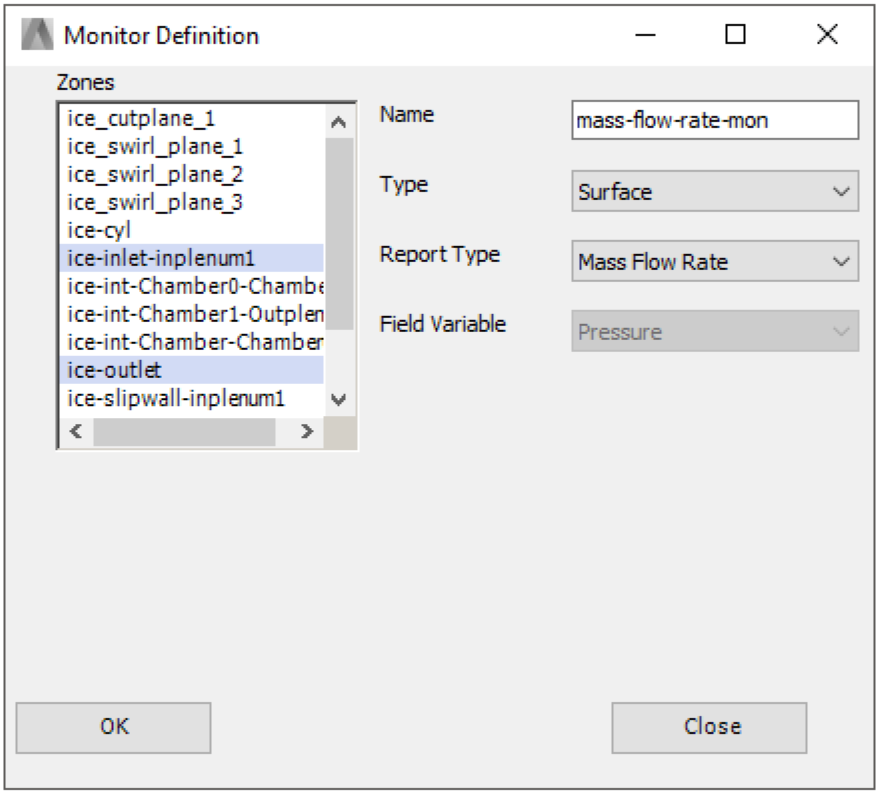
9.1.3. Monitor Definitions



In the **Monitor Definitions** tab you can see that at least one surface monitor has been set by default. It plots the variable **Mass Flow Rate** at the inlet and outlet. Depending upon the number of postprocessing planes (defined by **Post Planes Dist. From Ref.** in the **Input Manager**) those many swirl monitors will be created. Select each monitor and click **Edit** to check the details.

For information on how the swirl ratio is monitored on the swirl planes see the section on swirl definition (p. 582).

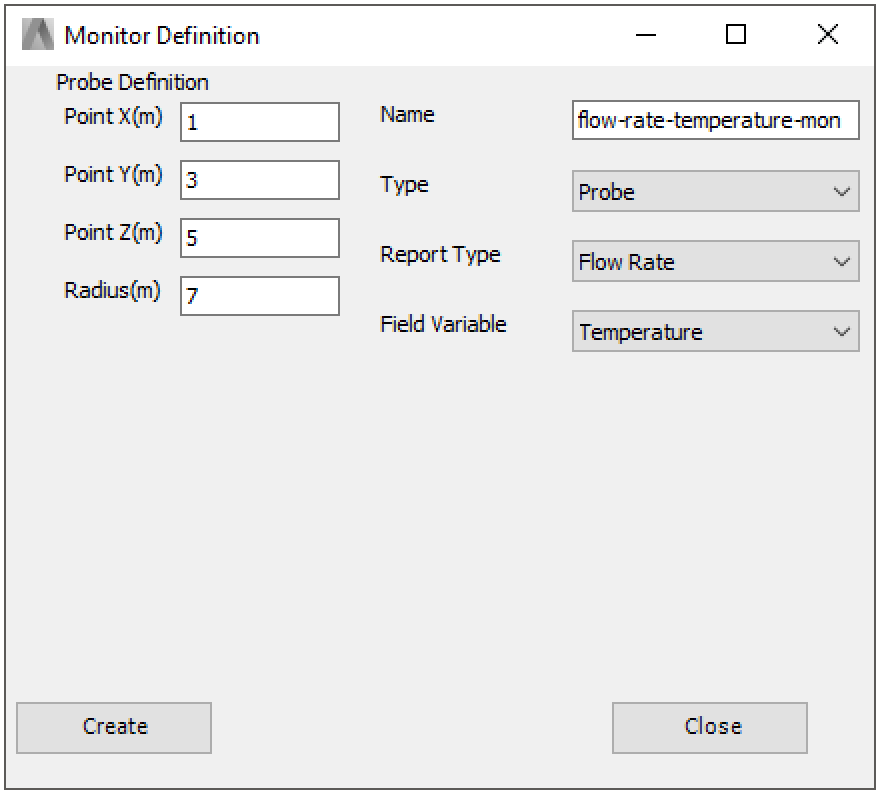
- **mass-flow-rate-mon** is a surface monitor.



It plots the **Mass Flow Rate** on **ice-inlet-inplenum1** and **ice-outlet**. This monitor is automatically set for a port flow simulation. You can change the settings here in the **Monitor Definition** dialog box.

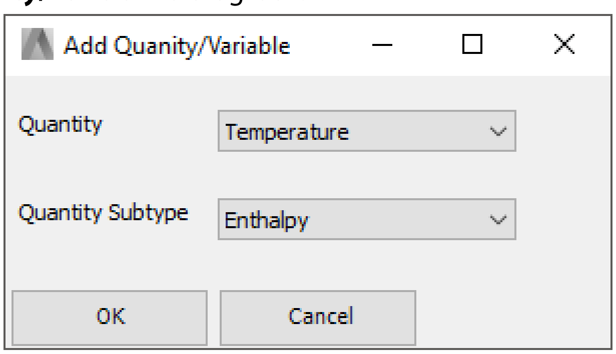
You can see additional details in the **Monitors** (p. 334) task page in Ansys Fluent. You can create additional monitors by clicking **Create**. Then select the **Type**, **Report Type**, and the **Field Variable** from the respective drop-down lists.

You can select **Volume**, **Surface**, or **Probe** from the **Type** drop-down list. If you select **Probe** you can create a point or surface on which you can monitor a variable.

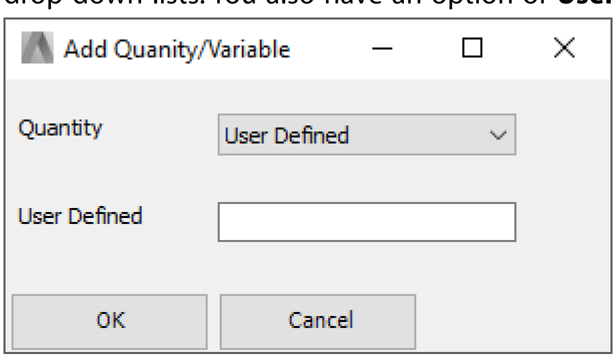


You need to enter values for **Point X**, **Point Y**, and **Point Z** to position the point. If you retain the value of **0** for **Radius** then a point monitor is created. If you enter a value for **Radius** then a circular surface with the given **Radius** and the center as the given **Point** is created. You can monitor the variables of your choice on this surface or point.

You can select **New Variable** from the **Field Variable** drop-down list. This opens the **Add Quantity/Variable** dialog box.



You can select the variable by selecting from the options under the **Quantity** and **Quantity Subtype** drop-down lists. You also have an option of **User Defined** under the **Quantity** drop-down list.

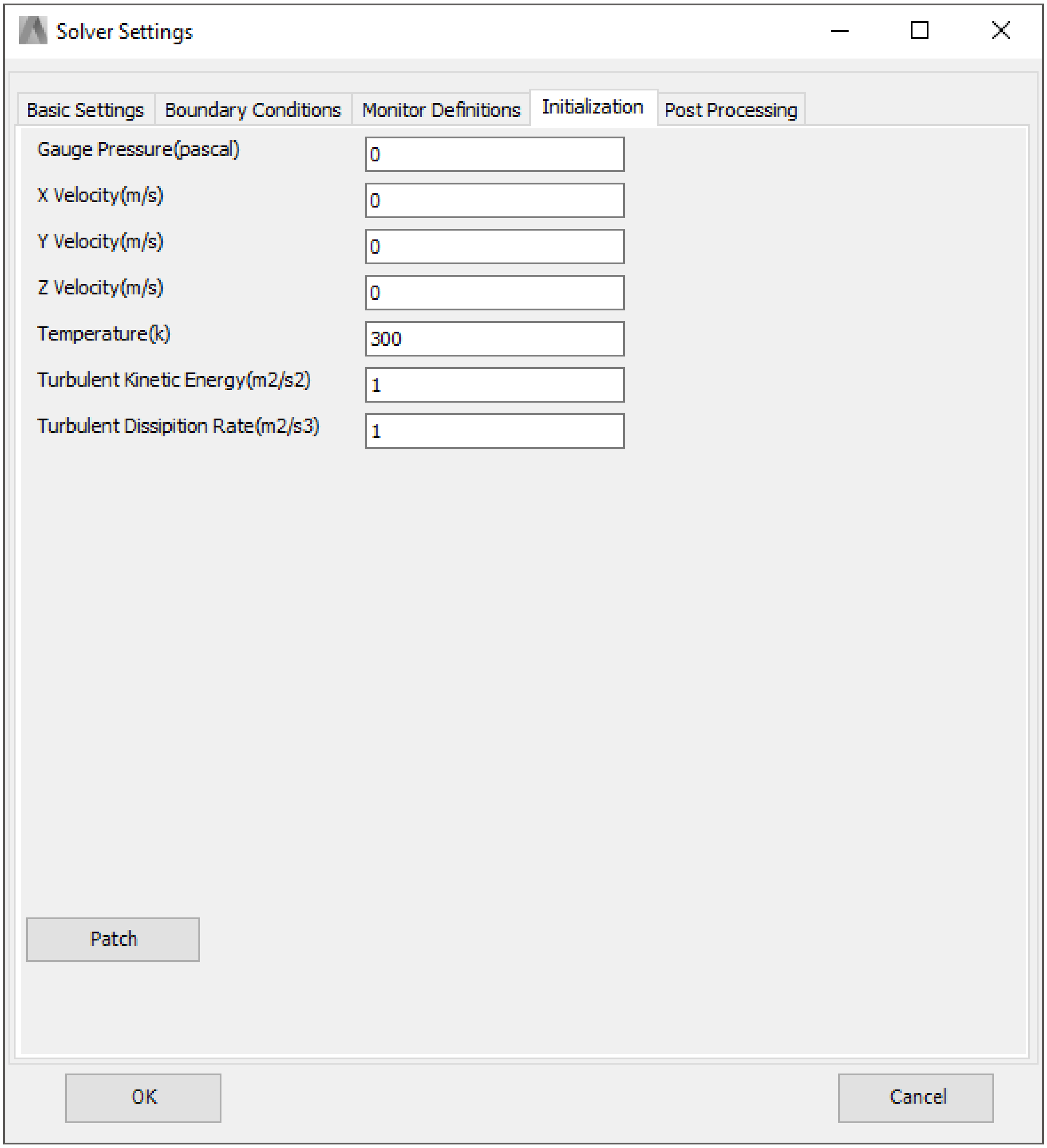


Here you can add the quantity or variable of your choice of which you would like postprocessing images, in the **User Defined** text box. You will have to check Fluent if the term is valid for the simulation. After you click **OK** this quantity will be available in the drop-down list of **Field Variable**.

The **Name** will be set according to your selections of monitor type and variable. You can override the proposed name. Click **Create** to create the monitor. It will now appear in the list in the **Monitor Definitions** tab.

9.1.4. Initialization

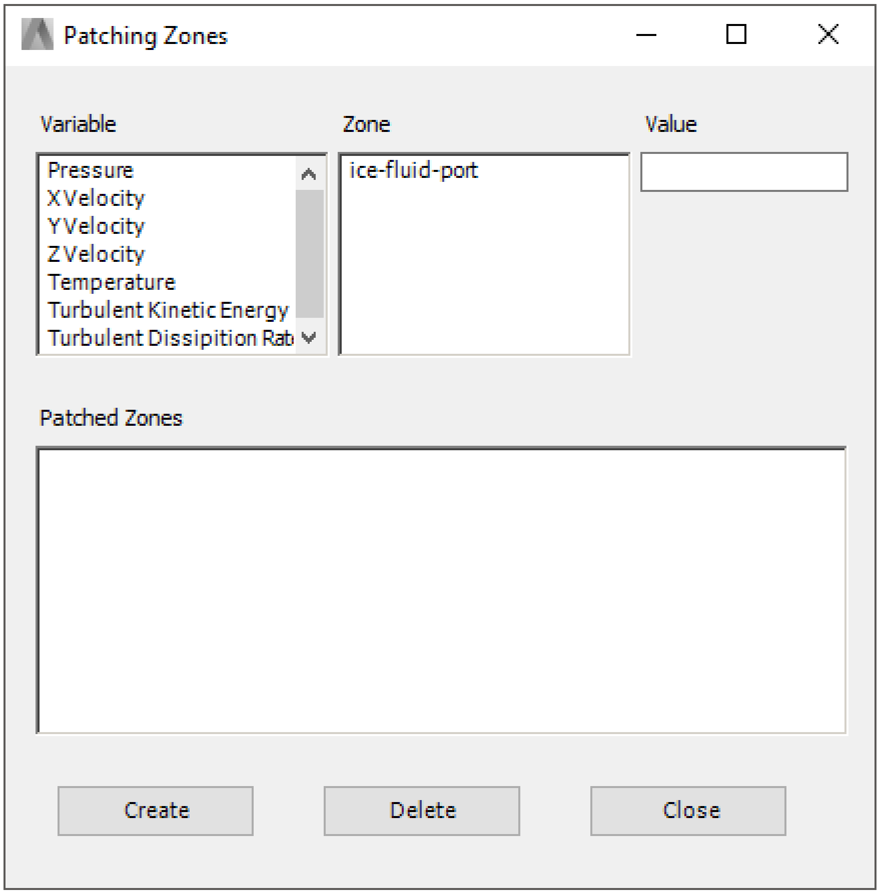
In the **Initialization** tab you can see the default set values for the various parameters.



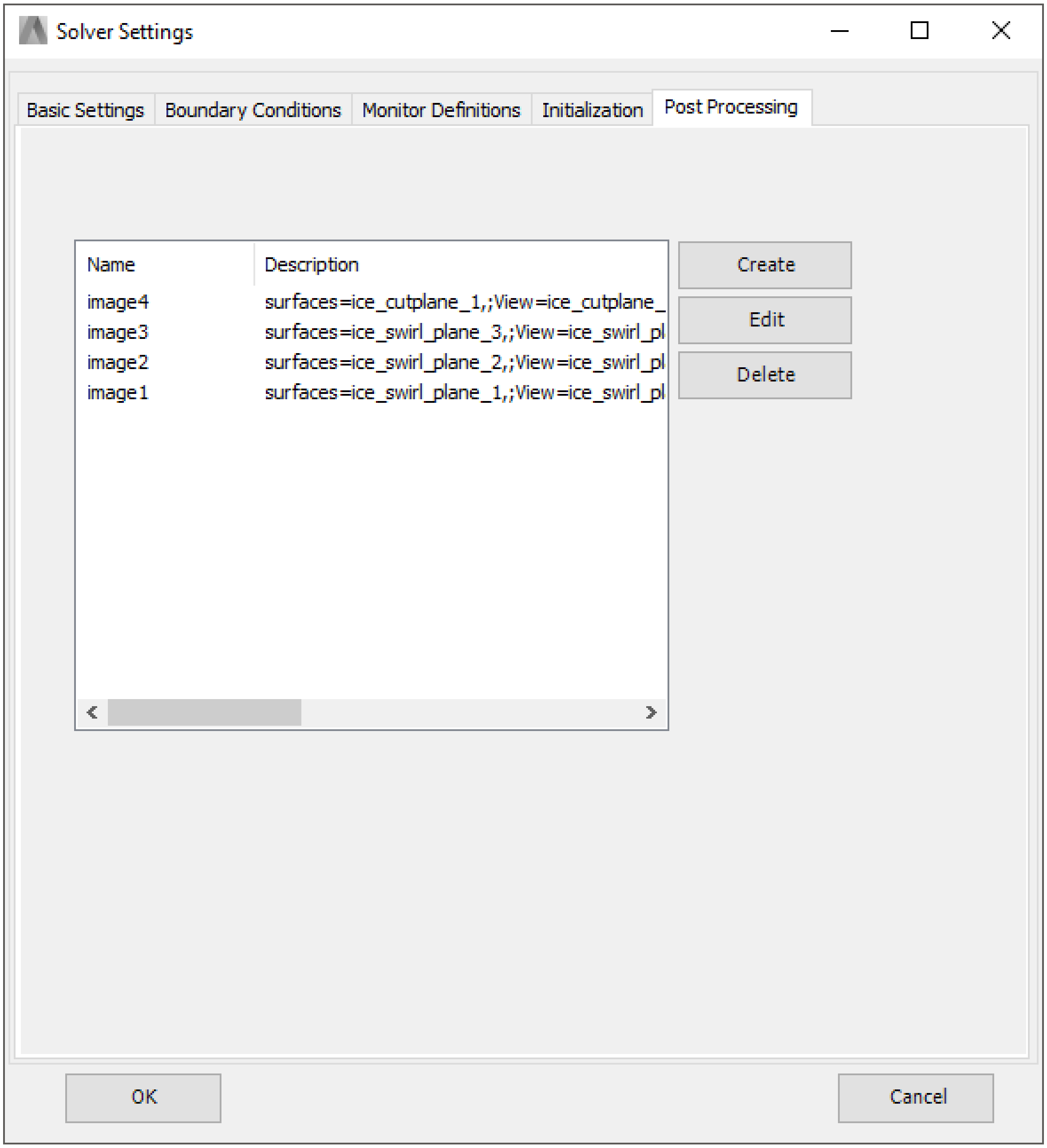
- **Gauge Pressure**, **X Velocity**, **Y Velocity**, and **Z Velocity** are all set to **0** by default.
- **Temperature** is set to **300**.

- **Turbulent Kinetic Energy** and **Turbulent Dissipation Rate** are set to **1** by default.

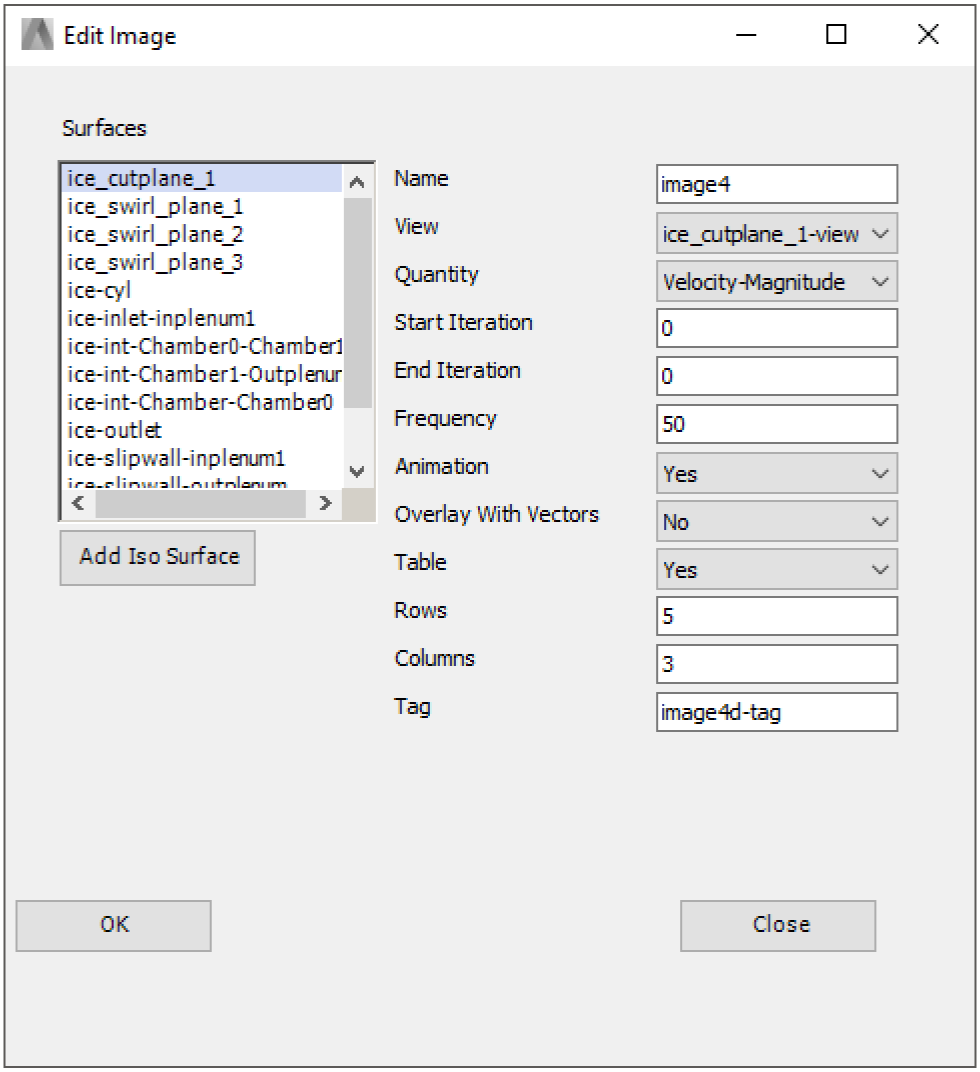
You can change the values as per your settings. Click **Patch** to open the **Patching Zones** dialog box. You can patch the various listed variables at your desired values to the **ice-fluid-port** zone.



9.1.5. Postprocessing

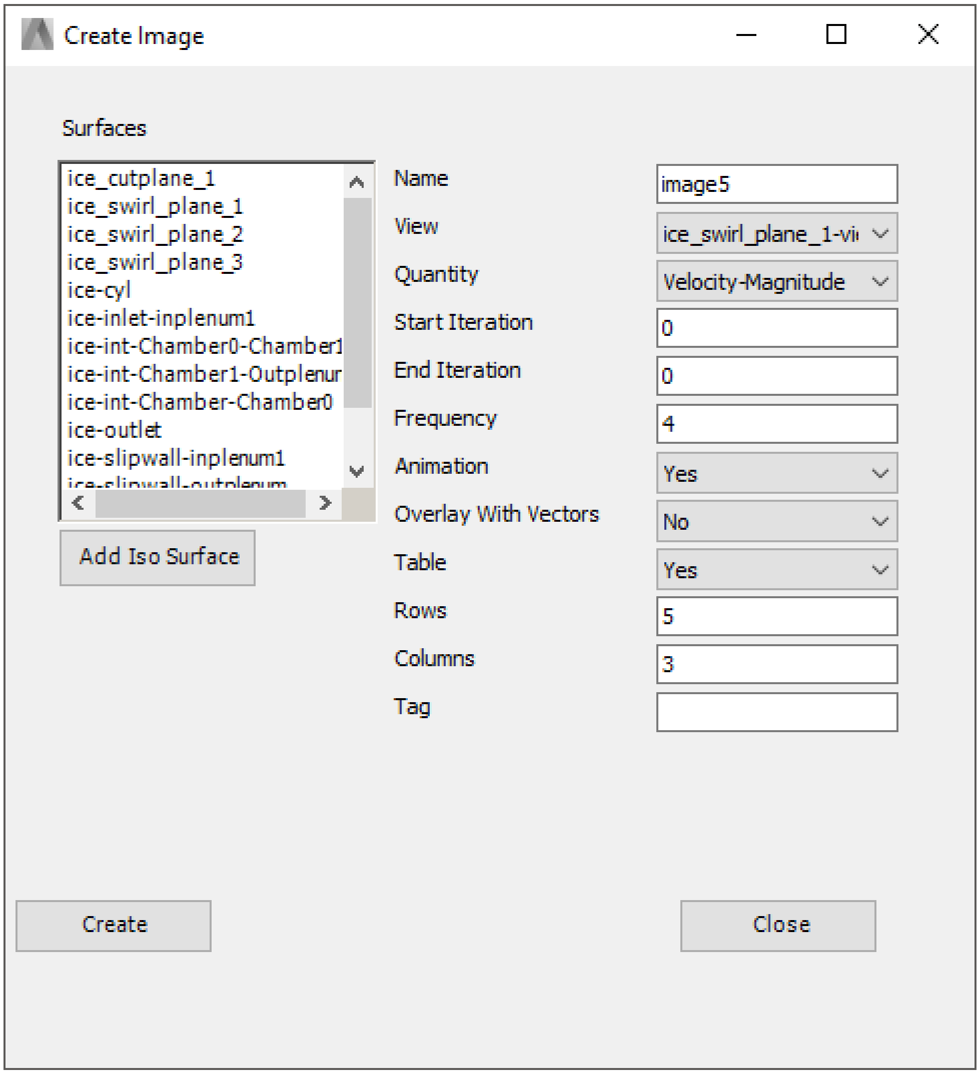


In the **Post Processing** tab you can see that one image is being automatically saved of the velocity contours plotted on the cut-plane surface. Additional images of swirl plotted on the swirl planes are saved. These will depending upon the number of **Post Planes Dist. From Ref.** you have defined in the **Input Manager**.

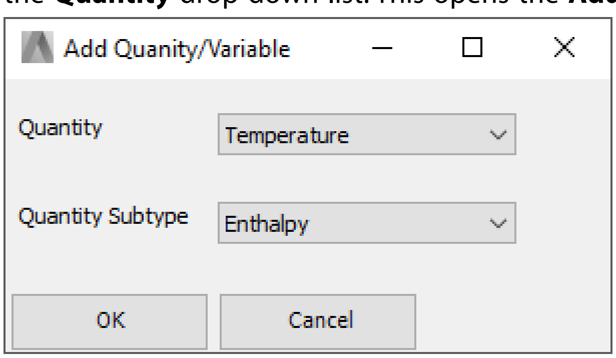


- The image is plotted on surface **ice_cutplane_1**.
- The image is captured at a saved view, **ice_cutplane_1–view**.
- **Velocity-Magnitude** contours are displayed.
- **Start Iteration** and **Stop Iteration** are set to `0` by default.
- Images are saved at a **Frequency** of **50** iterations.
- **Animation** is set to **Yes** signifying that animation of the saved velocity contour images will be created at the end of the simulation.
- **Overlay With Vectors** is set to **No**.
- **Table** is set to **Yes** signifying that the saved images will be displayed in the report in a table of **5 Rows** and **3 Columns**.
- **Tag** shows the name tag for the images.

In the **Edit Image** dialog box you can change the settings and values as required. To create additional postprocessing images click **Create**.



1. Select the **Surfaces** on which you want to plot the contours.
2. Select a view from the **View** drop-down list.
3. Select the parameter from the **Quantity** drop-down list. You can select **New Variable** from the **Quantity** drop-down list. This opens the **Add Quantity/Variable** dialog box.



Here you can add the quantity or variable of your choice of which you would like postprocessing images. You will have to check Fluent if the term is valid for the simulation.

4. Enter the **Start Iteration** and **End Iteration**. The images will be saved only within these points.

5. Enter a value for **Frequency**. The images will be captured at the entered frequency.

6. If you need to create an animation from the saved images you can select **Yes** from the **Animation** drop-down list.

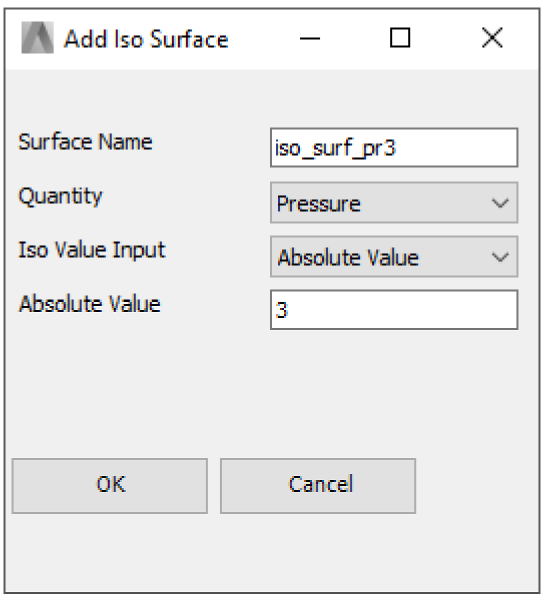
7. The option **Overlay With Vectors** is by default set to **No**. You can set it to **Yes** if you want the vectors to overlap the contour images.

8. If you want the saved images to be displayed in the report in a table format select **Yes** from the **Table** drop-down list.

9. Enter the values for **Rows** and **Columns** depending on how you want to format the table.

10. You can add a tag for the image. You can use this **Tag** if you want only the final images of different surfaces in a single table. In this case you have to provide the same tag to all the images.

11. Click **Add Iso Surface** if you want to create an iso-surfaces. These surfaces are isovalued sections of the entire domain.



   a. Enter a name for the isosurface you want to create at **Surface Name**.

   b. Select the **Quantity** from the drop-down list. You can select a quantity from the list or create a new variable by selecting **New Variable**.

   c. From the **Iso Value Input** drop-down list you can select either of **Absolute Value** or **Percentage of Range**.

      - **Absolute Value**: If you select this from the drop-down options, you can enter a value for **Absolute Value**. The iso-surafce created will be of this value.

      - **Percentage of Range**: If you select this from the drop-down options, then you need to enter a percentage value for **Percentage of Range**. The iso-surface created will be of the percentage value of the minimum and maximum values obtained of the quantity.

An iso-surface of the given name and of the selected quantity will be created and appear in the list of **Surfaces**. This isosurface will be of the **Absolute Value** entered or of the percentage of the minimum and maximum values of the quantity. You can select this isosurface to create images in postprocessing.

12. Click **Create** to create the image.

13. Click **Close** to close the **Create Image** dialog box.

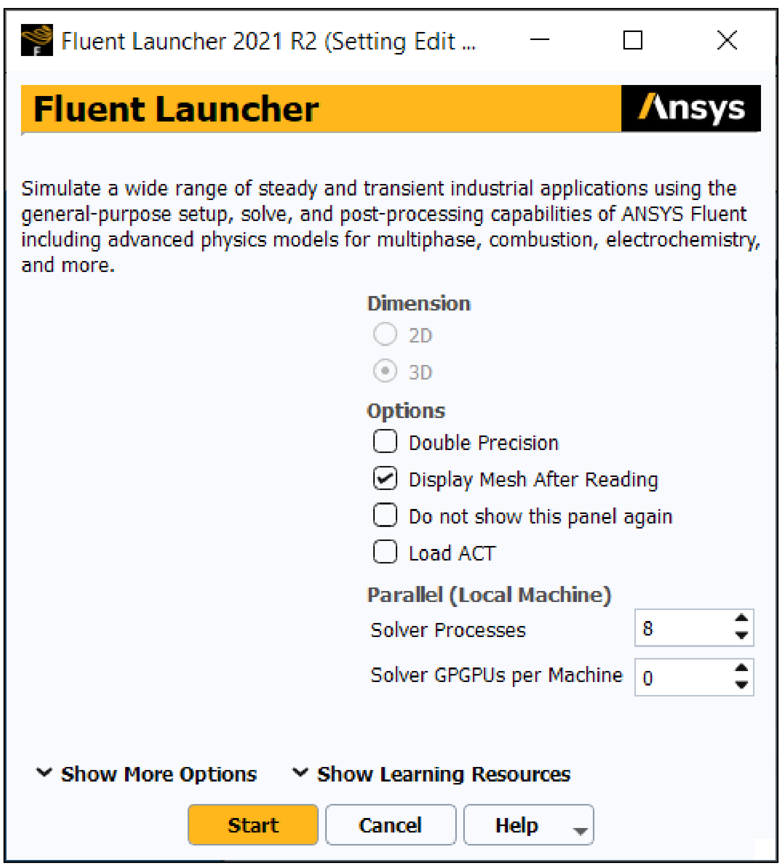
Once the changes are done in the **Solver Settings** dialog box you can close it and then update the **ICE Solver Setup** cell by choosing **Update** from the context menu.

9.2. Solver Default Settings

When you double-click the **Setup** cell, **FLUENT Launcher** opens.

Note:

Depending on your machine configuration, you may be able to increase the **Number of Processes** to decrease the time taken to arrive at a solution. A reasonable value for the **Number of Processes** is the number of cores on your machine.

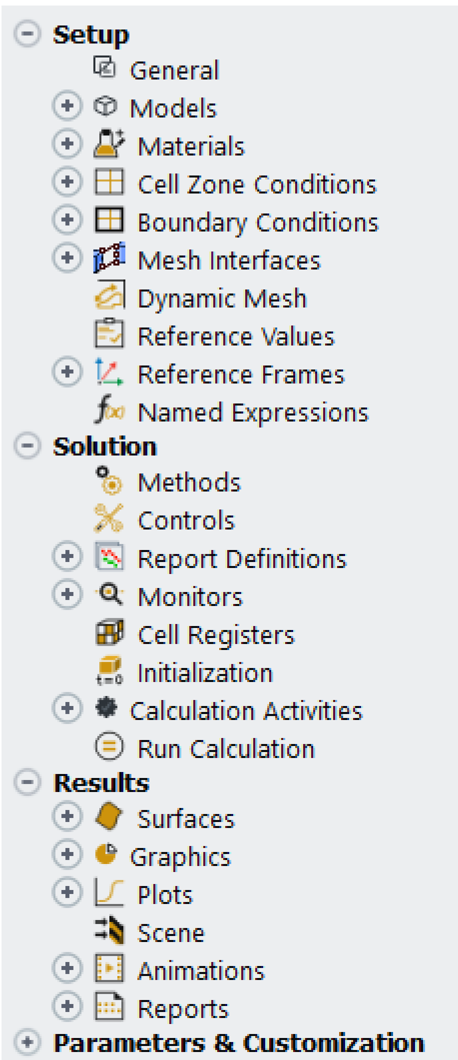


When you click **OK**, Ansys Fluent reads the mesh file and sets up the IC Engine case. It will:

- Set up the required models.
- Set up the default boundary conditions and material.
- Set up the default monitors.
- Use Full Multigrid (FMG) initialization. For more information, refer to Full Multigrid (FMG) Initialization in the Fluent Theory Guide.

In the Ansys Fluent application, you can check the default settings by highlighting the items in the navigation pane.

Figure 9.1: The Ansys Fluent Navigation Pane



The details about the settings in the different task pages are described in the following sections:

9.2.1. General Settings

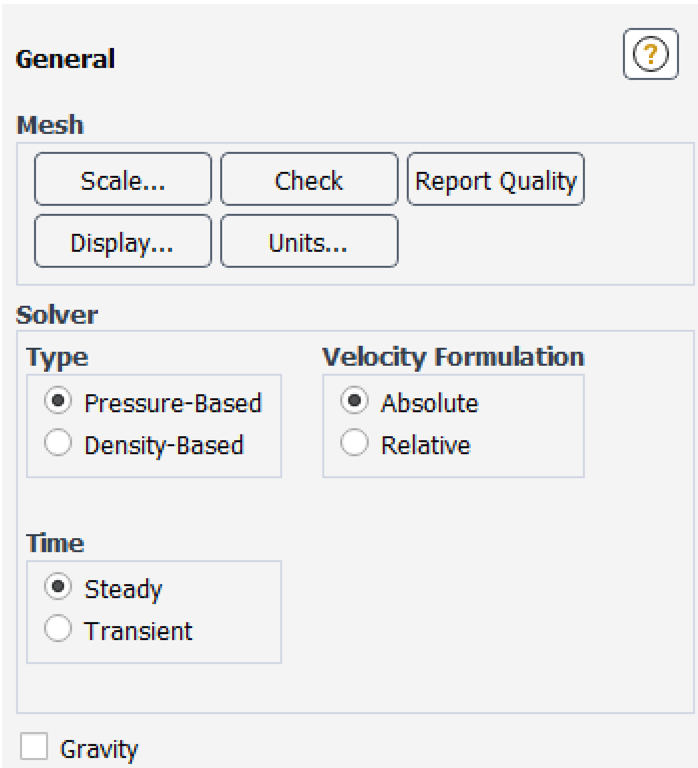
9.2.2. Models

9.2.3. Materials

9.2.4. Boundary Conditions

9.2.1. General Settings

The **General** task page has the following settings:

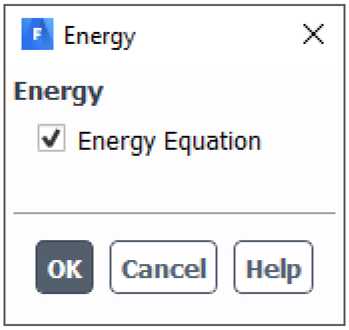


- c
- Solver **Type** is set to **Pressure-Based**.
- Solver **Time** is set to **Steady**.

9.2.2. Models

The following models are selected for the analysis:

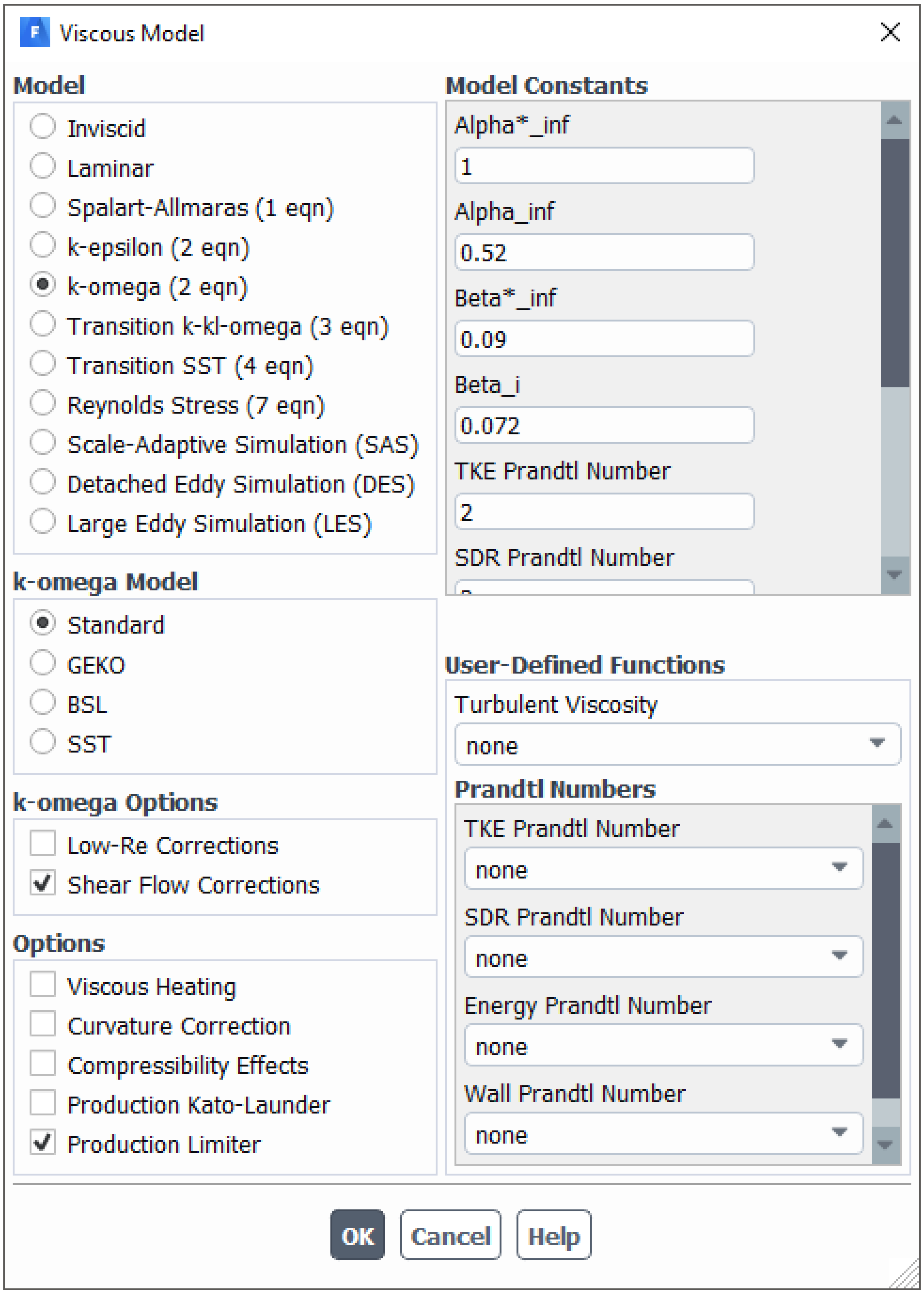
- The **Energy** model is enabled.



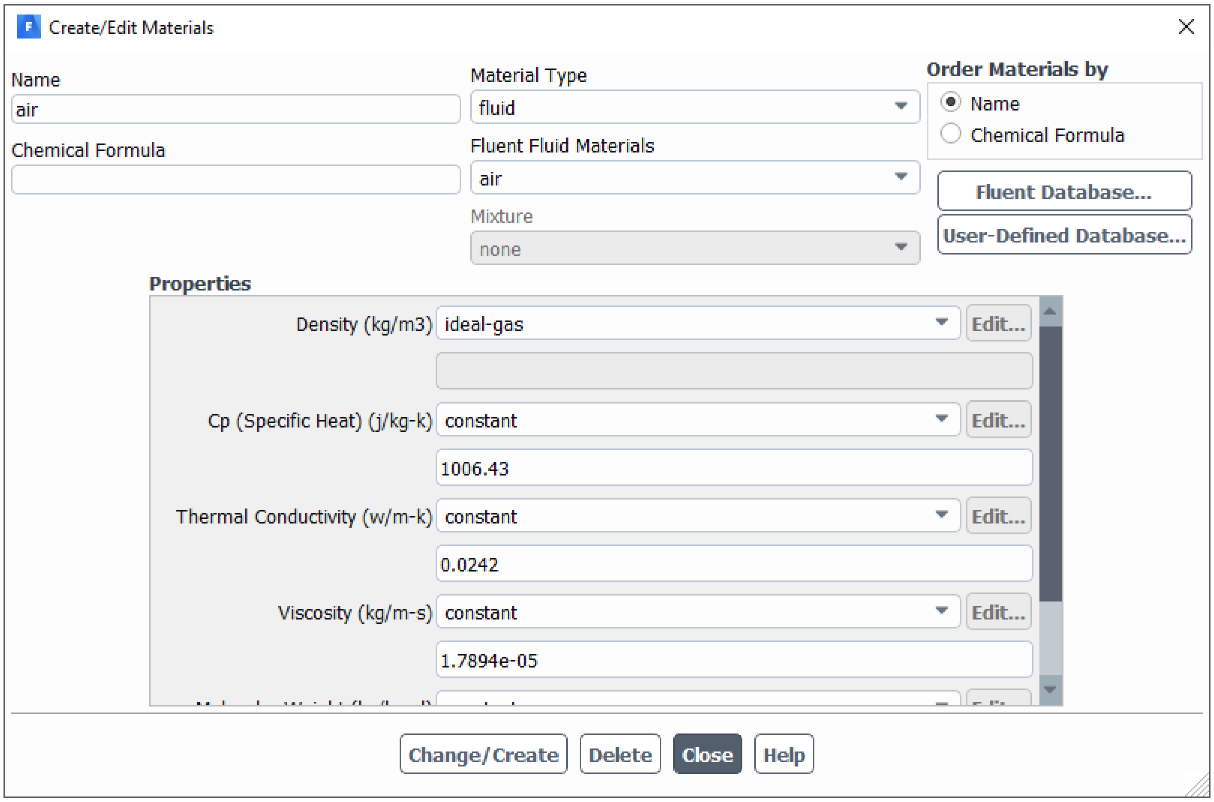
- From the **Viscous** models, the **Standard k-omega** model is selected. By default **Compressibility Effects** and **Shear Flow Corrections** are enabled.

Note:

The **k-omega** model internally adjusts the wall treatment based on mesh refinement near the walls. Validation tests have determined that this model is more stable and predicts correct results for port flow analyses. For more details about the **k-omega** model, refer to Standard and SST k- ω Models in the Fluent Theory Guide.


Viscous Model
Model
Inviscid
Laminar
Spalart-Allmaras (1 eqn)
k-epsilon (2 eqn)
k-omega (2 eqn)
Transition k-kl-omega (3 eqn)
Transition SST (4 eqn)
Reynolds Stress (7 eqn)
Scale-Adaptive Simulation (SAS)
Detached Eddy Simulation (DES)
Large Eddy Simulation (LES)
k-omega Model
Standard
GEKO
BSL
SST
k-omega Options
Low-Re Corrections
Shear Flow Corrections
Options
Viscous Heating
Curvature Correction
Compressibility Effects
Production Kato-Launder
Production Limiter
Model Constants
Alpha*_inf
1
Alpha_inf
0.52
Beta*_inf
0.09
Beta_i
0.072
TKE Prandtl Number
2
SDR Prandtl Number
User-Defined Functions
Turbulent Viscosity
none
Prandtl Numbers
TKE Prandtl Number
none
SDR Prandtl Number
none
Energy Prandtl Number
none
Wall Prandtl Number
none
OK
Cancel
Help

9.2.3. Materials

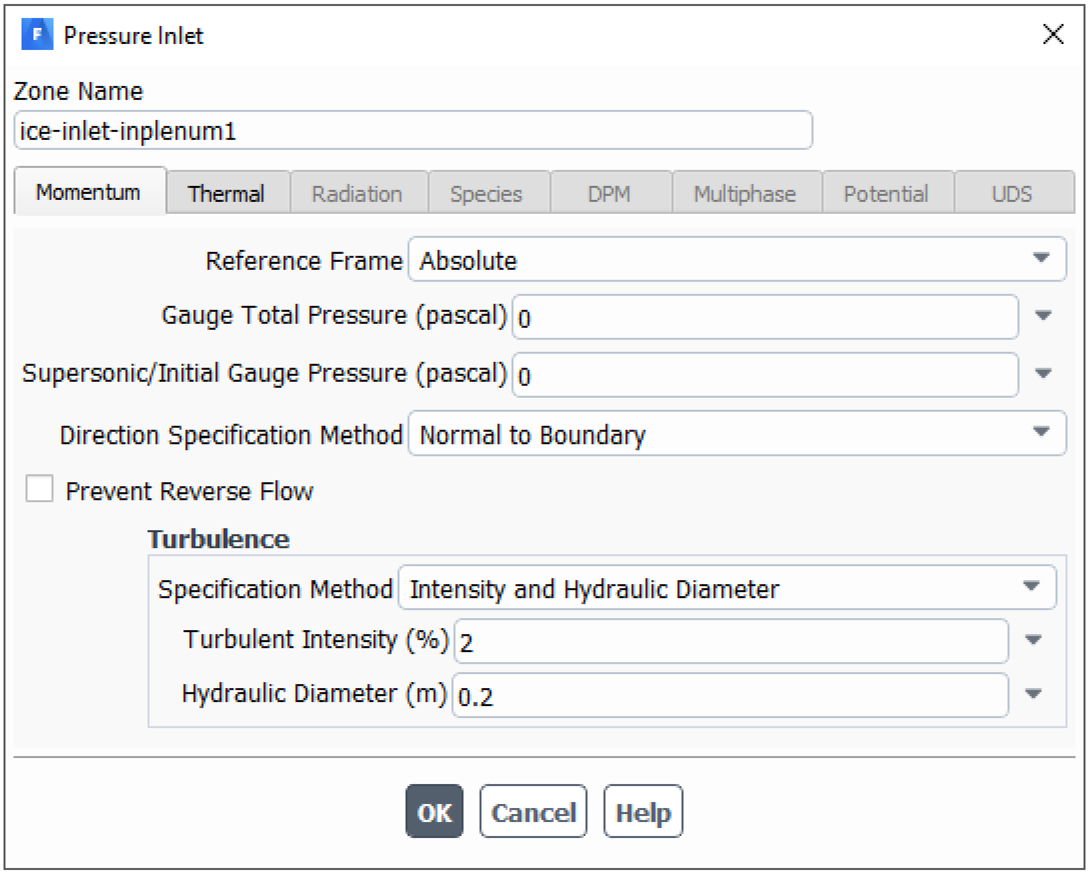


- **Air** is set as the material.
- The **Density** of air is set to **ideal-gas**.
- The **Specific Heat (Cp)** of air is set to **constant**.

9.2.4. Boundary Conditions

You can check the boundary conditions and monitors settings for port flow simulation in the `icBC-Settings` file. The values shown for the boundary conditions are the defaults set by the IC Engine System. If any of these values are different for your problem, you can change them in Fluent or you can edit the `icUserSettings.txt` file before launching Fluent to set the new default values.

- For the **ice-inlet-inplenum1** boundary conditions, the following settings are done in the **Pressure Inlet** dialog box:



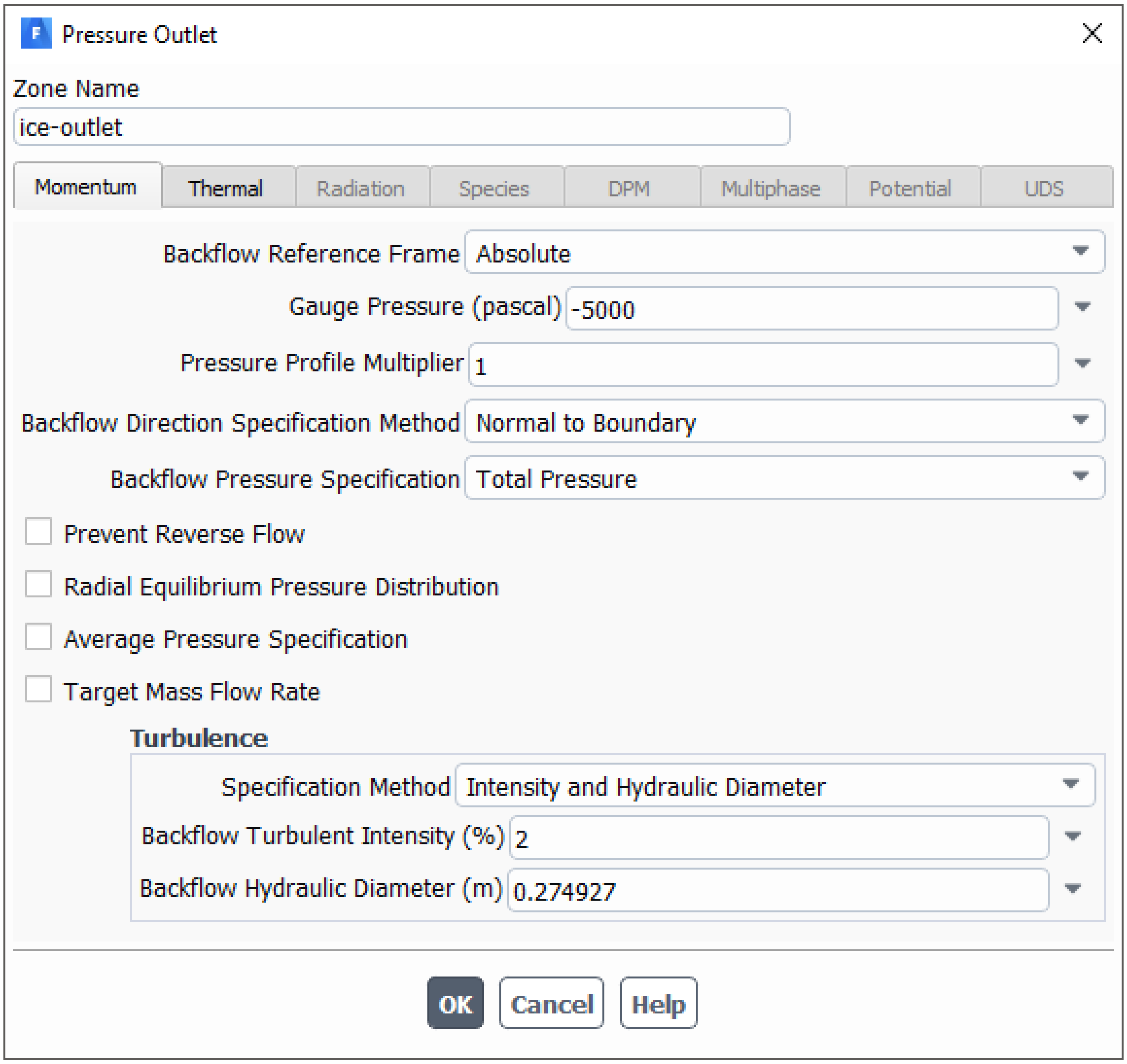
- **Gauge Total Pressure** is set to `0` Pa.
- The **Specification Method** in the **Turbulence** group box is set to **Intensity and Hydraulic Diameter**.

Note:

The **Specification Method** can also be set to **Intensity and Viscosity Ratio**.

- **Turbulent Intensity** is set to 2%, and **Hydraulic Diameter** is automatically calculated.

• For the **ice-outlet** boundary condition the following settings are done in the **Pressure Outlet** dialog box.



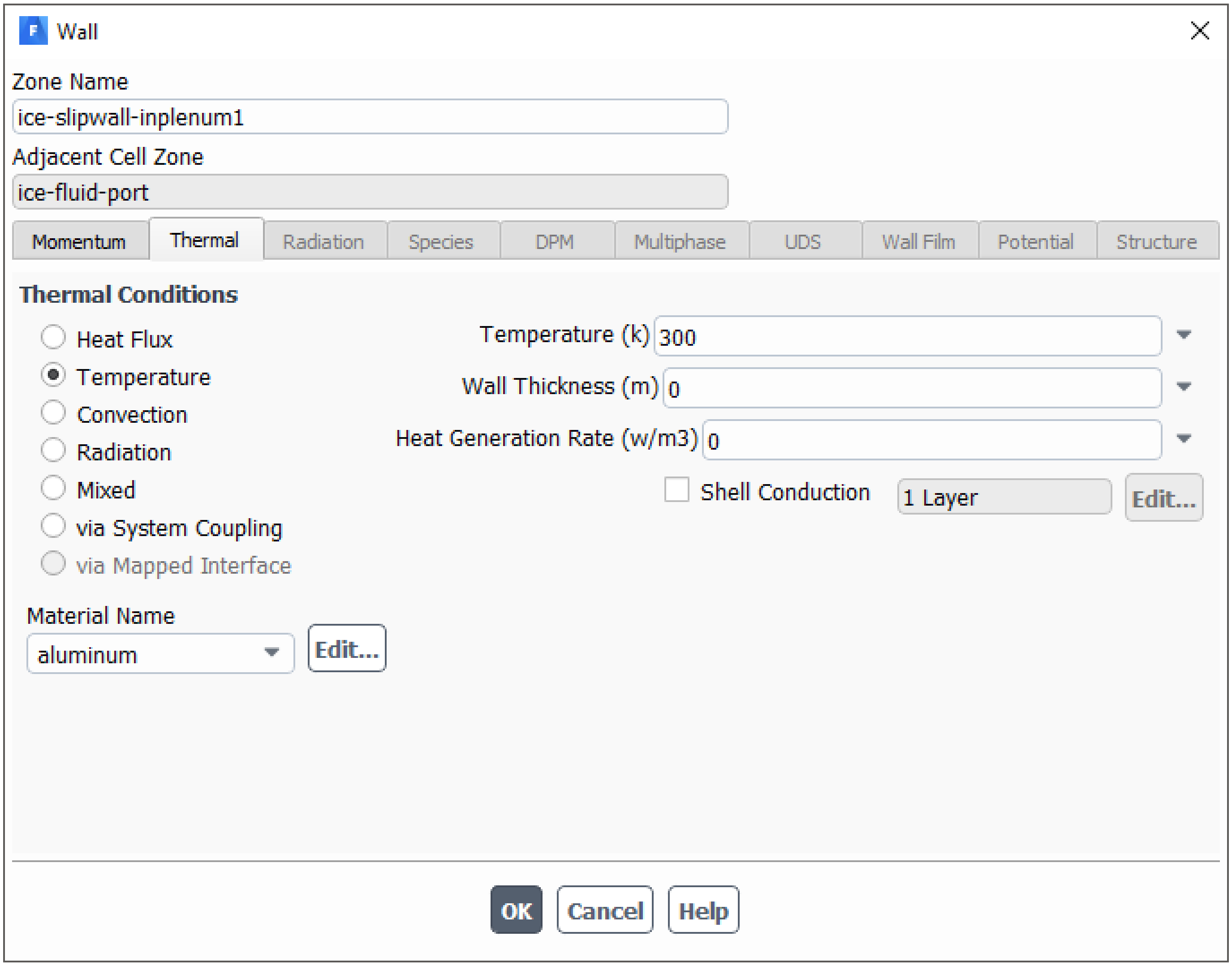
– **Gauge Pressure** is set to `-5000` Pa.

Note:

This is set based on the assumption that inlet and outlet pressure difference is 5000 Pa. If you have data on the actual values you can change the settings.

– **Backflow Total Temperature** is set to `300` K.

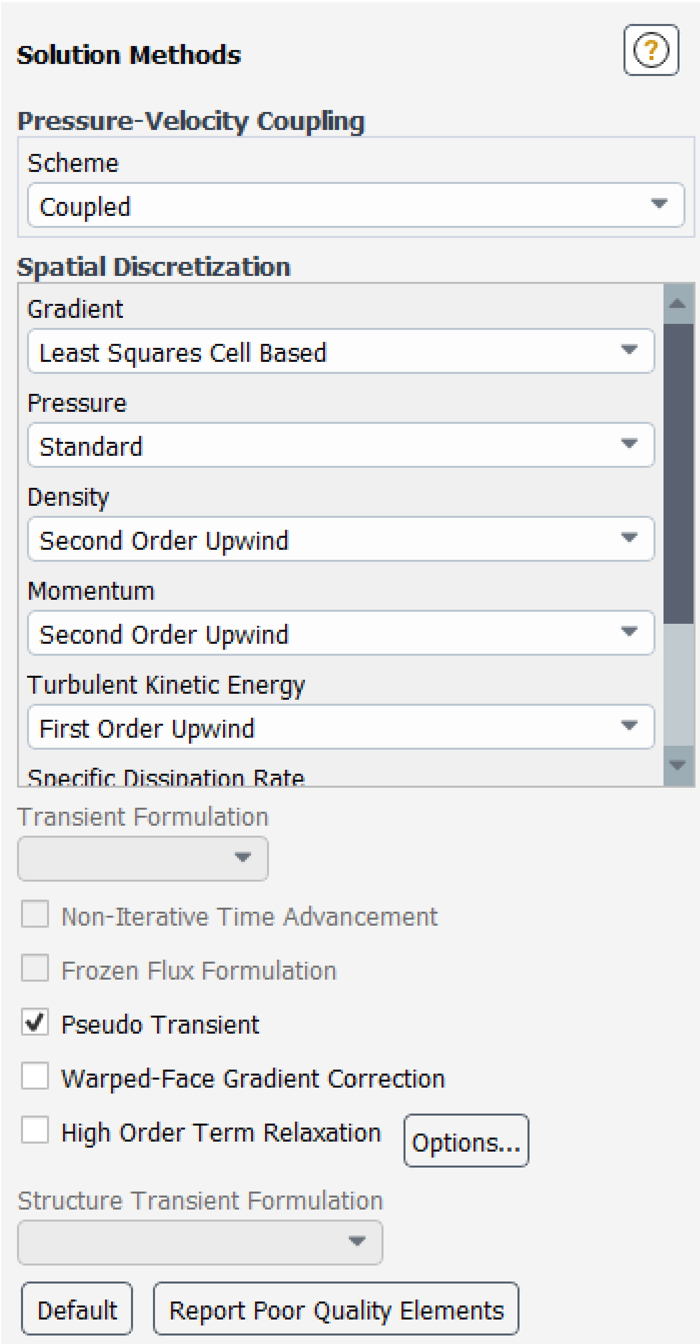
- For the **wall** boundary condition **Specified Shear** is selected from the **Shear Condition** group box. In the **Thermal** tab **Temperature** is set to `300` K.



9.2.5. Solution Methods

The **Solution Methods** are set differently for **Hybrid** and **CutCell** mesh type. The following are the settings for **Hybrid** mesh type.

Figure 9.2: Solution Methods for Hybrid Mesh Type

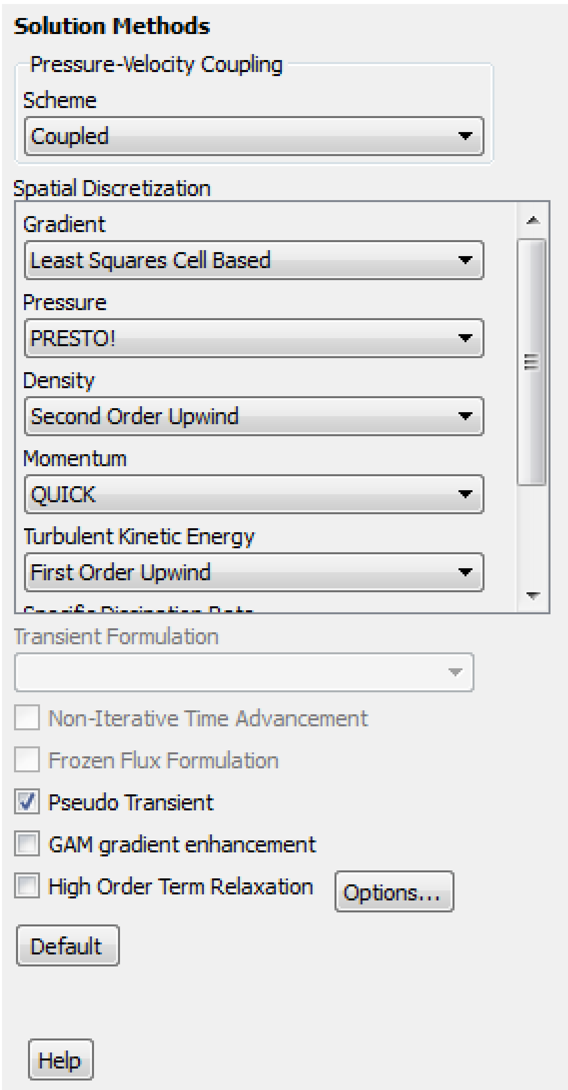


- The **Scheme** for the analysis is set to **Coupled** in the **Pressure-Velocity Coupling** group box.
- The **Gradient** in the **Spatial Discretization** group box is set to **Least Squares Cell Based**.
- **Pressure** is set to **Standard**.
- **Density**, **Momentum**, and **Energy** are all set to **Second Order Upwind**.
- **Turbulent Kinetic Energy** and **Specific Dissipation Rate** are set to **First Order Upwind**.

- **Pseudo Transient** is enabled. This increases the robustness and speed of solution convergence. For more information about this option refer to, Solving Pseudo-Transient Flow in the Fluent Theory Guide.

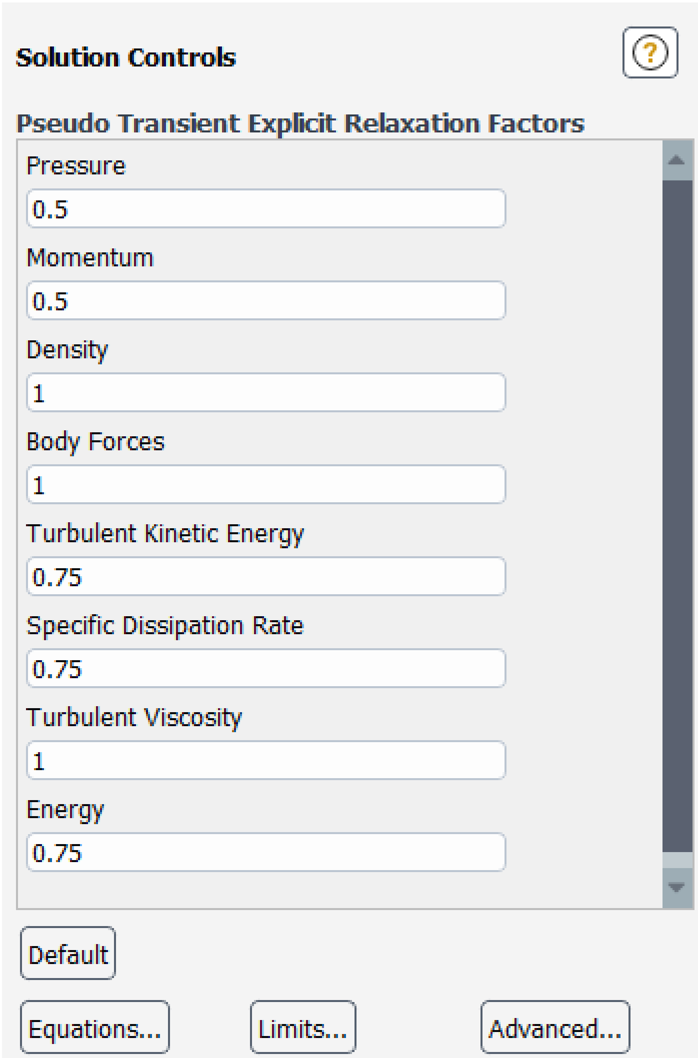
The following are the settings for **Cutcell** mesh type.

Figure 9.3: Solution Methods for CutCell Mesh Type



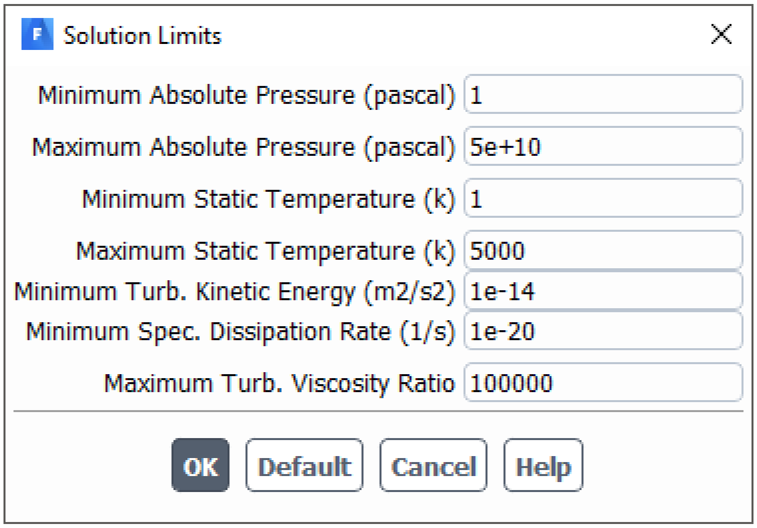
- The **Scheme** for the analysis is set to **Coupled** in the **Pressure-Velocity Coupling** group box.
- The **Gradient** in the **Spatial Discretization** group box is set to **Least Squares Cell Based**.
- **Pressure** is set to **Presto!**.
- **Density** and **Specific Dissipation Rate** are set to **Second Order Upwind**.
- **Momentum**, **Turbulent Kinetic Energy** and **Energy** are all set to **QUICK**.
- **Pseudo Transient** is enabled. This increases the robustness and speed of solution convergence. For more information about this option refer to, Solving Pseudo-Transient Flow in the Fluent User's Guide.

9.2.6. Solution Controls



The **Under-Relaxation Factors** are set as following:

- **Pressure**: `0.5`
- **Momentum**: `0.5`
- **Density**, **Body Forces**, and **Turbulent Viscosity** are all set to `1`.
- **Turbulent Kinetic Energy**, **Specific Dissipation Rate**, and **Energy** are set to `0.75`.
- **Solution Limits** for pressure and temperature are set as shown below:



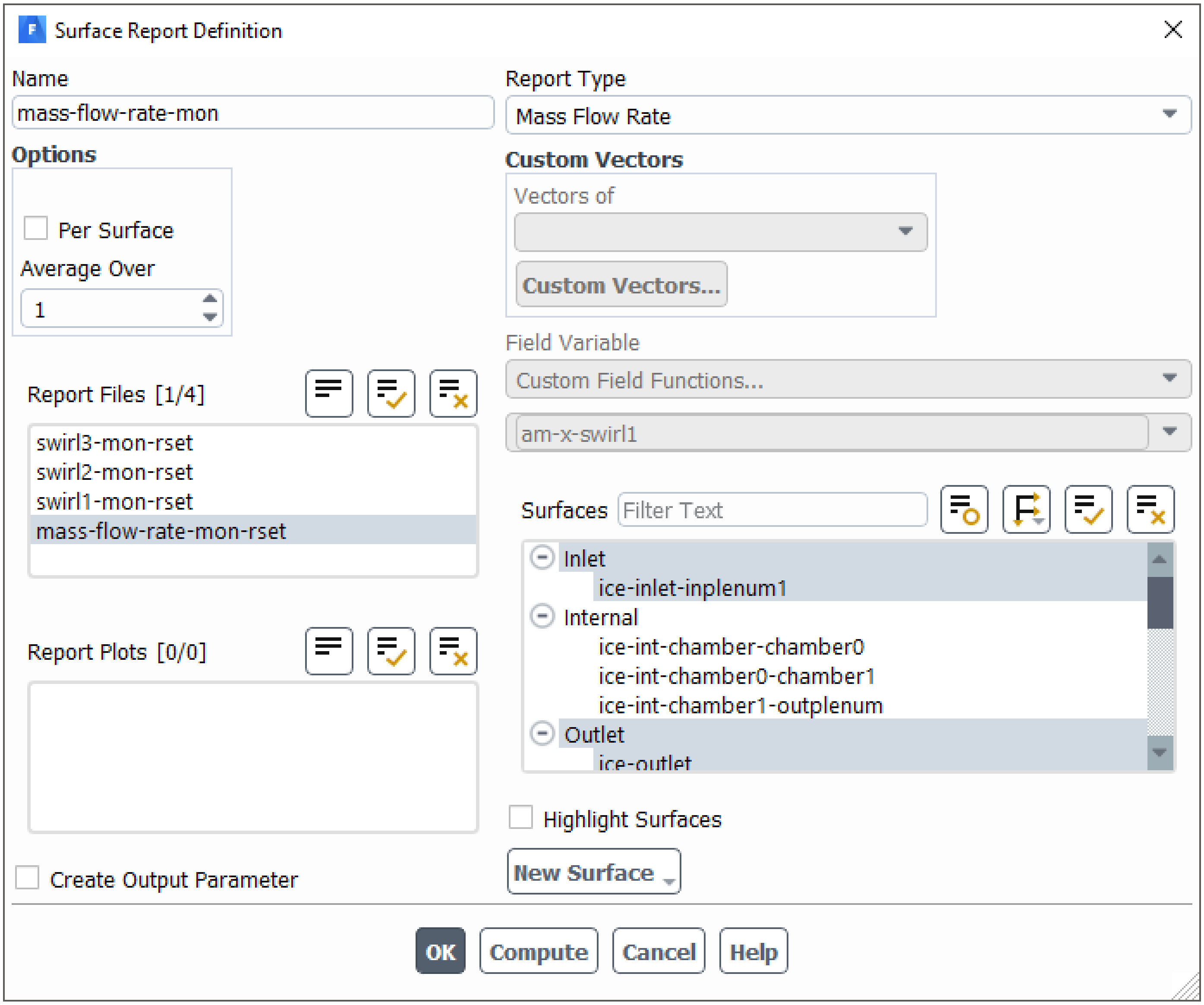
- **Minimum Absolute Pressure (pascal)** is set to `1`.
- **Maximum Absolute Pressure (pascal)** is set to `5e+10`.
- **Minimum Static Temperature (k)** is set to `1`.
- **Maximum Static Temperature (k)** is set to `5000`.

Note:

The temperature and pressure limits are restricted as indicated above because the values of these variables should not cross this specified range in some cells. However, if you are modeling combustion or gas exchange, or if you know that these limits are too aggressive for your case, then you can modify them appropriately.

9.2.7. Monitors

To check the output results of the parameters, some monitors have been created.



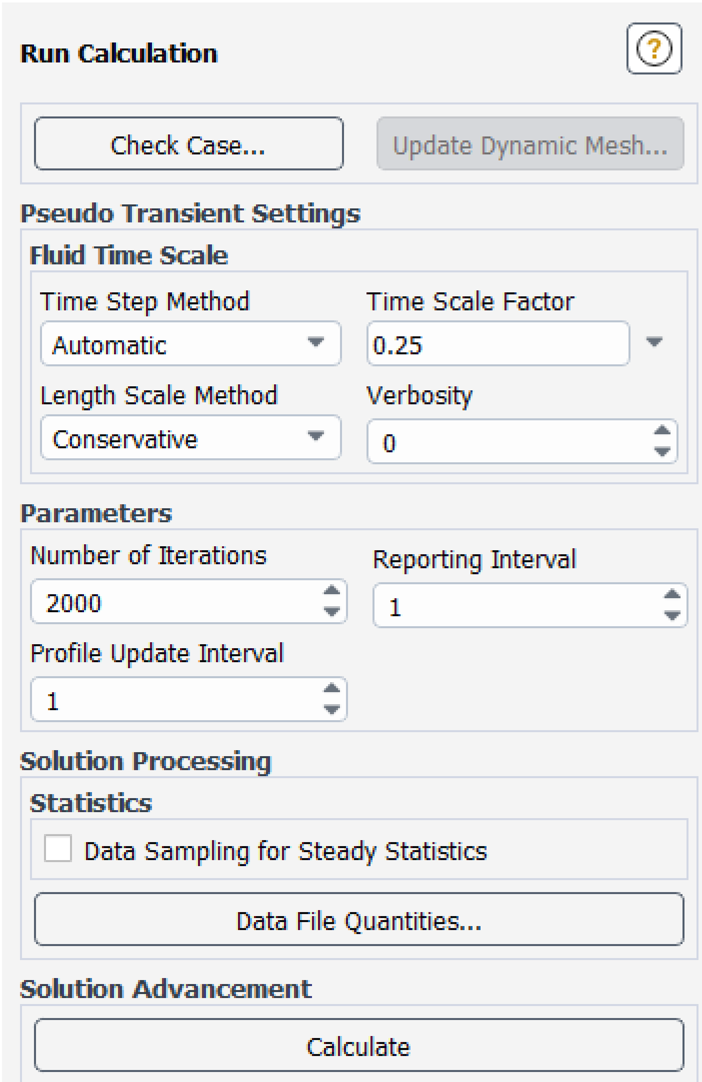
One monitor defined by default is a **Surface Monitor**. It plots the variable **Mass Flow Rate** at the inlet and outlet.

You can create your own **Surface Monitors** or **Volume Monitors** from the **Monitors** task page. Enable **Write** so that an output file is written, which can be loaded and viewed later. You can see the monitor settings that are done by default in the `icBcSettings.txt` file.

9.2.8. Solution Initialization

Initially **Standard Initialization** method is used for initialization and then an FMG initialization is performed.

9.2.9. Run Calculation



Note:

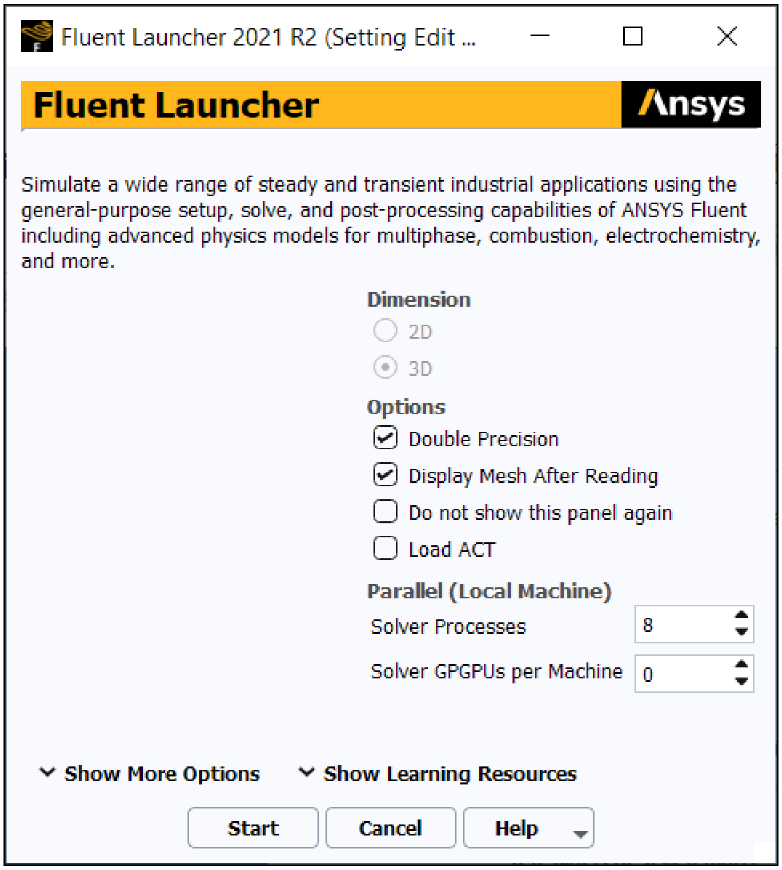
For more information on the settings done in the **Pseudo Transient** group box, refer to Solving Pseudo-Transient Flow in the Fluent User's Guide.

You can enter the **Number of Iterations** you want.

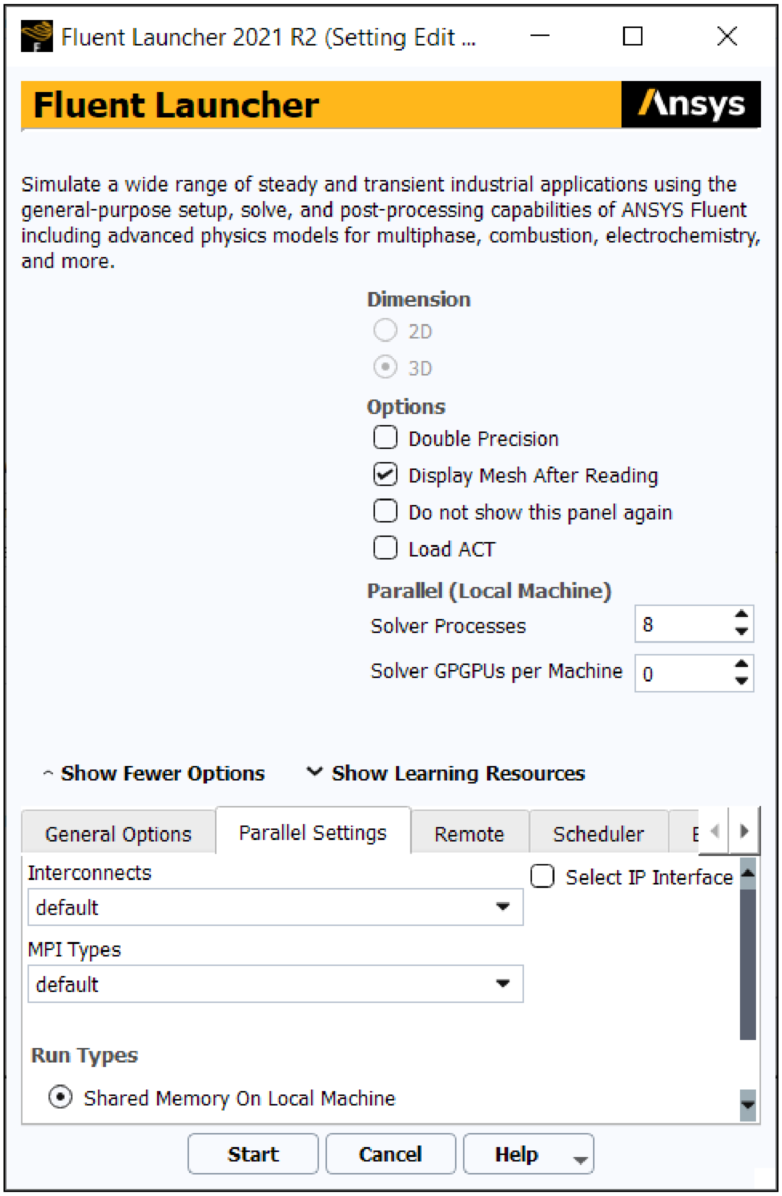
Click **Calculate** to run the simulation.

Running the Calculation from Windows on Remote Linux Machines

To run the simulation on Linux, some changes in the setup are required.



1. Set the number of **Solver Processes** in the **FLUENT Launcher**.
2. Click **Show More Options** to display the tabs.
3. Click the **Parallel Settings** tab.



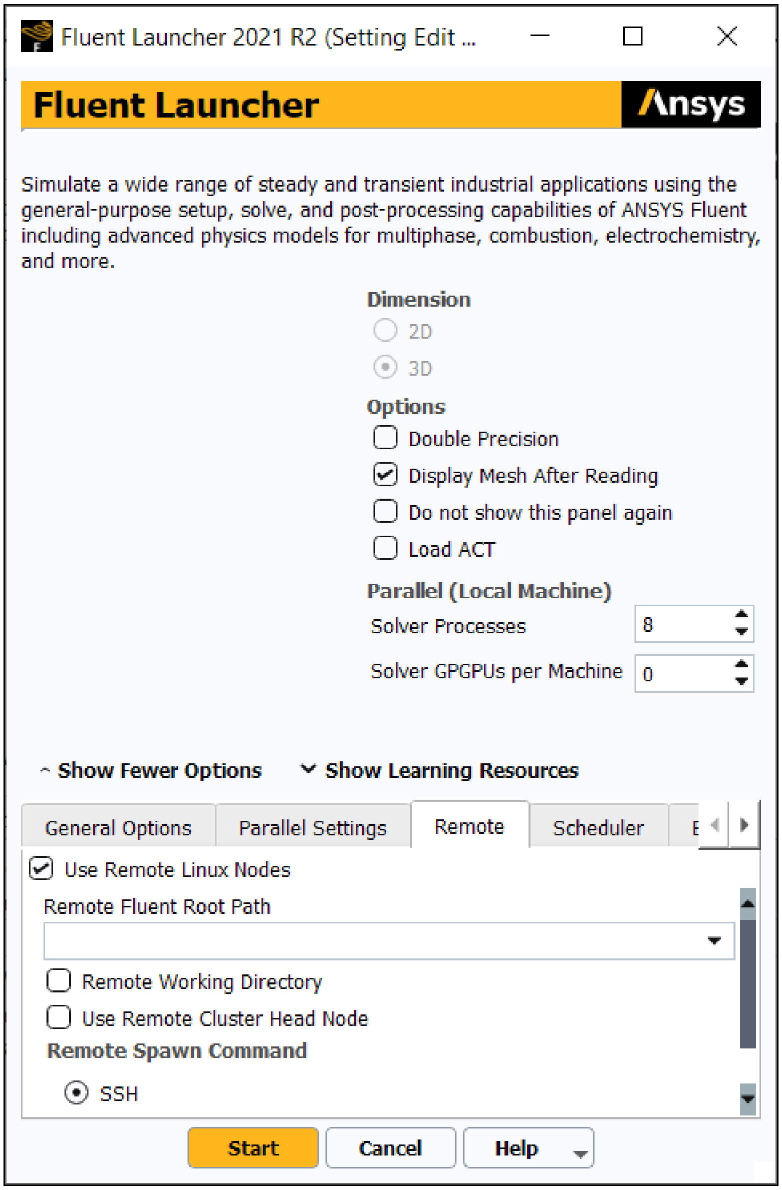
a. Select the **Run Types** from the list.

Note:

If you selected **Distributed Memory on a Cluster**, then you have to provide the **Machine Names**.

---

4. Click the **Remote** tab.



a. Enable **Use Remote Linux Nodes**.

b. Enter the path for **Remote FLUENT Root Path**. It is the root directory of Fluent installation.

c. Select your choice from the list of **Remote Spawn Command**.

> **Note:**
>
> For **SSH**, you need to set up a password-less connection before running the solution remotely.

d. Enable **Use Remote Cluster Head Node** and provide the node.

e. Enter the number for **Solver Processes**.

When the remote connection is set, it will use the nodes of the remote Linux machine. The details are displayed in the Fluent console.

```
Host spawning Node 0 on machine "pundevrhel64r6" (lnamd64).
/usr/local/fluent/develop/fluent14.0.0/bin/fluent -r14.0.0 3d -pdefault -node -t4 -ssh -mport 10.14.6.198:10.14.2.108:2673:0
Starting /usr/local/fluent/develop/fluent14.0.0/multiport/mpi/lnamd64/pcmpi/bin/mpirun -np 4 /usr/local/fluent/develop/fluent14.0.0/l
Platform-MPI licensed for FLUENT.

--------------------------------------------------------------------------------
ID     Comm.   Hostname        O.S.         PID     Mach ID  HW ID   Name
--------------------------------------------------------------------------------
host   net     punptstxp64t1   Windows-x64  177216  1        5       Fluent Host
n3     pcmpi   pundevrhel64r6. Linux-64     17969   0        3       Fluent Node
n2     pcmpi   pundevrhel64r6. Linux-64     17968   0        2       Fluent Node
n1     pcmpi   pundevrhel64r6. Linux-64     17967   0        1       Fluent Node
n0*    pcmpi   pundevrhel64r6. Linux-64     17966   0        0       Fluent Node

Selected system interconnect: shared-memory
```

All the files will be saved in the working directory. Once the solution is completed, the project is updated and you can proceed with your postprocessing.

For details, see Setting Additional Options When Running on Remote Linux Machines in the Fluent User's Guide.

Note:

You can also submit the run on different Linux machines though RSM.

Running on Standalone Fluent

1. Open Fluent and read the case and data file from the `~project-name_files\dp0\ICE\Fluent` directory.

   File → **Read** → **Case & Data...**

   Note:

   If the case is setup in the previously released versions, then you might have to read the scheme file, (`WB-ICE-Solver-Setup.scm`) from `~ANSYS Inc\v150\Addins\ICEngine\CustomizationFiles` folder in the mapped directory before running it in the present version.

   ---

2. In **Run Calculation** task page enter the required **Number of Time Steps** and click **Calculate**.

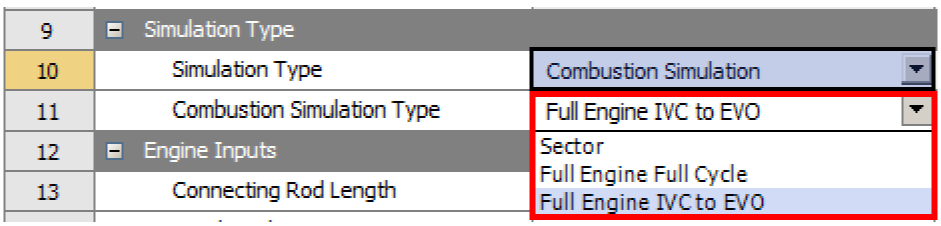
Note:

If you want the generate the report from IC Engine System report template, then you need to first make sure that **Solution** cell is updated. This can be done by running the solution for 1 timestep. Then, copy all the files generated by your standalone run into the Fluent directory and update the **Results** cell.

Chapter 10: Combustion Simulation: Preparing the Geometry in IC Engine

This chapter provides instructions and information about preparing the IC engine geometry for combustion simulations.

For combustion simulation you need to select **Combustion Simulation** from the **Simulation Type** drop-down list in the **Properties** pane.



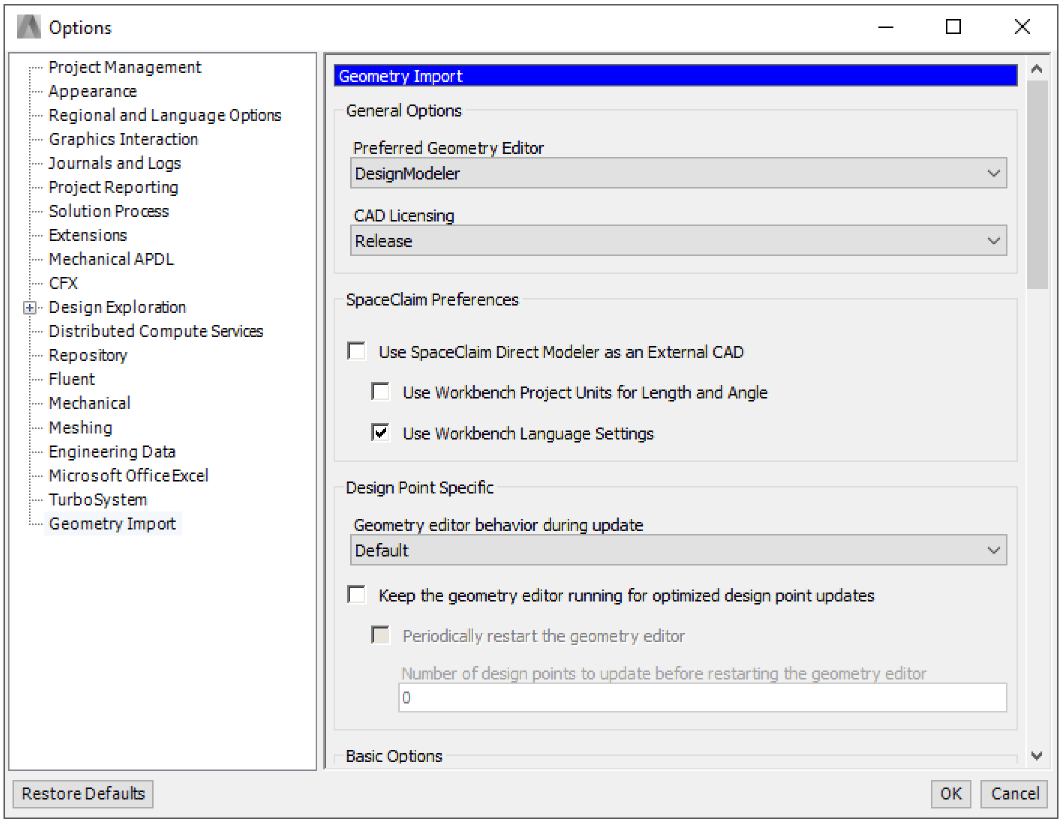
After selection of **Combustion Simulation** you need to select the subtype from the list of:

- **Sector** (Simulation of a sector of an engine),
- **Full Engine Full Cycle** (Simulation of complete engine during complete cycle), or
- **Full Engine IVC to EVO** (Simulation of complete engine during compression phase).

Then you are required to fill all the necessary information about the engine under **Engine Inputs**. After that you can open DesignModeler from **Geometry**, cell 3 in the **IC Engine (Fluent)** analysis system.

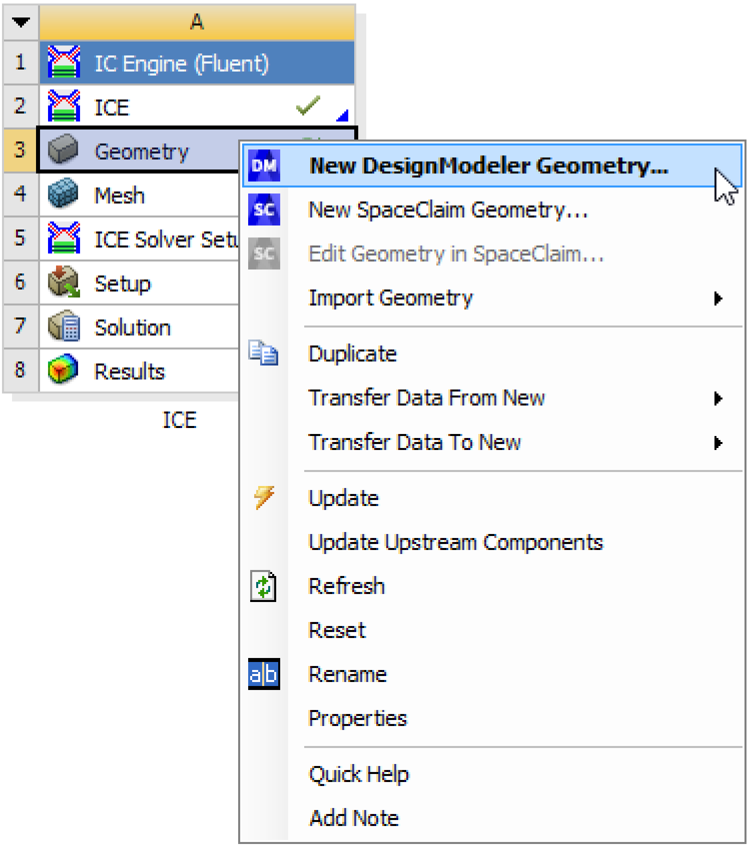
Note:

To open DesignModeler from the **Geometry** cell, you should set the **Preferred Geometry Editor** to **DesignModeler** in the **Geometry Import** section in the **Options** dialog box. The **Options** dialog box can be opened from the **Tools** → **Options...**menu.



Then you can double-click **Geometry**, cell 3, to open DesignModeler.

One more way to open DesignModeler is, to right-click on the **Geometry** cell, and select **New DesignModeler Geometry...** from the context menu.



10.1. Geometry Decomposition for Sector Combustion Simulation

Note:

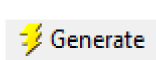
It is a good practice to check the geometry before decomposing; there are a few things that you should verify so that fewer problems occur later on. For more information, see Geometry Check (p. 517).

Selection of **Sector** is recommended for engines with axisymmetric piston. The shape of the dome geometry or the valve seats is not important as they will be trimmed later. It is also recommended for CI (compression ignition) engines as they do not contain a spark plug geometry. **Sector** selection can be used for SI engines if the spark plug is defined as a single point and is not represented in the geometry.

1. Set the units, depending upon the geometry units, in ICE-DesignModeler.

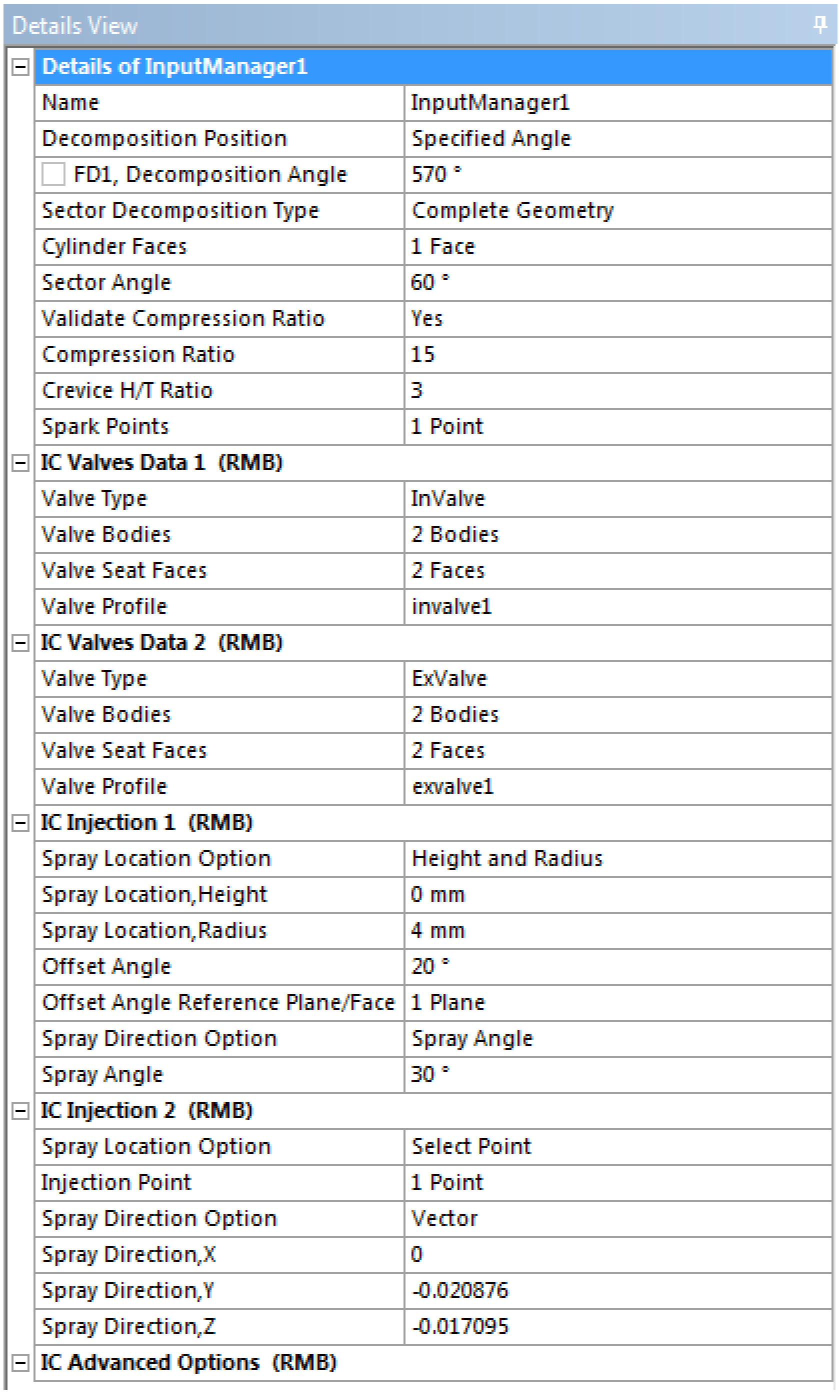
2. Load the geometry file.

   File → Import External Geometry File...

3. Click **Generate**  to complete the import.

4. Set up the **Input Manager** by clicking **Input Manager** (Input Manager located in the **IC Engine** toolbar).

> **Note:**
>
> The **IC Engine** toolbar is displayed in the ICE DesignModeler only after installing IC Engine Analysis System.



Details View	
Details of InputManager1	
Name	InputManager1
Decomposition Position	Specified Angle
FD1, Decomposition Angle	570 °
Sector Decomposition Type	Complete Geometry
Cylinder Faces	1 Face
Sector Angle	60 °
Validate Compression Ratio	Yes
Compression Ratio	15
Crevice H/T Ratio	3
Spark Points	1 Point
IC Valves Data 1 (RMB)	
Valve Type	InValve
Valve Bodies	2 Bodies
Valve Seat Faces	2 Faces
Valve Profile	invalve1
IC Valves Data 2 (RMB)	
Valve Type	ExValve
Valve Bodies	2 Bodies
Valve Seat Faces	2 Faces
Valve Profile	exvalve1
IC Injection 1 (RMB)	
Spray Location Option	Height and Radius
Spray Location,Height	0 mm
Spray Location,Radius	4 mm
Offset Angle	20 °
Offset Angle Reference Plane/Face	1 Plane
Spray Direction Option	Spray Angle
Spray Angle	30 °
IC Injection 2 (RMB)	
Spray Location Option	Select Point
Injection Point	1 Point
Spray Direction Option	Vector
Spray Direction,X	0
Spray Direction,Y	-0.020876
Spray Direction,Z	-0.017095
IC Advanced Options (RMB)	

Details of InputManager

This section in the **Input Manager** dialog box takes inputs of the engine to set it up for decomposition.

- For **Decomposition Position** you can select the angle at which you want the geometry to be decomposed. You can choose from:
  - **Specified Angle**: If you choose this option then you can specify the particular angle at which you want to decompose the geometry.
  - You can set the angle at which you want the geometry to be decomposed by entering the value in **Decomposition Angle**. It is set to **0** by default. When you want the solver to start the simulation from a specific crank angle you can do so by changing the **Decomposition Angle**. If you enable **FD1** next to it, a new parameter will be created.

    Note:

    Setting **Decomposition Angle** to a value will move the piston and valves to the specified crank angle before decomposition. For details, see Moving the Piston to a Specified Crank Angle (p. 183).

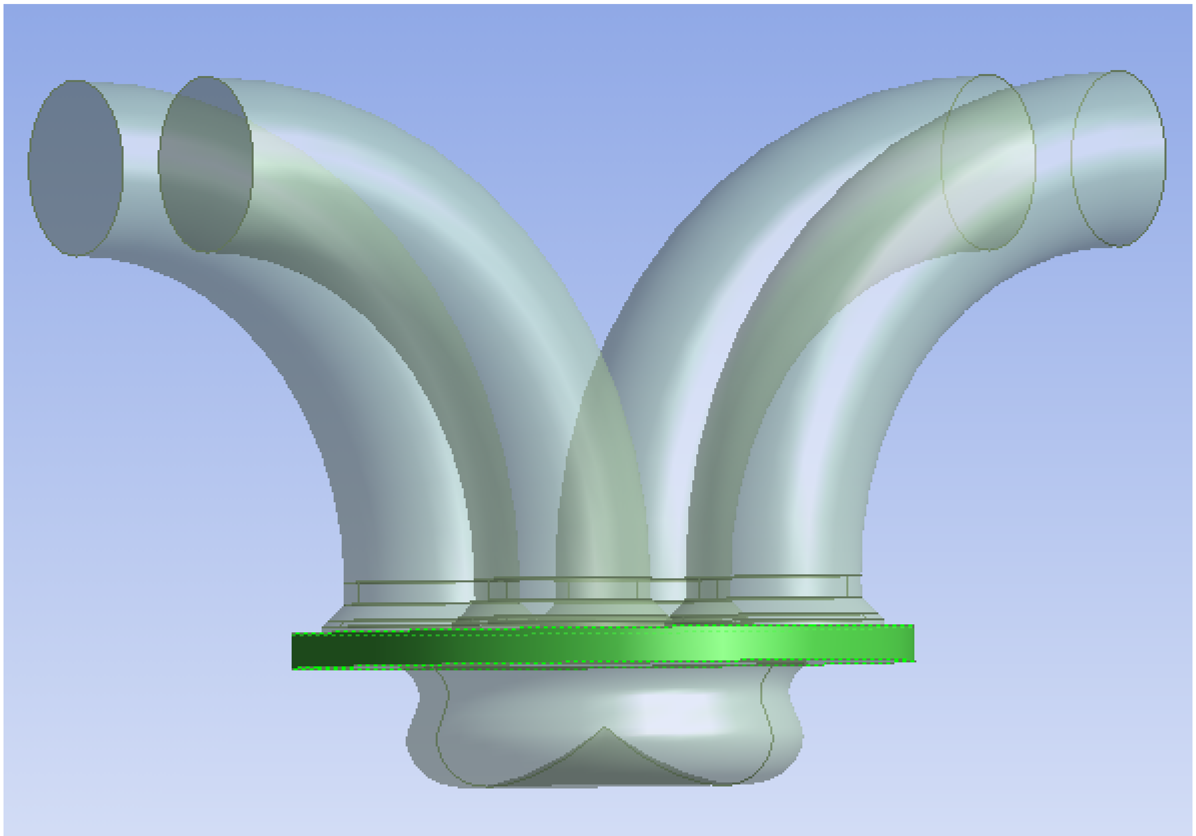
  - **FTDC**: This is the firing top dead center angle.
  - **IVC**: This is the inlet valve close angle.
- From the **Sector Decomposition Type** drop-down list, you can select
  - **Complete Geometry**, or
  - **Sector Geometry**

  Complete Geometry is selected when the input geometry is a complete geometry, including one flow volume and solid valves.

  Sector Geometry is selected when the input geometry is a sector.

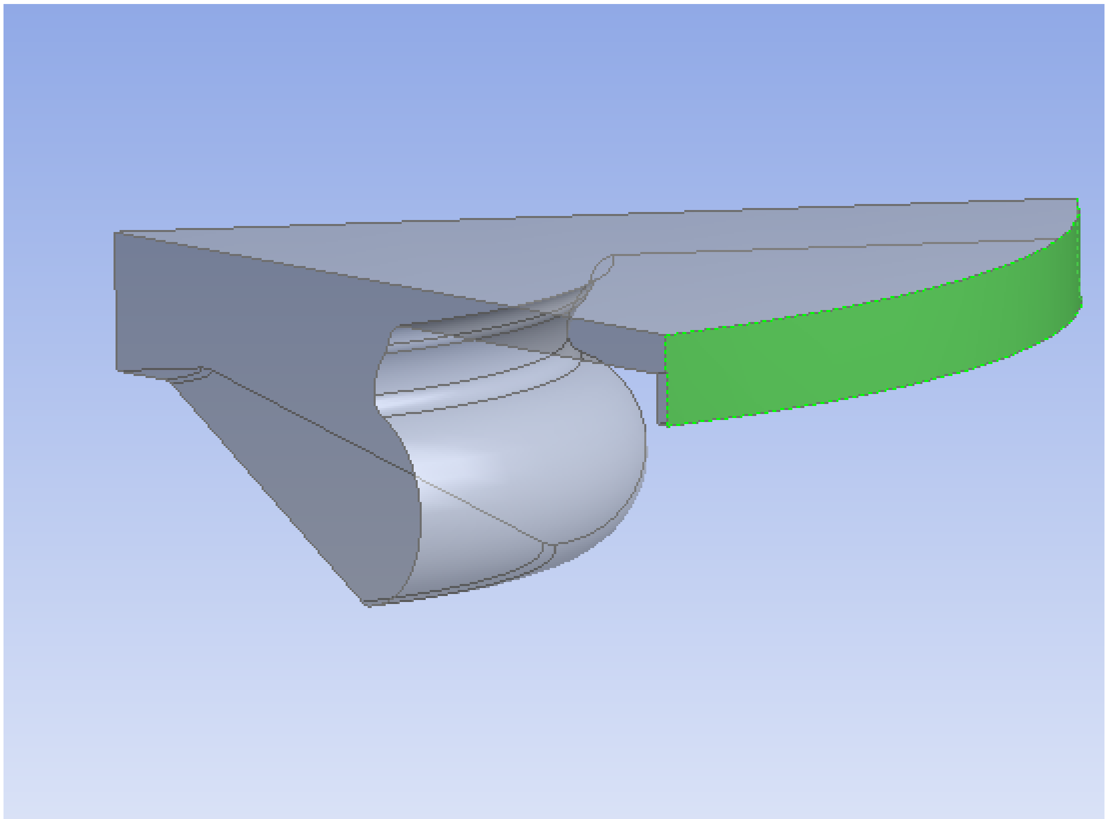
  Depending upon your selection further options will be available.
- Select all the faces of the engine cylinder for **Cylinder Liner Faces** and click **Apply**.

Figure 10.1: Cylinder Face Selection for a Complete Geometry

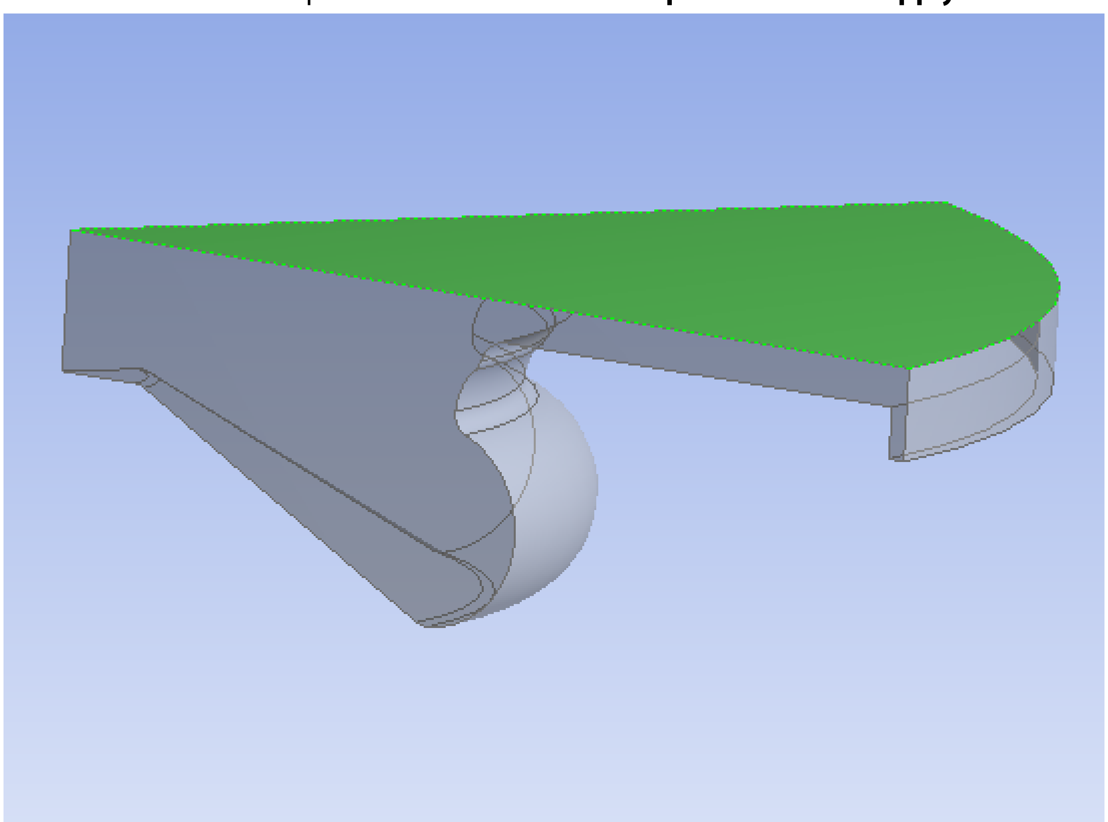


The cylinder radius and the cylinder axis are displayed in the status bar at the bottom of the window.

Figure 10.2: Cylinder Face Selection for a Sector Geometry

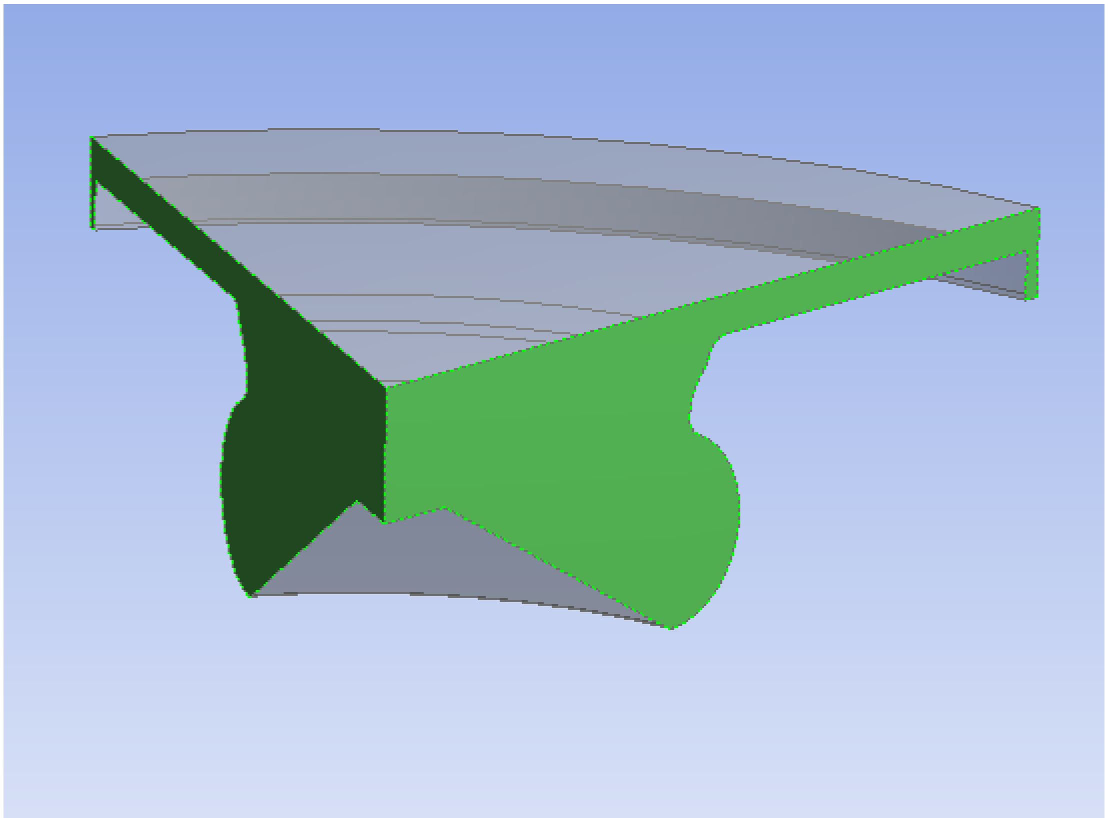


- Select the face at the top of the sector for **Sector Top Face** and click **Apply**.



This option is present only when you select **Sector Geometry** from **Sector Decomposition Type** drop-down list.

- Select the faces at the sides of the sector for **Sector Periodic Faces** and click **Apply**.



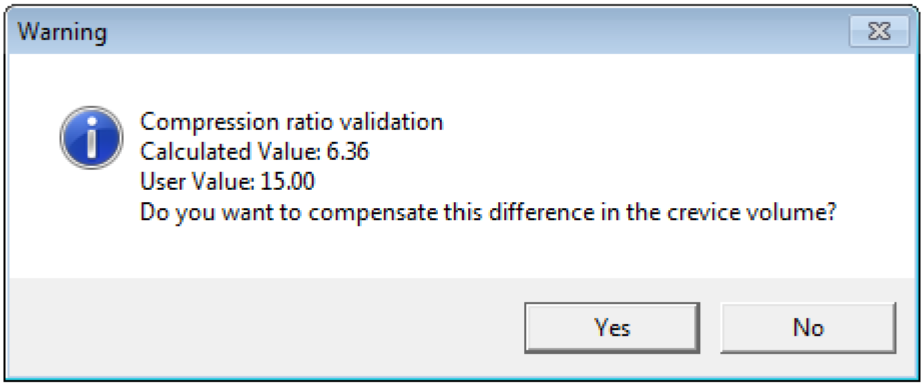
This option is also present only when you select **Sector Geometry** from **Sector Decomposition Type** drop-down list.

- Select **Yes** or **No** for **Crevice Option** depending up on whether the imported geometry has a crevice or not. If a crevice is present then the tool will calculate the appropriate sizes for meshing the crevice. This option is present only if you select **Sector Geometry** from **Sector Decomposition Type** drop-down list in the **Input Manager** or if you select **Full Engine Full Cycle** or **Full Engine IVC to EVO** from the **Combustion Simulation Type** drop down list in the **Properties** dialog box

- Enter the angle of the sector to be formed for **Sector Angle**.

- You can select **Yes** or **No** from the **Validate Compression Ratio** drop-down list. If you select **Yes** then a **Compression Ratio** option appears below it.

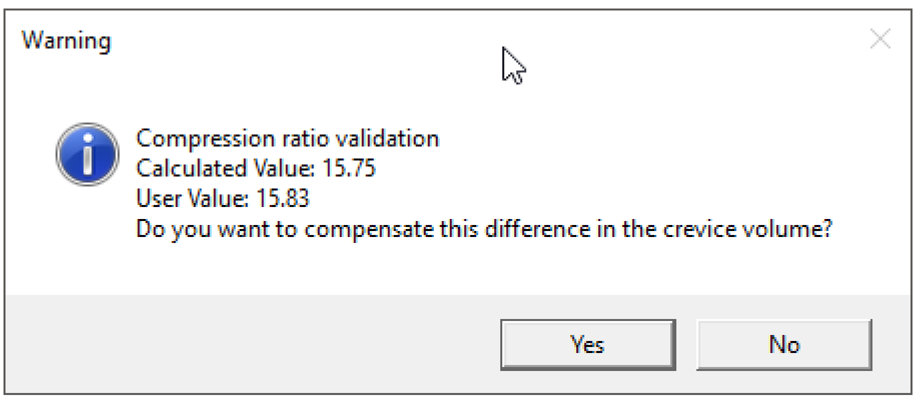
- Enter the value for **Compression Ratio**. By default it is set to **15**. The program will internally calculate the compression ratio from the geometry and give a warning if the values do not match.



Note:

For details on how the compression ratio is calculated internally see Calculating Compression Ratio (p. 590).

The warning will also allow you to compensate for the difference in the compression ratio by clicking **Yes**. The program will try to increase the crevice volume for a lower user value of compression ratio and vice versa. But if the user value is too high and the crevice volume reduction is calculated in negative volume then another warning is given and no correction is made.



- **Crevice H/T Ratio** option is present when you select **Complete Geometry** from the **Sector Decomposition Type** drop down list. After decomposition some dome bodies are deleted. To maintain the clearance volume at TDC a crevice is added at the outer edge of the sector. Here you can specify the height to thickness ratio of the crevice which will be created. By default it is set to **3**.

- You can select points on the geometry in the graphic window for the **Spark Points** option.

IC Valves Data1

Note:

This option is not present only when you select **Sector** from **Sector Decomposition Type** drop-down list.

In this section you are required to give details about the valves.

- Select **InValve** or **ExValve** as the **Valve Type** from the drop-down list.
- Select the valves from the figure and click **Apply** next to **Valve Bodies**.

Note:

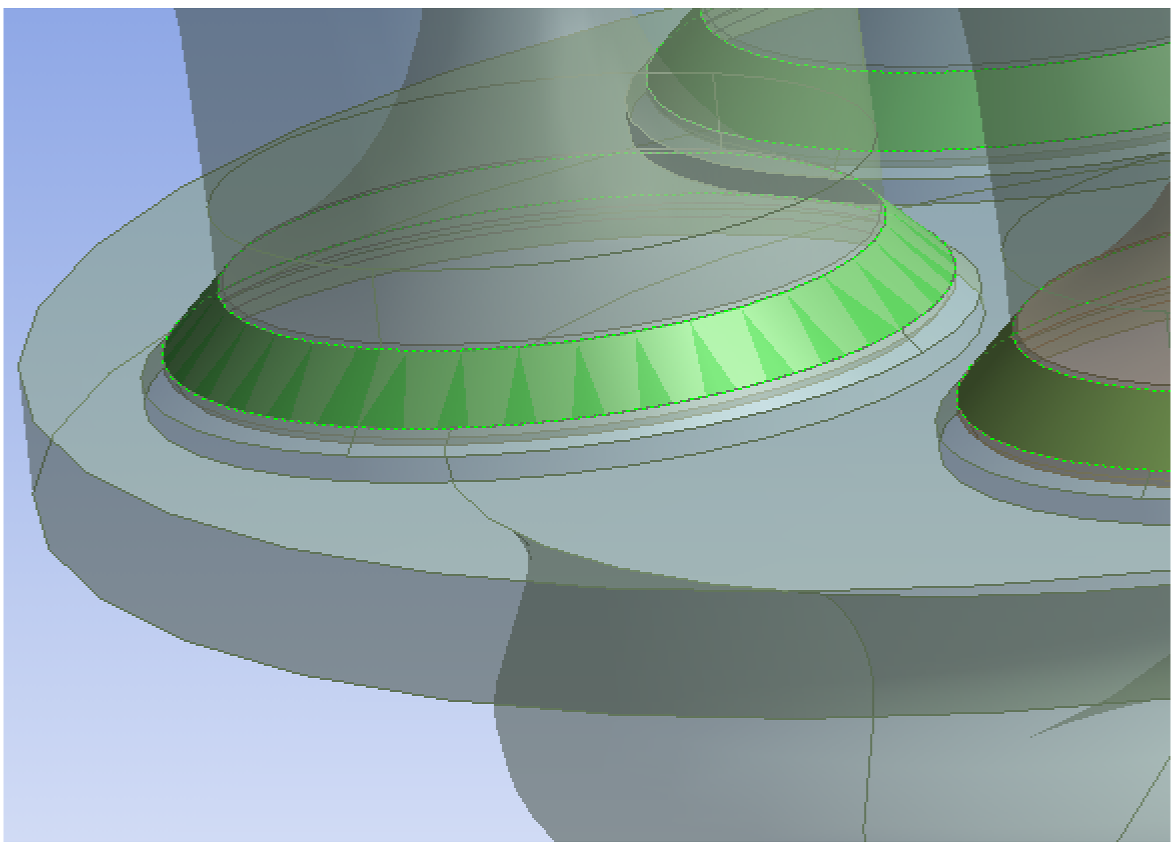
– You can select multiple valves at a time and set them to the desired type.

– Confirm that **Selection Filters** are set to **Bodies**.

– Remember the order of selection of the valve bodies. This is important especially while using meshes from another setup for the KeyGrid option.

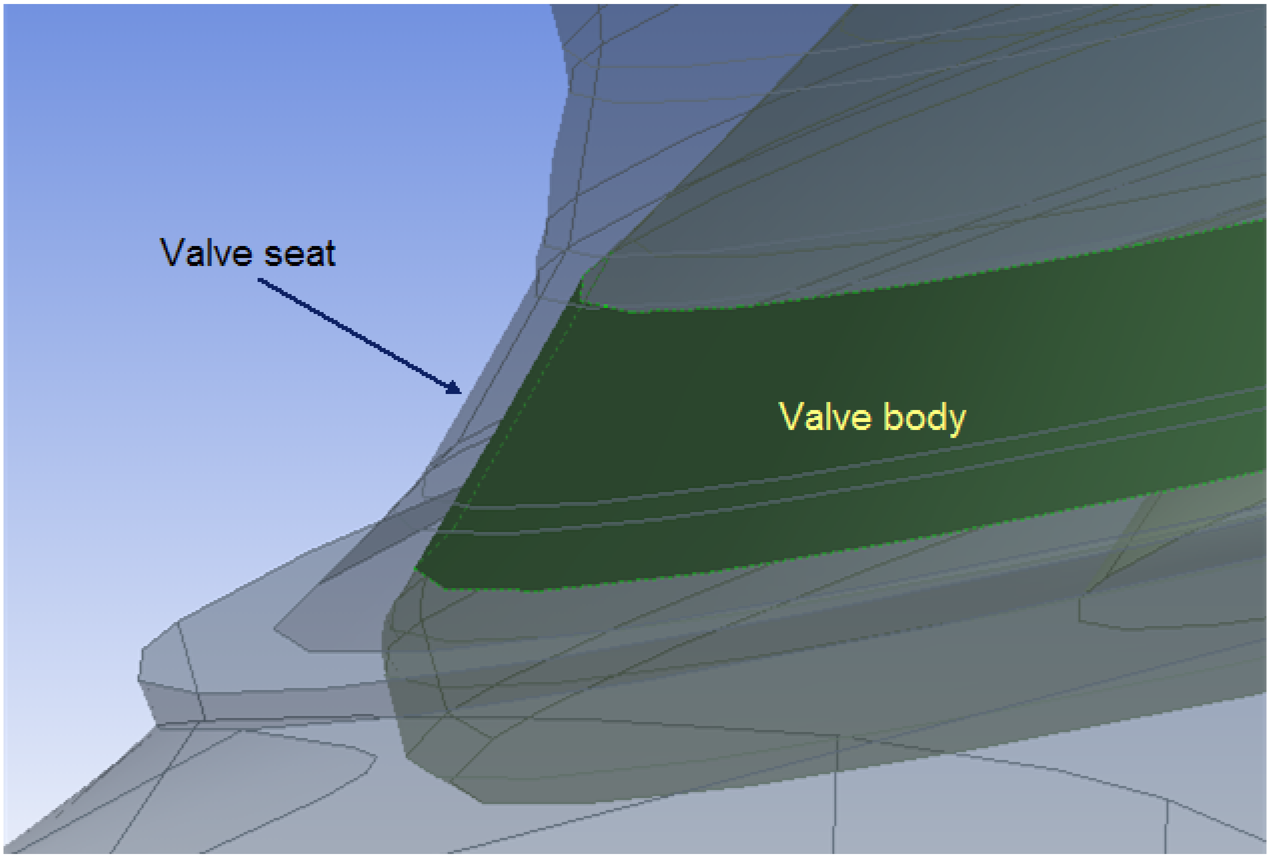
- Select the faces on which the valves rest for **Valve Seat Faces** and click **Apply**.

Figure 10.3: Valve Seats



The valve seat is that face of the port body which comes in contact with the valve body. The Figure 10.4: Valve Seat Selection (p. 351) shows that the highlighted face of the valve body will make contact with the valve seat face of the port body.

Figure 10.4: Valve Seat Selection



Note:

Confirm that **Selection Filters** are set to **Model Faces**. Some engines might have line contact between the valve and the valve seat. In such cases you can select the face which is near the valve and includes the line of contact.

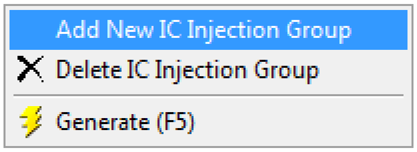
- Select the valve from the **Valve Profile** drop-down list. This will set that particular valve to the selected valve profile.

Note:

The profile file (**Valve Lift and Piston Motion Profile**) (p. 146) contains a table of crank angle against the valve lift for different valves. Each table type has a name. The names are loaded in the **Valve Profile** drop-down list. This term is valid only when you select **By Lift Curve Profile** for **Input option for IVC and EVO** in the **Properties** pane.

IC Injection 1

In this section you are required to give details about the injection. If you choose **Multiple** from the **No. of Injection Rows** drop-down list, then you can add injection groups by right-clicking on **IC Injection 1** and selecting **Add New IC Injection Group**.



- For **Spray Location Option** you can choose from either of:
  - **Height and Radius**
  - **Coordinates**
  - **Select Point**
  - **Beam Origin, Footprint**: This will be available only for **Full Engine Full Cycle**

  from the drop-down list. After selection of each, different options are available.

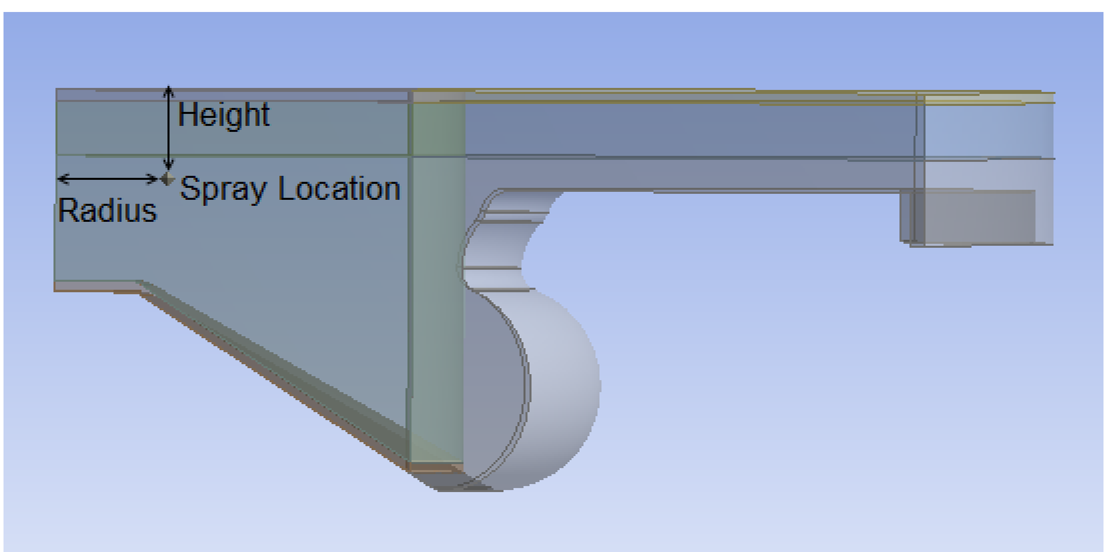
  - When you choose **Height and Radius** option, you will have to provide the following inputs.

Spray Location Option	Height and Radius
Spray Location,Height	4 mm
Spray Location,Radius	3 mm

→ **Spray Location, Height**: The location of the spray in terms of its height from the chamber top

→ **Spray Location, Radius**: The distance of the spray from the cylinder axis.

Figure 10.5: Spray Location from Height and Radius



→ Enter the value for the **Offset Angle**, which is the angle for which you would like to form a sector which can be further used for solving the simulation. It should ideally be (360 degrees / Number of nozzles). Internal calculations are done assuming you have entered the sector angle accordingly. This option is present only when you select **Complete Geometry** from **Sector Decomposition Type** drop-down list.

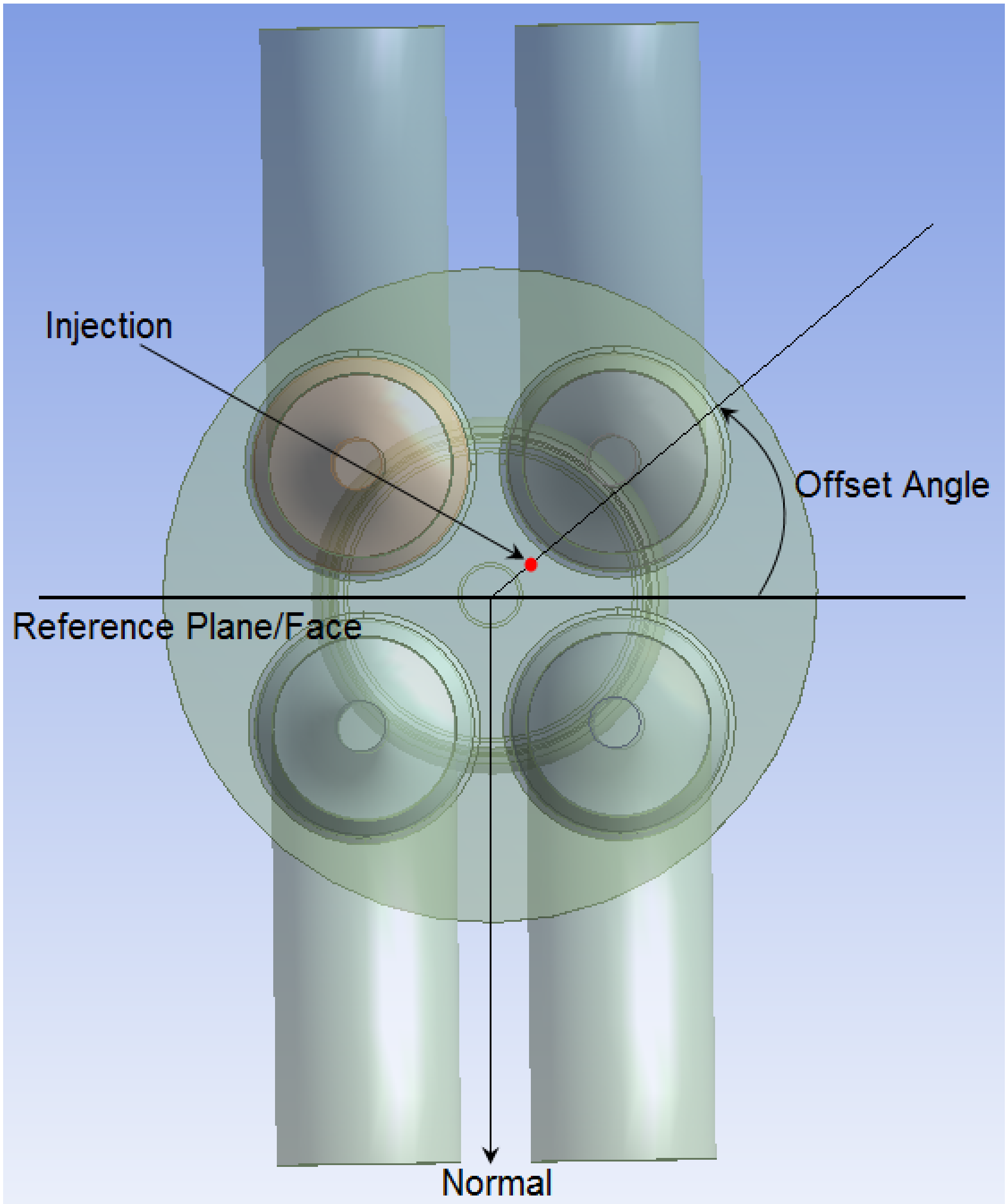
→ You can select a plane or a face as the **Offset Angle Reference Plane/Face**. This option will be present only when you have selected **Height and Radius** from the **Spray Location Option** drop-down list and more than one injection group is present. This reference plane should contain the cylinder axis. The injection positions are given with reference to this plane. If you do not select a face or plane, then the **Offset Angle Reference Plane/Face** will be auto created during decomposition.

Note:

When a sector is imported then one of the **Sector Periodic Faces** will be selected as the **Offset Angle Reference Plane/Face** if you have not selected any.

For multiple groups of **Height and Radius** the **Offset Angle Reference Plane/Face** will be the same. So you can select in any one group and it will be reflected in the rest.

→ **Offset Angle** gives the location of the injection from the **Reference Plane/Face**.



As shown in the figure, the **Offset Angle** is calculated in an anti-clockwise direction from the **Reference Plane/Face**, in the opposite direction of the normal. Generally, the normal is the Z axis in the local coordinate system. So the Y axis is on the **Reference Plane/Face**. Thus, the **Offset Angle** is the angle of the injection from the Y axis of the local coordinate system. This angle is also known as the azimuthal angle. This term is present only if **Multiple** is chosen for **No. of Injection Rows**.

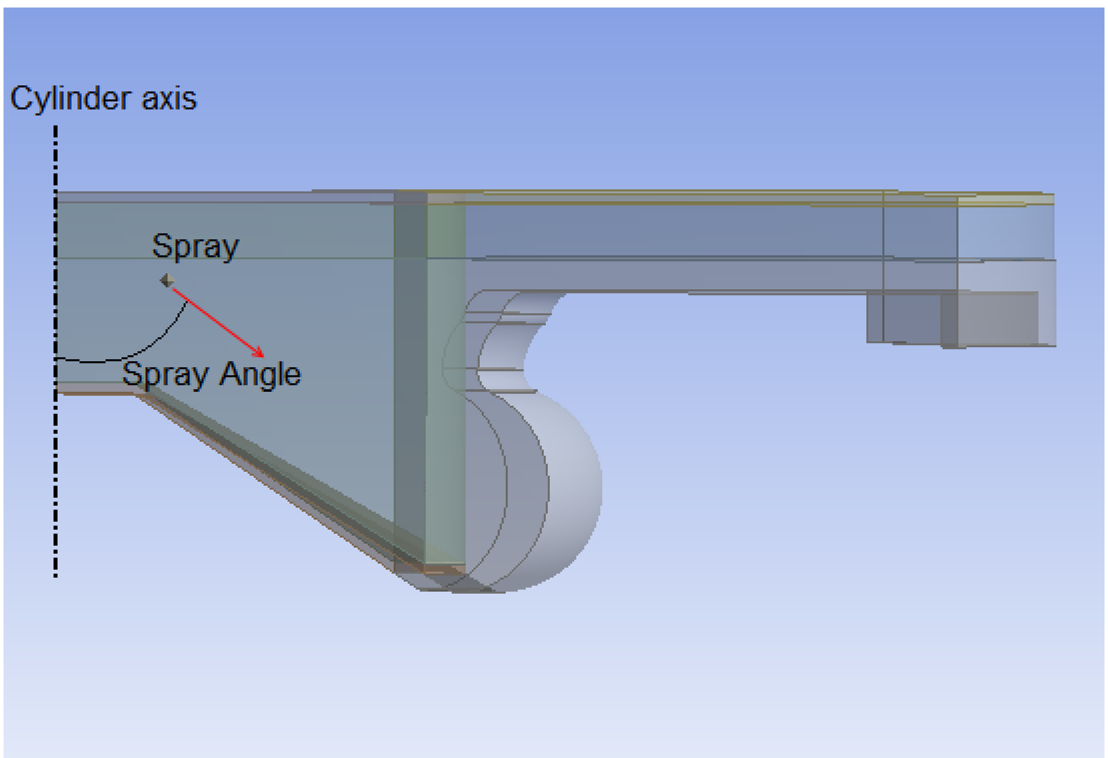
- When you choose **Coordinates** option, you will have to provide the location of the spray in terms of coordinates.

Spray Location Option	Cordinates
Spray Location,X	-2 mm
Spray Location,Y	-2 mm
Spray Location,Z	-2 mm

- **Spray Location, X**
- **Spray Location, Y**
- **Spray Location, Z**

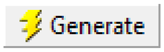
- For **Select Point** you can select the point in the geometry.
- You can choose **Spray Angle** or **Vector** for **Spray Direction Option**. The **Spray Angle** is the angle between the direction of spray and the cylinder axis. If you select and enter the **Spray Angle** then direction vector is automatically calculated.

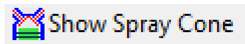
Figure 10.6: Spray Angle



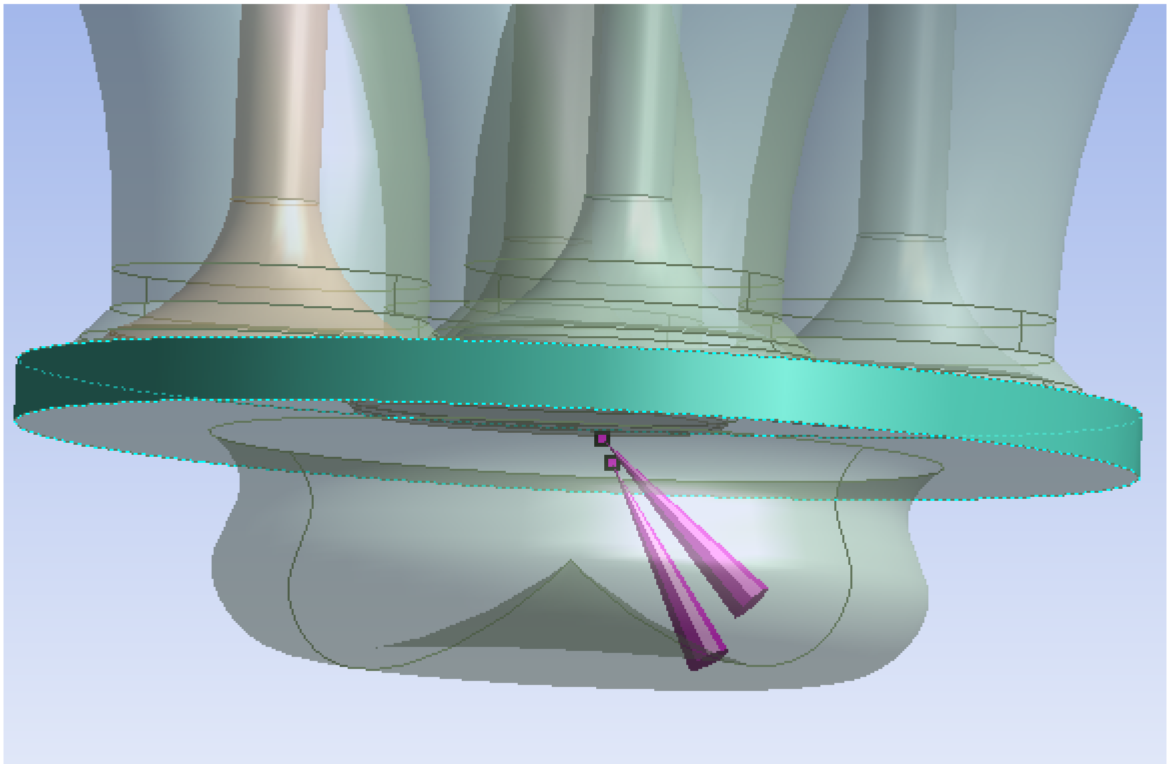
You can also provide the spray direction in terms of **Vector**.

Spray Direction Option	Vector
Spray Direction,X	1
Spray Direction,Y	1
Spray Direction,Z	1

To observe the spray position and cone in the graphics window after entering the values click **Generate** 

and then click the button **Show Spray Cone** 

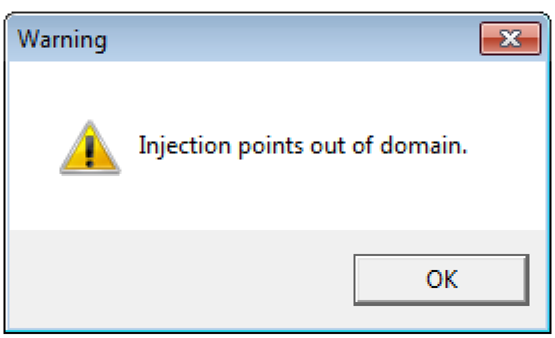
on the **IC Engine** toolbar. This button is a toggle button.



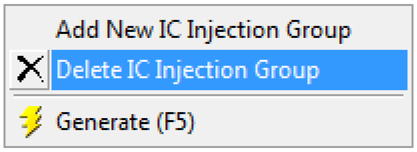
After decomposition injection planes are created, which pass through the injection. You can use these to observe and plot variables.

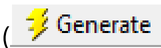
Note:

All the injections provided should lie within the sector domain. You will get a warning if the program calculates that the injections you entered are outside the domain.

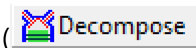


If your simulation does not include injections then delete the **IC Injection (RMB)** group.

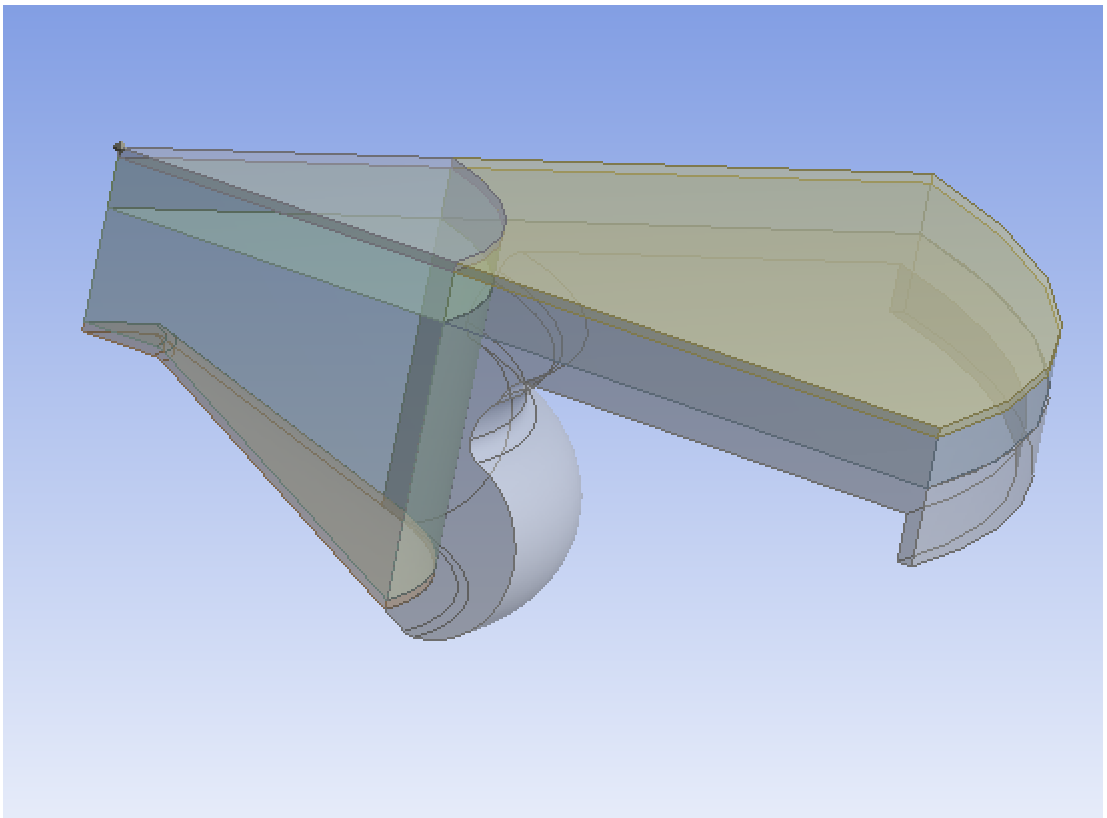


Click **Generate** (

located in the Ansys DesignModeler toolbar).

5. Click **Decompose** (

located in the **IC Engine** toolbar).

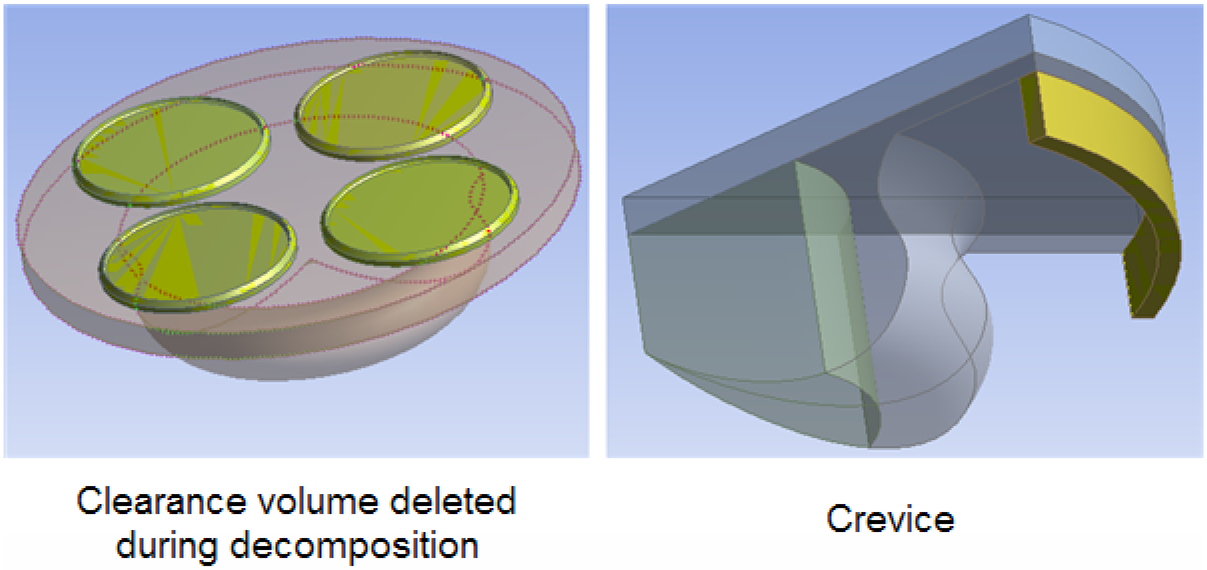


6. Close the Ansys DesignModeler.

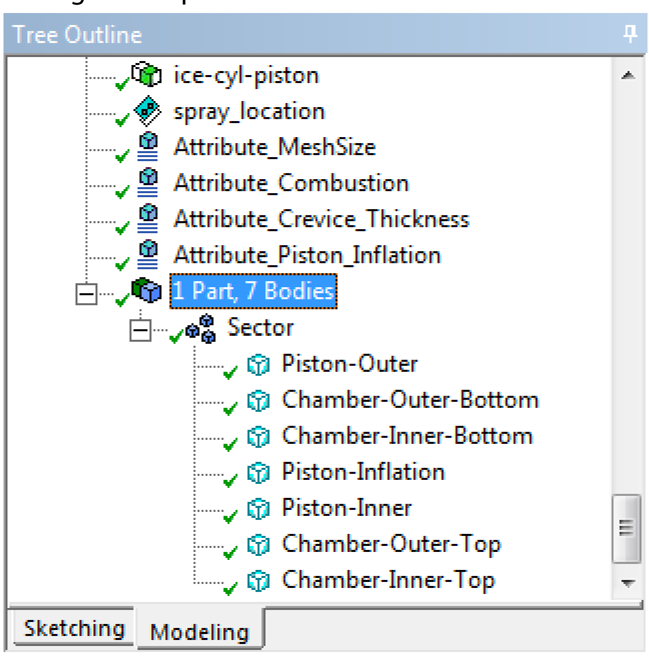
10.2. Viewing the Bodies and Parts

After decomposition the engine is divided into one part, **Port** and several bodies. Some dome bodies are deleted during decomposition of the geometry into a sector. To maintain the clearance volume at TDC a crevice is added at the outer edge of the sector.

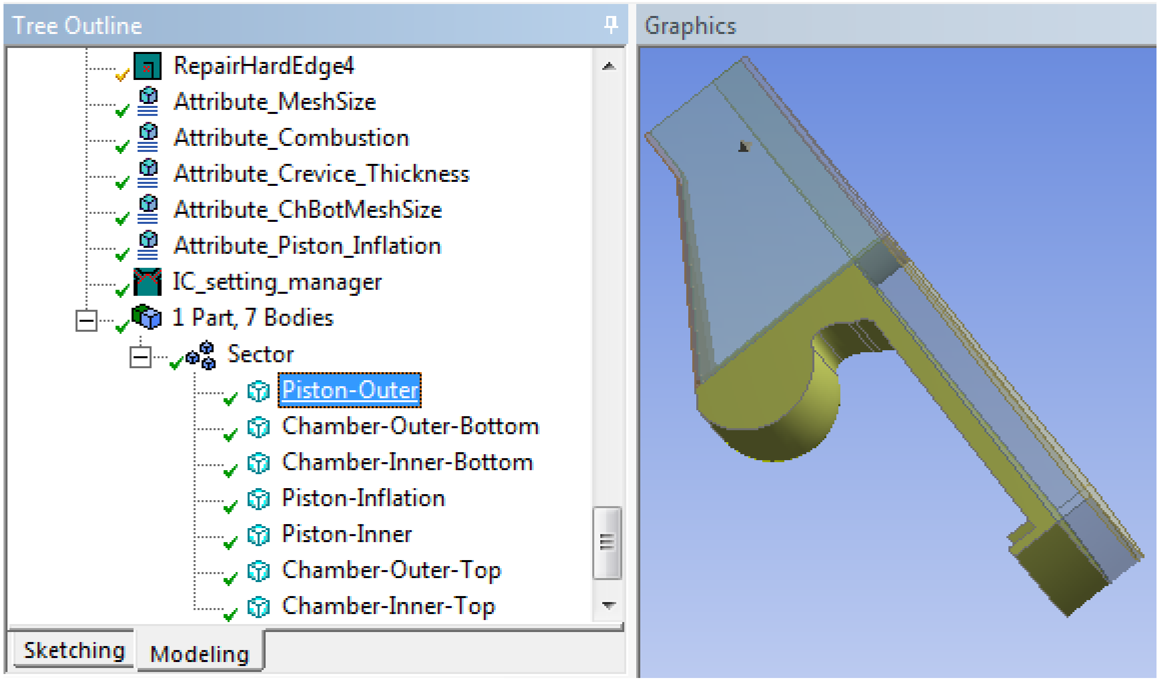
Figure 10.7: Crevice Formation



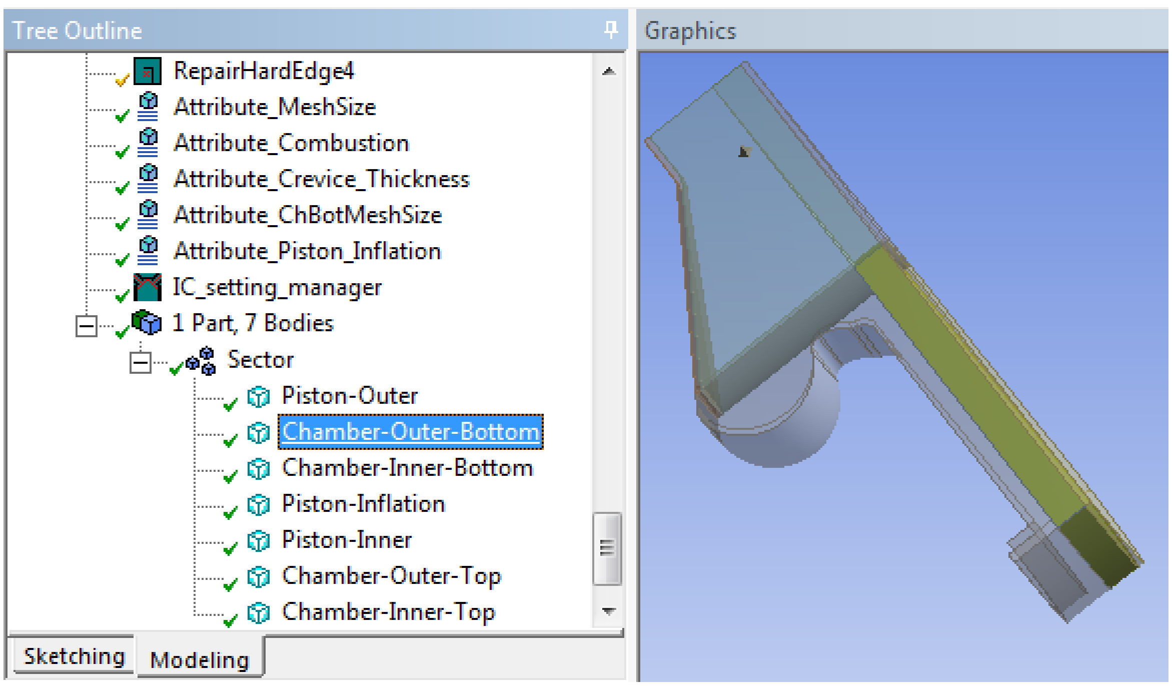
The volume of the crevice is equal to the volume of the dome bodies or the clearance volume deleted during decomposition.



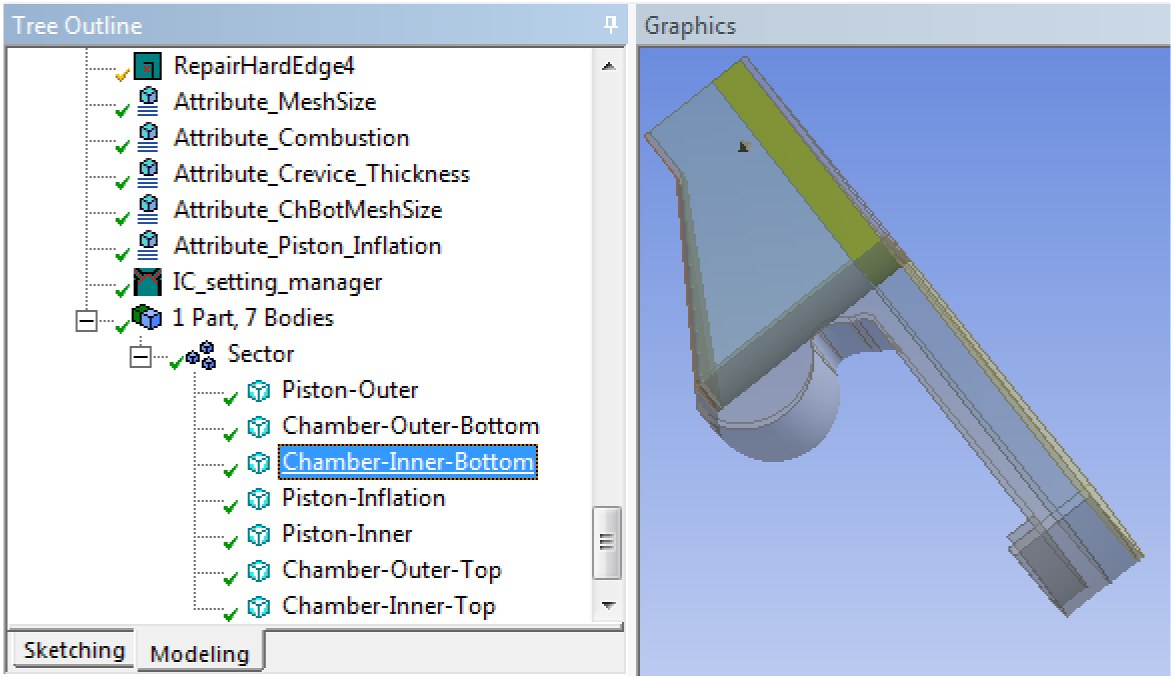
- **Piston-Outer**: Body created at the outer edge of the sector. It includes the crevice.



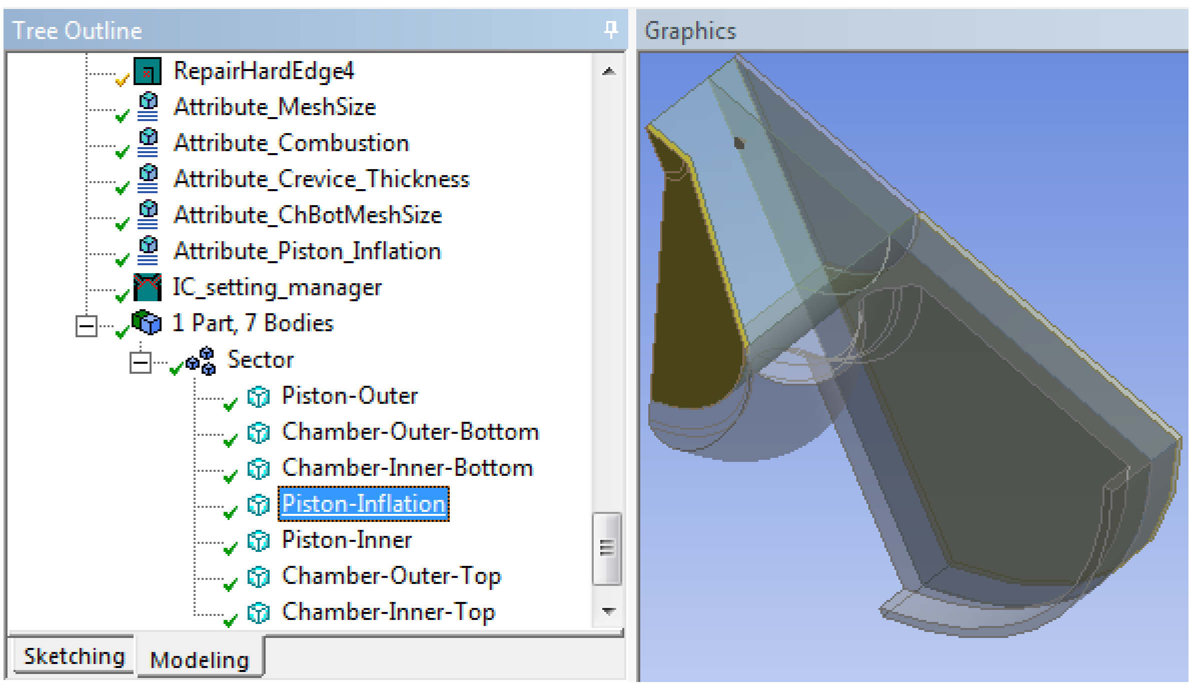
- **Chamber-Outer-Bottom**: Its height is equal to (Mesh Size/3).This body will have layered mesh.



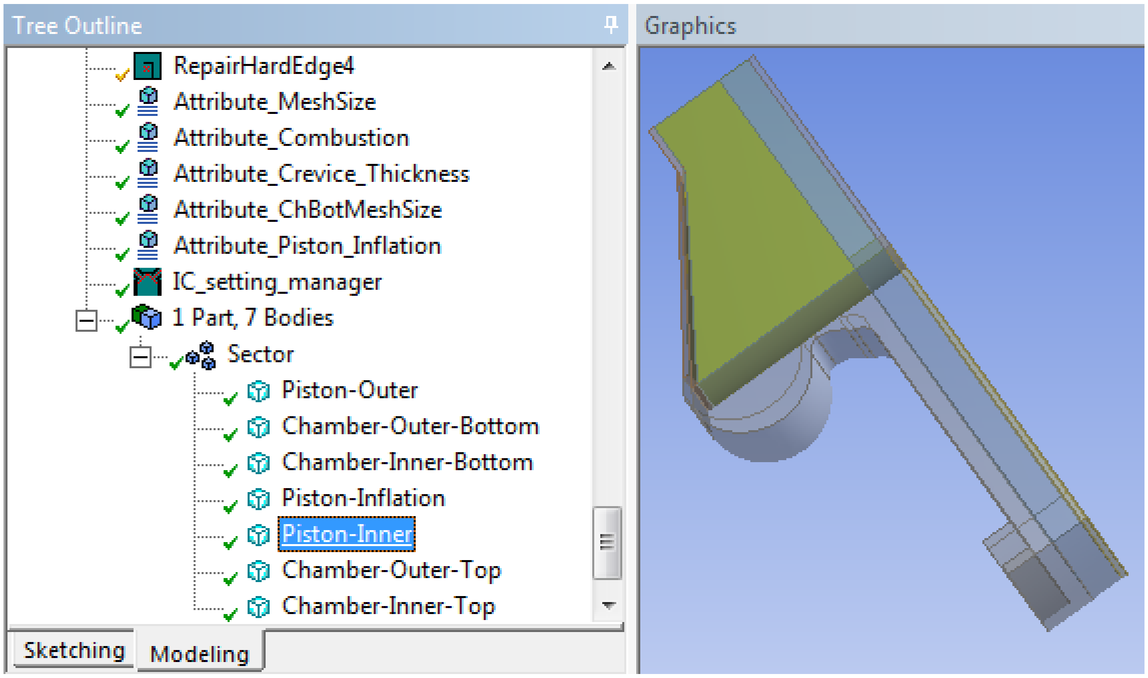
- **Chamber-Inner-Bottom**: Its height is equal to (Mesh Size/3).This body will have layered mesh.



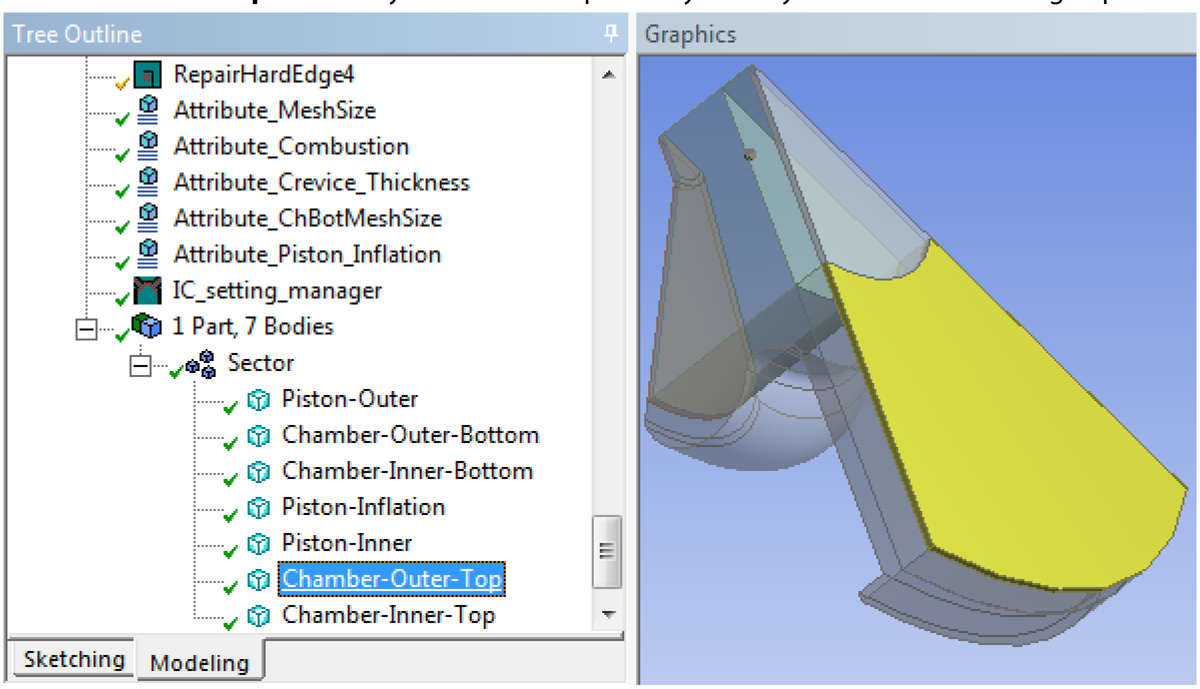
- **Piston-Inflation**: This body is created for prism layers.



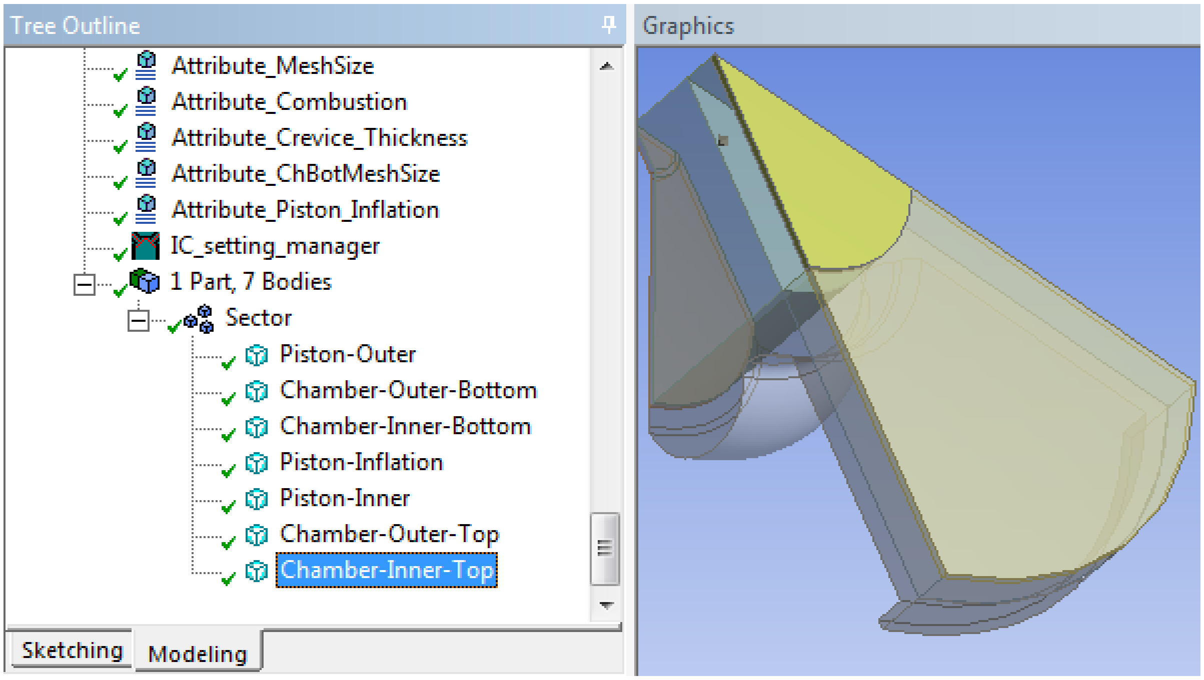
- **Piston-Inner**: The radius of this body is equal to 0.4 times the cylinder radius.



- **Chamber-Outer-Top**: This body is created for prism layers only when there is enough space.



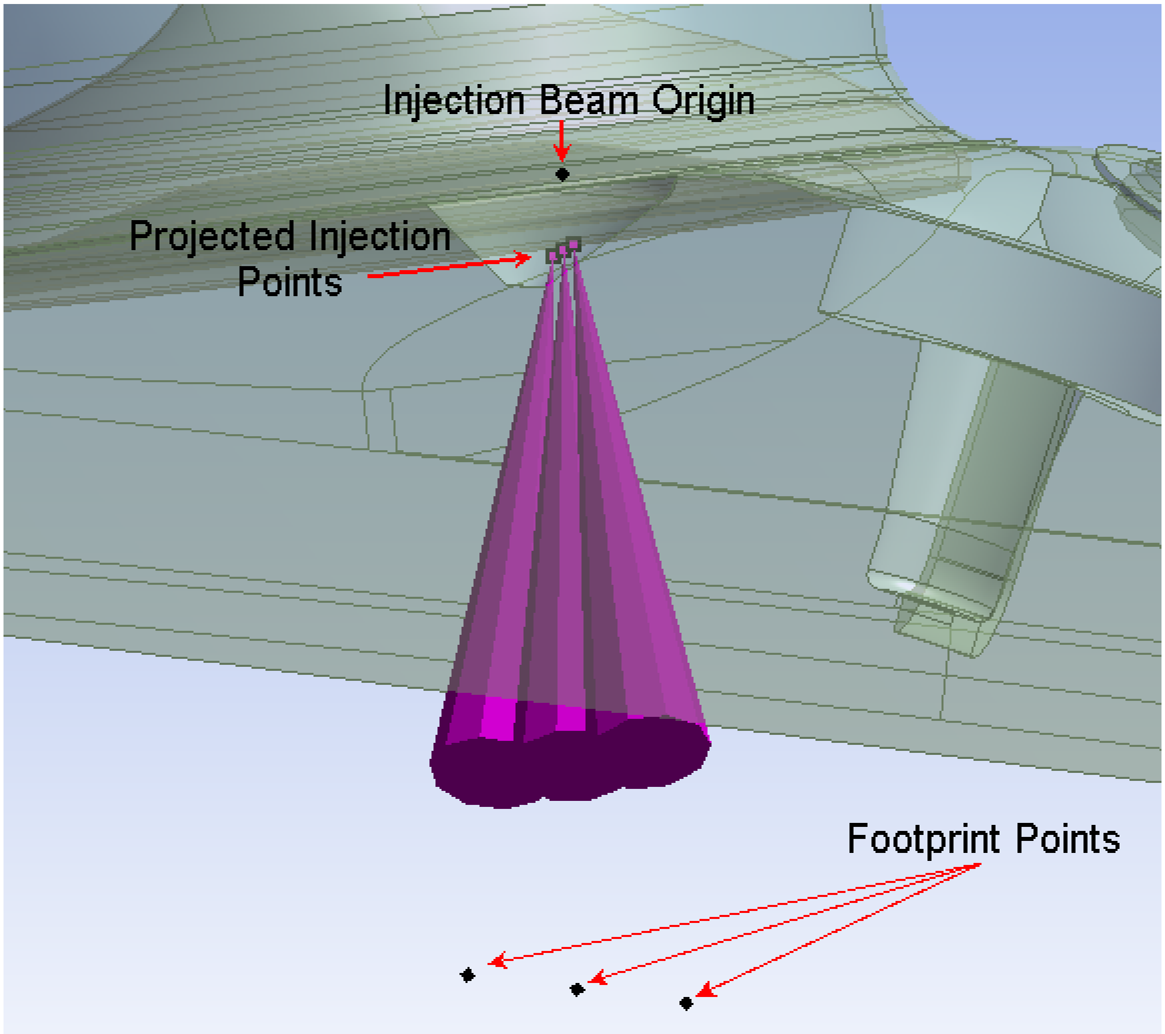
- **Chamber-Inner-Top**: This body is created for prism layers only when there is enough space.



10.3. Geometry Decomposition for Full Engine Full Cycle

When **Full Engine Full Cycle** option is chosen from the **Combustion Simulation Type** drop-down list the decomposition and meshing process is same as for a cold flow simulation. The injection inputs taken are similar to the **Sector Decomposition Type**.

For **Spray Location Option** one more choice is available — **Beam Origin, Footprint**. For this option you have to select two or more points. These points should be defined before invoking the **InputManager**.



- **Injection Beam Origin**: It is assumed that all the spay beams start at one point. This is the **Injection Beam Origin**. This is the single point from which multiple injections points are assumed to start. It is required for the projection of the **Footprint Points** on the chamber body.

- **Footprint Points**: These points are the footprints of spray on any plane. It is required to define spray axis. If **Injection Beam Origin** point is outside the domain, multiple injection points are projected on cylinder head along the direction of footprint points. If the **Injection Beam Origin** point is inside the chamber then it will be taken as the spray location without projection.

  The **Footprint Points** have to be created relative to the **Injection Beam Origin** point. You have to create a coordinate system while importing a footprint file. You can use a local plane or a global plane to import **Footprint Points**.

  From these points the spray location and direction will be transferred to further components.

The term, **No. of Holes per Injection Row** is added for **Full Engine Full Cycle** and **Full Engine IVC to EVO** in the **Input Manager**.

IC Injection 1 (RMB)	
Spray Location Option	Height and Radius
Spray Location,Height	2 mm
Spray Location,Radius	3 mm
Offset Angle	25 °
Offset Angle Reference Plane/Face	1 Plane
No. of Holes per Injection Row	3
Spray Direction Option	Spray Angle
Spray Angle	30 °

Since it is expected for a sector to have only one injection per injection row this term is not present for **Sector Combustion Simulation**. This term decides the number of injections in a row and is used to find the periodic angle. Periodic angle is the angle between two consecutive injections in the same row.

For more information about the other terms in **Input Manager** see Cold Flow Simulation: Preparing the Geometry (p. 149). For terms related to injection see Geometry Decomposition for Sector Combustion Simulation (p. 343). For information related to meshing see Cold Flow Simulation: Meshing (p. 185).

10.4. Geometry Decomposition for Full Engine IVC to EVO

When **Full Engine IVC to EVO** option is chosen from the **Combustion Simulation Type** drop-down list the decomposition process is similar to that for a cold flow simulation. The only difference is that valve and port regions are removed. So only the chamber portion of the geometry remains. For more information see the Chamber (p. 179) section in the Cold Flow Simulation: Preparing the Geometry (p. 149) chapter.

The injection inputs taken are similar to the **Full Engine Full Cycle Decomposition Type**, see Geometry Decomposition for Full Engine Full Cycle (p. 362). For more information about the other terms in **Input Manager** see Cold Flow Simulation: Preparing the Geometry (p. 149). For terms related to injection see Geometry Decomposition for Sector Combustion Simulation (p. 343). For information related to meshing see Cold Flow Simulation: Meshing (p. 185).

Chapter 11: Combustion Simulation: Meshing in IC Engine

The decomposed geometry is used to generate the mesh. The goal of the IC Engine meshing tool is to minimize the effort required to generate a mesh for the IC Engine specific solver. It uses the named selection created in the decomposition to identify different zones and creates the required mesh controls. Since you can choose different types for combustion simulation the meshing process for each type also differs.

- This chapter explains the meshing process when **Sector** is chosen from the **Combustion Simulation Type** (p. 143) drop-down list.
- When **Full Engine Full Cycle** is chosen from the **Combustion Simulation Type** (p. 143) drop-down list, the decomposition (p. 149) and meshing (p. 185) process is the same as for cold flow simulation.
- When **Full Engine IVC to EVO** is chosen from the **Combustion Simulation Type** (p. 143) drop-down list, the decomposition is similar as for cold flow simulation. The valve and port regions are removed. Only the chamber portion of the geometry is retained. Meshing process therefore has slight differences from the cold flow meshing process. The differences will be explained in Meshing for Full Engine IVC to EVO Combustion Simulation (p. 389) section.

The information in this chapter is divided into the following sections:

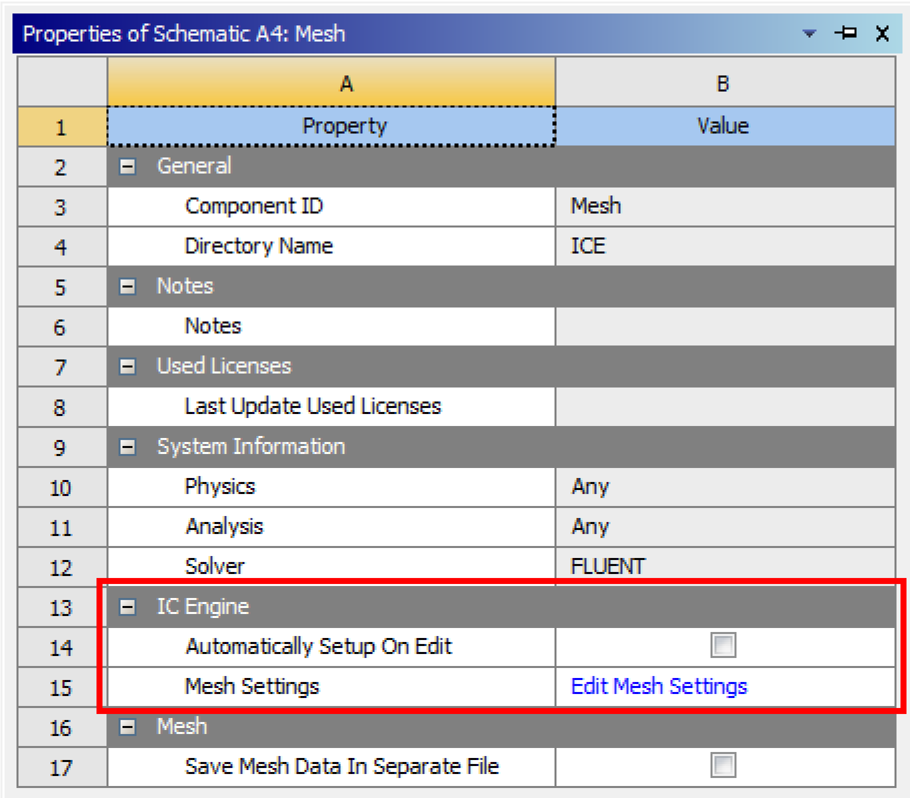
11.1. Meshing Procedure for Sector Combustion Simulation

There are two ways to generate the mesh.

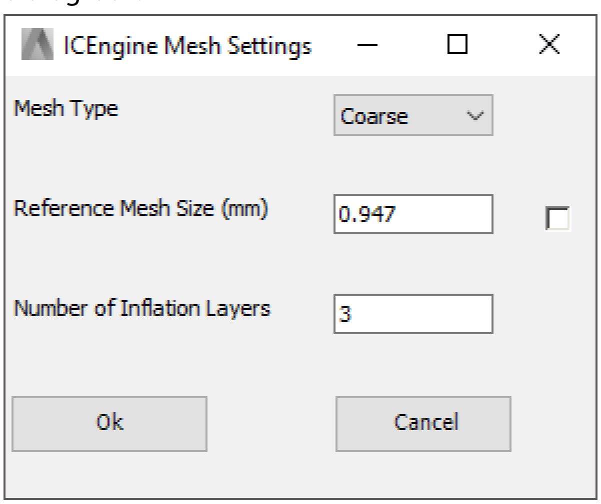
Meshing directly from the Workbench Window

After decomposition is done you can then directly generate the mesh from the Workbench window without opening the Ansys Meshing application. If you want to use the default mesh settings then right-click on the **Mesh** cell and select **Update** from the context menu. This will first create the mesh controls and then generate the mesh.

You can also change the mesh settings from the Workbench window.



In the **Properties** box which is displayed after selecting the **Mesh** cell you can click on **Edit Mesh Settings** next to the **Mesh Settings** property, under **IC Engine**. This will open a **ICEngine Mesh Settings** dialog box.



You can change the settings for the parameters:

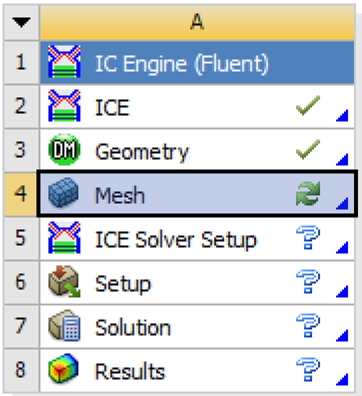
- **Mesh Type**
- **Reference Mesh Size (mm)** (This can be parametrized by enabling the check box next to it.)
- **Number of Inflation Layers**

For more information on these check the section on **IC Mesh Parameters** (p. 187).

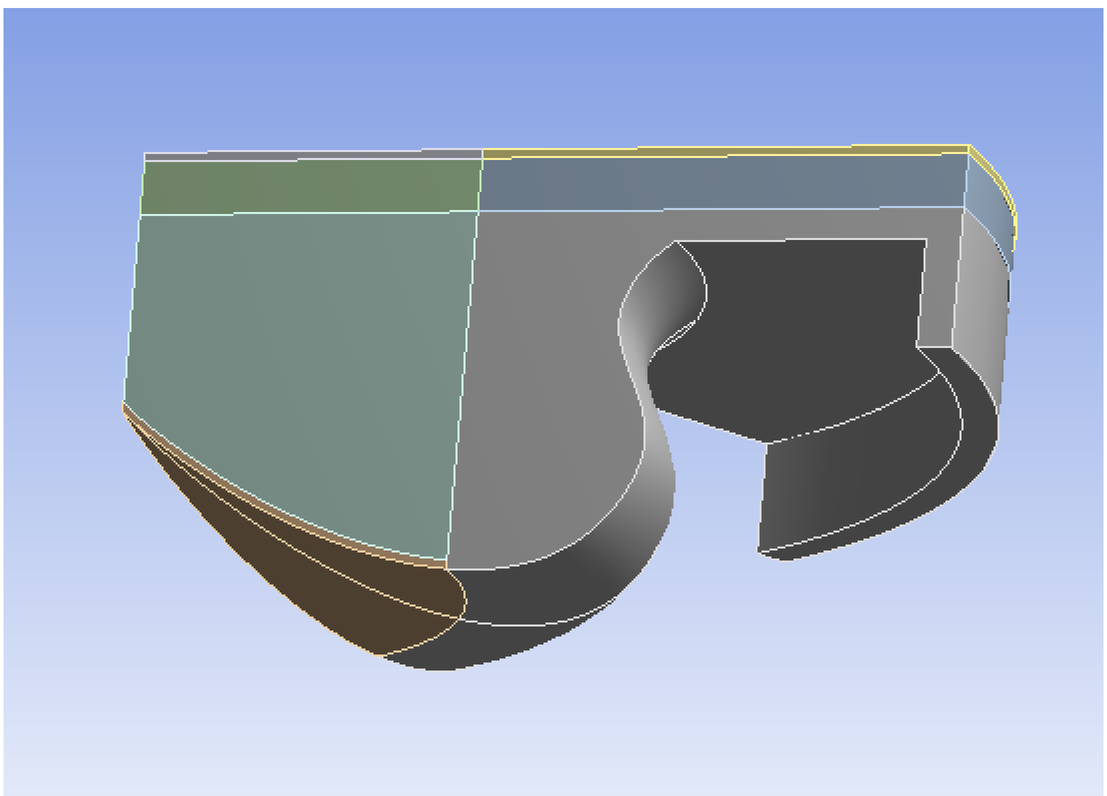
Meshing using Ansys Meshing Application

In this method of meshing you can check the mesh controls and settings in details.

1. Double-click the **Mesh** cell in the **IC Engine** analysis system to open the Ansys Meshing application.



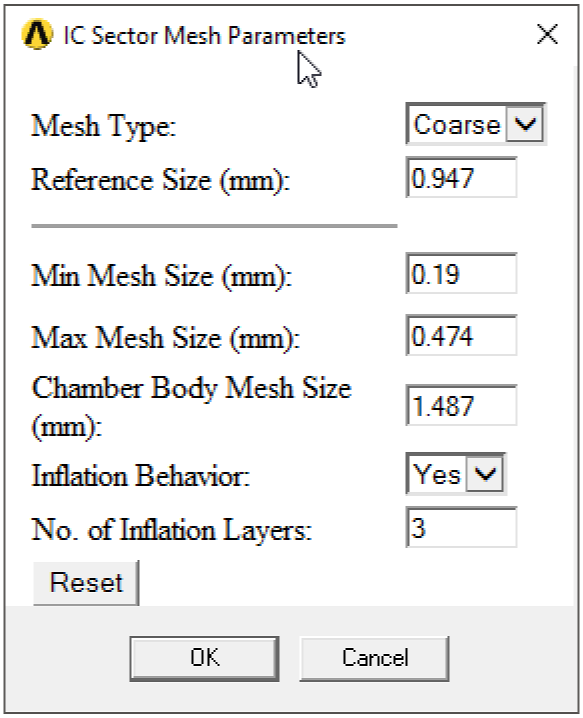
ICE



2. Click **IC Setup Mesh** (

located in the **IC Engine** toolbar). This opens the **IC Sector Mesh Parameters** dialog box.

   a. Here you can define different mesh settings for the different parts and virtual topologies.



In the **IC Sector Mesh Parameters** dialog box you can see the default mesh settings. You can change the settings or use the default ones.

- **Mesh Type**: You can select **Coarse** or **Fine** from the drop-down list.
- **Reference Size**: This is a reference value. Some global mesh settings and local mesh setting values are dependent on this term. The value changes depending upon the selection of the **Mesh Type**.

  Reference Size = (Valve margin perimeter) /100
- **Min Mesh Size**: This value is set to **Reference Size**/5.
- **Max Mesh Size**: This value is set equal to the **Reference Size**/2.
- **Chamber Body Mesh Size**: This value is set to the **0.633 mm** by default.
- **Inflation Behavior**: It is set to **Yes** by default.
- **No. of Inflation Layers**: This value is set to **3** by default.

Note:

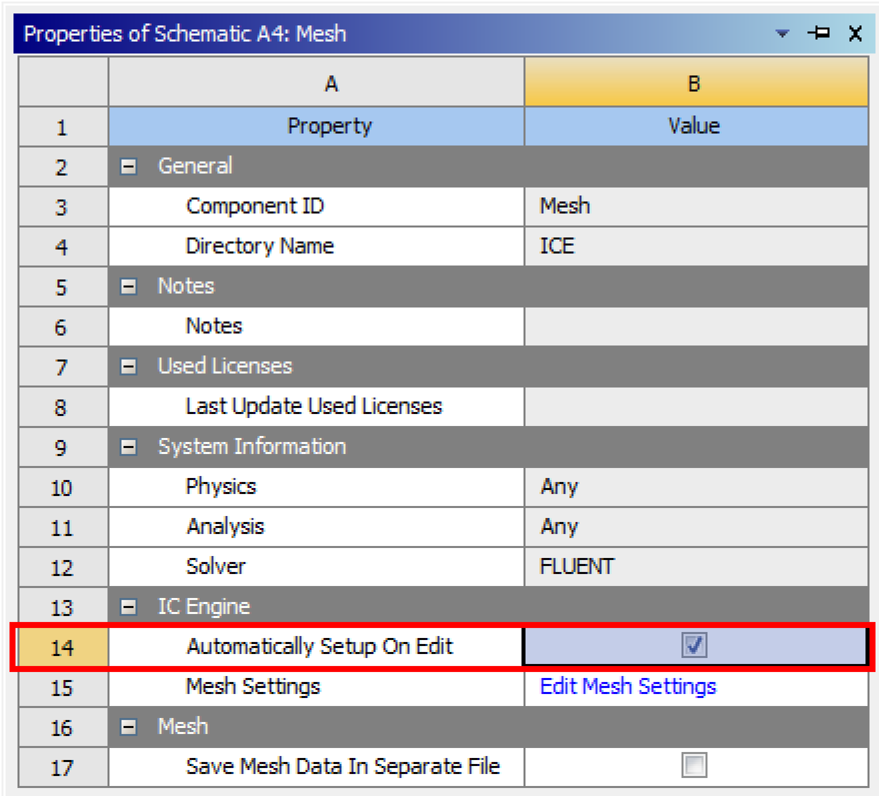
If **Reference Size** is changed, all the other parameters will change, depending on their relation with it.

---

b. If after changing the values of the parameters you would like to go back to the default values, click **Reset**.

c. Click **OK** to set and close the **IC Sector Mesh Parameters** dialog box.

You can generate the mesh controls before opening Ansys Meshing application. To do this enable **Automatically Setup On Edit** under **IC Engine** in the **Properties** box which is displayed after selecting the **Mesh** cell in the Workbench window.

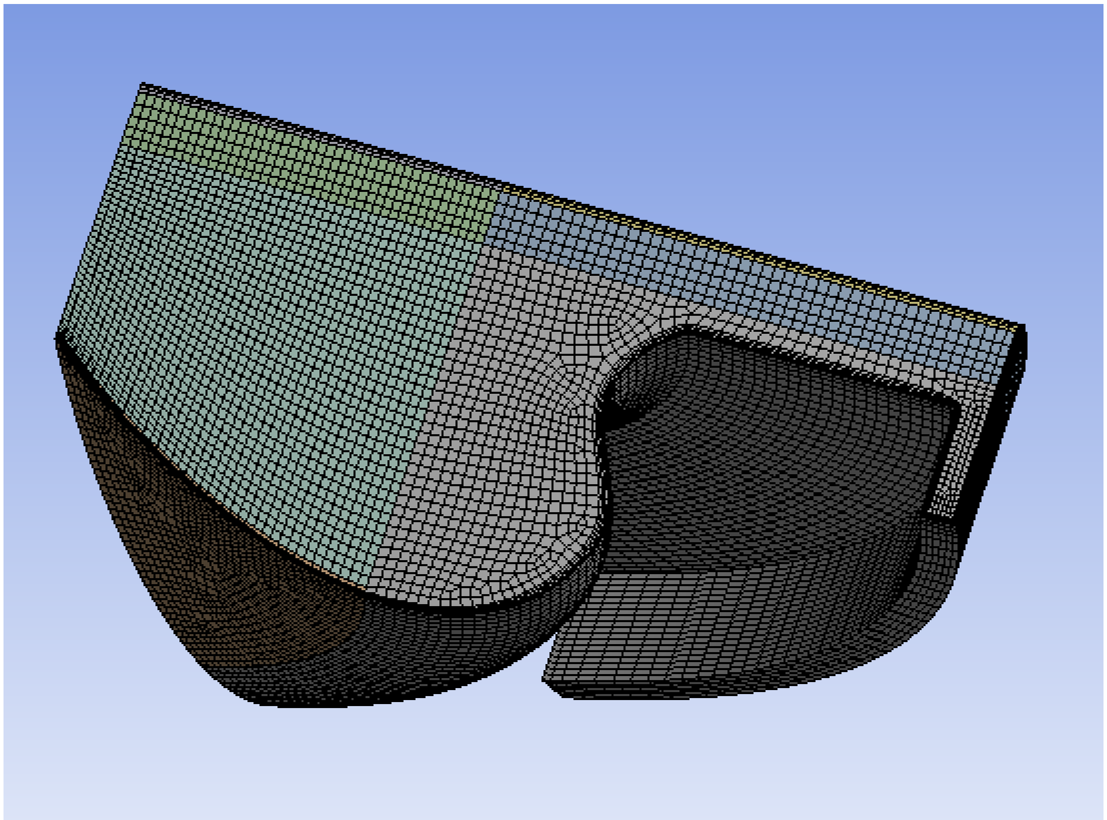

Properties of Schematic A4: Mesh

	A	B
1	Property	Value
2	⊟ General	
3	Component ID	Mesh
4	Directory Name	ICE
5	⊟ Notes	
6	Notes	
7	⊟ Used Licenses	
8	Last Update Used Licenses	
9	⊟ System Information	
10	Physics	Any
11	Analysis	Any
12	Solver	FLUENT
13	⊟ IC Engine	
14	Automatically Setup On Edit	☑
15	Mesh Settings	Edit Mesh Settings
16	⊟ Mesh	
17	Save Mesh Data In Separate File	☐

So after you double-click the **Mesh** cell to open Ansys Meshing application the mesh controls are already set.

3. Click **IC Generate Mesh** (

located in the **IC Engine** toolbar) to generate the mesh.



When you complete the setup through **IC Setup Mesh**, it does the following:

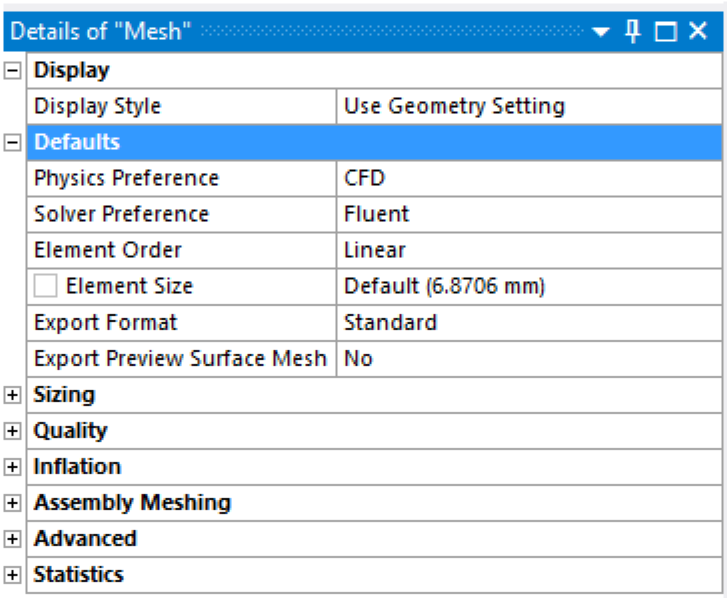
- It creates virtual topologies for the Piston-Inner body.
- It makes the required changes in **Global Mesh Settings**.
- It creates **Local Mesh Settings**.

11.2. Global Mesh Settings for Sector Combustion Simulation

IC Engine meshing tool creates **Global Mesh Settings** based on the information provided in the **IC Sector Mesh Parameters** (p. 367) dialog box. Following sections describe these settings:

Defaults Group

Under **Details of Mesh**, the following are the global mesh settings defined under **Defaults**.



- **Physics Preference**: This is set to **CFD**.

  This option allows you to establish how Workbench will perform meshing based on the physics of the analysis type that you specify.

- **Solver Preference**: Ansys Fluent is set as the solver for the simulation.

  Since **CFD** is chosen as your **Physics Preference**, it causes a **Solver Preference** option to appear in the **Details View** of the **Mesh** folder. The chosen value sets certain defaults that will result in a mesh that is more favorable to the respective solver.

- **Element Order**: This is set to **Linear**. This option allows you to control whether meshes are to be created with midside nodes (quadratic elements) or without midside nodes (linear elements). Reducing the number of midside nodes reduces the number of degrees of freedom. Choices include **Program Controlled**, **Linear**, and **Quadratic**.

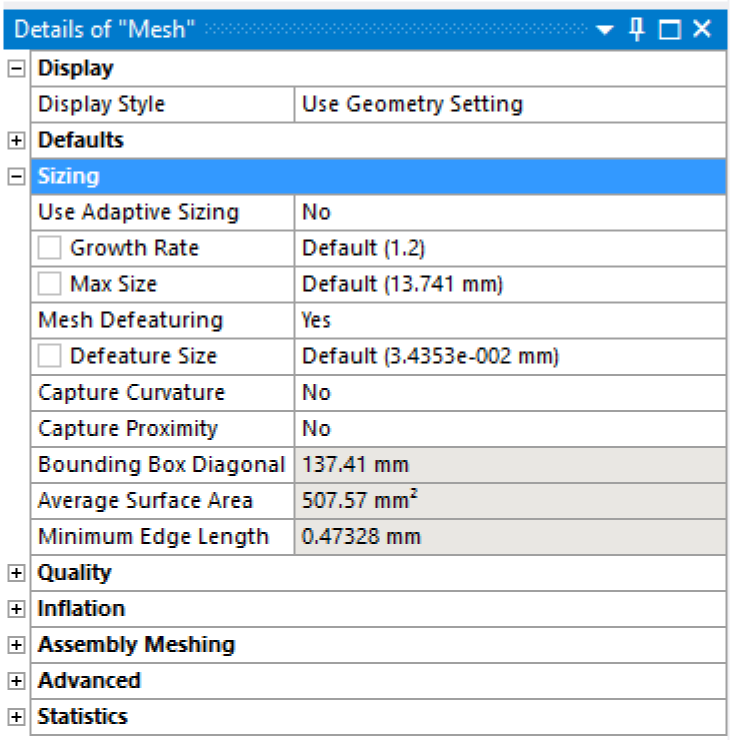
- **Element Size**: This allows you to specify the element size used for the entire model. This size will be used for all edge, face, and body meshing.

- **Export Format**: This option defines the format for the mesh when exported to Ansys Fluent. The default is **Standard**. You can change this to **Large Model Support** to export the mesh as a cell-based Fluent mesh.

- **Export Preview Surface Mesh** : This option controls the export of the preview surface mesh elements. This option can be used when the bodies have been meshed only partially, that is, not all volumes have been filled with elements and only previewing of surface meshes was done. The default is **No**, which results in export of only volume mesh elements to the Fluent mesh file. You can change this to Yes to export both the volume mesh and the preview surface meshes to the Fluent mesh file.

Sizing Group

Under the **Details of Mesh**, the following are the global mesh settings defined under **Sizing**.



- **Use Adaptive Sizing**: This option refers to a 2D curvature and proximity-based refinement approach which refines edges based on curvature and/or proximity but does not propagate the refined mesh along the face. When set to **Yes**, the mesher uses the value of the element size property to determine a starting point for the mesh size. The value of the element size property can be set by the user or automatically computed using defaults. When meshing begins, edges are meshed with this size initially, and then they are refined for curvature and 2D proximity. Next, mesh based defeaturing and pinch control execution occurs. The final edge mesh is then passed into a least-squares fit size function, which guides face and volume meshing.

- **Growth Rate**: It is equal to default value of 1.20.

- **Max Size**: It is equal to the value of **Max Mesh Size**, which is set in the **IC Sector Mesh Parameters** (p. 187) dialog box.

- **Mesh Defeaturing**: This option automatically defeatures small features and dirty geometry according to the **Defeature Size** you specify here.

- **Transition**: When **Use Adaptive Sizing** is set to **Yes**, this option affects the rate at which adjacent elements will grow. **Slow** produces smooth transitions while **Fast** produces more abrupt transitions.

- **Span Angle Center**: It is set to **Fine**.

  When **Use Adaptive Sizing** is set to **Yes**, this option sets the goal for curvature based refinement. The mesh will subdivide in curved regions until the individual elements span this angle. The following choices are available:

  - **Coarse** — 91^o to 60^o

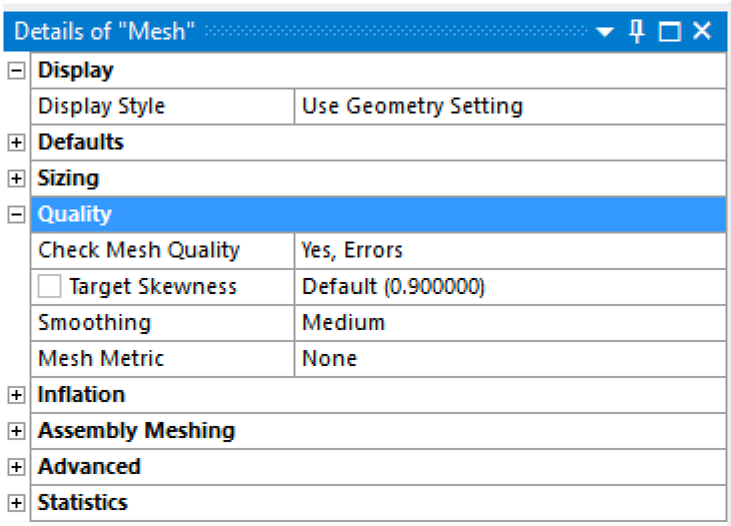
  - **Medium** — 75^o to 24^o

  - **Fine** — 36^{o} to 12^{o}

- **Initial Size Seed**: When **Use Adaptive Sizing** is set to **Yes**, this option allows you to control the initial seeding of the mesh size for each part.
- **Bounding Box Diagonal**: This option provides a read-only indication of the length of the assembly diagonal.
- **Average Surface Area**: This option provides a read-only indication of the average surface area of the model.
- **Minimum Edge Length**: This option provides a read-only indication of the smallest edge length in the model.
- **Capture Curvature**: This option allows you to take into account curvature effects.
- **Curvature Min Size**: When **Capture Curvature** is set to **Yes**, this option is equal to the value of **Min Mesh Size**, which is set in the **IC Sector Mesh Parameters** (p. 187) dialog box.
- **Curvature Normal Angle**: When **Capture Curvature** is set to **Yes**, this option is set to the value of the **Normal Angle**.
- **Capture Proximity**: This option allows you to account for proximity effects.
- **Proximity Min Size**: When **Capture Proximity** is set to **Yes**, this option allows you to specify a global minimum size to be used in proximity sizing calculations.
- **Num Cells Across Gap**: When **Capture Proximity** is set to **Yes**, this option is the minimum number of layers of elements to be generated in the gaps. You can specify a value from 1 to 100, or accept the default (3).
- **Proximity Size Function Sources**: When **Capture Proximity** is set to **Yes**, this option determines whether regions of proximity between faces and/or edges are considered when proximity size function calculations are performed. You can specify **Edges**, **Faces**, or **Faces and Edges**.

Quality Group

Quality is useful for configuring mesh quality.



- **Check Mesh Quality**: This option determines how the software behaves with respect to error and warning limits

- **Target Skewness**: This option allows you to set a target skewness that you would like the mesh to satisfy.

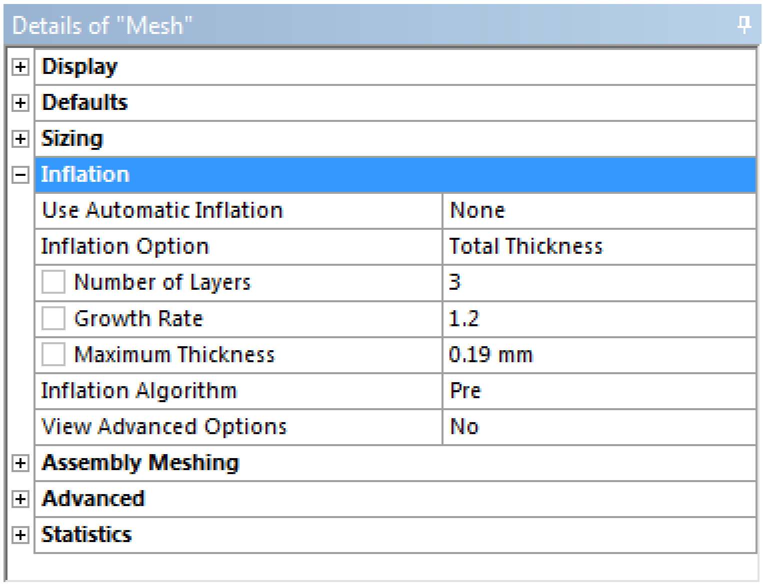
- **Smoothing**: This option attempts to improve element quality by moving locations of nodes with respect to surrounding nodes and elements. The **Low**, **Medium**, or **High** option controls the number of smoothing iterations along with the threshold metric where the mesher will start smoothing.

- **Mesh Metric**: This option allows you to view mesh metric information and thereby evaluate the mesh quality.

  When you select a mesh metric, its **Min**, **Max**, **Average**, and **Standard Deviation** values are reported in the Details View, and a bar graph is displayed under the Geometry window. The graph is labeled with color-coded bars for each element shape represented in the model's mesh, and can be manipulated to view specific mesh statistics of interest.

Inflation Group

Inflation is useful for CFD boundary layer resolution.



- **Use Automatic Inflation**: It is set to **None** for **Combustion Simulation**.
- **Inflation Option**: It is set to **Total Thickness**.
  - **Number of Layers**: It is set to **3** which is equal to the **No. of Inflation Layers** set in the **IC Sector Mesh Parameters** dialog box.
  - **Growth Rate**: This value is set to `1.2`.
  - **Growth Rate**: This value is approximately equal to the **Min Mesh Size**.
- **Inflation Algorithm** is set to **Pre**.
- **View Advanced Options**: This control determines whether advanced inflation options appear in the **Details View**. Choices are **No** and **Yes**. It is set to **No** for **Combustion Simulation**.

Assembly Meshing

Method is set to **None**.

Advanced Group

The **Advanced** group allows you to use the following features and controls:

Pinch Tolerance

- **Number of CPUs for Parallel Part Meshing**: This option sets the number of processors to be used for parallel part meshing. Using the default for specifying multiple processors will enhance meshing performance on geometries with multiple parts. For parallel part meshing, the default is set to Program Controlled or 0. This instructs the mesher to use all available CPU cores. The default setting inherently limits 2 GB memory per CPU core. An explicit value can be specified between 0 and 256, where 0 is the default.

- **Straight Sided Elements**: This option specifies meshing to straight edge elements when set to **Yes**. This option may affect the placement of midside nodes if the **Element Order** option is set to **Quadratic**.
- **Rigid Body Behavior**: This option determines whether a full mesh is generated for a rigid body, rather than a surface contact mesh. **Rigid Body Behavior** is applicable to all body types. Valid values for **Rigid Body Behavior** are **Dimensionally Reduced** (generate surface contact mesh only) and **Full Mesh** (generate full mesh).
- **Triangle Surface Mesher**: This option determines which triangle surface meshing strategy will be used by patch conforming meshers. In general, the advancing front algorithm provides a smoother size variation and better results for skewness and orthogonal quality. This option is inaccessible when an assembly meshing algorithm is selected.
- **Topology Checking**: This option controls what happens when a user scopes an object (such as loads, boundary conditions, named selections and so on) to geometry (bodies, faces, edges, and vertices) after the mesh has been generated. If **Topology Checking** is set to **Yes** (default), the software will check to see if the scoped geometry has mesh properly associated to it. If the associations are incorrect, the scoping of the object will force the mesh to be out of date. The mesh would need to be re-generated to get proper associations. If the associations are correct, the scoping is performed without any change to the mesh and the mesh stays up to date. Set **Topology Checking** to **No** to avoid the checks and always keep the mesh up to date.
- **Pinch Tolerance**: This is set to the default value.

  This control allows you to specify a tolerance for the Meshing application to use when it generates automatic pinch controls.
- **Generate Pinch on Refresh**: This option determines whether pinch controls will be regenerated following a change made to the geometry (such as a change made via a DesignModeler application operation such as a merge, connect, etc.). If **Generate Pinch on Refresh** is set to **Yes** and you change the geometry, all pinch controls that were created automatically will be deleted and recreated based on the new geometry. If **Generate Pinch on Refresh** is set to **No** and you update the geometry, all pinch controls related to the changed part will appear in the Tree Outline but will be flagged as undefined.

11.3. Local Mesh Settings for Sector Combustion Simulation

Based on information provided in the **IC Sector Mesh Parameters** dialog box, IC Engine meshing tool creates some mesh settings at local level. Following section describes these settings for combustion simulation:

For more information on **Local Mesh Settings**, see Local Mesh Controls in the Meshing User's Guide.

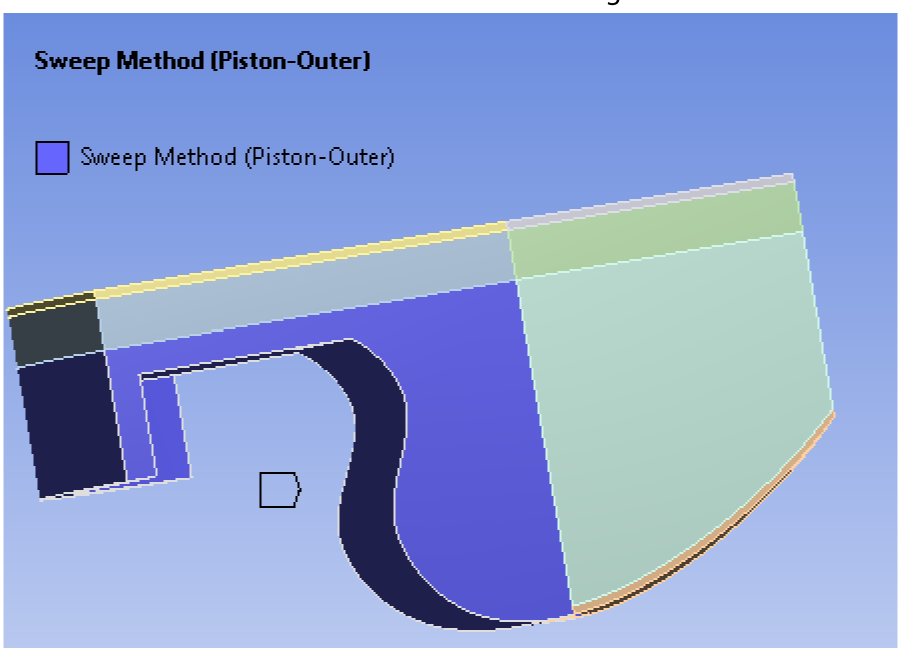
11.3.1. Sweep Method (Piston-Outer)

When you click **Sweep Method (Piston-Outer)** under **Mesh** in the **Outline**, you can see the details. In this method, a swept mesh is forced on the “sweepable” bodies.

Details of "Sweep Method (Piston-Outer)" - Method

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Method	Sweep
Element Midside Nodes	Use Global Setting
Src/Trg Selection	Manual Source
Source	1 Face
Target	Program Controlled
Free Face Mesh Type	Quad/Tri
Type	Number of Divisions
Sweep Num Divs	Default
Sweep Bias Type	No Bias
Element Option	Solid
Constrain Boundary	No

- **Geometry**: It shows the selected body.
- **Manual Source** is chosen from the **Src/Trg Selection** drop-down list.
- **Source**: It is the face selected as shown in the figure.

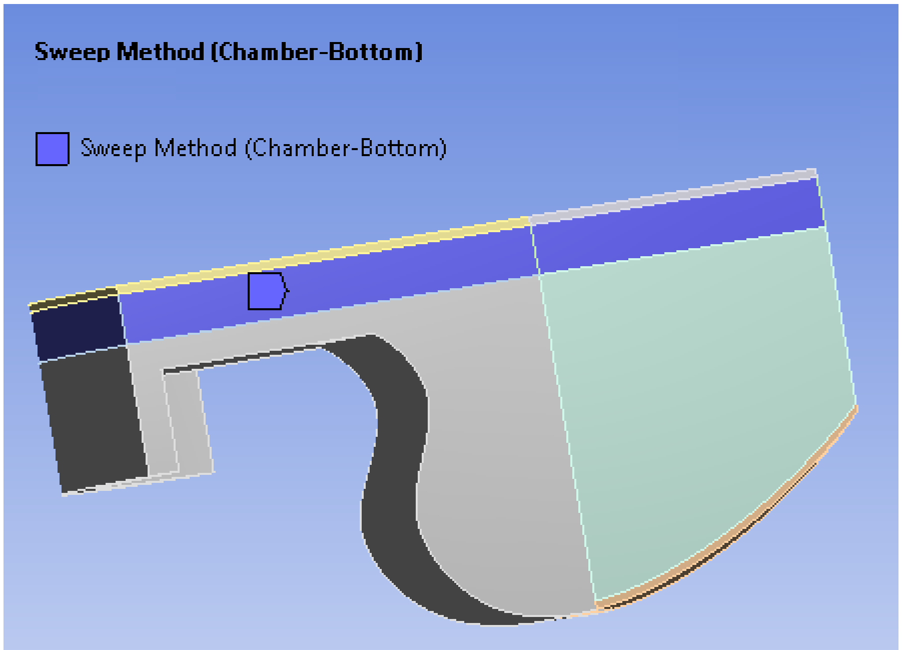


- **Target**: It is **Program Controlled**. This means that the program determines the target for the body.

- **Free Face Mesh Type**: It is set to **Quad/Tri**. This determines the shape of the elements used to fill the swept body.
- **Sweep Bias Type**: It adjusts the spacing ratio of nodes on the edge. **No Bias** is chosen from the drop-down list.

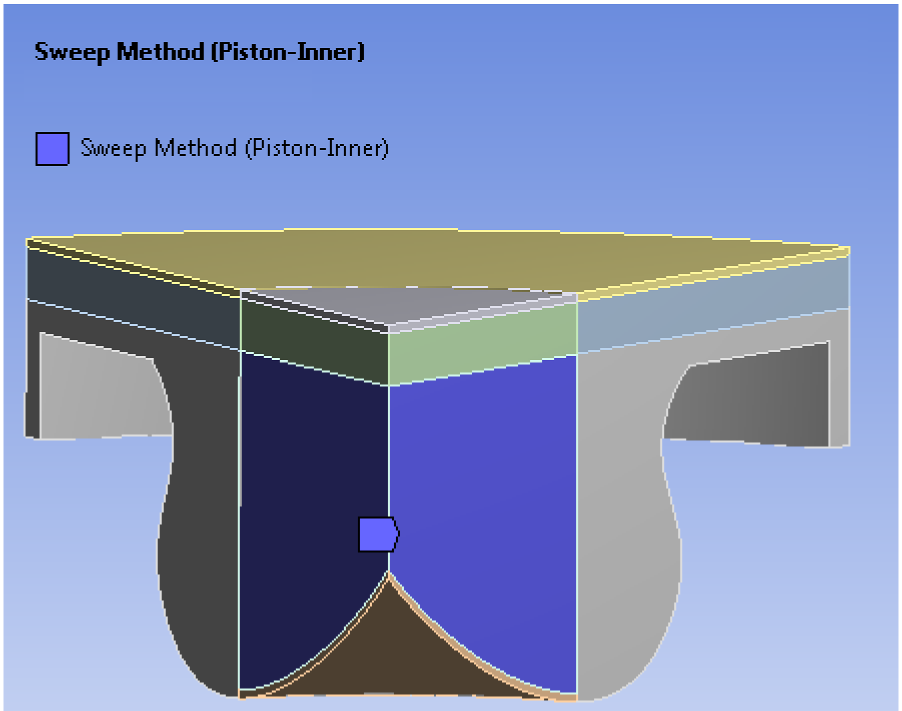
11.3.2. Sweep Method (Chamber-Bottom)

When you click **Sweep Method (Chamber-Bottom)** under **Mesh** in the **Outline**, you can see the details. Settings are almost similar to **Piston-Outer**.



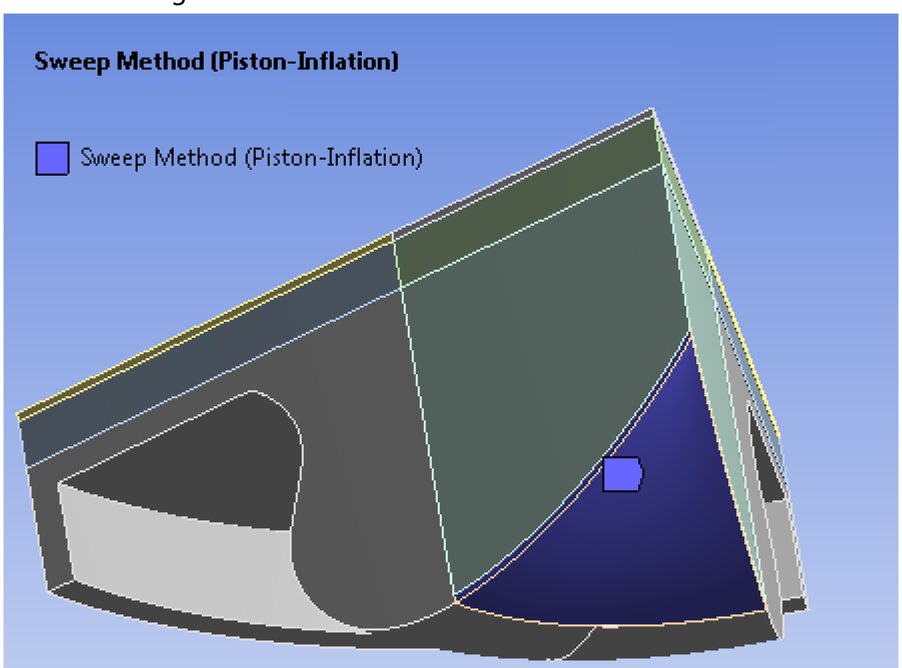
11.3.3. Sweep Method (Piston-Inner)

When you click **Sweep Method (Piston-Inner)** under **Mesh** in the **Outline**, you can see the details. Settings are same as for **Piston-Outer**.



11.3.3.1. Sweep Method (Piston-Inflation)

When you click **Sweep Method (Piston-Inflation)** under **Mesh** in the **Outline**, you can see the details. Settings are same as for **Piston-Outer**.



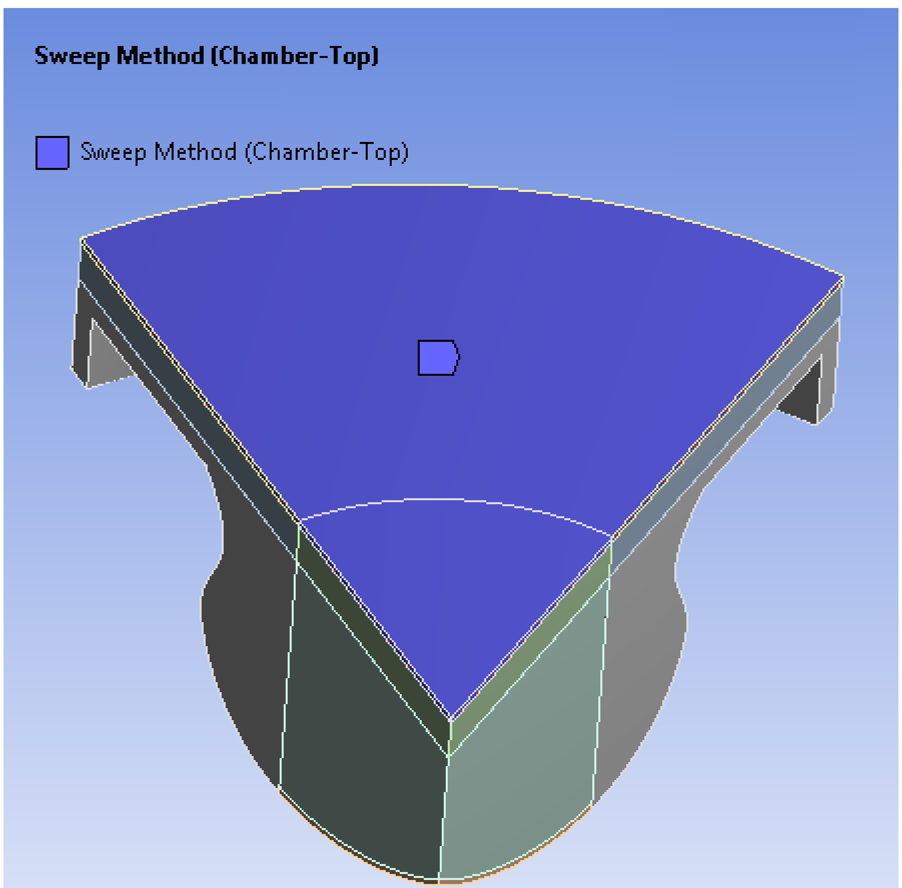
11.3.3.2. Sweep Method (Chamber-Top)

When you click **Sweep Method (Chamber-Top)** under **Mesh** in the **Outline**, you can see the details. In this method, a swept mesh is forced on the "sweepable" bodies.

Details of "Sweep Method (Chamber-Top)" - Method

Scope	
Scoping Method	Geometry Selection
Geometry	2 Bodies
Definition	
Suppressed	No
Method	Sweep
Element Midside Nodes	Use Global Setting
Src/Trg Selection	Manual Source
Source	2 Faces
Target	Program Controlled
Free Face Mesh Type	Quad/Tri
Type	Number of Divisions
Sweep Num Divs	3
Sweep Bias Type	_____ ___ _ _
Sweep Bias	2.
Element Option	Solid
Constrain Boundary	No

- **Geometry**: It shows the selected bodies.
- **Manual Source** is chosen from the **Src/Trg Selection** drop-down list.
- **Source**: The faces selected are as shown in the figure.



- **Target**: It is **Program Controlled**. This means that the program determines the target for the body.
- **Free Face Mesh Type**: It is set to **Quad/Tri**. This determines the shape of the elements used to fill the swept body.
- **Sweep Bias Type**: It adjusts the spacing ratio of nodes on the edge. **No Bias** is chosen from the drop-down list.

11.3.3.3. Face Sizing(Src-PistonOuter)

When you click **Face Sizing (Src-PistonOuter)** under **Mesh** in the **Outline**, you can see the details.

Details of "Face Sizing(Src-PistonOuter)" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Suppressed	No
Type	Element Size
Element Size	1.4867 mm
Behavior	Soft
Curvature Normal Angle	Default
Growth Rate	Default
Local Min Size	Default (6.8981e-002 mm)

- **Geometry**: This shows the selected face.
- **Type**: It is set to **Element Size**.
- **Element Size**: This is approximately equal to **Chamber Body Mesh Size** as set in the **IC Sector Mesh Parameters** dialog box.
- **Behavior**: For this body part it is set to **Soft**.

11.3.3.4. Face Sizing(Src-PistonInflation)

When you click **Face Sizing (Src-PistonInflation)** under **Mesh** in the **Outline**, you can see the details.

Details of "Face Sizing(Src-PistonInflation)" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Suppressed	No
Type	Element Size
Element Size	0.47328 mm
Behavior	Hard

- **Geometry**: This shows the selected face.

- **Type**: It is set to **Element Size**.
- **Element Size**: This is approximately equal to **Max Mesh Size** as set in the **IC Sector Mesh Parameters** dialog box.
- **Behavior**: For this body part it is set to **Hard**.

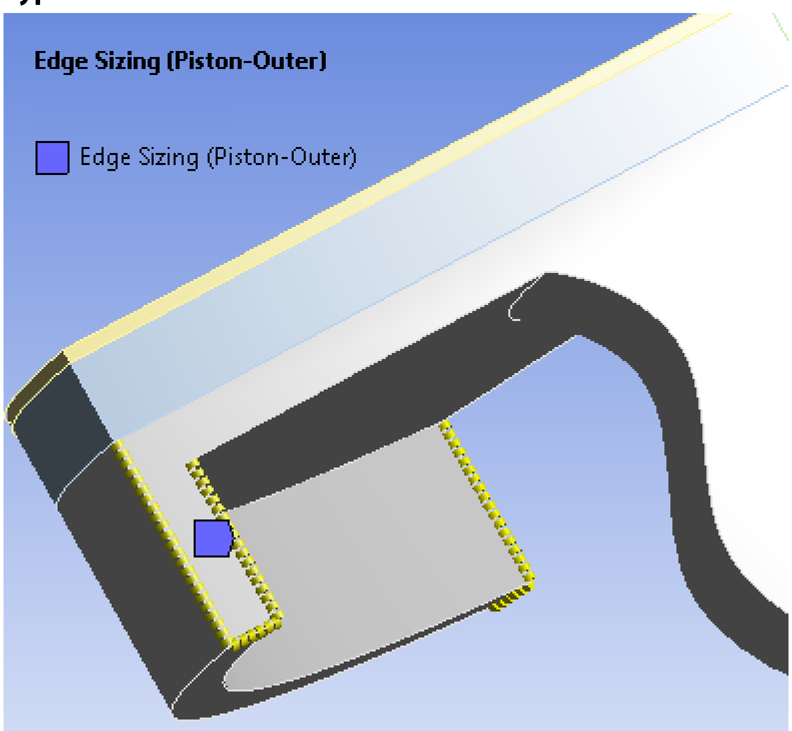
11.3.3.5. Edge Sizing(Piston-Outer)

When you click **Edge Sizing (Piston-Outer)** under **Mesh** in the **Outline**, you can see the details.

Details of "Edge Sizing (Piston-Outer) " - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	6 Edges
Definition	
Suppressed	No
Type	Element Size
Element Size	0.18784 mm
Behavior	Hard
Bias Type	No Bias

- **Geometry**: This shows the number of selected edges.
- **Type**: It is set to **Element Size**.



- **Element Size**: This is approximately equal to crevice thickness/3.
- **Behavior**: For this body part it is set to **Hard**.

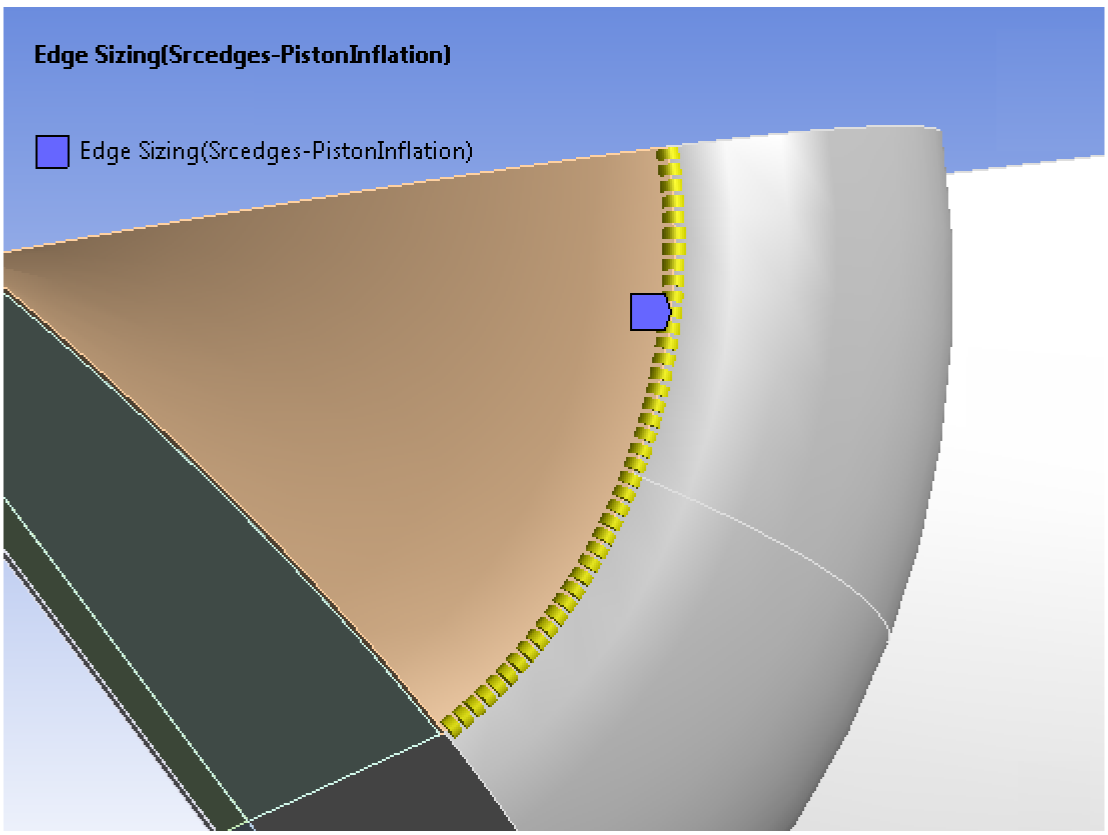
11.3.3.6. Edge Sizing(Srcedges-PistonInflation)

When you click **Edge Sizing (Piston-Outer)** under **Mesh** in the **Outline**, you can see the details.

Details of "Edge Sizing(Srcedges-PistonInflation)" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	2 Edges
Definition	
Suppressed	No
Type	Element Size
Element Size	0.47328 mm
Behavior	Hard
Bias Type	No Bias

- **Geometry**: This shows the number of selected edges.
- **Type**: It is set to **Element Size**.



- **Element Size**: This is approximately equal to crevice thickness/3.
- **Behavior**: For this body part it is set to **Hard**.

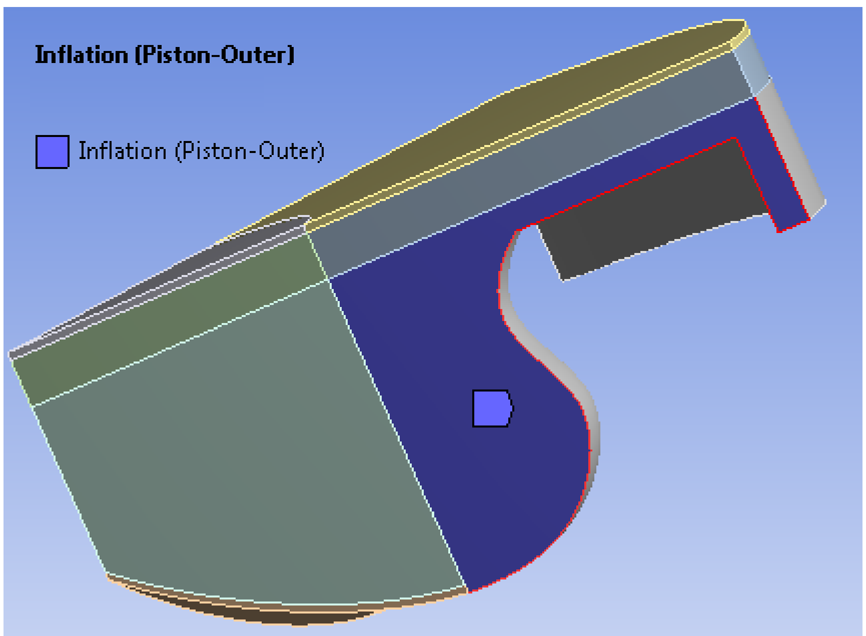
11.3.3.7. Inflation(Piston-Outer)

When you click **Inflation (Piston-Outer)** under **Mesh** in the **Outline**, you can see the details.

Details of "Inflation (Piston-Outer)" - Inflation

Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Suppressed	No
Boundary Scoping Method	Geometry Selection
Boundary	11 Edges
Inflation Option	Total Thickness
Number of Layers	3
Growth Rate	1.2
Maximum Thickness	0.23675 mm
Inflation Algorithm	Pre

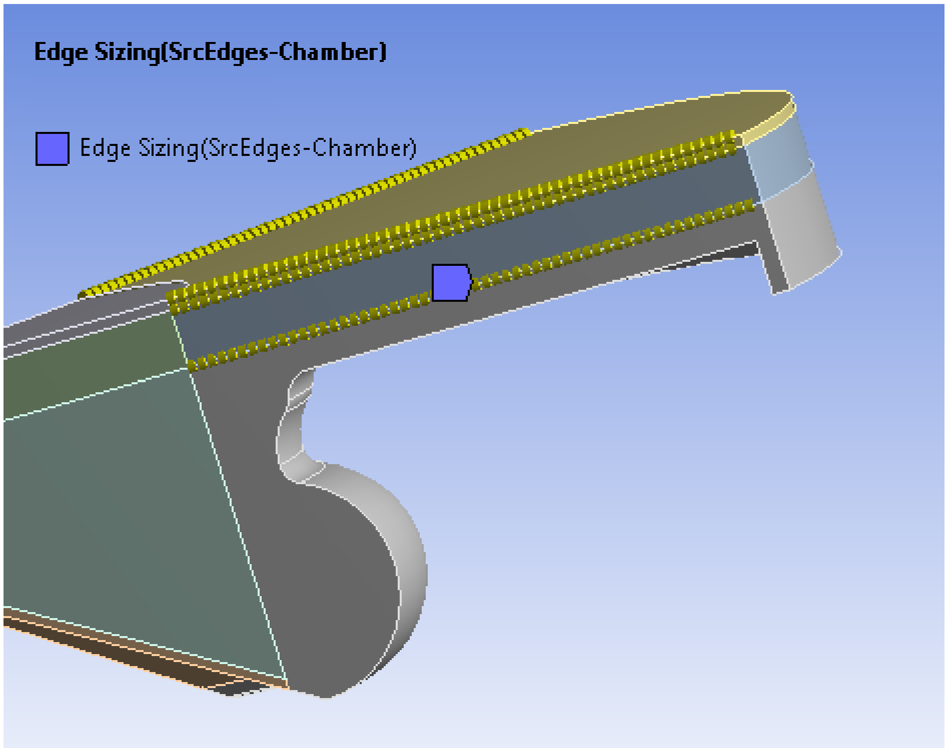
- **Geometry**: This shows the selected face.



- **Boundary**: This shows the selected edges.
- **Inflation Option**: it is set to **Total Thickness**.

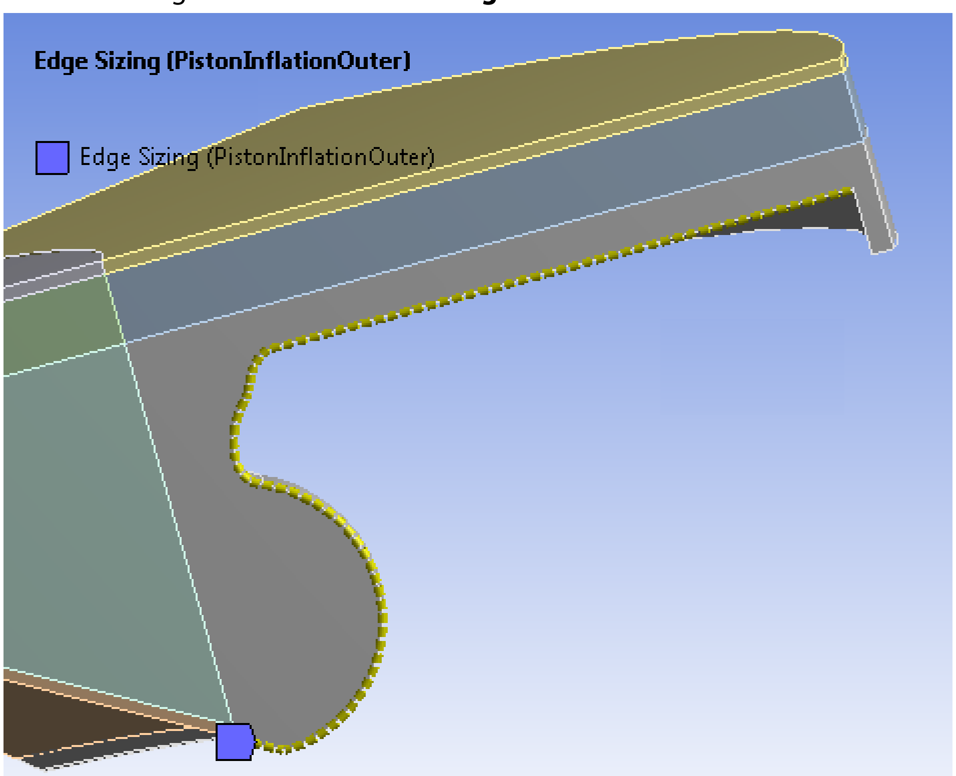
11.3.3.8. Edge Sizing(SrcEdges-Chamber)

When you click **Edge Sizing(SrcEdges-Chamber)** under **Mesh** in the **Outline**, you can see the details. Settings are same as for **SrceEdges-PistonInflation**.



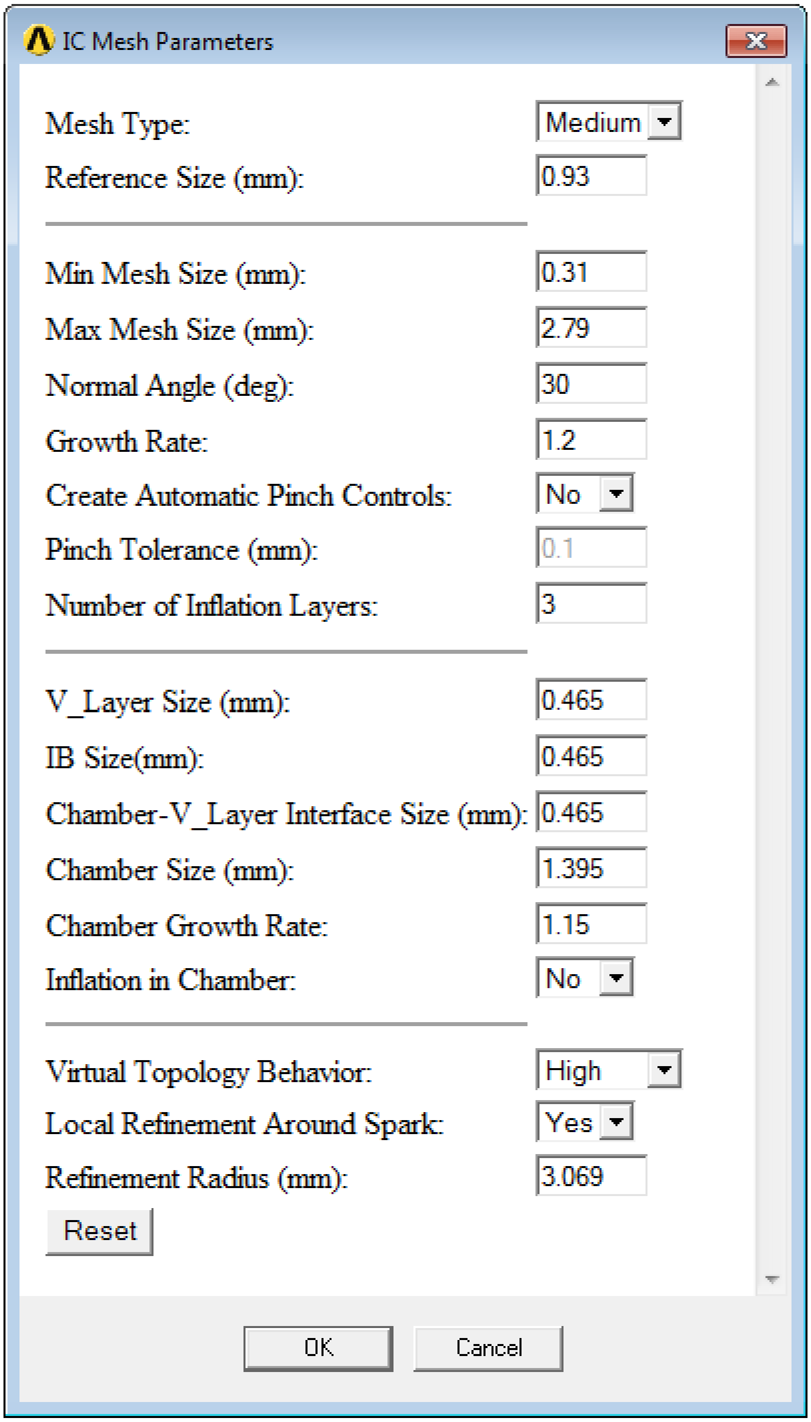
11.3.3.9. Edge Sizing(PistonInflationOuter)

When you click **Edge Sizing(PistonInflationOuter)** under **Mesh** in the **Outline**, you can see the details. Settings are same as for **SrcEdges-PistonInflation**.



11.4. Meshing for Full Engine Full Cycle Combustion Simulation

The meshing process for **Full Engine Full Cycle** combustion simulation is almost similar to the cold flow meshing process. Since combustion can involve spark, some additional spark refinement options are present in meshing.



When **Spark Points** are selected in the **Input Manager** then the following spark related options will be additionally visible in the **IC Mesh Parameters** dialog box.

- **Local Refinement Around Spark**: You can select **Yes** or **No** from the drop-down list. The default selection will be **Yes** only if the crank angle is between IVC and (TDC+10). Spark refinement for the mesh will only be done if **Yes** is selected for **Local Refinement Around Spark**.

Figure 11.1: Local Refinement Around Spark

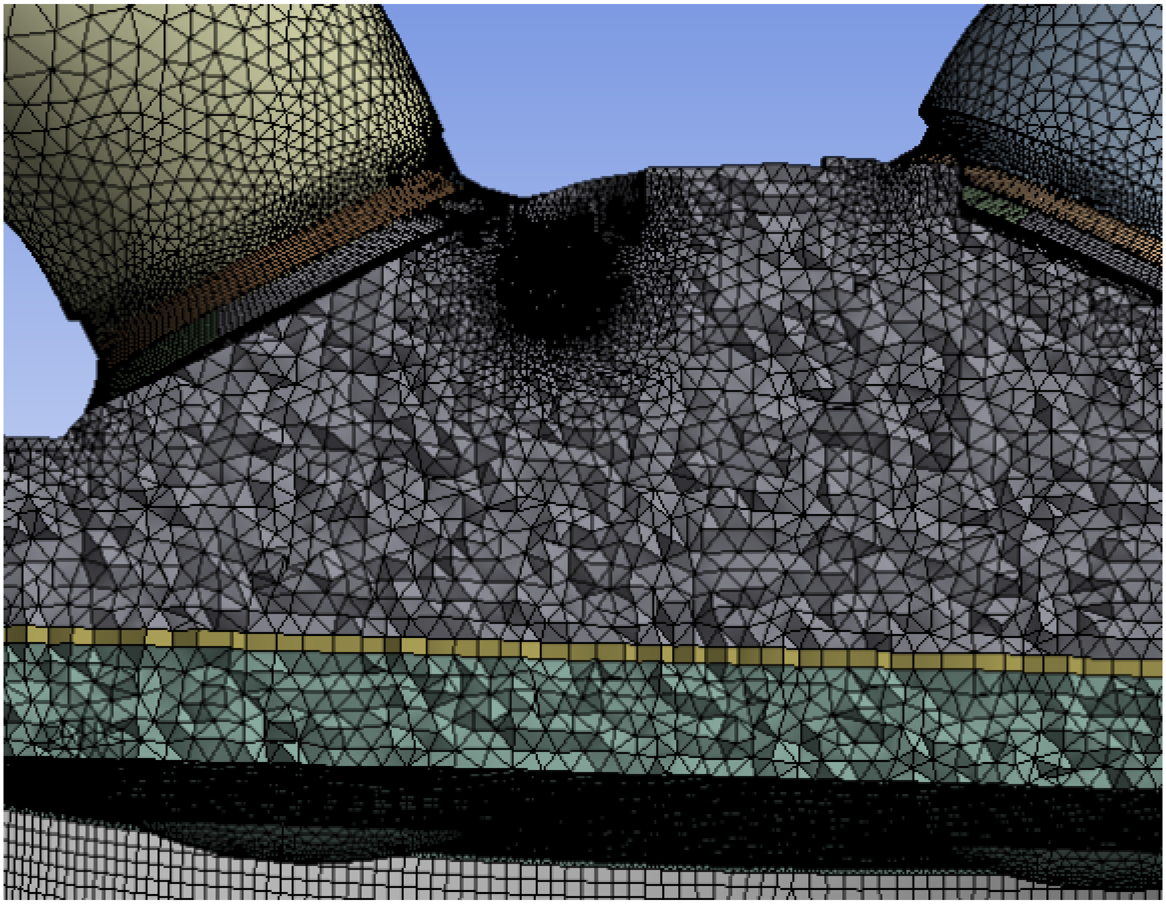
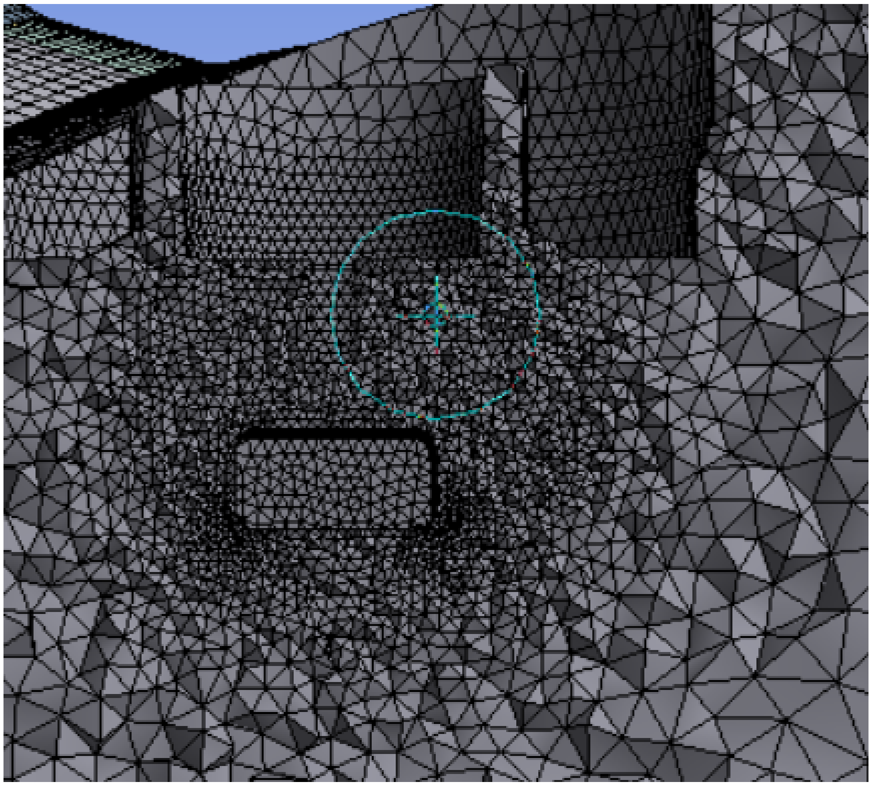


Figure 11.2: Closer Look at Spark Refinement



- **Refinement Radius**: By default this value is set to 3.3 X **Reference Size**.

The following mesh control will be defined under **Mesh** in the tree.

Details of "Body Sizing(ice_spark_center_0)" - Sizing

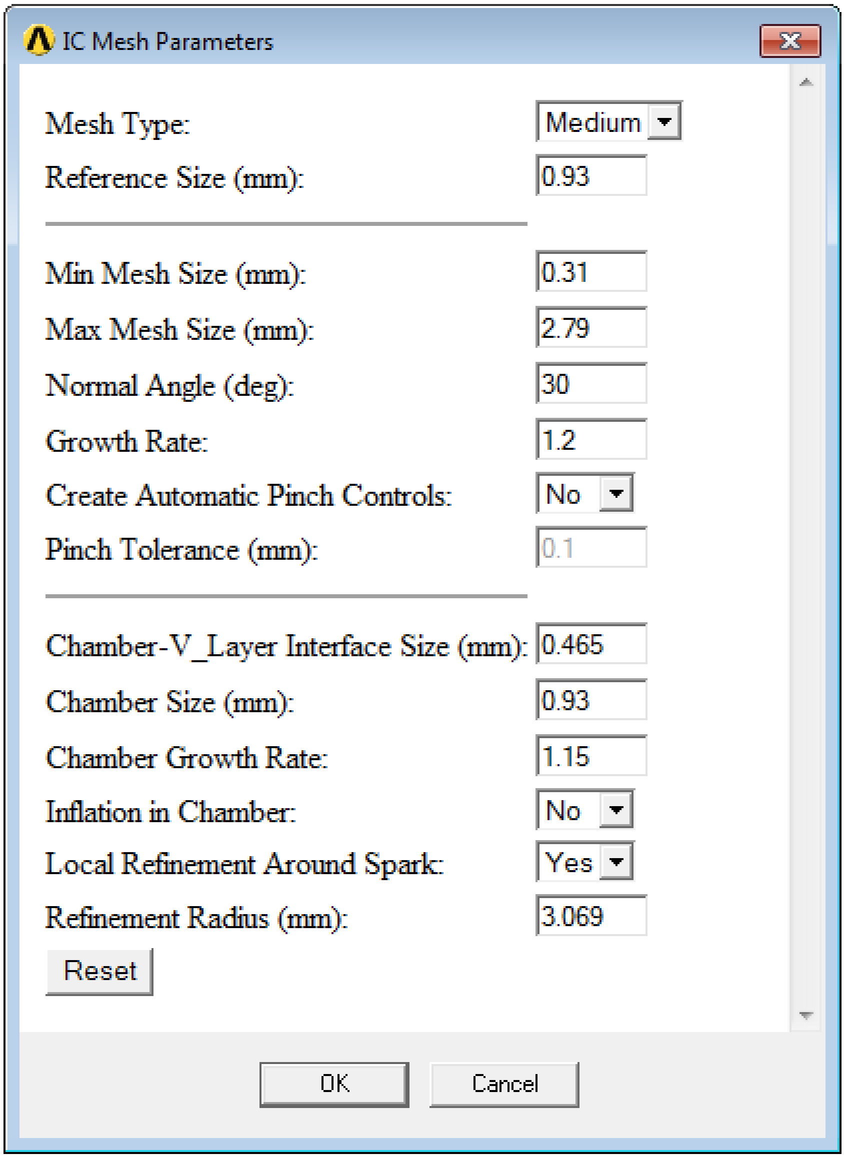
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	Yes
Active	No, Suppressed
Type	Sphere of Influence
Sphere Center	ice_spark_center_0
Sphere Radius	3.069 mm
Element Size	0.186 mm
Local Min Size	Default (0.186 mm)

The **Element Size** is set equal to **Reference Size**/5.

For details about the mesh settings see the sections about mesh settings in Cold Flow Simulation: Meshing (p. 185) chapter.

11.5. Meshing for Full Engine IVC to EVO Combustion Simulation

The meshing process for **Full Engine IVC to EVO** combustion simulation is almost similar to the cold flow meshing process. Since only the chamber part needs to be meshed the mesh settings are slightly different.



- **Mesh Type**: You can select **Fine**, **Medium**, or **Coarse** from the drop-down list. By default it is set to **Medium**.

- **Reference Size**: This is a reference value. Some global mesh settings and local mesh setting values are dependent on this term.

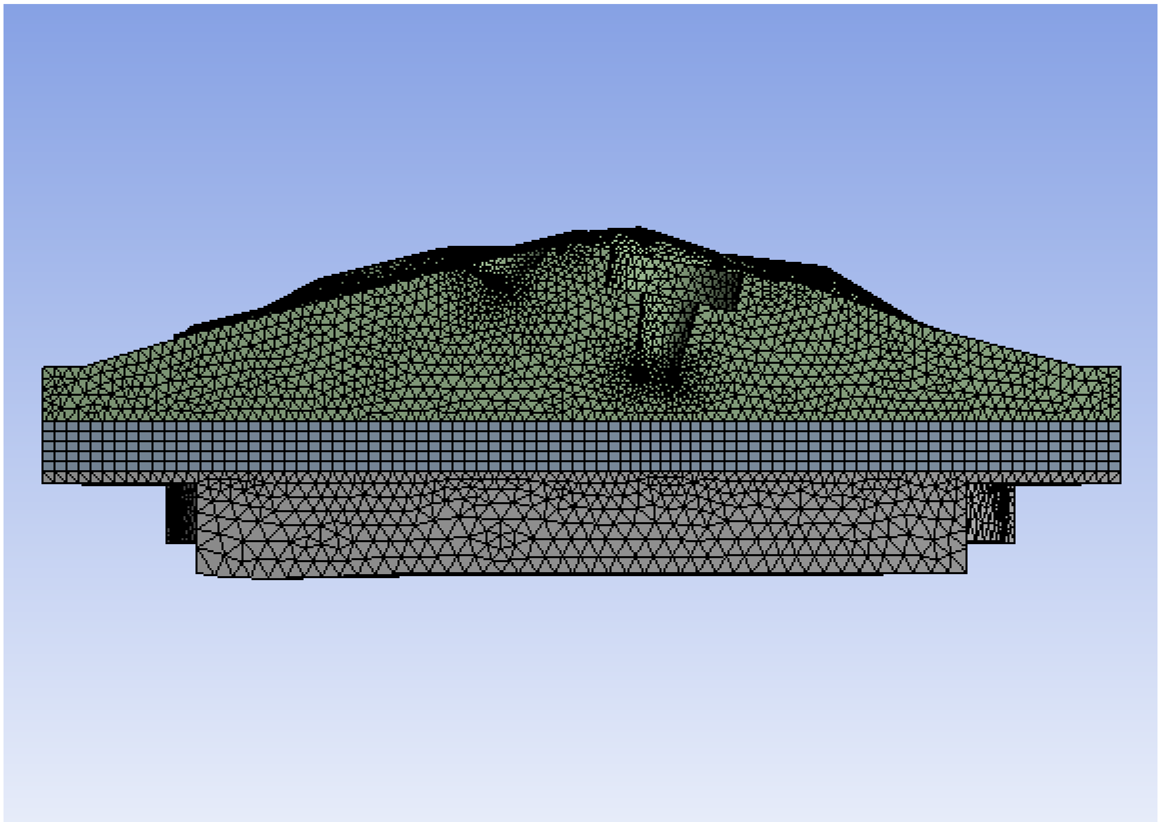
  Reference Size = (Valve margin perimeter) /100

Note:

Choosing **Fine**, **Medium**, or **Coarse** changes the **Reference Size** and thus all other dependent parameters are also affected.

- **Min Mesh Size**: This value is set to **Reference Size**/3.
- **Max Mesh Size**: This value is set to **Reference Size** × 3.
- **Normal Angle**: This value changes depending upon the chosen **Mesh Type**.
- **Growth Rate**: This value is set to **1.2**.
- **Create Automatic Pinch Controls**: This is set to **No** by default.
- **Pinch Tolerance**: Automatic pinch controls are not created by default. If you would like to create pinch control select **Yes** from the **Create Automatic Pinch Controls** drop-down list. You can change the default value of **0.1**.
- **Chamber-V_Layer Interface Size**: This value is set to **Reference Size**/2.
- **Chamber Size**: This value is set to **Reference Size**.
- **Chamber Growth Rate**: This value is set to **1.15**.
- **Inflation in Chamber**: This is set to **No** by default. If you select **Yes** —
  - For canted valve case, inflation will be created on liner, dome, valve bottom, and vlayer-ch interface.
  - For straight valve case inflation will be created on liner.

  A new inflation control for ch faces is created.
- **Local Refinement Around Spark**:The default selection will be **Yes** only if the crank angle is between IVC and (TDC+10). You can select **Yes** or **No** from the drop-down list. Spark refinement for the mesh will only be done if **Yes** is selected for **Local Refinement Around Spark**. When **Spark Points** are selected in the **Input Manager** then the spark related options will be additionally visible in the **IC Mesh Parameters** dialog box.
- **Refinement Radius**: By default this value is set to 3.3 X **Reference Size**. You can edit this value.



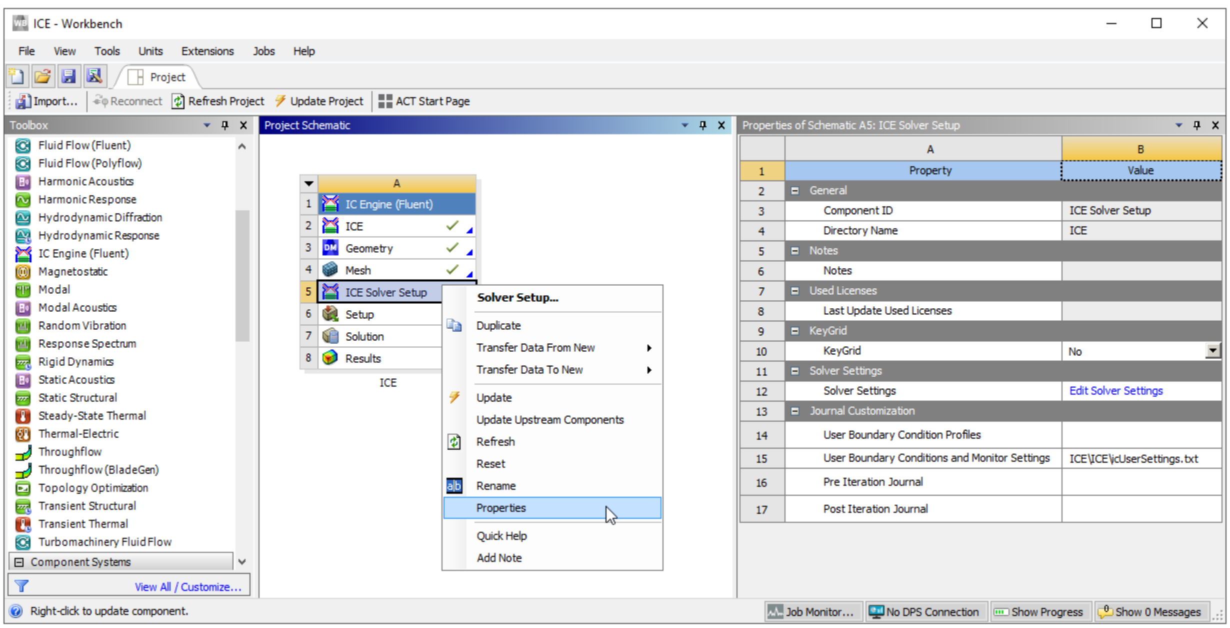
For details about the mesh settings see the sections about chamber meshing (p. 209) in Cold Flow Simulation: Meshing (p. 185) chapter.

Chapter 12: Combustion Simulation: Setting Up the Analysis in IC Engine

This chapter describes how to configure the analysis for a combustion sector simulation. The boundary conditions and solver parameters are set as per the **Decomposition Crank Angle** set in the **Input Manager**. All the settings described in this chapter are done automatically, but you can change them by accessing the dialog boxes and task pages.

> **Note:**
>
> The piston should be at TDC while decomposing even though you want to start the simulation from an angle other than zero.

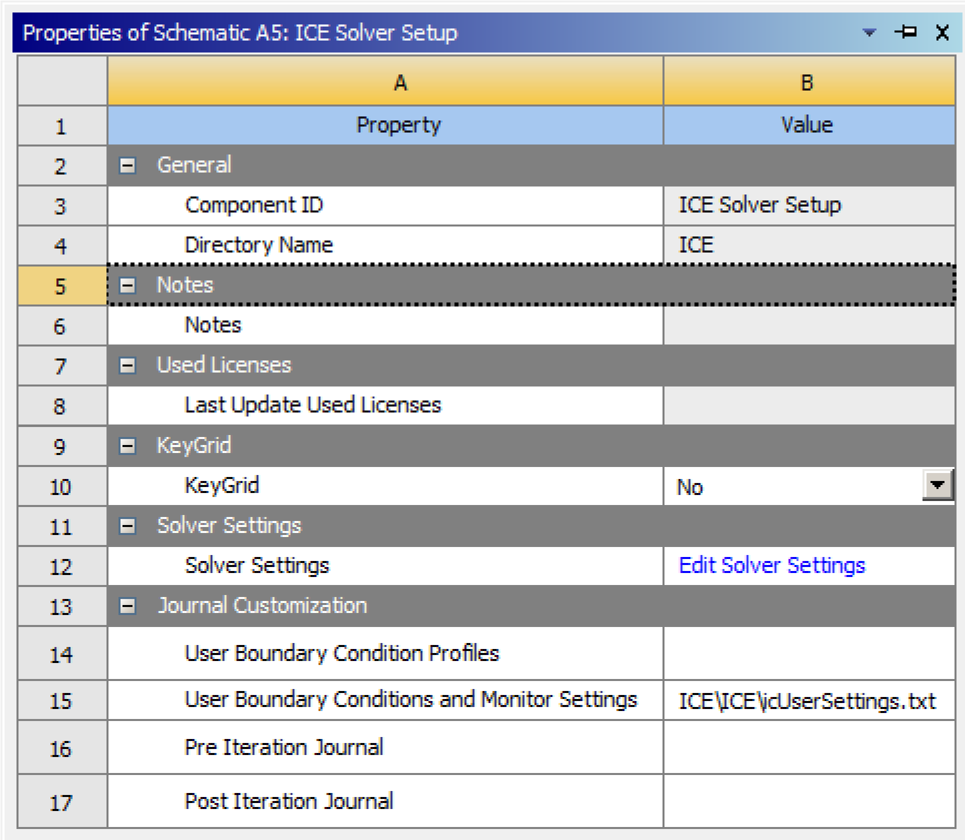
After meshing you can set up the solver. In the **Properties** pane of the **ICE Solver Setup** cell you can make changes to the solver settings or set the keygrid crank angles.



Click **Edit Solver Settings** to open the **Solver Settings** dialog box where you can check the default settings and change any if required.

12.1. ICE Solver Settings

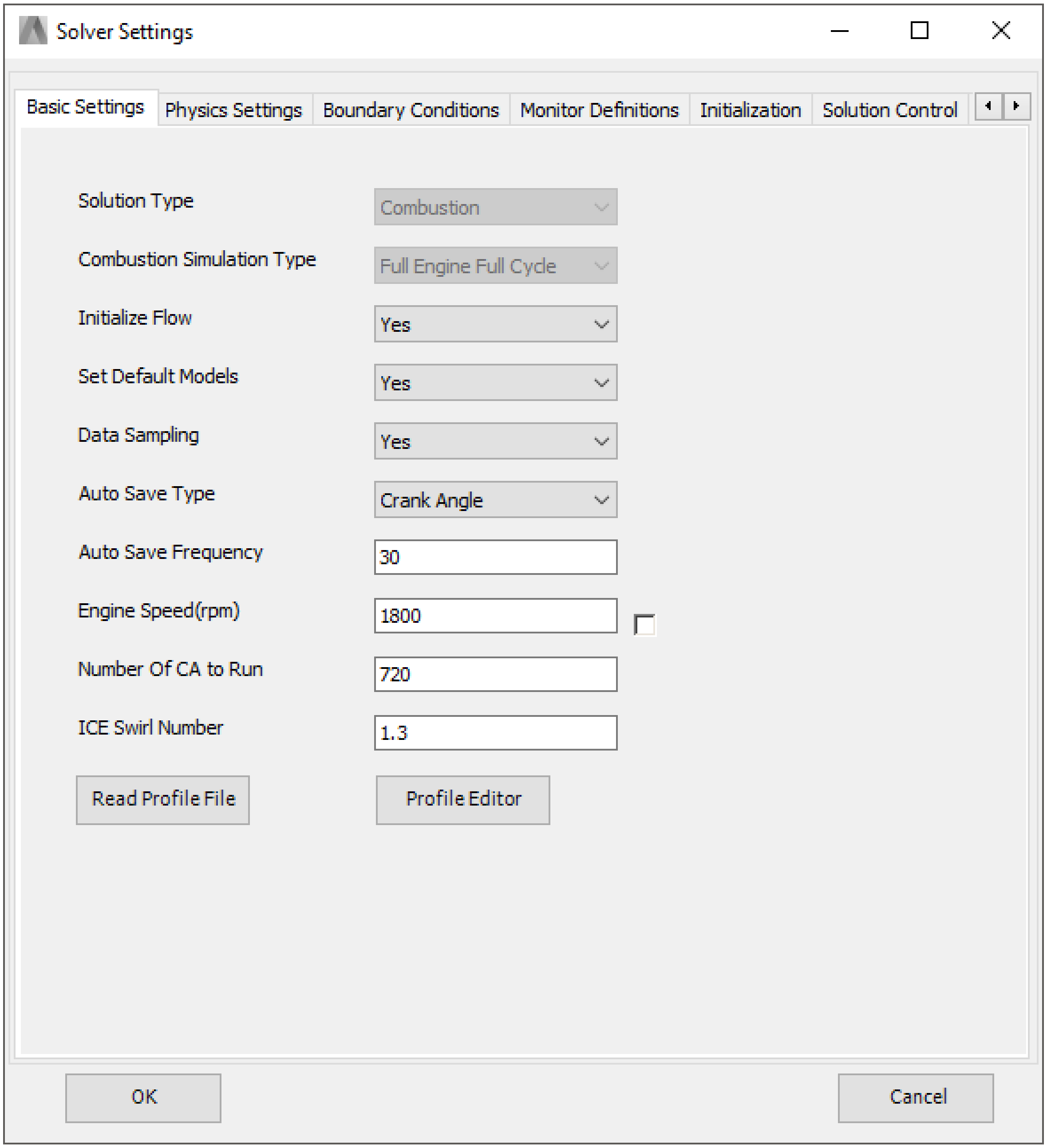
Click **Edit Solver Settings** to open the **Solver Settings** dialog box.



Properties of Schematic A5: ICE Solver Setup

	A	B
1	Property	Value
2	General	
3	Component ID	ICE Solver Setup
4	Directory Name	ICE
5	Notes	
6	Notes	
7	Used Licenses	
8	Last Update Used Licenses	
9	KeyGrid	
10	KeyGrid	No
11	Solver Settings	
12	Solver Settings	Edit Solver Settings
13	Journal Customization	
14	User Boundary Condition Profiles	
15	User Boundary Conditions and Monitor Settings	ICE\ICE\icUserSettings.txt
16	Pre Iteration Journal	
17	Post Iteration Journal	

12.1.1. Basic Settings



In the **Basic Settings** tab you have the following settings:

Solution Type

shows the **Solution Type** you have selected in the **Properties** pane of the **ICE** cell. For the present case it is **Combustion**. You cannot make any changes here.

Combustion Simulation Type

shows the type of combustion simulation you have selected. You cannot make any changes here.

Initialize Flow

is set to **Yes** by default. This will initialize the flow. To check the initialization settings see the **Solution Initialization** (p. 454) task page in Ansys Fluent.

Set Default Models

is set to **Yes** by default. Some models of Ansys Fluent have been chosen as default for better results. These are the **Energy** model, and the **Standard k-epsilon** model from the list of **Viscous** models with **Standard Wall Functions**. You can check them at the **Models** (p. 438) task page of Ansys Fluent. Additional models will be selected on the basis of further selection of options.

Data Sampling

is set to **Yes** by default. This enables **Data Sampling for Time Statistics** in the **Run Calculation** task page of Ansys Fluent. It will enable the sampling of data during the unsteady calculation. See the **Run Calculation** (p. 455) task page of Ansys Fluent.

Auto Save Type

is set to **Crank Angle** by default. This means that the intermediate case and data files will be saved at the entered frequency crank angles. You can also select **Time** from the drop-down menu if you want to save the case and data files at a specific frequency of time steps.

Auto Save Frequency

shows the number of crank angles or time steps, after which the case and data file will be saved in Ansys Fluent. Here the default value is **30**.

Engine Speed

It is set to a default value of **1800rpm**. The engine speed is used along with the crank angle step size to calculate the time step size. You can enter the speed of your engine here. You can enable the check box next to it to parameterize it.

Number of CA to run

is set to **720** by default. This corresponds to a full cycle. You can enter the number of crank angles you want to run the simulation

ICE Swirl Number

is used to patch the velocities in the chamber when you start the simulation at angles other than TDC. Here the default value is **1.3**. The following equations show how the swirl number is calculated.

$\vec{v} = \vec{r} \times \vec{w}$

where

$\vec{v}$ = velocity

$\vec{r}$ = radial distance

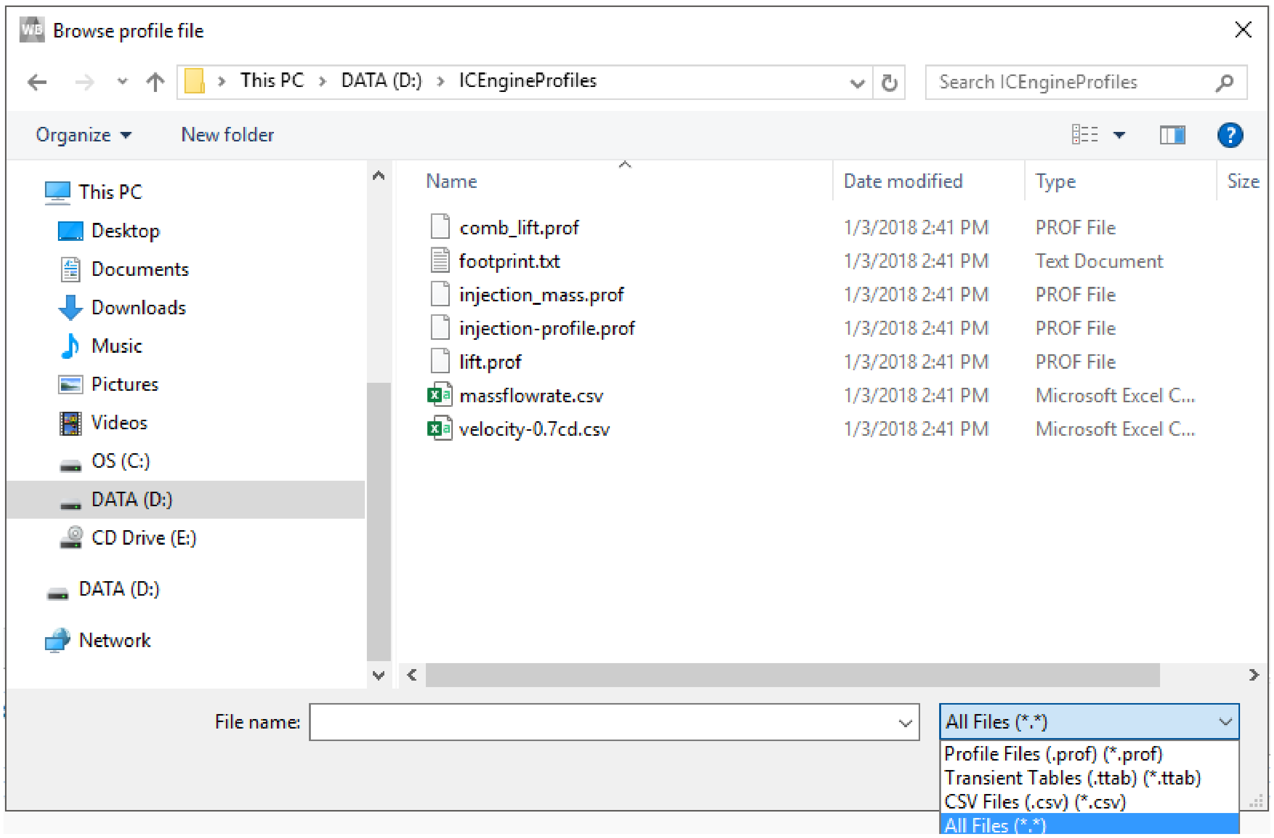
$$|\omega| = Swirl\ \ Number\ \times \frac{2\times\pi\times RPM}{60}$$

$$\vec{\omega} = |\omega|.\ cyl.axis$$

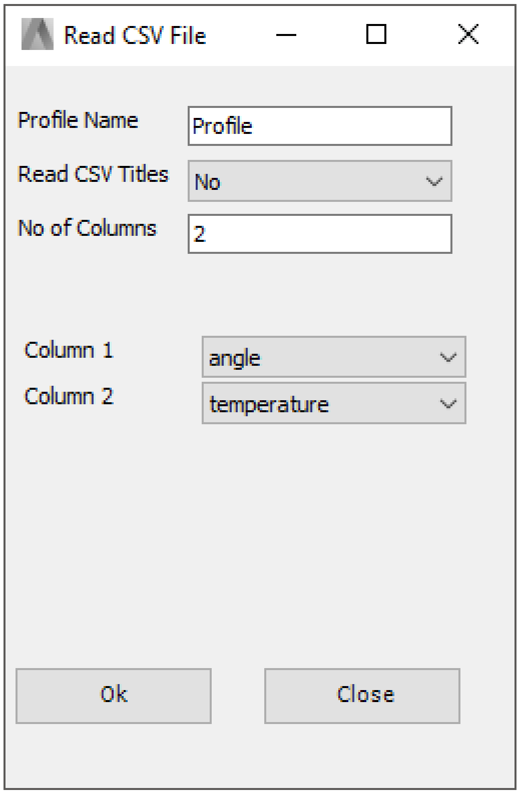
$$\vec{\omega} = (|\omega|)(cyl\_axis\_x\ \hat{i}\ +\ cyl\_axis\_y\ \hat{j}\ + cyl\_axis\_z\ \hat{k}\ )$$

Read Profile File

allows you to read multiple profile files. Clicking on **Read Profile File** opens **Browse profile file** dialog box. In the browsing window three types of file extensions will be supported `.prof` (profile file), `.ttab` (transient table), and `.csv`. **All Files** option is also present if you have a file extension which does not match with either of the given extensions. In this case the file type will be automatically identified, and an error will be thrown if the format is not supported.



If you select a `.csv` file and click **Open**, then a **Read CSV File** dialog box opens.



- You need to enter a name for **Profile Name**. This name will appear in the drop-down list of **Profiles** in the **Profile Editor** dialog box.
- If you retain the default setting of **Yes** for **Read CSV Titles** then the quantity or variable names will be as per the names in the CSV file.
- If you choose **No** for **Read CSV Titles** then you have to specify the **No. of Columns** of the CSV file you want to read. For each column you have to select a different variable name from the drop-down list. In this case the titles of the CSV columns will not be read. Your selections for the columns will be the titles.

Important:

– All the columns in CSV should have same number of values. Variable number of values and interpolation is not supported in the current version.

– Ensure that there are no empty spaces in the titles.

The format of the standard profile file is

```
((profile-name transient n periodic?)
(field_name-1 a1 a2 a3 .... an)
(field_name-2 b1 b2 b3 .... bn)
.
.
.
.
(field_name-r r1 r2 r3 .... rn))
```

The profile name as well as the field names have to be shorter than 64 characters. One of the `field_name` should be used for the `time` field, and the `time` field section *must* be in ascending order. n is the number of entries per field. The `periodic?` entry indicates whether or not the profile is time-periodic. Set it to `1` for a time-periodic profile, or `0` if the profile is not time-periodic.

An example is shown below:

```
((sampleprofile transient 8 0 0)
(angle
4.400000e+02 4.412000e+02
4.644900e+02 4.656900e+02
6.400000e+02
6.412000e+02 6.480800e+02 6.492800e+02)
(mass-flow
0.000000e+00 2.450040e-03 2.450040e-03 0.000000e+00 0.000000e+00
2.475870e-03 2.475870e-03 0.000000e+00)
(velocity
0.000000e+00 1.941520e+02 1.941520e+02 0.000000e+00 0.000000e+00
1.922430e+02 1.922430e+02 0.000000e+00)
)
```

Important:

All quantities, including coordinate values, must be specified in SI units because Ansys Fluent does not perform unit conversion when reading profile files. Also, profile names must have all lowercase letters (for example, `name`). Uppercase letters in profile names are not acceptable.

The format of the transient table file is

```
profile-name n_field n_data periodic?
field-name-1 field-name-2 field-name-3 .... field-name-n_field
v-1-1  v-2-1... ... ... ... v-n_field-1
v-1-2  v-2-2... ... ... ... v-n_field-2
.
.
.
.
.
v-1-n_data v-2-n_data ... ... ... ... v-n_field-n_data
```

The first field name (for example `field-name-1`) should be used for the `time` field, and the `time` field section, which represents the flow time, *must* be in ascending order. The `periodic?` entry indicates whether or not the profile is time-periodic. Set it to `1` for a time-periodic profile, or `0` if the profile is not time-periodic.

An example is shown below:

```
sampletprofile 3 8 0
angle        mass-flow    velocity
440          0.0          0.0
441.2        0.00245004   194.152
464.49       0.00245004   194.152
465.69       0.0          0.0
640          0.0          0.0
641.2        0.00247587   192.243
648.08       0.00247587   192.243
649.28       0.0           0.0
```

This file defines the same transient profile as the standard profile example above.

Important:

All quantities, including coordinate values, must be specified in SI units because Ansys Fluent does not perform unit conversion when reading profile files. Also, profile names must have all lowercase letters (for example, `name`). Uppercase letters in profile names are not acceptable. When choosing the field names, spaces or parentheses should not be included.

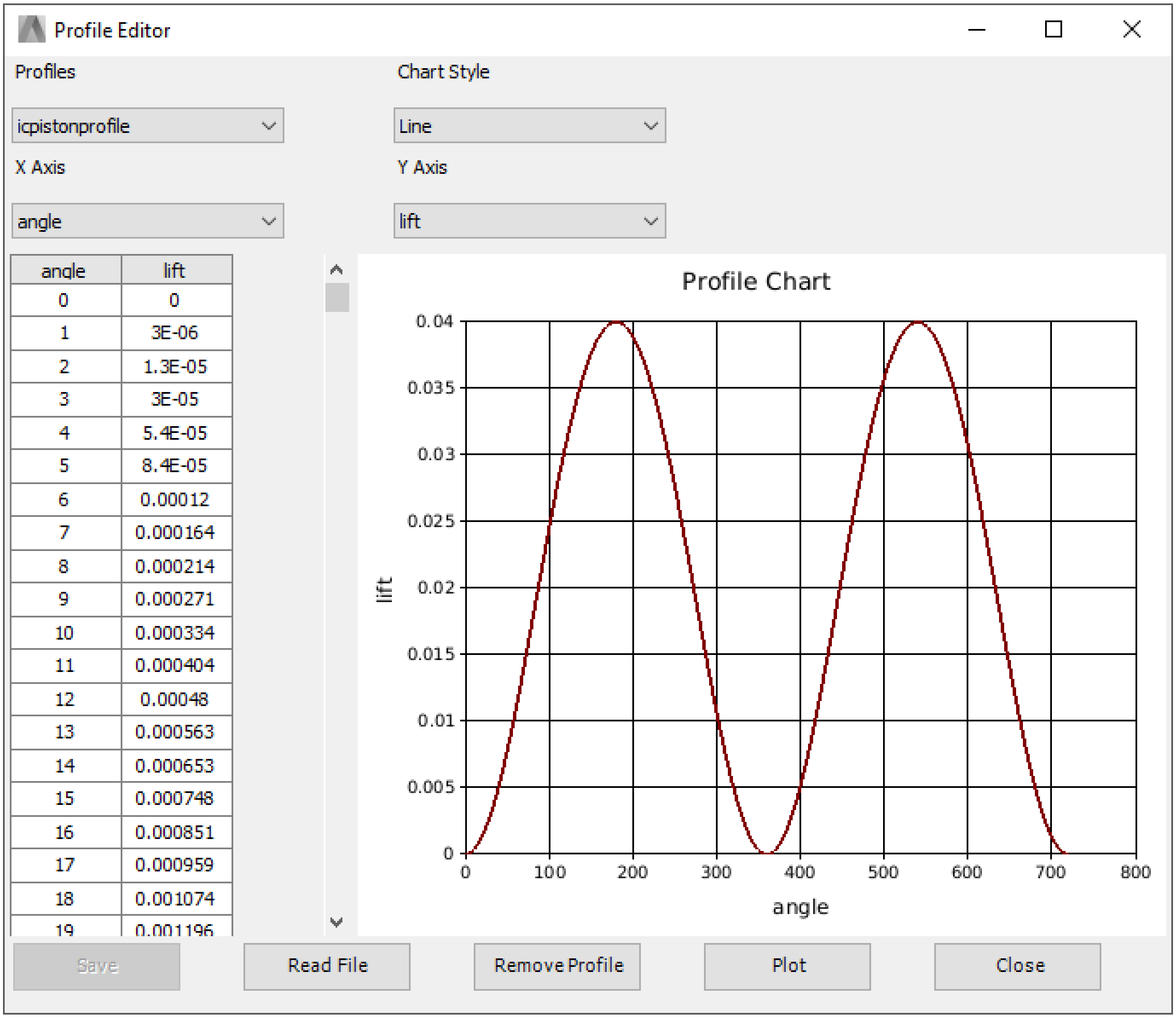
Note:

The independent variable or **field-name–1** for all the types of profile files should be either angle or x or y or z coordinate.

After reading the files the profiles will be available in the boundary condition drop-down lists.

Profile Editor

allows you to view the plot of the profiles.



- Select a profile from the drop-down list of **Profiles**. The **TIMESTEP** profile is added to the list by default.
- When a particular profile is selected all corresponding variables will be displayed in the **X Axis** and **Y Axis** drop-down lists.
- Select the variable for **X Axis** and **Y Axis** and click **Plot**. The plot will be displayed in the area of **Profile Chart**.
- You can select the type of chart to be displayed from the **Chart Style** drop-down list. Options of **Spline**, **Step** and **Line** are provided.
- You can also read a profile by clicking on **Read File**.
- The read profile which is presently selected can be deleted from the list by clicking on **Remove Profile**.

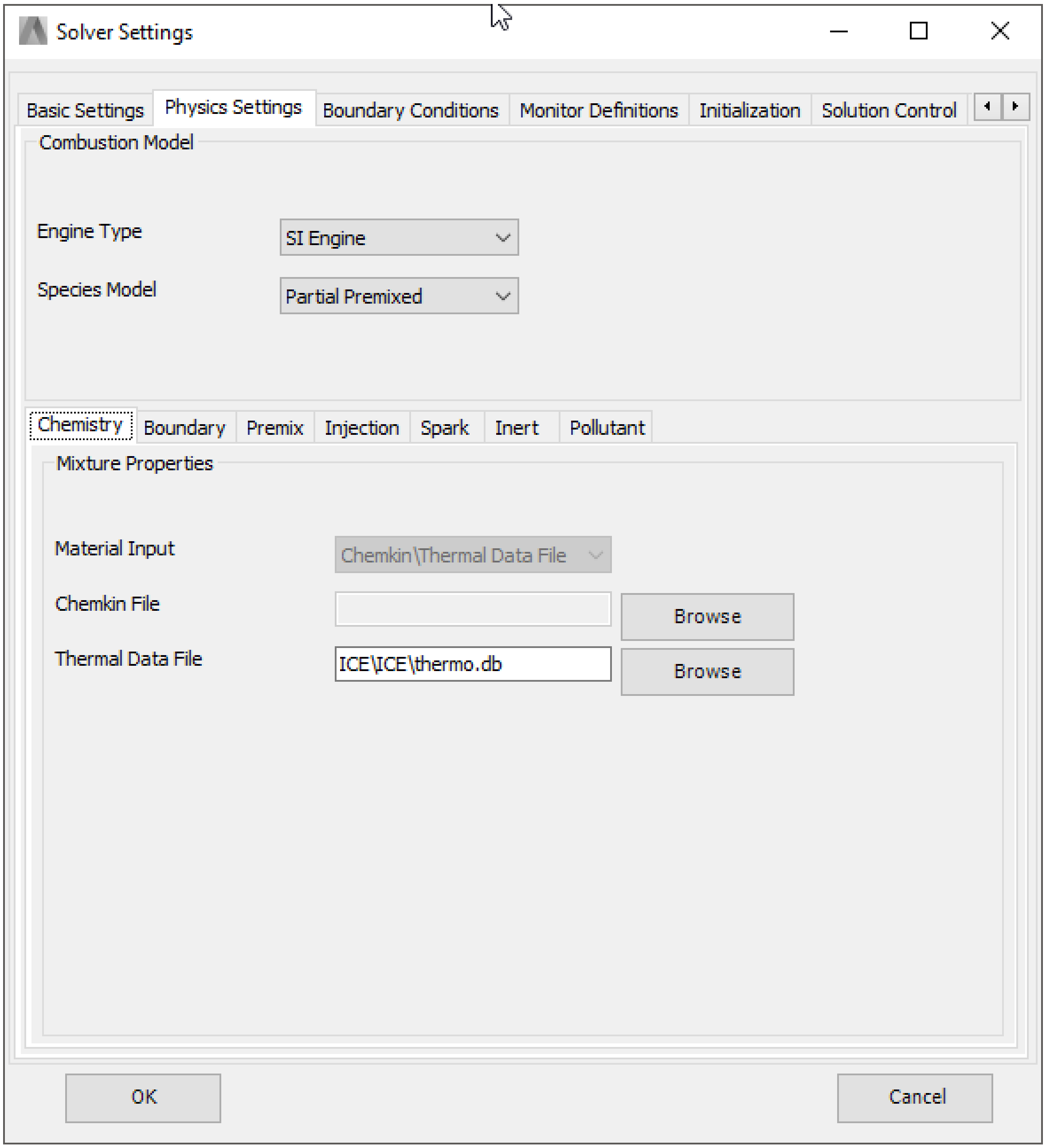
- You can make changes to the values in the profile table displayed. Click **Plot** to view the **Profile Chart** with the changed values. You can save the profile with the edited values by clicking on **Save**.
- To manipulate the chart:

Table 12.1: Chart Manipulation

	Operation
Rotate Middle Mouse	Zoom
Shift + Middle Mouse	Zoom
Ctrl + Middle Mouse	Pan
Drag Right Mouse	Box Zoom
F key	Fit to Window

12.1.2. Physics Settings

The **Physics Settings** tab is present only for combustion simulation.



Combustion Model

Engine Type

If you select Full Engine Full Cycle (p. 143) from the **Combustion Simulation Type** in the **Properties** pane of **ICE** cell, then you can select the **Engine Type** as **CI Engine** or **SI Engine**. Else, if you select **Sector** or **Full Engine IVC to EVO** the **Engine Type** selected by default is **CI Engine**.

Species Model

Depending upon the selection of the **Engine Type** the choice for **Species Model** changes.

- **Laminar Finite Rate** option is present for **CI Engine** as well as for **SI Engine**. For more information about this model see The Laminar Finite-Rate Model in the Fluent Theory Guide.
- **Diesel Unsteady Flamelet** option is present only for **CI Engine**. For more information about this model see Using the Diesel Unsteady Laminar Flamelet Model in the Fluent User's Guide.
- **Partially Premixed** option is present only for **SI Engine**. For more information about this model see the chapter on Modeling Partially Premixed Combustion in the Fluent User's Guide. Details are also present in the chapter Partially Premixed Combustion in the Fluent Theory Guide.

Depending upon your selection of the **Engine Type** and **Species Model**, the tabs activated in the lower half of the **Physics Settings** tab differ.

Chemistry

In the **Chemistry** tab you can set the **Mixture Properties**.

Material Input

For **Laminar Finite Rate** you can choose **Fluent** or **Chemkin\Thermal Data File** from the drop-down list. Else the default selection is **Chemkin\Thermal Data File** which you cannot modify.

Mixture Material

If you choose **Fluent** then a few mixture materials that have been chosen from the Fluent materials database will be available to choose from in the **Mixture Material** drop-down list.

Chemkin File

This allows you to read a chemkin file in which you can provide your own mixture material. This option is available when you choose **Chemkin File** from the **Material Input** drop-down list. Click **Browse** to read the file.

Thermal Data File

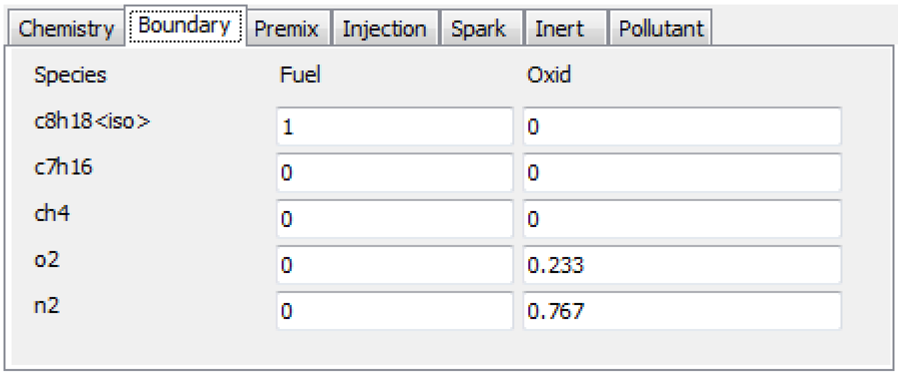
This allows you to read a thermodynamic data file. This option is available when you choose **Chemkin File** from the **Material Input** drop-down list or when the **Species Model** is set to **Partially Premixed**. By default the `Thermal Data File` from the Fluent installation is chosen. Click **Browse** to change the file name.

Depending upon the choices you have made for the **Mixture Properties**, the list in the **Initialization** tab changes.

Boundary

In this tab you can set the chemical boundary species in consideration. This tab is activated only for the following combinations:

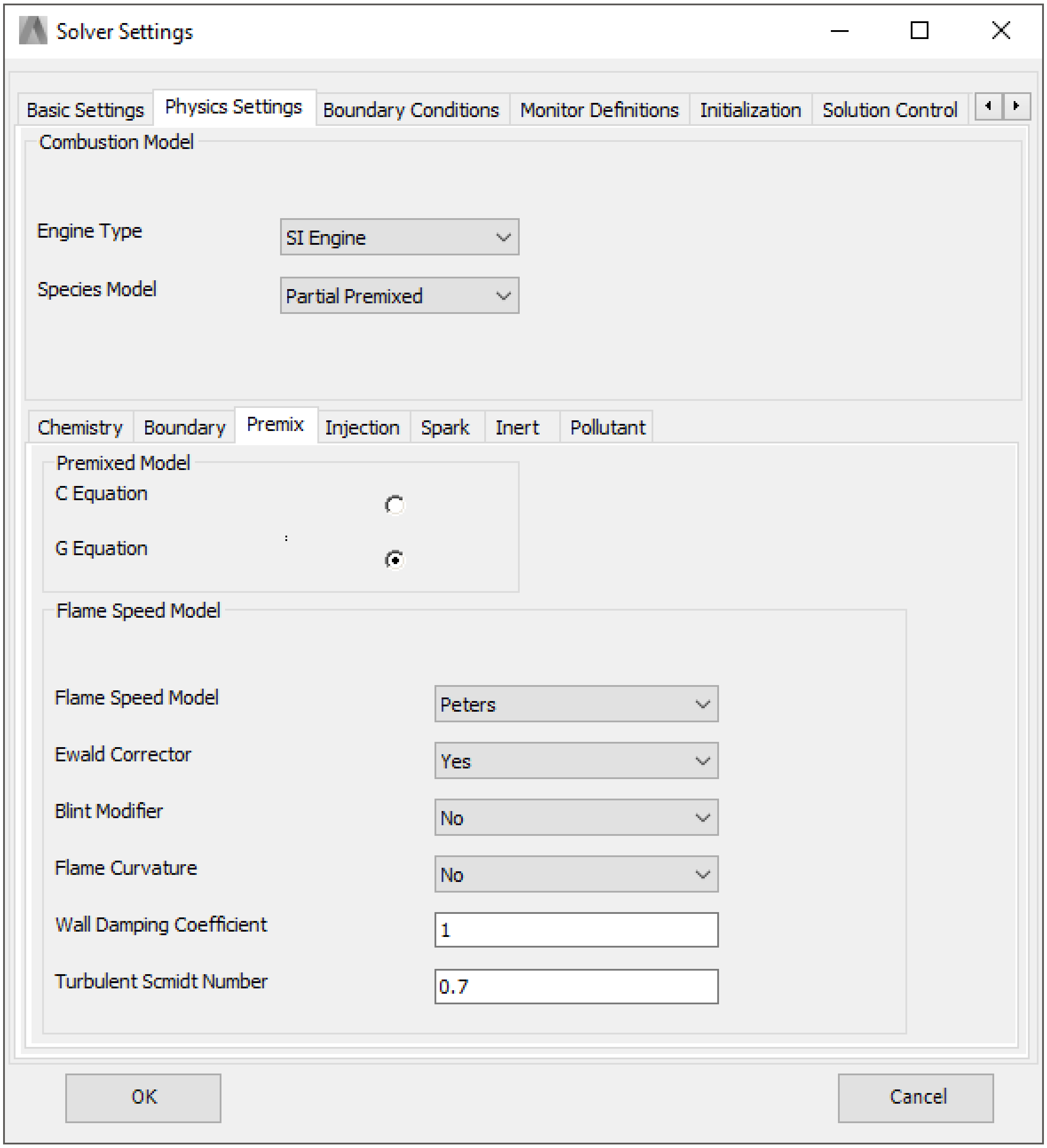
- Selection of **CI Engine** from **Engine Type** drop-down list and selection of **Diesel Unsteady Flamelet** from the **Species Model** drop-down list.
- Selection of **SI Engine** from **Engine Type** drop-down list and selection of **Partial Premixed** from the **Species Model** drop-down list.



Depending upon your selections in the **Chemistry** tab the species will be displayed in the **Boundary** tab. If you have chosen **Fluent** then the species of the selected **Mixture Material** will be displayed along with their set **Fuel** and **Oxid** values. For a Chemkin file you will have to set the values.

Premix

The **Premix** tab is activated when you select **Partial Premixed** from the **Species Model** drop-down list for **SI Engine**.



Premixed Model

You can select **C Equation** or **G Equation** in the **Premixed Model** group box.

Flame Speed Model

In the **Flame Speed Model** group box you can choose the turbulent flame speed model. Depending upon the choice the further options change.

Flame Speed Model

choose from **Peters** or **Zimont** from the drop-down list.

Ewald Corrector

is set to **Yes** by default and described in Peters Flame Speed Model in the Fluent Theory Guide. This option is present only when **Peters** is selected form the **Flame Speed Model** drop-down list.

Blind Modifier

is set to **No** by default. This term is present only for **Peters Flame Speed Model**.

Flame Curvature

is set to **No** by default. If it is set to **Yes** only then the curvature source term in the G-Equation is included. For details check G-Equation Model Theory in Fluent Theory Guide. This term is present only for **Peters Flame Speed Model**.

Turbulent Length Scale Constant

value is set to **0.37** by default. This term is present only for **Zimont Flame Speed Model**.

Turbulent Flame Speed Constant

value is set to **0.52** by default. This term is present only for **Zimont Flame Speed Model**.

Stretch Factor Coefficient

value is set to **0.26** by default. This term is present only for **Zimont Flame Speed Model**.

Wall Damping Coefficient

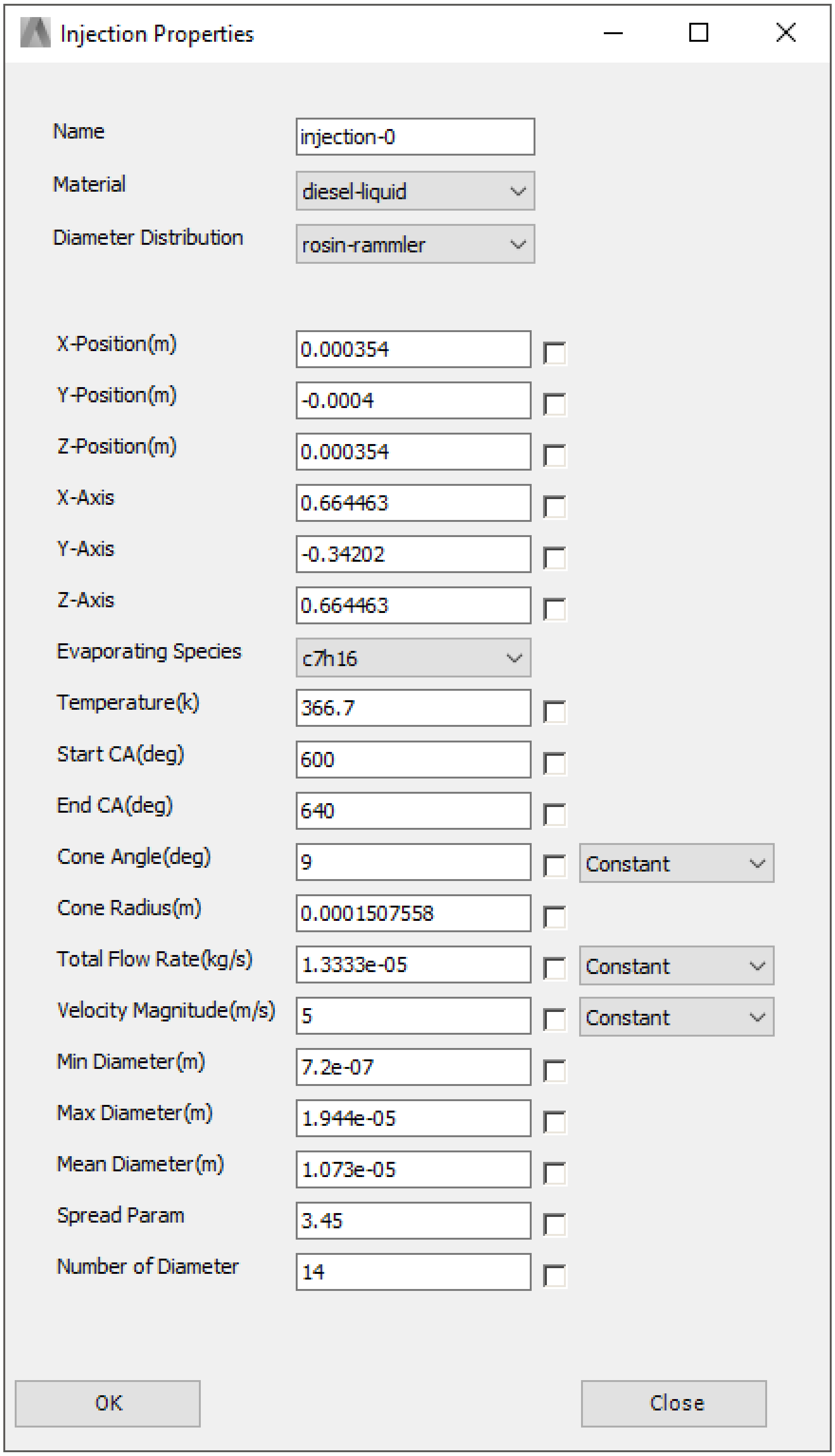
value is set to **1** by default.

Turbulent Schmidt Number

value is set to **0.7** by default.

Injection

In the **Injection** tab you can see that an injection has been setup. Select **injection-0** and click **Edit** to check the injection properties and settings. For more details on the injection settings check the Set Injection Properties (p. 441) dialog box in Ansys Fluent.



Material

indicates the material for the particles. You can choose from the drop-down menu. If this is the first time you select a particle of this type, you can choose from all of the materials by copying them from the database or creating them from scratch. **diesel-liquid** is the material chosen by default.

Diameter Distribution

allows you to change from **uniform** method used to determine the size of the particles in a **surface** injection, to the **rosin-rammler** method.

X, Y, and Z Position

indicates the x, y and z position of the injection source/nozzle tip.

X, Y, and Z Axes

indicates the x, y and z components of the vector defining the cone's axis.

Evaporating Species

(for **droplet** particles) specifies the gas-phase species created by the vaporization and boiling laws.

Temperature

indicates the initial (absolute) temperature of the injected particle stream.

Start CA

indicates the crank angle at which the injection will start.

End CA

indicates the crank angle at which the injection will stop.

Cone Angle

indicates half of the spray cone angle.

Cone Radius

indicates a non-zero inner radius to model injectors that do not emanate from a single point. The particles will be distributed about the axis with the specified radius.

Total Flow Rate

is the total mass flow rate of all the streams in the group.

Velocity Magnitude

is the velocity magnitude of the particle streams that will be oriented along the specified spray cone.

Min. Diameter

is the smallest diameter to be considered in the size distribution.

Max. Diameter

is the largest diameter to be considered in the size distribution.

Mean Diameter

is the size parameter in the Rosin-Rammler equation.

Spread Parameter

This is the exponential parameter, n, in the Rosin-Rammler equation.

Number of Diameter

indicates the number of diameters in each distribution (that is, the number of different diameters in the stream injected from each face of the surface).

If you have read a profile before then you can select the profile from the drop-down list while setting some of the variables like **Cone Angle**, **Total Flow Rate**, and **Velocity Magnitude**. You can also select **PROFILE EDITOR** option which will open the **Profile Editor** dialog box where you can check the plot of the profile or read a new profile.

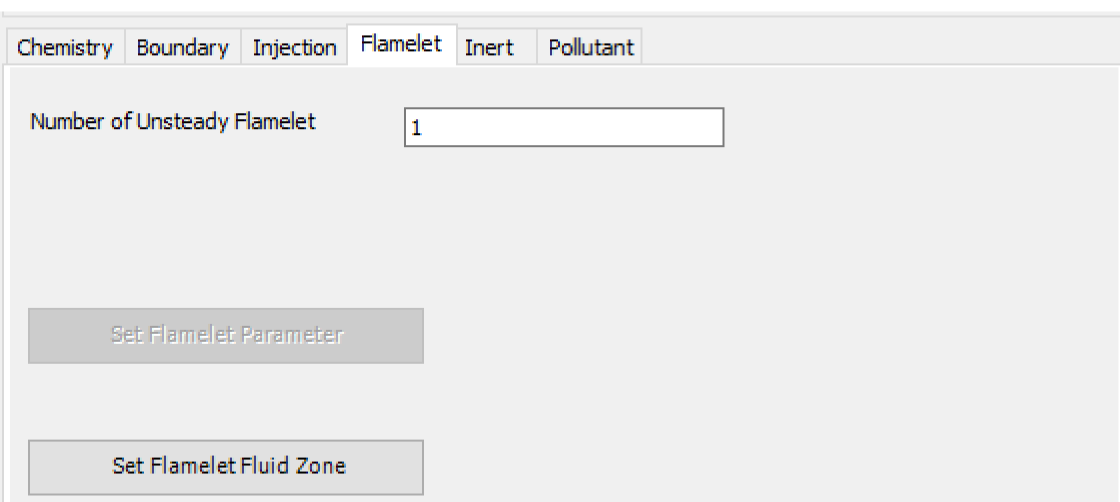
Default injection is defined based on the geometry information provided in the DesignModeler. For creating additional injections for the simulation use the **Create** or **Copy** options. Use **Edit** to change the properties of the injection.

Note:

If you need to edit multiple injections, you can select them all in the tab and click **Edit**. This will open up the **Injection Properties** dialog box which will allow you to edit the properties of all the selected injections at once. Other than the position of the injections all other properties can be edited together.

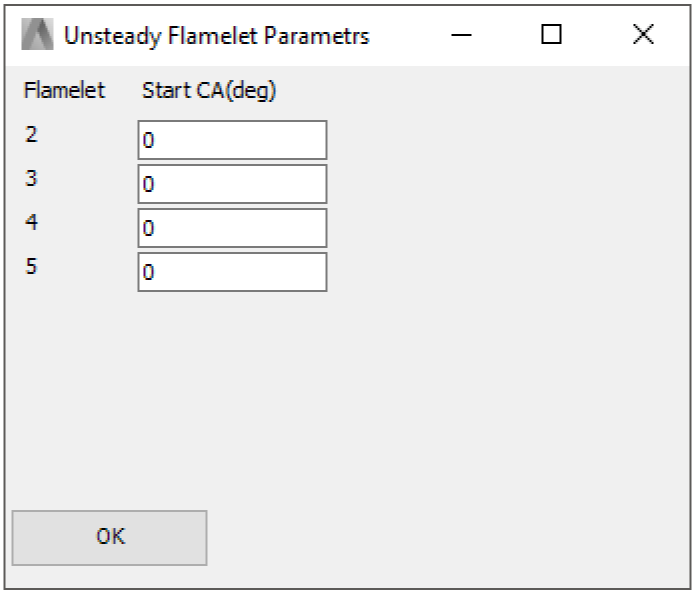
Flamelet

For the **Diesel Unsteady Flamelet** species model you will set the flamelet parameters in the **Flamelet** tab.



- One flamelet is set by default. You can set the **Number of Unsteady Flamelets** that will be generated during simulation.

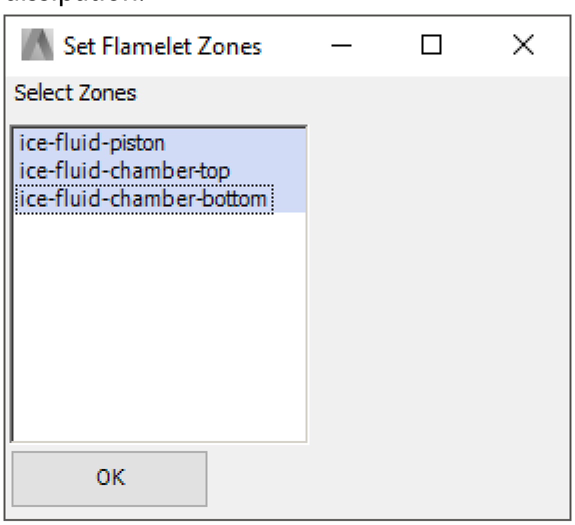
- Ansys Fluent automatically sets the start time for the first flamelet, but you must set the start time for each consecutive flamelet using the **Unsteady Flamelet Parameters** dialog box. Open it by clicking **Set Flamelet Parameter** and enter the start time for each flamelet in crank angles.

  Ansys Fluent starts simulation with only one flamelet, and then it automatically introduces new flamelets into the reacting domain at the times you have specified.

  Whenever injection is added/edited/deleted flamelet parameters will change. Flamelet will be created for each injection start angle (non repeating start crank angles when there are multiple injections).

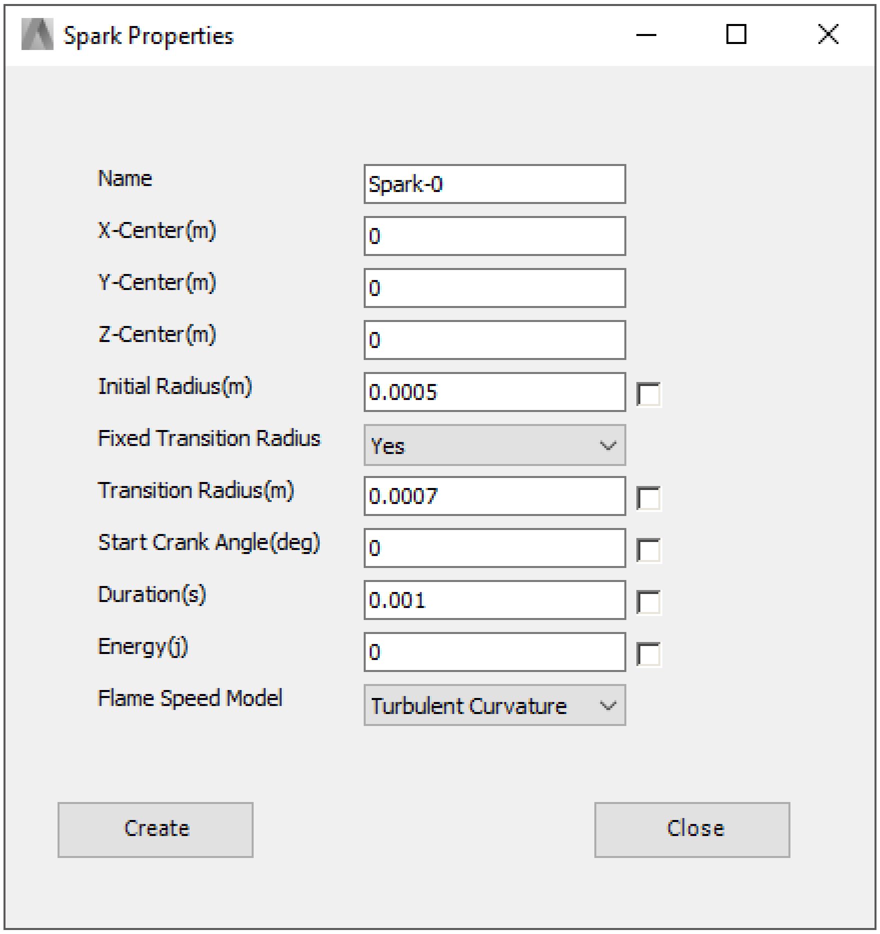
- Specify the flamelet fluid zones. Ansys Fluent calculates diesel unsteady flamelets using the zone-averaged pressure and scalar dissipation at every time step. By default, the averaging is performed over all fluid zones in the domain, but you can also select and/or deselect the fluid zones using the **Set Flamelet Zones** dialog box. Open this dialog box by clicking **Set Flamelet Fluid Zone** and select the fluid zones to be used for calculating average pressure and scalar dissipation.

  If no fluid zone is selected, Ansys Fluent will compute domain average pressure and scalar dissipation using all fluid zones.

Spark

This tab is present only when **SI Engine** is selected from the **Engine Type** drop-down list. It allows you to define multiple sparks. Click **Create** to open the **Spark Properties** dialog box where you can enter the details required for the spark.



Name

displays the name of the spark being defined.

X, Y, and Z-Center

specifies x, y, and z coordinates of the spark center. These parameters are needed to define the location of the spark.

Initial Radius

specifies the initial spark radius. You can parameterize it by enabling the check box next to it.

Fixed Transition Radius

is set by default to **Yes**. This will allow you to specify the **Transition Radius**.

If you select **No** then Ansys Fluent will compute the transition radius. For more information see equation 13–1 and 13–2 in Spark Model Theory in the Fluent Theory Guide

Transition Radius

is set to a value here. When the radius reaches this value, Ansys Fluent automatically switches the spark flame speed model off, so that the flame speed is modeled using the flame speed model that you have selected in the **Premix** tab. You can parameterize it by enabling the check box next to it.

Start Crank Angle

is the time of spark ignition initialization in crank angle degrees. You can parameterize it by enabling the check box next to it.

Duration

is used to calculate the rate of spark energy input and the time when this input ceases. You can parameterize it by enabling the check box next to it.

Energy

contains the total energy input by the spark. The default energy value is 0. A positive value for energy will cause the temperature of the spark kernel to rise above the combustion process temperature. You can parameterize it by enabling the check box next to it.

Flame Speed Model

allows you to select the turbulent flame speed model for controlling the rate at which the flame front moves.

Turbulent Curvature

includes the effect of flame curvature as specified in Turbulent Curvature in the Fluent Theory Guide.

Turbulent Length

neglects the effects of flame curvature on the flame speed as described in Turbulent Length in the Fluent Theory Guide.

Herweg-Maly

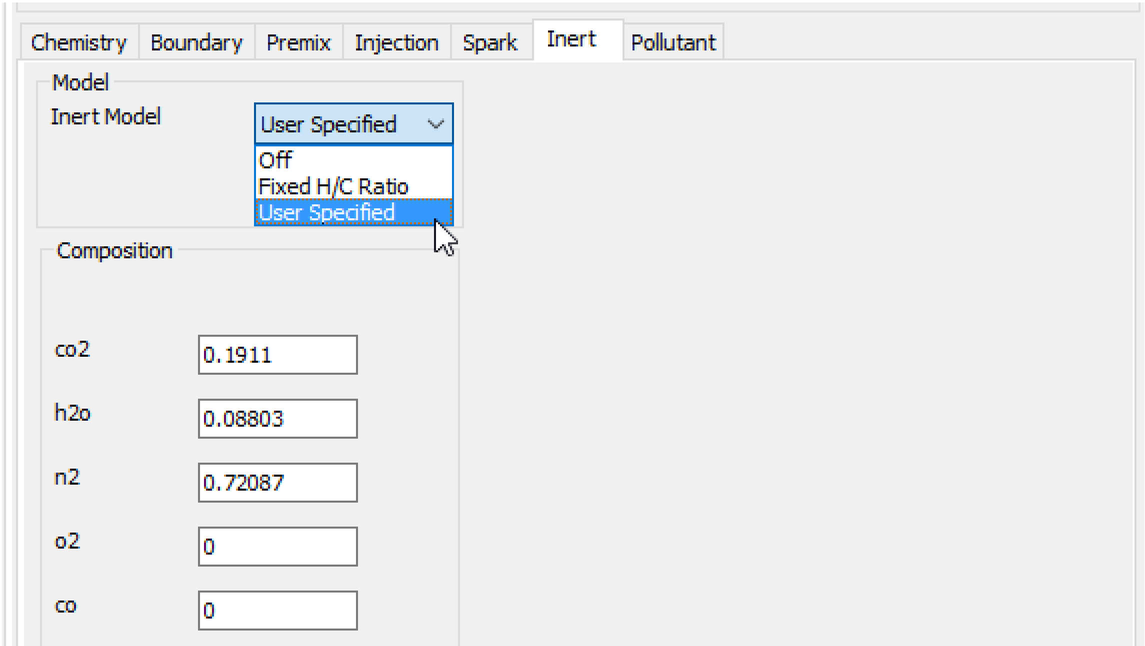
calculates the turbulent flame speed using Herweg-Maly model. See Herweg-Maly in the Fluent Theory Guide.

Laminar

specifies the turbulent flame speed as the laminar flame speed.

Inert

The **Inert** tab is activated for **Partially Premixed** and **Diesel Unsteady Flamelet** Species Model.



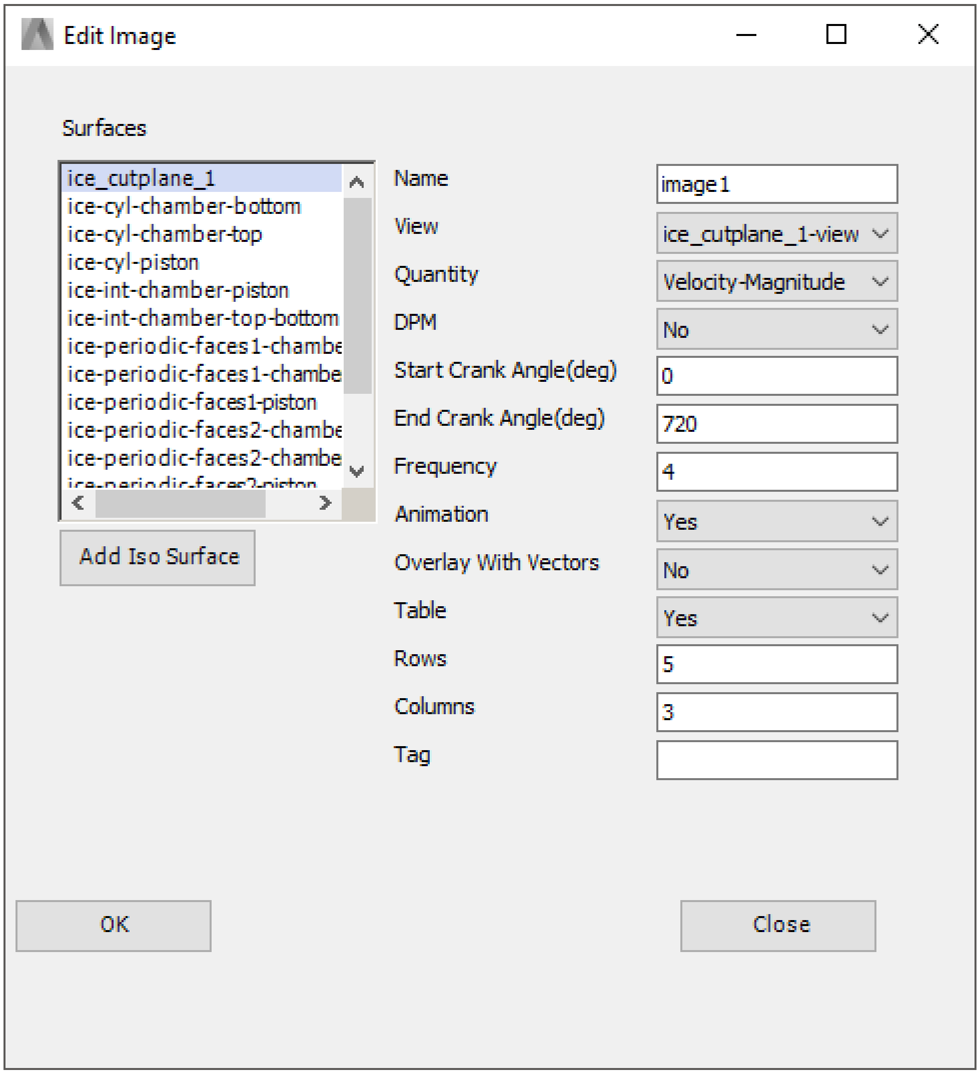
In the **Inert** tab you have three options to choose from the **Inert Model** drop-down list.

Off

Inert Model is not activated.

Fixed H/C Ratio

Select this option if the hydrogen to carbon ratio is known. For example, for methane (CH_4) enter 4 for **Fixed H/C Ratio**.



Setting the H/C Ratio assumes that the burned gas resulted from the complete, stoichiometric combustion of that hydrocarbon fuel with air, and the only products of the combustion are CO_2, H_2) and N_2.

User Specified

Select this option if you want to specify an arbitrary composition for the inert stream. You can specify the mass fraction for the four species in the default list: CO_2, H_2O, N_2, and O_2. Ensure that the sum of the mass fractions add up to 1.

You will need to set appropriate boundary conditions at flow inlets and exits for the inert tracer mass fraction. The tracer species mass fraction must be between zero and one, with the value of one meaning that all of the material entering the domain comes from the inert stream. The values for flow boundaries are set in the **Inert Stream** field. This is under the **Species** tab of the inlet or outlet boundary condition dialog box which can be accessed in the **Boundary Conditions** tab.

You will also need to initialize the **Inert Variable** in the **Initialization** tab. For details check Initializing the Inert Stream in Fluent User's Guide.

You can also patch the inert mass fraction by entering an appropriate value. The **Inert Variable** is available under the **Variable** list in the **Patching Zones** dialog box. For details check Inert Fraction in Fluent User's Guide.

Ansys Fluent allows burned gases to be converted to an inert gas. This has been designed with in-cylinder combustion in mind to aid the simulation of multiple cycles of such engines using the partially-premixed combustion model with exhaust gas recirculation (EGR). At the end of a combustion stroke, the premixed progress variable of the burnt gases is unity. When fresh charge, with progress variable of zero, is mixed with the burnt trapped gases, combustion will be initiated unless these trapped gases are converted to inert.

This facility is accessed via the **Dynamic Mesh Events**. There, you will need to set the crank angle at which the event occurs (usually shortly before the inlet valves open) and at the fluid zones (usually the combustion chamber) where the burnt gases are converted to inert.

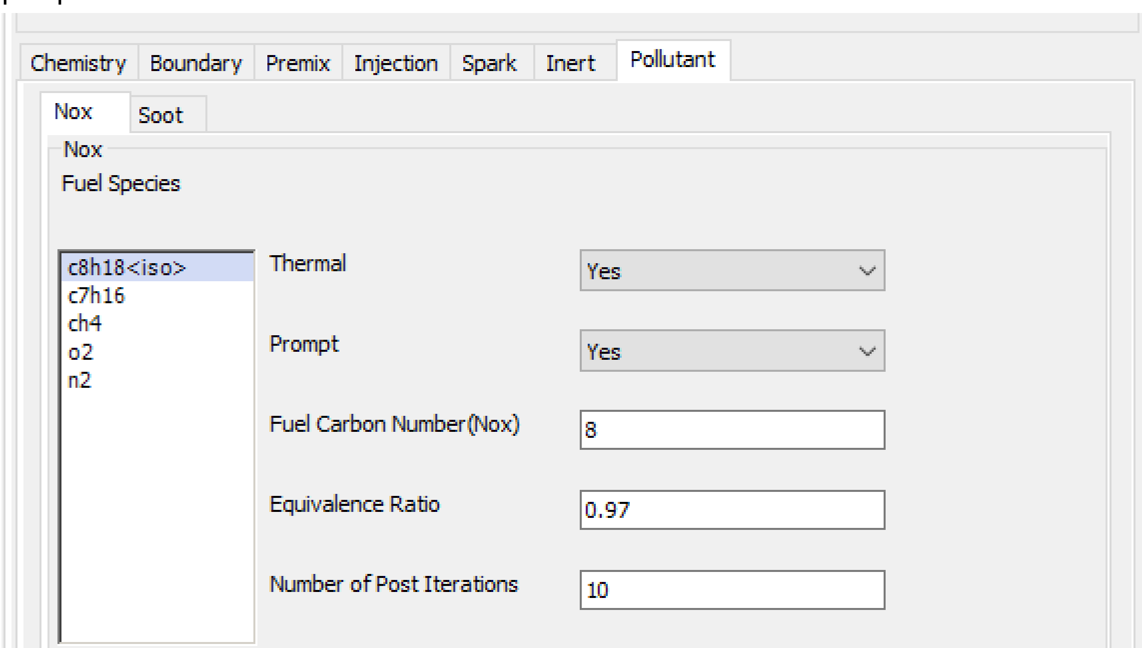
In Fluent in **Dynamic Mesh Events** an **Inert EGR Reset** (p. 450) event is added. The event is set before IVO. When multiple invalves are present then this event is set for the invalve whose opening crank angle comes first. The **Zones** set for this event are the chamber zones. For more information go to Resetting Inert EGR in Fluent User's Guide.

Pollutant

In the **Pollutant** tab you can use the Nox and Soot Model for modeling Soot formation. Under the **Pollutant** tab are two tabs: **Nox** and **Soot**.

Nox

Nox concentrations generated in combustion systems are generally low. As a result, chemistry has negligible influence on the predicted flow field, temperature, and major combustion product concentrations. It follows that the most efficient way to use the Nox model is as a postprocessor to the main combustion calculation.



Thermal

To enable thermal Nox select **Yes** from the **Thermal** drop-down list.

Prompt

To enable prompt Nox select **Yes** from the **Prompt** drop-down list.

- Set the **Fuel Carbon Number** to specify the number of carbon atoms per fuel molecule.
- Set the **Equivalence Ratio** as follows:

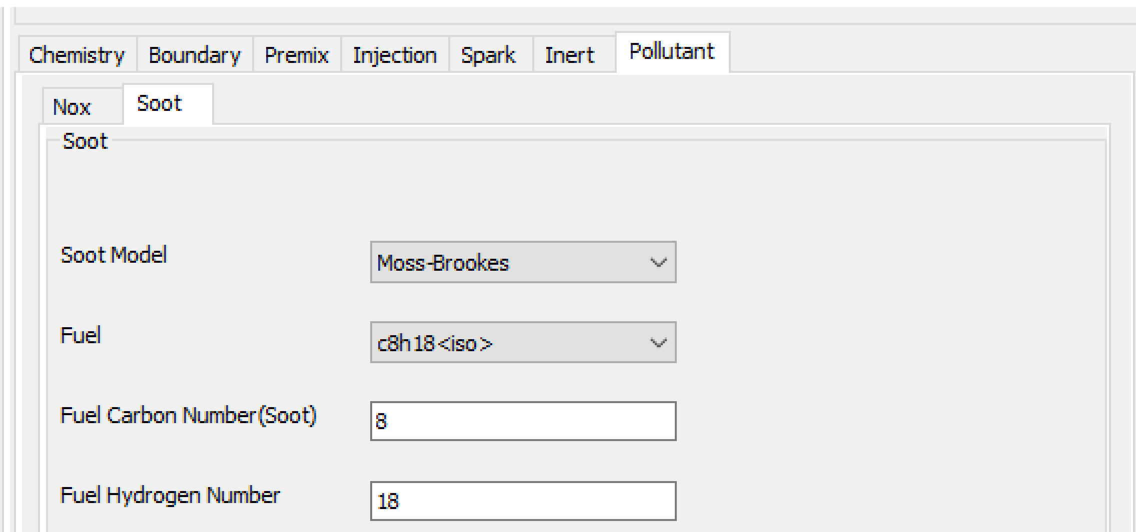
$$\text{Equivalence Ratio}=\frac{\text{actual fuel-to-air ratio}}{\text{stoichiometric fuel-to-air ratio}} \tag{12.1}$$

Number of Post Iterations

When either of **Thermal** or **Prompt** is enabled then the **Number of Post Iterations** is set to **10**. This will be set in the Ansys Fluent **Run Calculation** task page.

Soot

When the mass fraction of soot is relatively large (for example, 10%) the soot formation should be computed as part of the main combustion solution and not through postprocessing.



Soot Model

You can enable and set up the Moss-Brookes soot formation models by selecting **Moss-Brookes** from the **Soot Model** drop-down list.

Fuel

Select the fuel from the **Fuel** drop-down list. If you are using the non-premixed model for the combustion calculation and your fuel stream consists of a mixture of components, you should choose the most appropriate species as the **Fuel** species for the soot formation model.

Fuel Carbon Number (Soot)

After selecting **Fuel** species enter the related **Fuel Carbon Number** for use in the mixture fraction calculation.

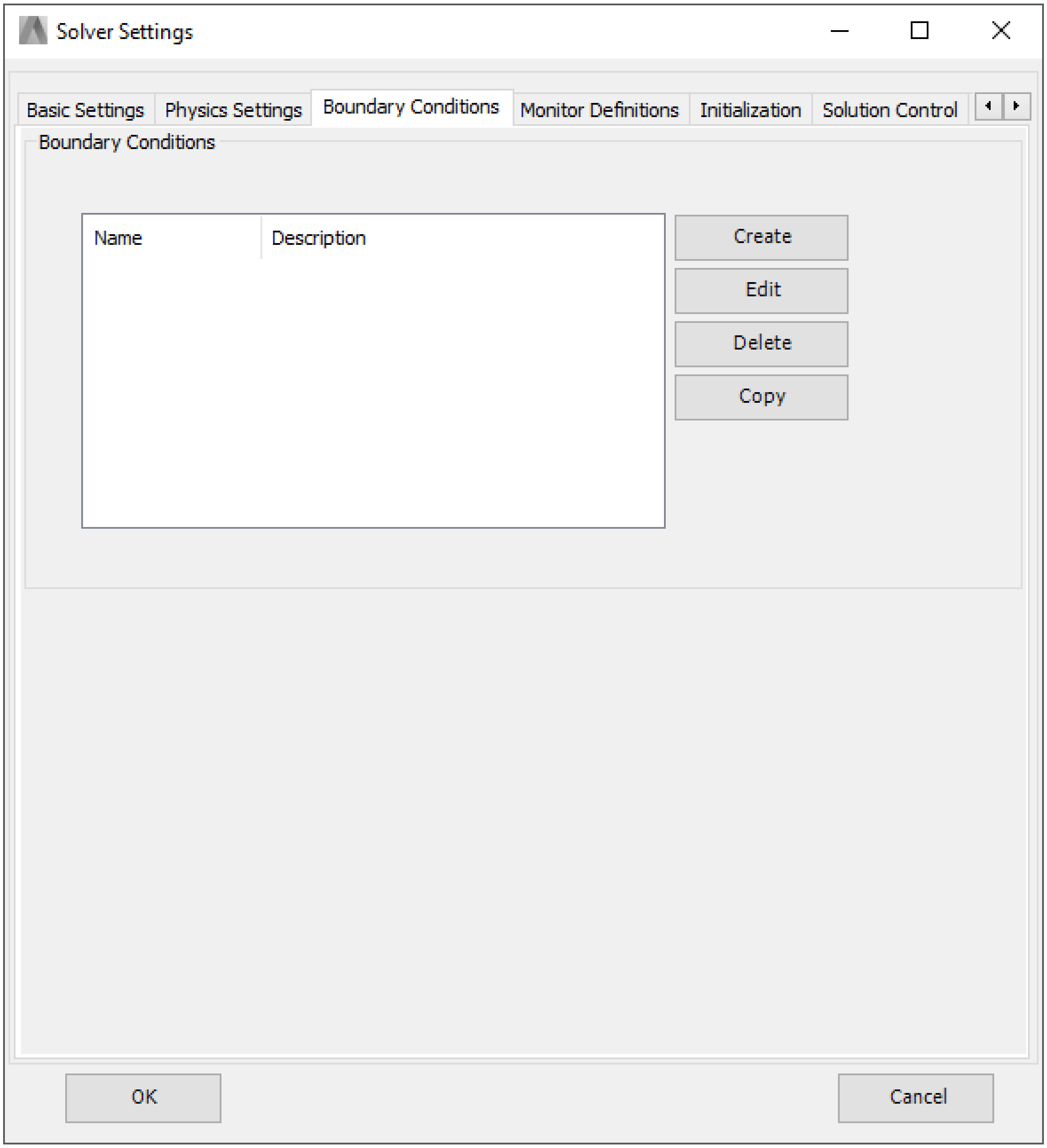
Fuel Hydrogen Number

After selecting **Fuel** species enter the related **Fuel Hydrogen Number** for use in the mixture fraction calculation.

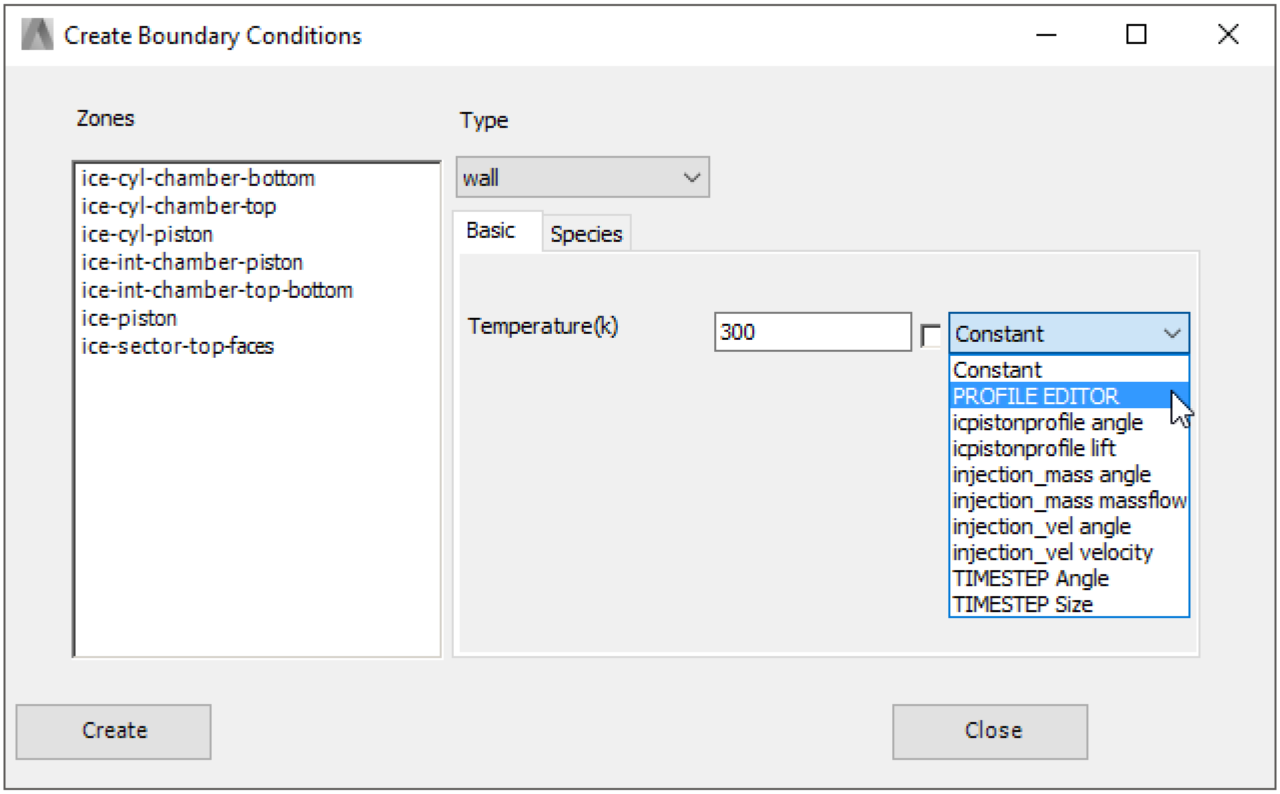
Note:

When **Partially Premixed** combustion model is chosen then both Soot and Nox will be PDF mixture. If Chemkin is chosen then both Soot and Nox will be PDF temperature beta 20 with transported variance. These settings are done internally.

12.1.3. Boundary Conditions



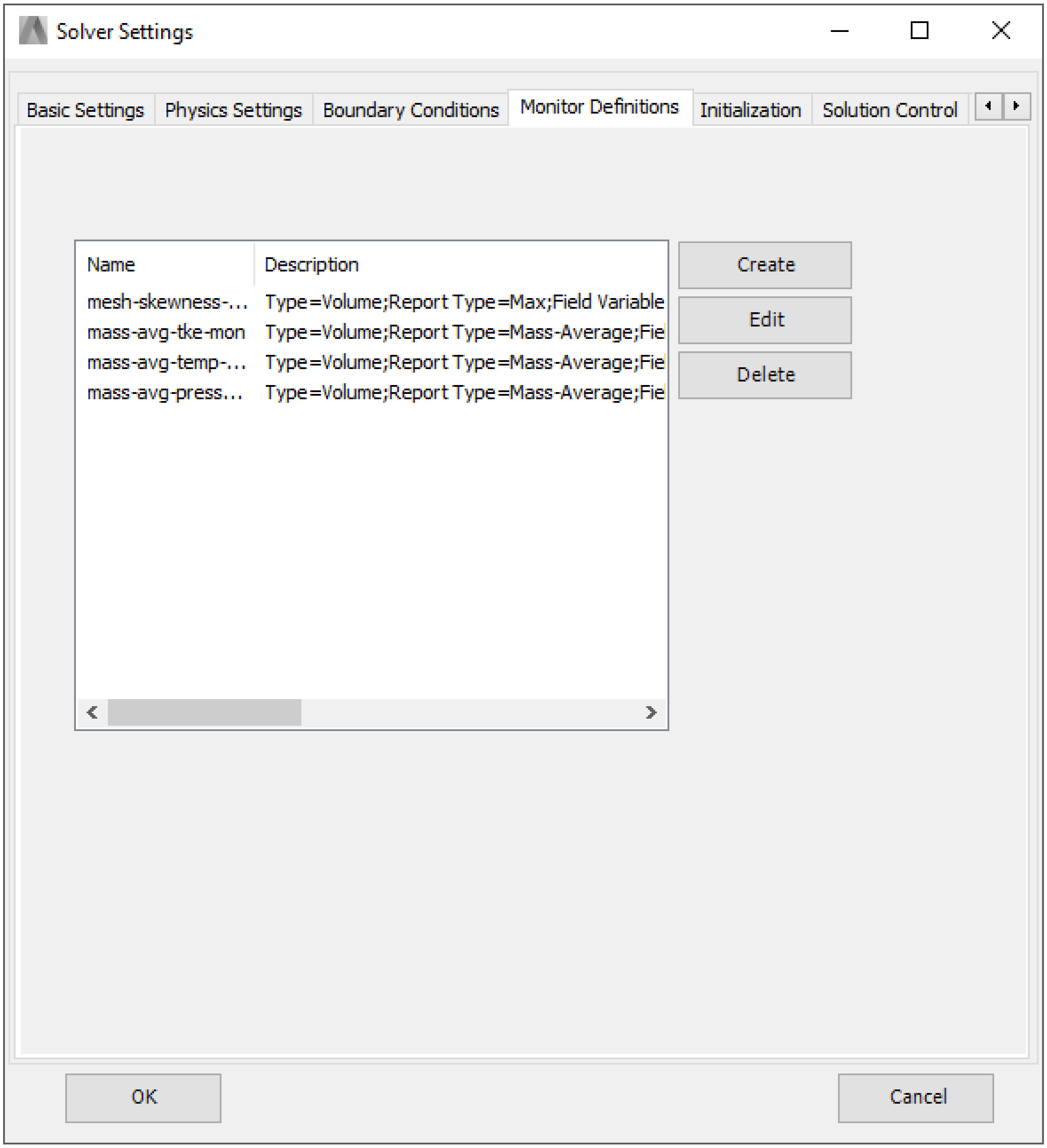
In the **Boundary Conditions** tab you can see the boundary conditions set depending upon the type of combustion simulation you have chosen. You can use the **Create**, **Edit**, or **Copy** options to change the default settings.



In the **Create Boundary Conditions** dialog box select the zone to which you like to apply the boundary conditions to from the list under **Zones**. Then select **Type** and enter the required values for the variables, either a constant value or a variable profile from the drop-down list. You can select a profile in case you have read a profile before, or you can select **PROFILE EDITOR** option which will open the **Profile Editor** dialog box where you can check the plot of the profile or read a new profile. After you click **OK** you can see the zone name and the details in the **Boundary Conditions** tab.

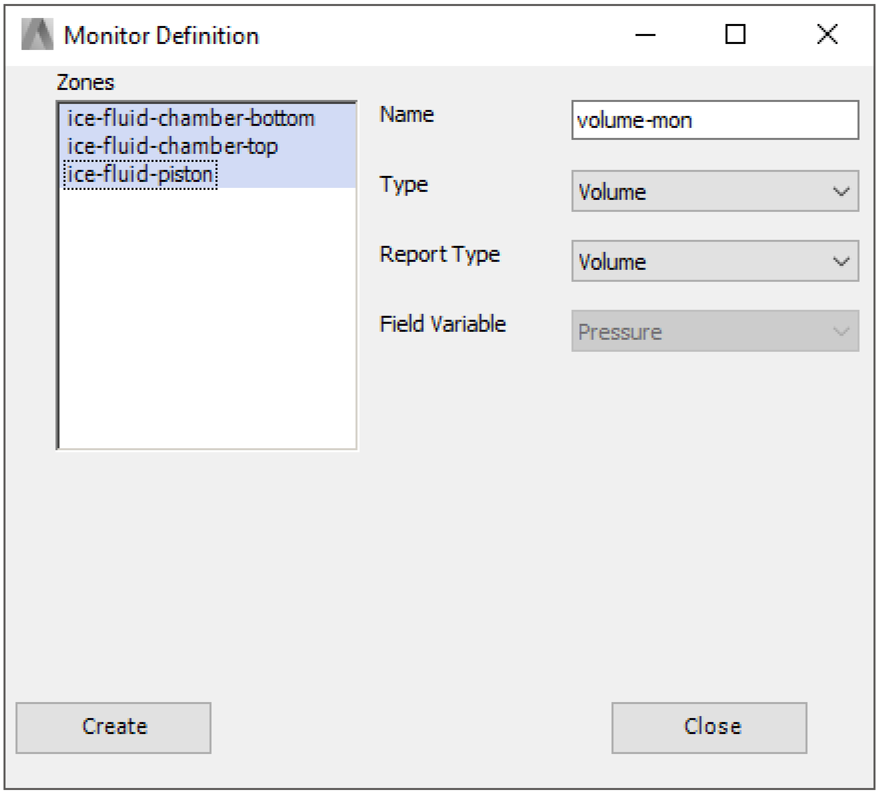
For the boundary conditions, you can parameterize the quantity by enabling the check box next to it.

12.1.4. Monitor Definitions



In the **Monitor Definitions** tab you can see that several monitors have been set. Depending upon the selection of **Combustion Simulation Type** the list will vary. For **Full Engine Full Cycle** and **Full Engine IVC to EVO** most monitor types will be same as for **Cold Flow Simulation**. Select each monitor and click **Edit** to check the details. Listed below are some default monitors for **Sector** type.

- **vol-mon** plots the **Volume**.

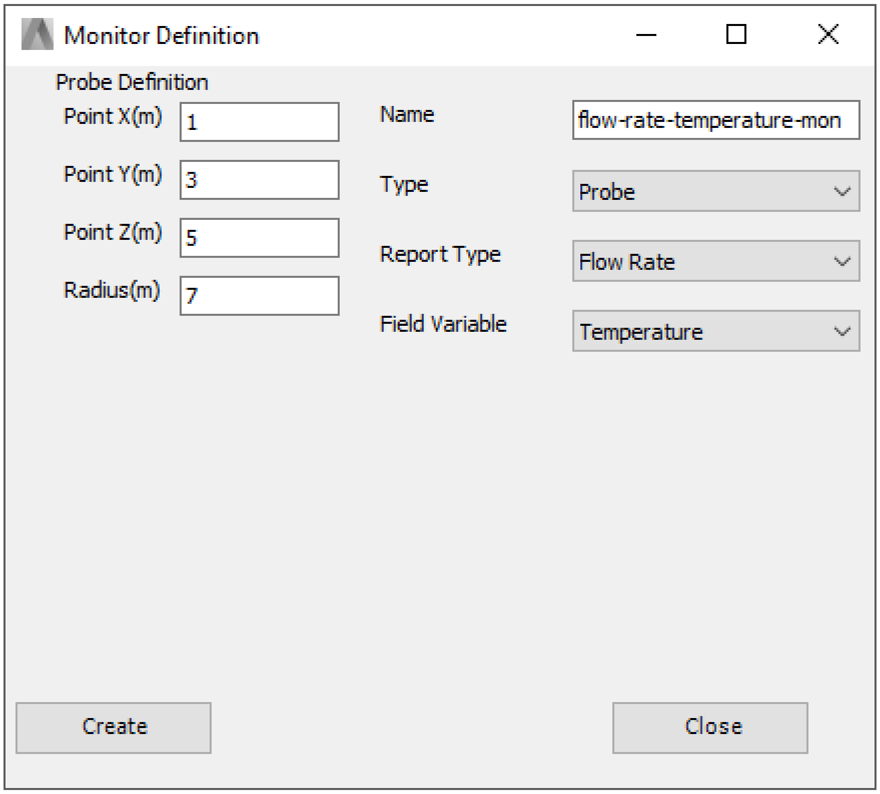


- **vol-avg-temp-mon** plots the **Volume Integral** of **Temperature**.
- **vol-avg-pres-mon** plots the **Volume Integral** of **Pressure**.
- **max-vel-mon** plots the **Max Velocity**.
- **max-temp-mon** plots the **Max Temperature**.
- **max-pres-mon** plots the **Max Pressure**.
- **mass-mon** plots the **Volume Integral** of **Density**.
- **mass-avg-tke-mon** plots the **Mass Average** of **Turbulent Kinetic Energy**.

You can see additional details in the **Monitors** (p. 453) task page in Ansys Fluent. You can create additional monitors by clicking **Create**. Then select the **Type**, **Report Type**, and the **Field Variable** from the respective drop-down lists.

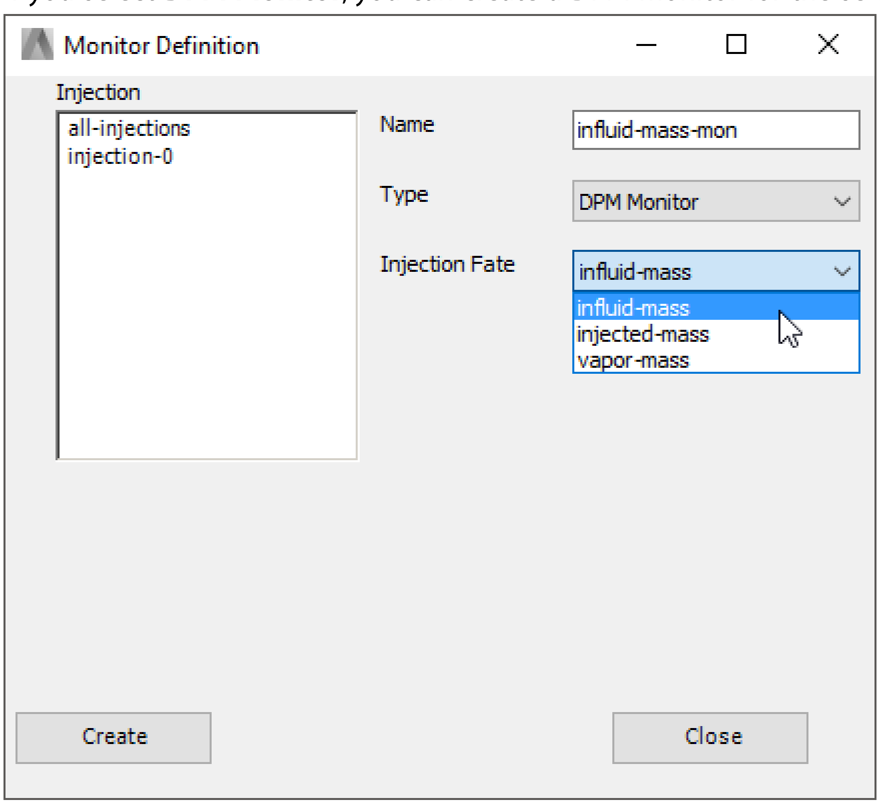
You can select **Volume**, **Surface**, **Probe**, or **DPM Monitor** from the **Type** drop-down list.

If you select **Probe** you can create a point or surface on which you can monitor a variable.



You need to enter values (in meters) for **Point X**, **Point Y**, and **Point Z** to position the point. If you retain the value of **0** for **Radius** then a point monitor is created. If you enter a value for **Radius** then a circular surface with the given **Radius** and the center as the given **Point** is created. You can monitor the variables of your choice on this surface or point.

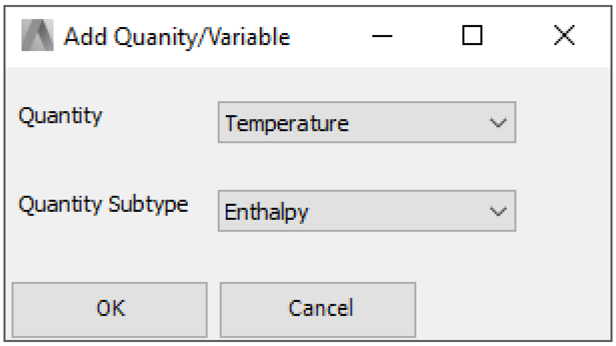
If you select **DPM Monitor**, you can create a DPM monitor for the selected injections.



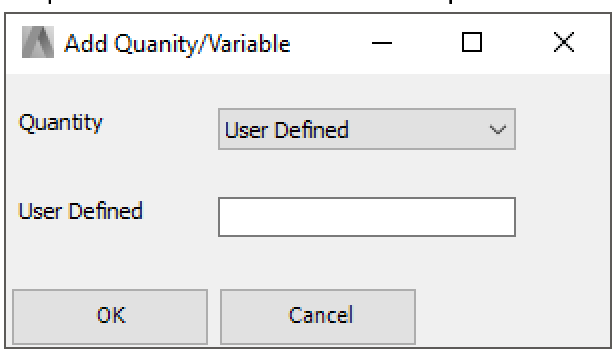
The monitoring variable is referred to as the term **Injection Fate**. You can select from

- **influid-mass**: These are the unsteady particle parcels that are currently in the computational domain, but are not a part of the Lagrangian wall film.
- **injected-mass**: These are entirety of all DPM particles/unsteady particle parcels that have been injected. In steady-state tracking, these correspond to the dispersed phase flow rate defined in the injection(s). In unsteady tracking, it refers to the amount of unsteady particle parcels that have been injected since the beginning of the calculation.
- **vapor-mass**: These are the particle parcels that have (already) evaporated completely.

You can select **New Variable** from the **Field Variable** drop-down list. This opens the **Add Quantity/Variable** dialog box.



You can select the variable by selecting from the options under the **Quantity** and **Quantity Subtype** drop-down lists. You also have an option of **User Defined** under the **Quantity** drop-down list.

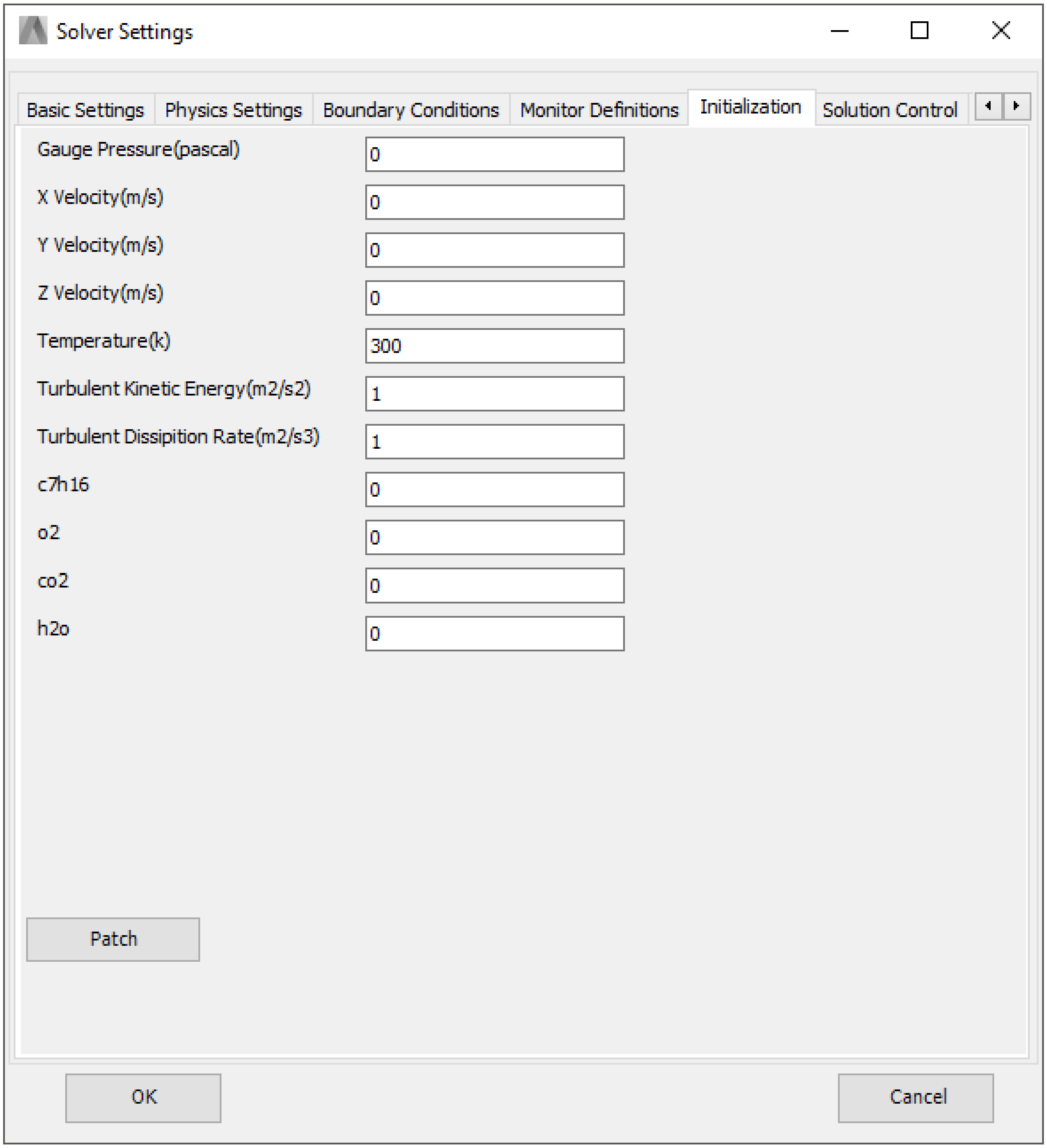


Here you can add the quantity or variable of your choice of which you would like postprocessing images, in the **User Defined** text box. You will have to check Fluent if the term is valid for the simulation. After you click **OK** this quantity will be available in the drop-down list of **Field Variable**.

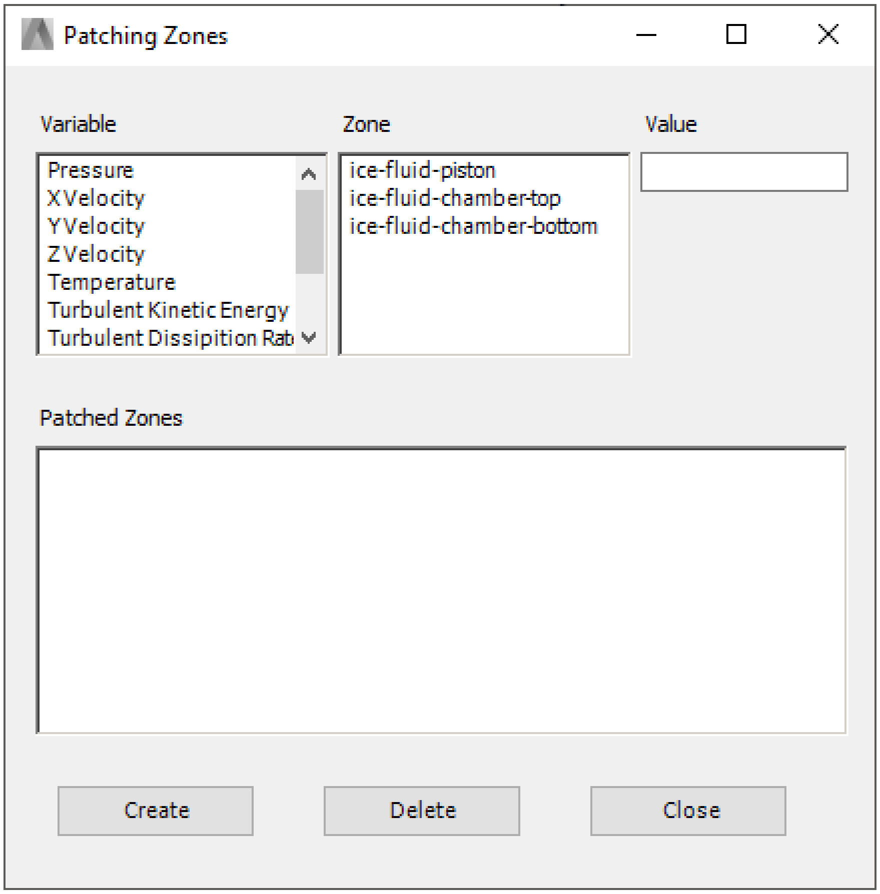
The **Name** will be set according to your selections. You can enter a name of your choice. Click **OK** to create the monitor. It will now appear in the list in the **Monitor Definitions** tab.

12.1.5. Initialization

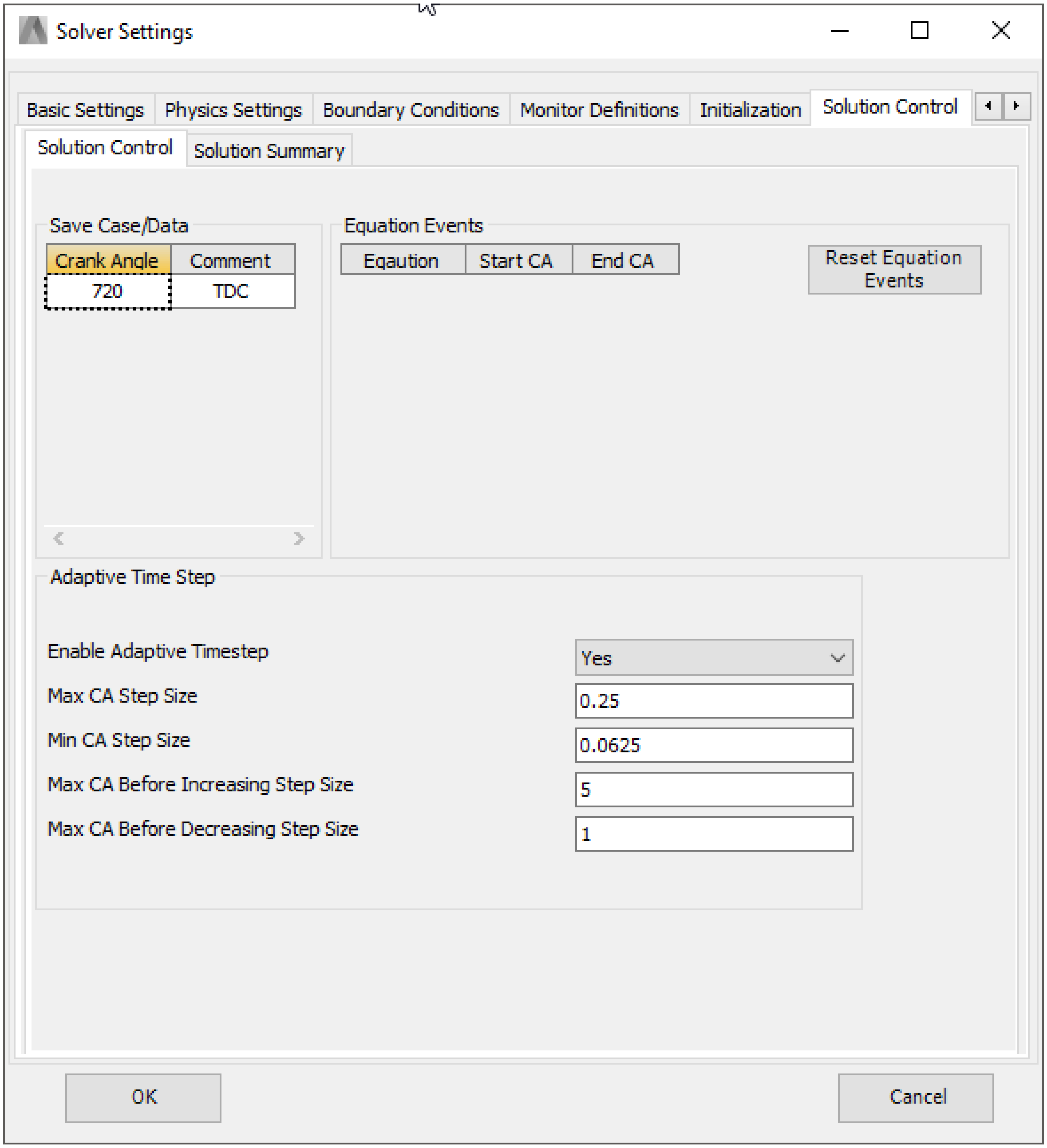
Depending upon the mixture material you have chosen some terms in this tab will vary. Here you can define the initial values for the flow variables.



Click **Patch** to open the **Patching Zones** dialog box. You can patch the various listed variables at your desired values to the listed zones.


Patching Zones
Variable
Zone
Value
Pressure
X Velocity
Y Velocity
Z Velocity
Temperature
Turbulent Kinetic Energy
Turbulent Dissipition Rat
ice-fluid-piston
ice-fluid-chamber-top
ice-fluid-chamber-bottom
Patched Zones
Create
Delete
Close

12.1.6. Solution Control



In the **Solution Control** tab you can set additional specific **Crank Angles** at which the case and data files should be saved and also set the starting and ending crank angles for activating the equations. There are two tabs under the **Solution Control** tab.

12.1.6.1. Solution Control

Save Case/Data

The case and data are saved at a specific frequency which you have entered in the **Basic Settings** (p. 396) tab. In the **Save Case/Data** list you can see the additional crank angles at which the case and data are saved while running the solution. These are at the events of valve opening, valve closing, TDC, etc. You can change these angles. In the **Comment** column you can see the event or definition for the crank angle. You can enter your own definition for the comment column. If you right-click on the list you get options of **Insert Row Below**, **Insert Row Above**, and **Delete Row**. You can use these options to customize your solution.

Equation Events

As per your selection of the combustion models the list of **Equations Events** will change. Here you can enter the starting and ending crank angles for which you want the specific equations activated. The starting crank angle (**Start SA**) is set to the decomposition crank angle by default. If you do not enter anything for the ending crank angle (**End CA**) then the equation will be activated throughout the simulation cycle. To reset the **Equations Events** settings to default values click **Reset Equation Events**.

Adaptive Time Step

By using the options provided under **Adaptive Time Step** you can control how the solver changes the time step size within the specified limits based on convergence criteria. This will help in achieving a better and faster convergence of the solution.

During the solution run, if for one time step the maximum number of iterations are used to converge then it means that the solution is far from converging. In this case the solution will execute the number of time steps according to the crank angle size specified as **Max CA Before Decreasing Step Size** and then reduce the time step by half of the present value. If the solution still uses the maximum number of iterations in the time step then it will again reduce the time step by half. This will go on until **Min CA Step Size** is reached.

On the other hand if the solution is converging in less than half of the maximum number of iterations in the time step, then the time step is doubled after executing the number of time steps according to the crank angle size specified as **Max CA Before Increasing Step Size**. This will go on until **Max CA Step Size** is reached.

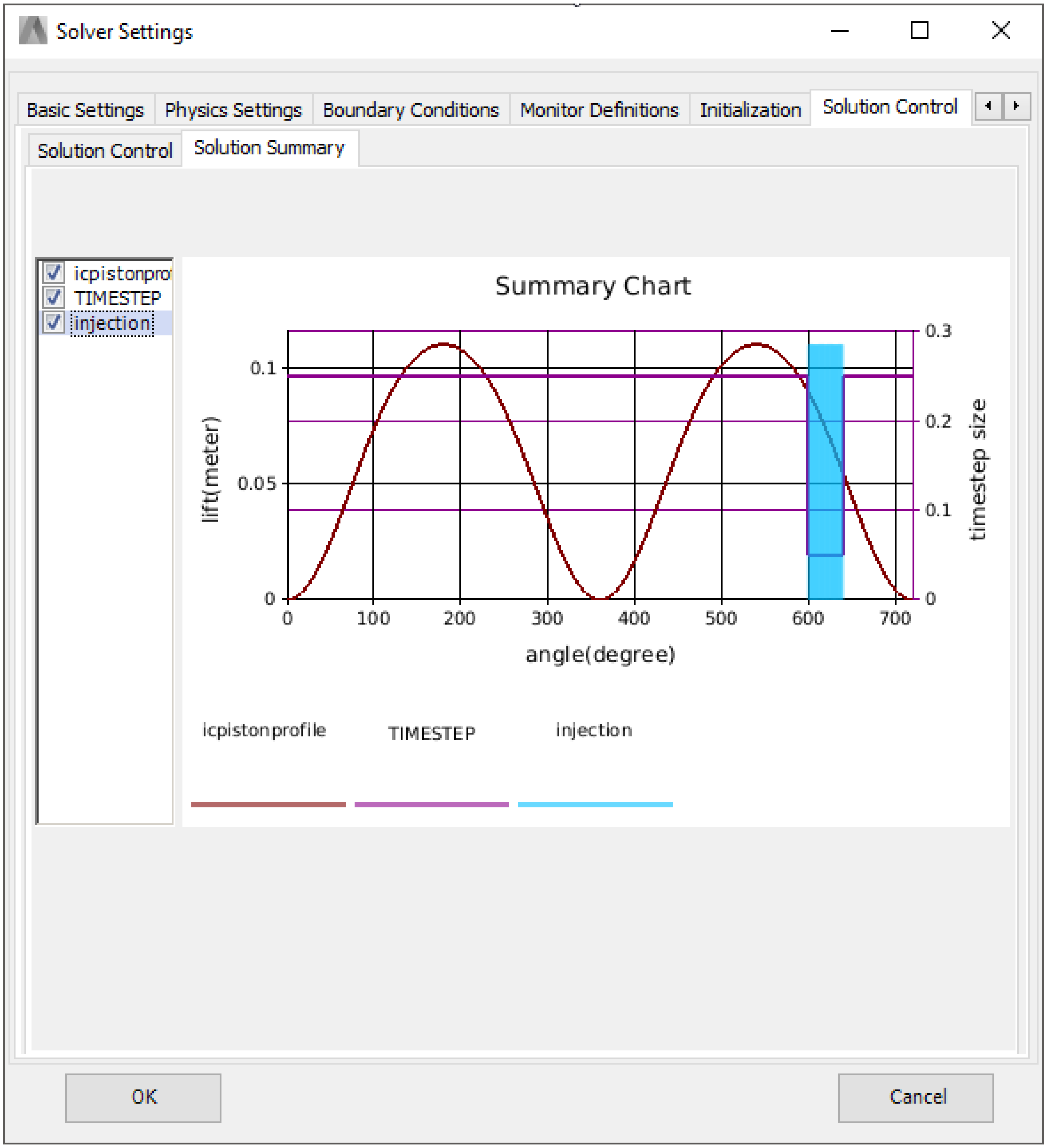
In case there is an event of KeyGrid replacement then the time step is reduced such that the event is within the tolerance limit of **Min CA Step Size**. This will take priority over any other criteria which will try to increase the time step. After the KeyGrid replacement the solution will go back to run on the settings of **Adaptive Time Step**.

Also, if there are any direct events which try to change time step size, the program will respect them but from next time step the **Adaptive Time Step** criteria will control the solution.

- For using the option select **Yes** from the **Enable Adaptive Timestep** drop-down list.
- When the solution starts to converge you can increase the time step size. Enter the crank angle step size to which it can increase in **Max CA Step Size**.
- Similarly if the solution reaches the maximum iterations in a time step then the time step size is decreased. Enter the crank angle step size to which the time step can decrease in **Min CA Step Size**.
- The number of angles the solution will wait before increasing the step size is entered for **Max CA Before Increasing Step Size**.
- The number of angles the solution will wait before decreasing the step size is entered for **Max CA Before Decreasing Step Size**.

.

12.1.6.2. Solution Summary



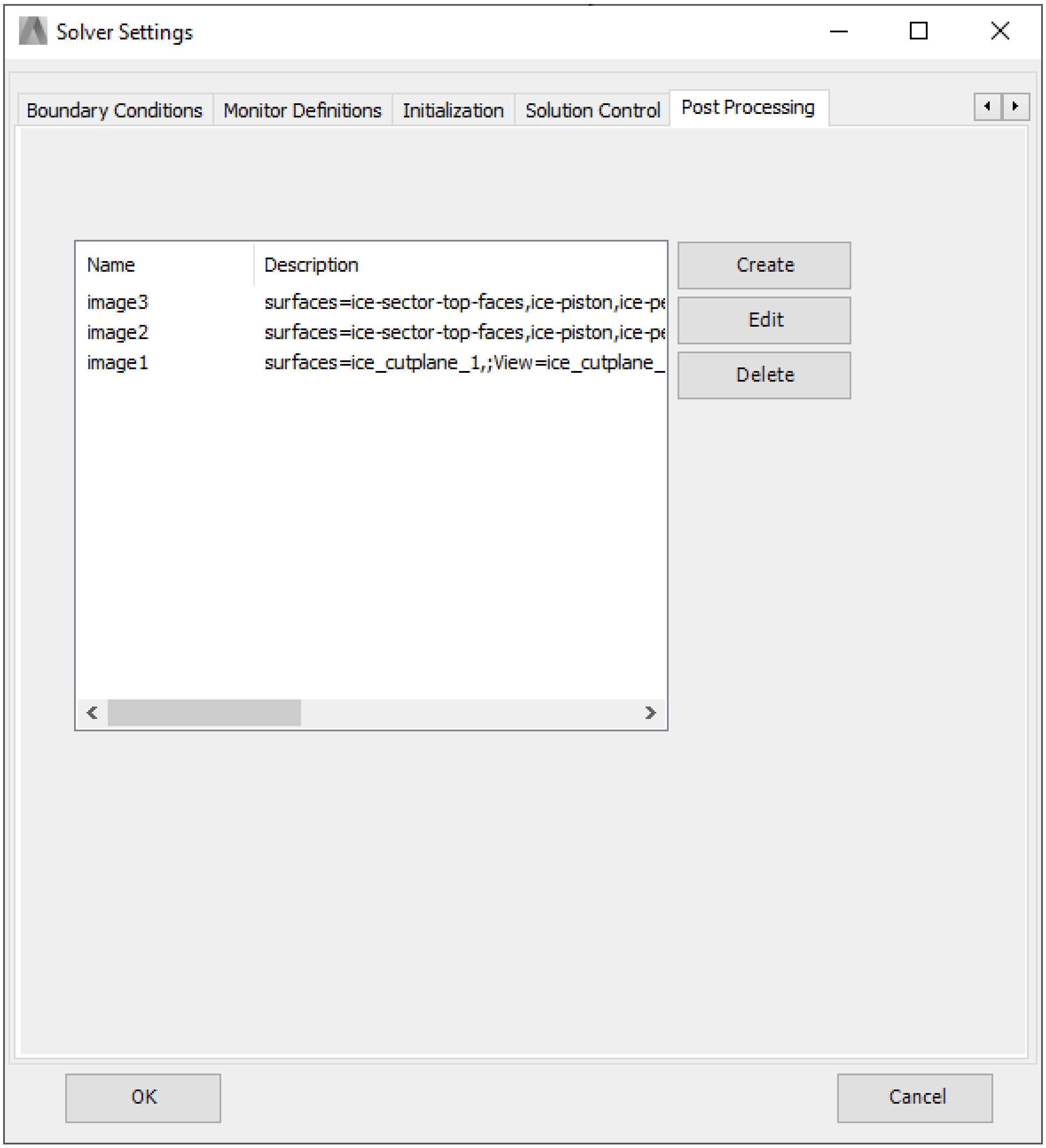
In this group box you can view different profiles overlapping each other. You can enable the profiles you would like to see on the chart from the list on the left hand side. All the profiles are plotted on the base of crank angles. The **Summary Chart** gives a graphic view of the valve profiles, piston profiles, timesteps, etc. Observing the chart gives you an account of the position of the piston and valve at the injection time. This will help if you want to manipulate the default time step settings, or any other events. The chart can be manipulated using the following table.

Table 12.2: Chart Manipulation

	Operation

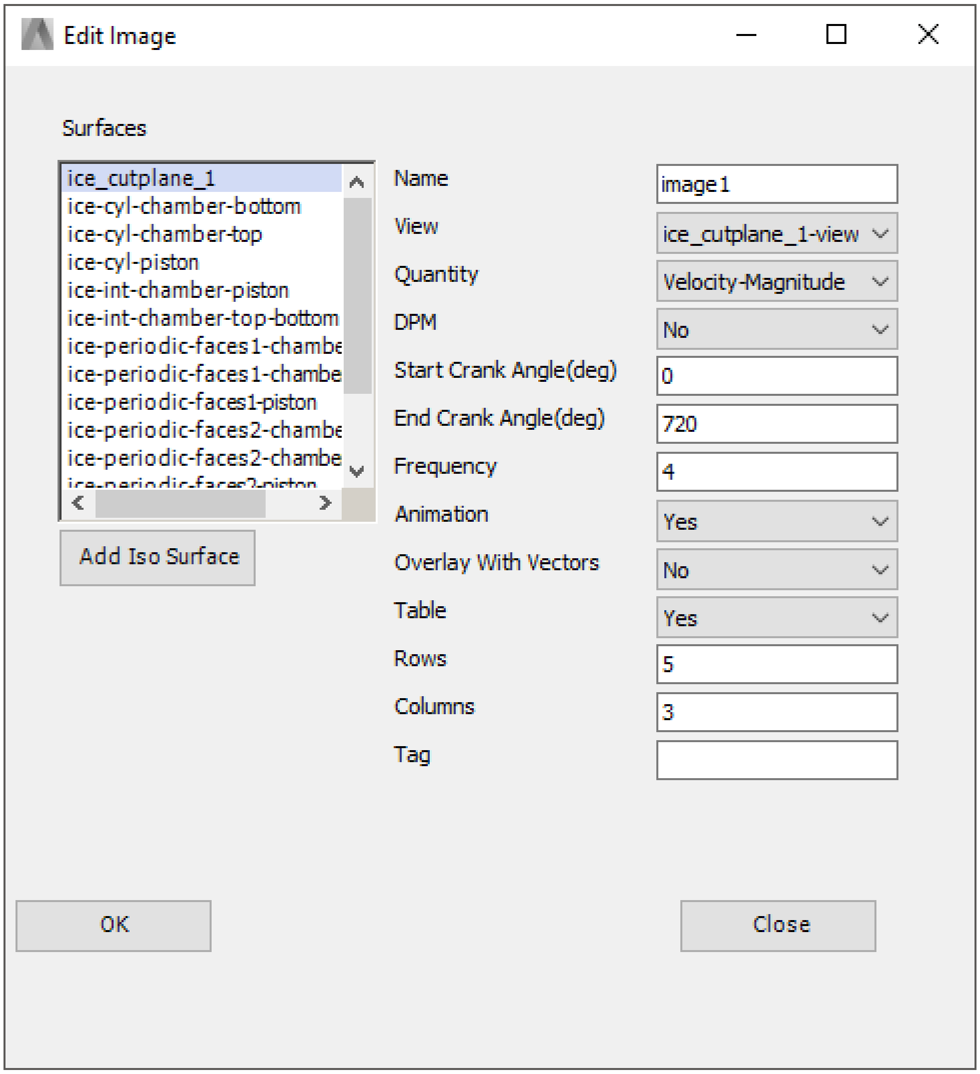
Rotate Middle Mouse	Zoom
Shift + Middle Mouse	Zoom
Ctrl + Middle Mouse	Pan
Drag Right Mouse	Box Zoom
F key	Fit to Window

12.1.7. Postprocessing



In the **Post Processing** tab you can see some of the default images that will be automatically saved during simulation Depending upon the combustion type simulation these images may vary. Select any and click **Edit** to check the details.

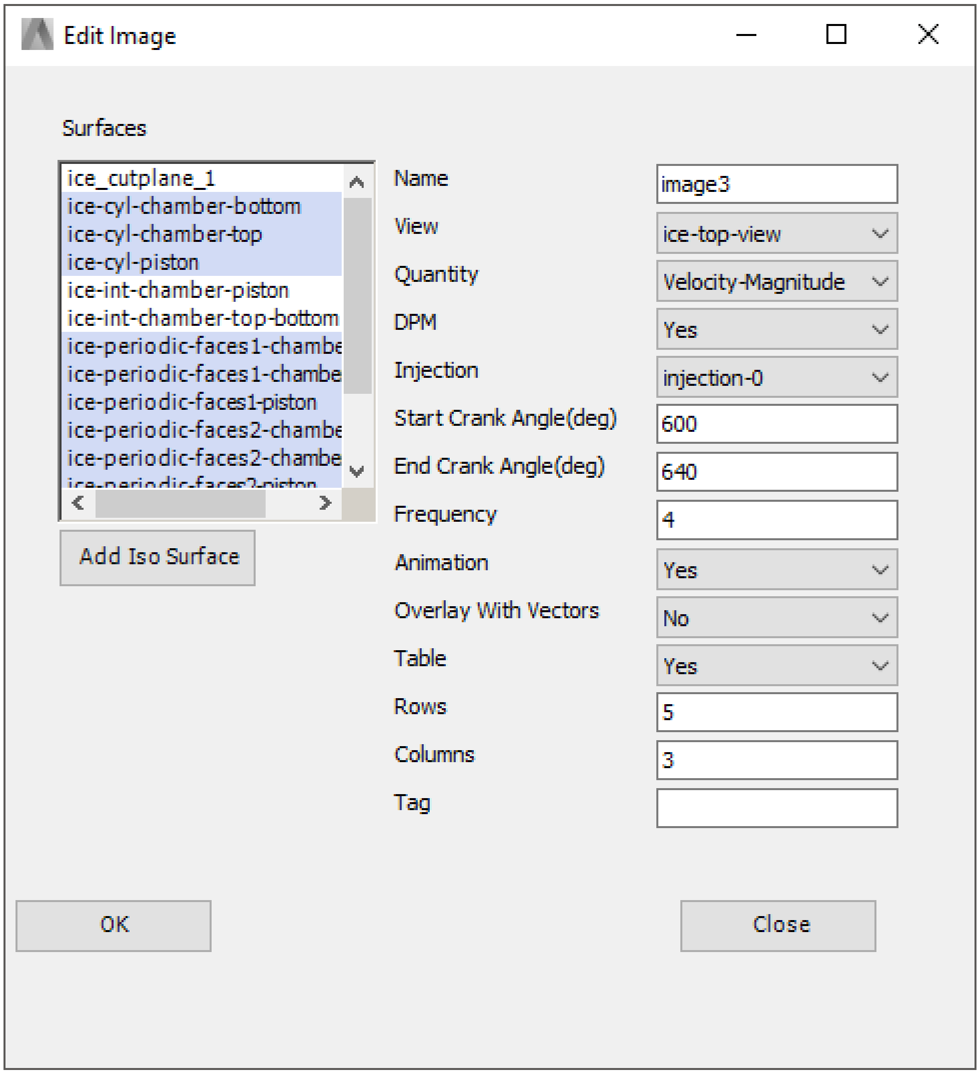
1. Here we have an example of one of the images created.



Here we have an example of one of the images created.

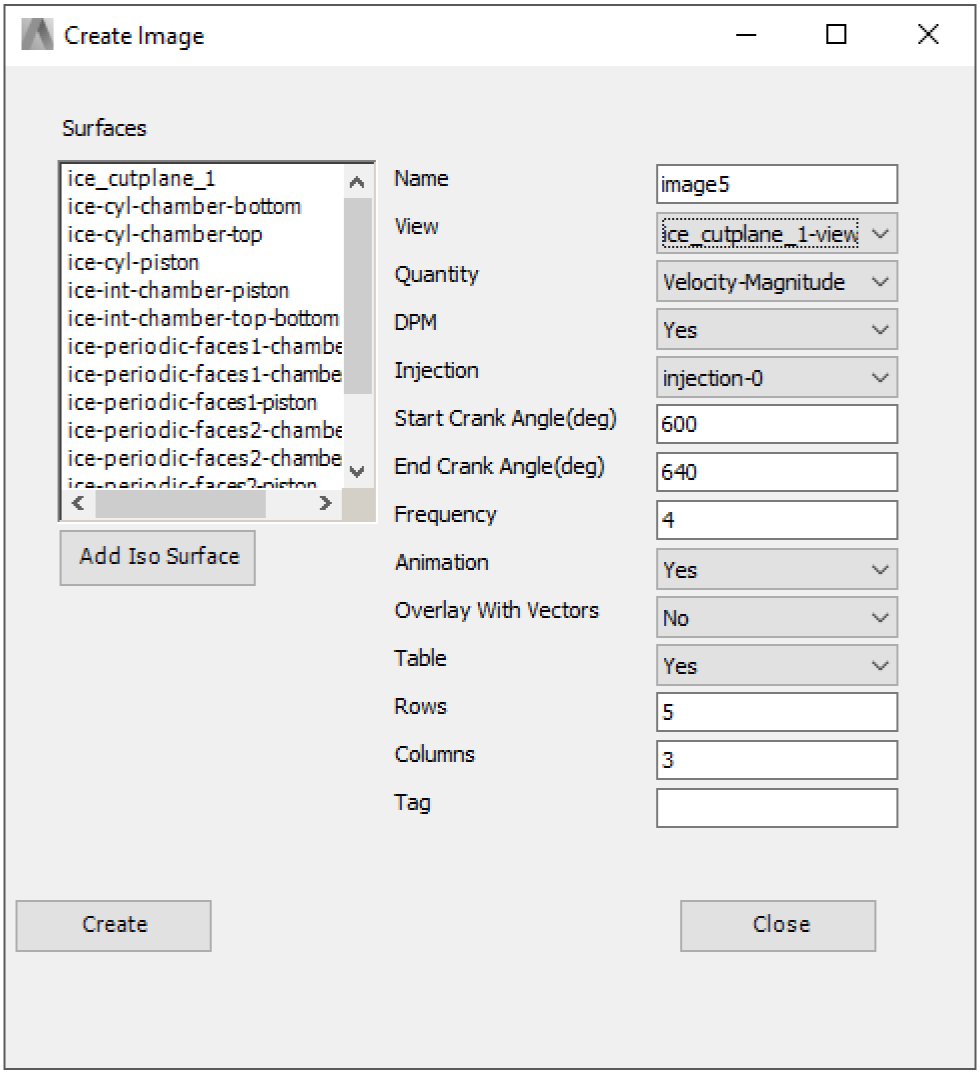
- **image2** is plotted on **ice-cutplane_1**.
- The image is captured at a saved view, **ice-top-view**.
- **Velocity-Magnitude** contours are displayed.
- Images are saved from the **Start Crank Angle** of **0** to the **End Crank Angle** of **720** at a **Frequency** of every **4** time-steps.
- **Animation** is set to **Yes** signifying that animation of the velocity contour images will be created at the end of the simulation.
- **Overlay With Vectors** is set to **No**. You can select **Yes** if you want the vectors along with the contours to be displayed.
- **Table** is set to **Yes** signifying that the saved images will be displayed in the report in a table of **5 Rows** and **3 Columns**.

2. Another example is of **image3**.



Since **DPM** is set to **Yes** it plots the particles of the discrete phase colored by velocity magnitude. The contours are not displayed. The **Surfaces** will be used as an outline boundary.

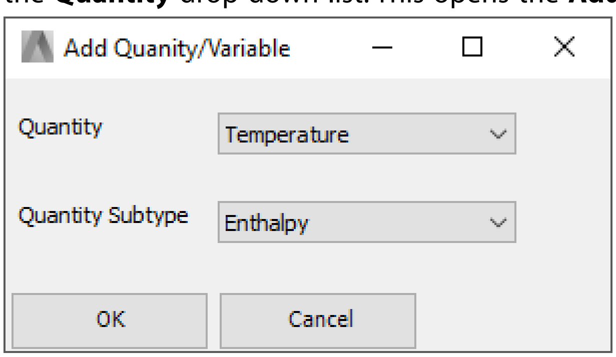
In the **Edit Image** dialog box you can change the settings and values as required. To create additional postprocessing images click **Create**.



1. Select the **Surfaces** on which you want to plot the contours or particle tracks.

2. Select a view from the **View** drop-down list.

3. Select the parameter from the **Quantity** drop-down list. You can select **New Variable** from the **Quantity** drop-down list. This opens the **Add Quantity/Variable** dialog box.

   Here you can add the quantity or variable of your choice of which you would like postprocessing images. You will have to check Fluent if the term is valid for the simulation.

4. Select **Yes** from the **DPM** drop-down list if you want to plot the particle tracks.

5. If you select **Yes** from the **DPM** drop-down list then the **Injection** option is displayed. Select the injection from the drop-down list.

6. Enter the **Start Crank Angle** and **End Crank Angle**. The images will be saved only within these two crank angles. Ensure that these are the crank angles during which spray injection occurs.

7. Enter a value for **Frequency**. The images will be captured at the entered frequency.

8. If you need to create an animation from the saved images you can select **Yes** from the **Animation** drop-down list.

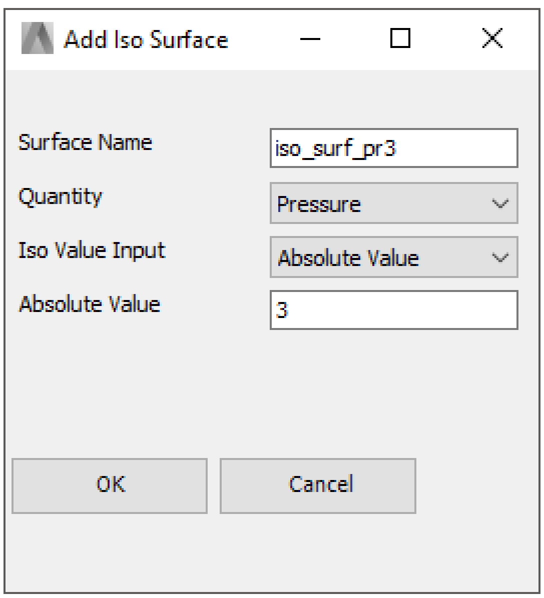
9. The option **Overlay With Vectors** is by default set to **No**. You can set it to **Yes** if you want the vectors to overlap the contour images.

10. If you want the saved images to be displayed in the report in a table format select **Yes** from the **Table** drop-down list.

11. Enter the values for **Rows** and **Columns** depending on how you want to format the table.

12. You can add a tag for the image. You can use this **Tag** if you want only the final images of different surfaces in a single table. In this case you have to provide the same tag to all the images.

13. Click **Add Iso Surface** if you want to create an iso-surfaces. These surfaces are isovalued sections of the entire domain.



    a. Enter a name for the isosurface you want to create at **Surface Name**.

    b. Select the **Quantity** from the drop-down list. You can select a quantity from the list or create a new variable by selecting **New Variable**.

    c. From the **Iso Value Input** drop-down list you can select either of **Absolute Value** or **Percentage of Range**.

- **Absolute Value**: If you select this from the drop-down options, you can enter a value for **Absolute Value**. The iso-surface created will be of this value.
- **Percentage of Range**: If you select this from the drop-down options, then you need to enter a percentage value for **Percentage of Range**. The iso-surface created will be of the percentage value of the minimum and maximum values obtained of the quantity.

An iso-surface of the given name and of the selected quantity will be created and appear in the list of **Surfaces**. This isosurface will be of the **Absolute Value** entered or of the percentage of the minimum and maximum values of the quantity. You can select this isosurface to create images in postprocessing.

14. Click **Create** to create the image.
15. Click **Close** to close the **Create Image** dialog box.

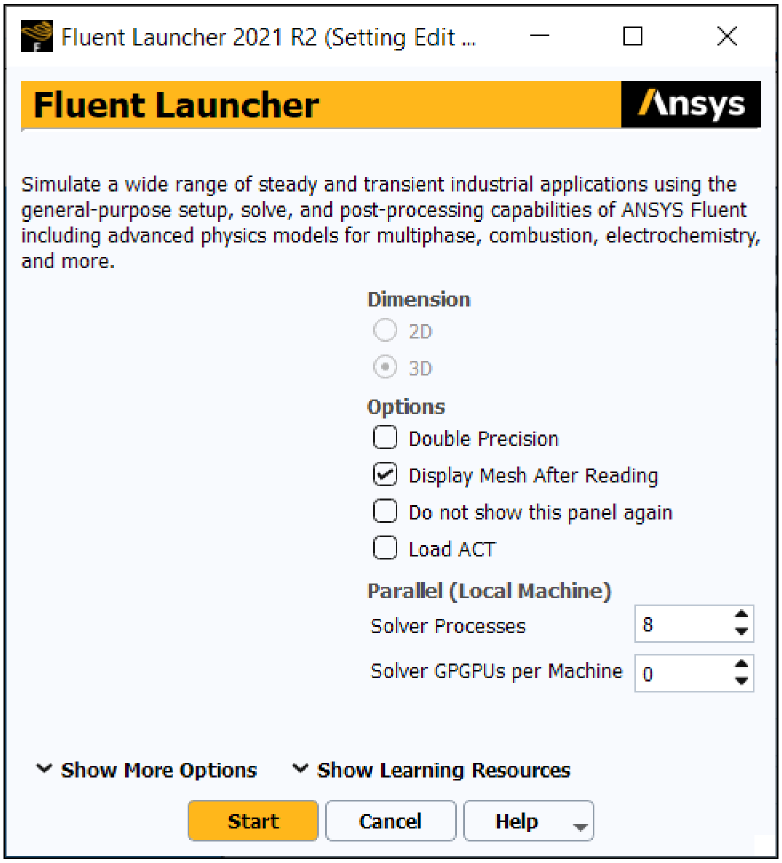
Once the changes are done in the **Solver Settings** dialog box you can close it and then update the **ICE Solver Setup** cell by choosing **Update** from the context menu.

12.2. Solver Default Settings

When you double-click the **Setup** cell, **FLUENT Launcher** opens.

Note:

Depending upon your machine configuration you may be able to increase the **Number of Processes** to decrease the time taken to arrive at a solution. A reasonable value for the **Number of Processes** is the number of cores on your machine.

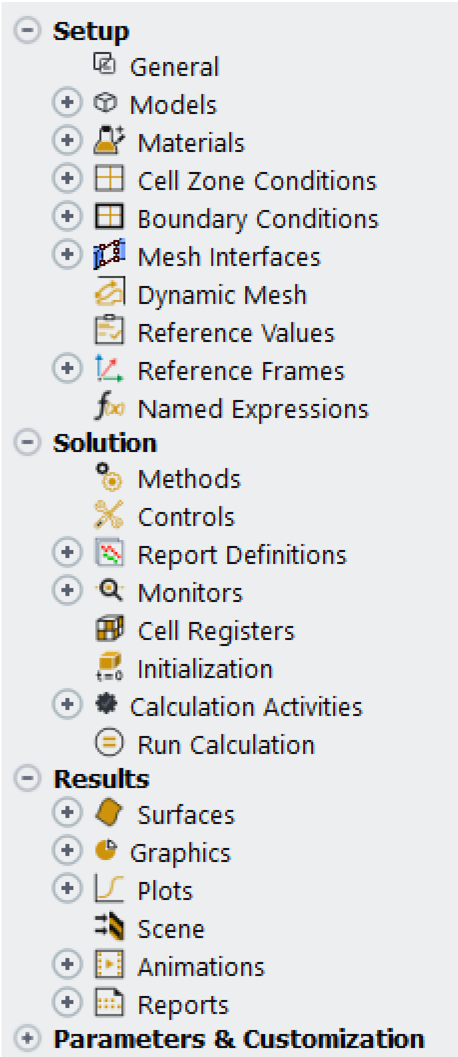


When you click **OK**, Ansys Fluent reads the mesh file and sets up the IC Engine case. It will:

- Read the valve and piston profile.
- Create various dynamic mesh zones.
- Create interfaces required for dynamic mesh setup.
- Set up the dynamic mesh parameters.
- Set up the required models.
- Set up the default boundary conditions and material.
- Create all the required events.
- Sets up the under-relaxations factors.
- Set up the default monitors.
- Initialize and patch the solution.

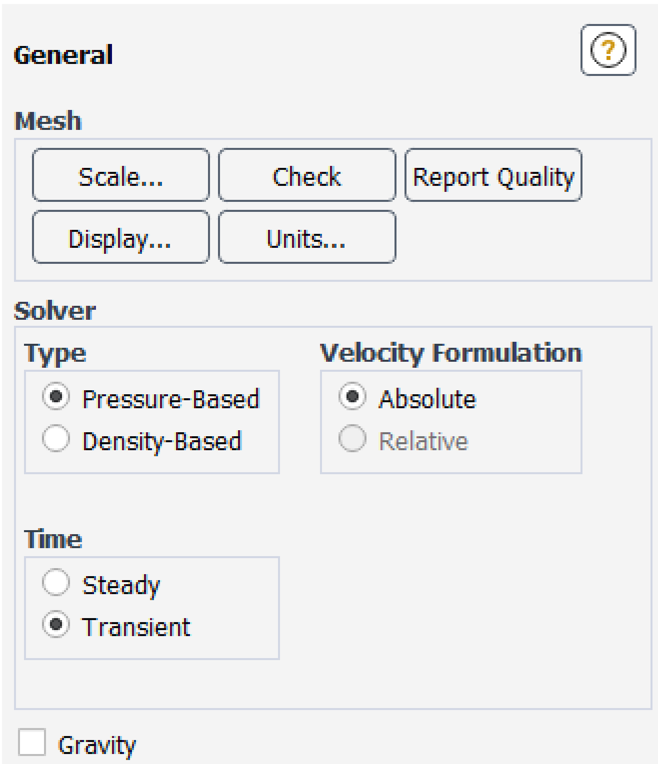
In the Ansys Fluent application, you can check the default settings by highlighting the items in the navigation pane.

Figure 12.1: The Ansys Fluent Navigation Pane



12.2.1. General Settings

In the **General** task page the following settings are done:

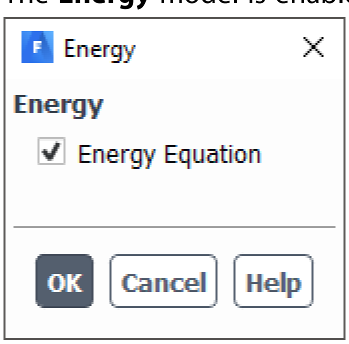


- Solver **Type** is set to **Pressure-Based**.
- Solver **Time** is set to **Transient**.

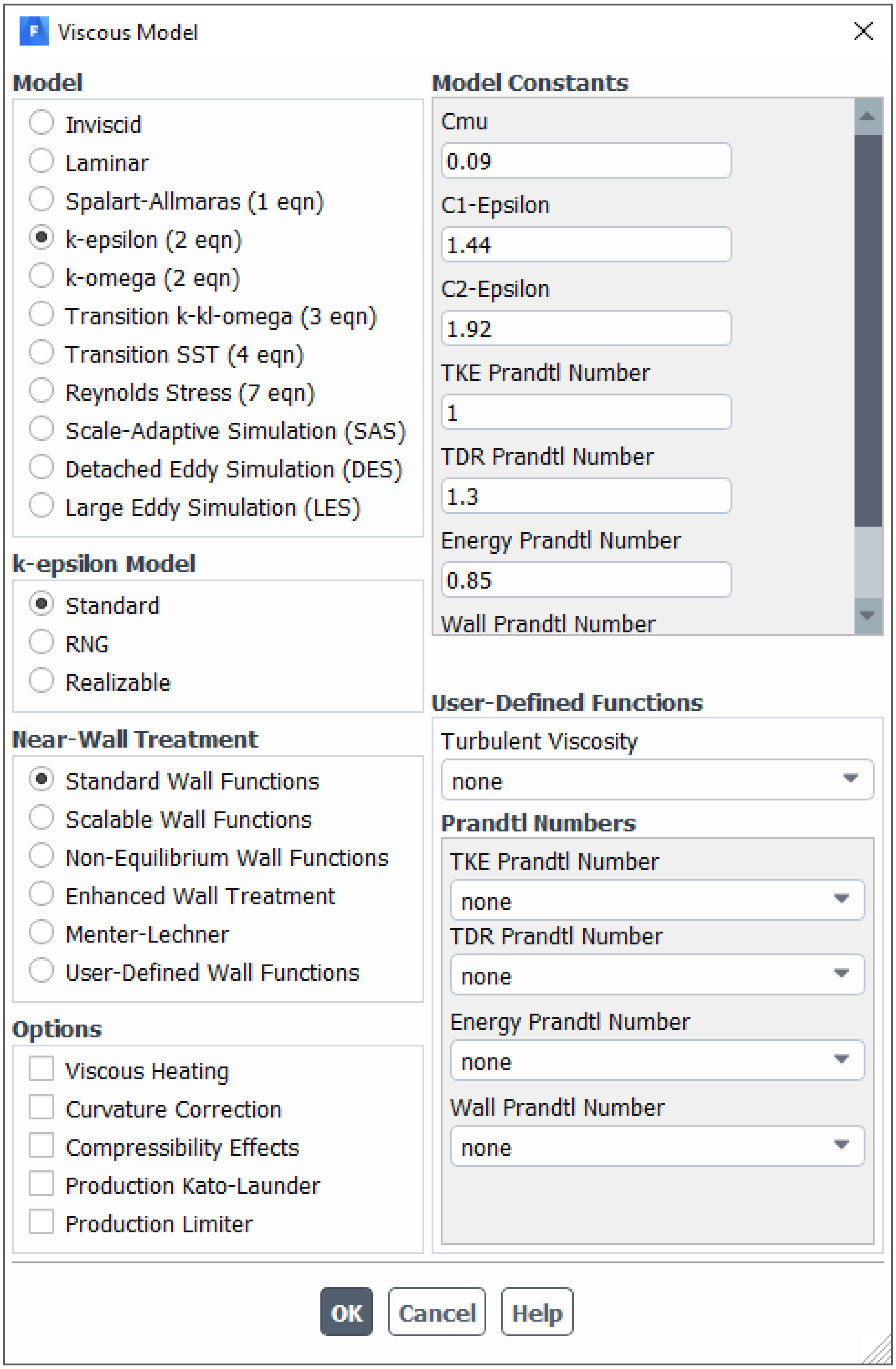
12.2.2. Models

The models selected for combustion will vary upon your selection in the **Solver Settings**. These are some of the default models selected for the analysis for a sector combustion:

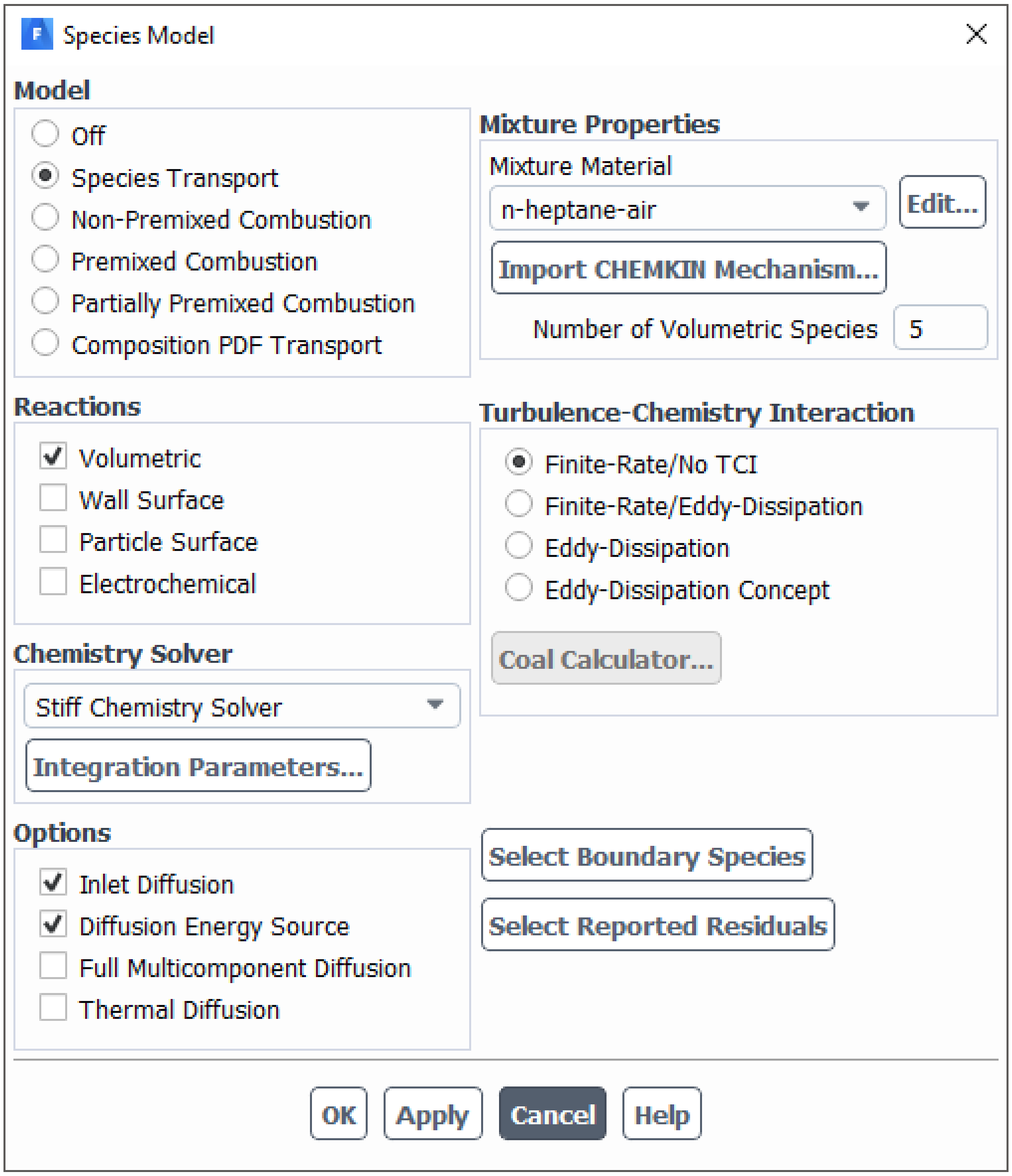
- The **Energy** model is enabled.



- From the **Viscous** models, the **Standard k-epsilon** model is selected, with **Standard Wall Functions** as **Near-Wall Treatment**.

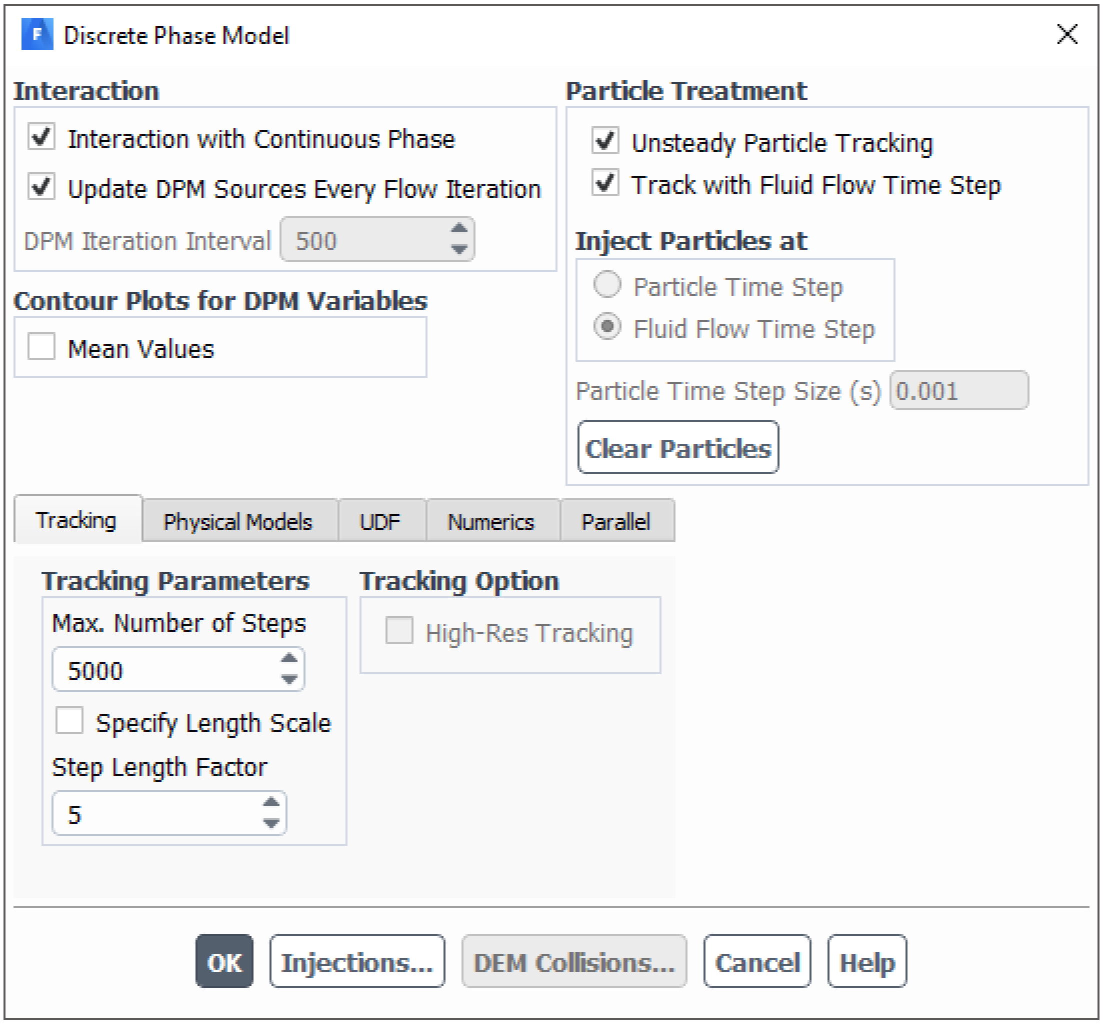


- **Species Transport** is selected from the list of models under **Species**. Depending upon your selection in the **Solver Settings** dialog box the options will vary.



- **Volumetric** is enabled from the **Reactions** group box.
- From the **Options** group box **Inlet Diffusion**, **Diffusion Energy Source**, and **Stiff Chemistry Solver** are enabled.
- If you have provided a `CHEMKIN` file, then **chemkin-import** is chosen from the **Mixture Material** drop-down list. Else a default material **n-heptane-air**, is provided form the Fluent database of materials.
- From the **Turbulence-Chemistry Interaction** group box **Laminar Finite-Rate** is selected.

• The **Discrete Phase Model** is enabled. Depending upon your selection in the **Solver Settings** dialog box the options will vary.

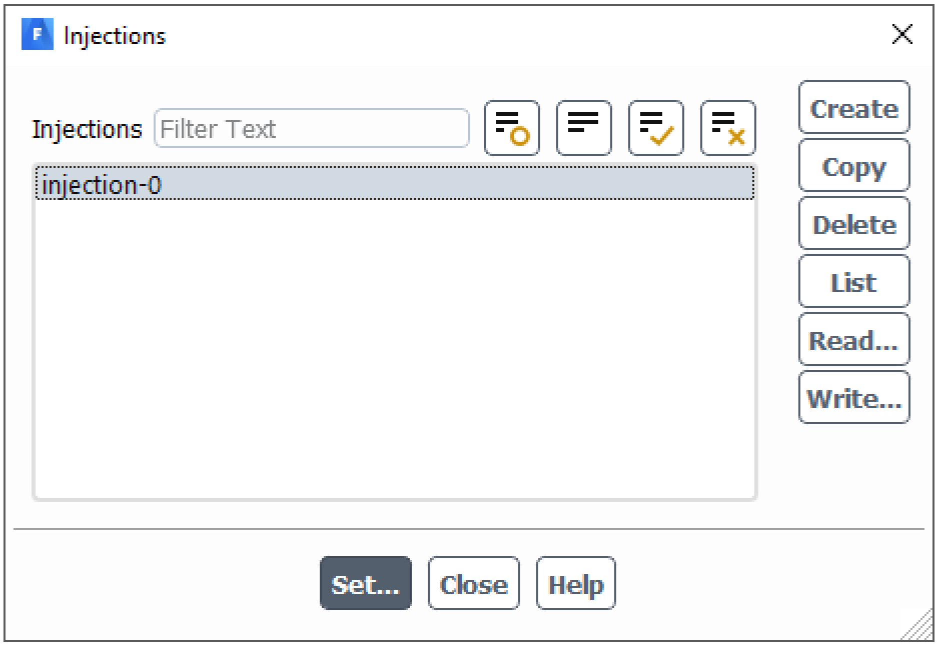


- **Interaction with Continuous Phase** and **Update DPM Sources Every Flow Iteration** are enabled in the **Interaction** group box.
- **Unsteady Particle Tracking** and **Track with Fluid Flow Time Step** are enabled in the **Particle Treatment** group box.
- In the **Tracking** tab, **Max. Number of Steps** is set to `5000` and **Step Length Factor** to 5.
- In the **Physical Models** tab **Stochastic Collision**, **Coalescence**, and **Breakup** are enabled from the list of **Options**.

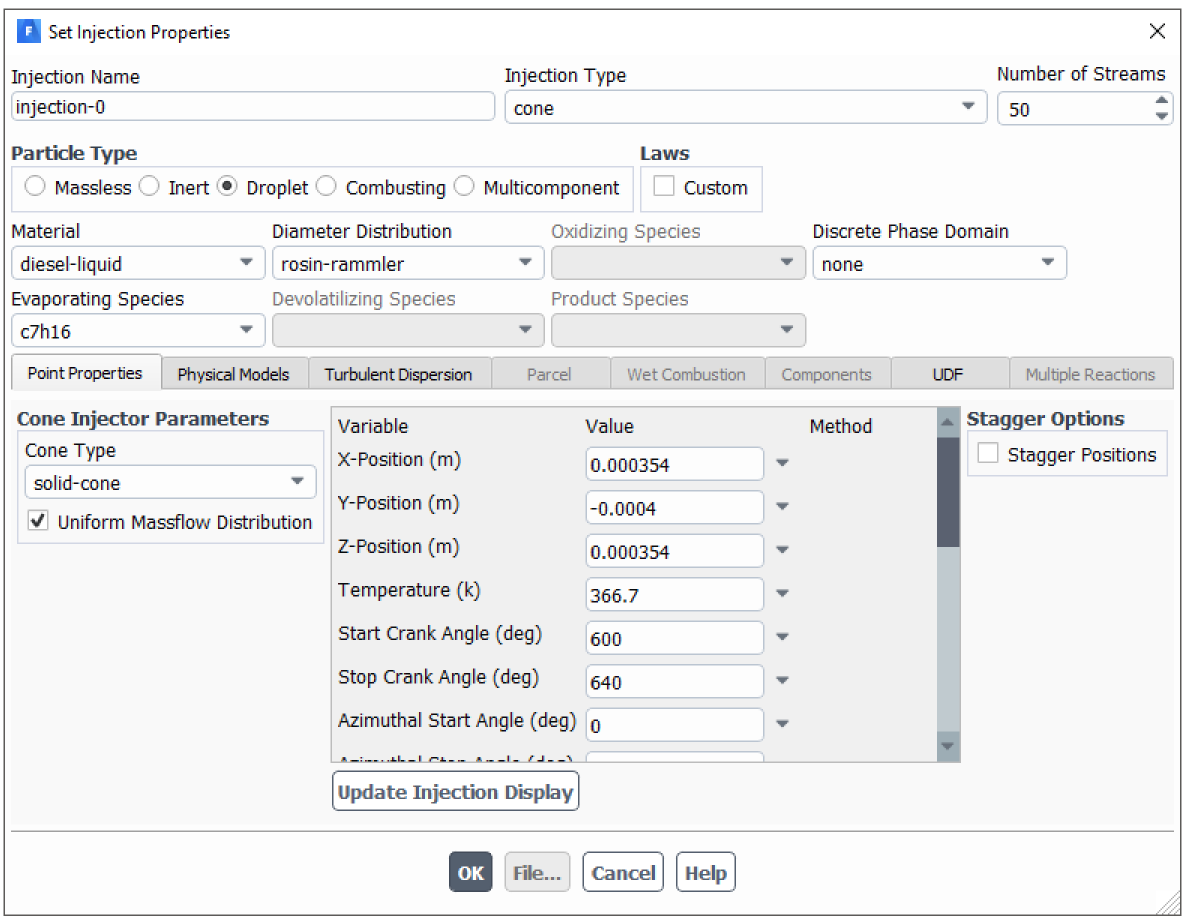
12.2.3. Injections

Depending upon your choices in the **Solver Settings** dialog box injections and their settings will vary. Here is an example of a default setting for sector. The spray injection properties are defined in the **Set Injection Properties** dialog box. You can open the dialog box through the **Define** menu or by the **Injections...** button in the **Discrete Phase Model** dialog box.**Define** → **Injections...**

This dialog box can also be opened through the **Discrete Phase Model** dialog box.



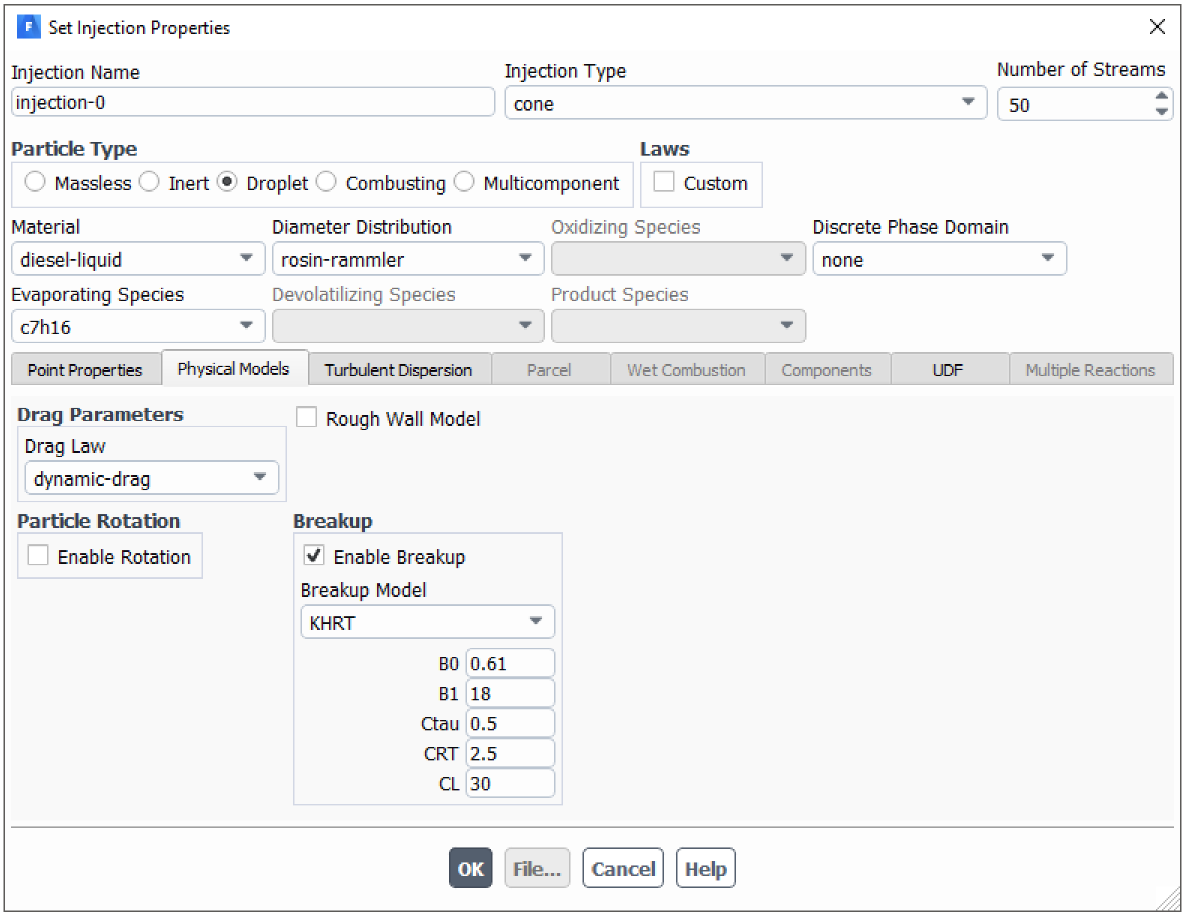
Depending upon the injections that you have set up in the **Solver Settings** dialog box the options will vary. You can selec the listed injections and click **Set...** to open the **Set Injection Properties** dialog box.



- **Particle Type** selected is **Droplet**.

- **Material** of injection is selected as **diesel-liquid**.
- The default **Evaporating Species** selected is **c7h16**.
- You can check the rest of the values of the variables in the **Point Properties** tab.

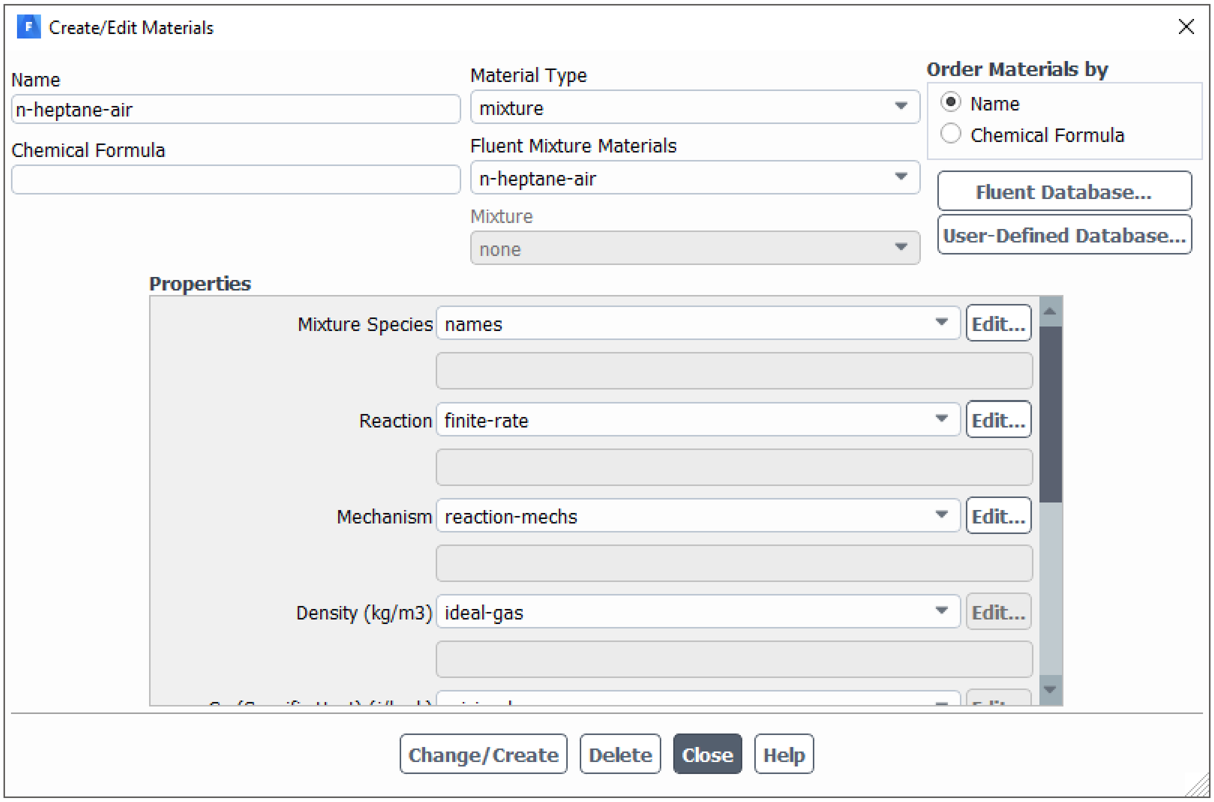
Check the settings In the **Physical Models** tab.



- In the **Drag Parameters** group box, **dynamic-drag** is chosen from the **Drag Law** drop-down list.
- In the **Breakup** group box **KHRT** is chosen as the **Breakup Model**.

12.2.4. Materials

Material will differ upon your selection in the **Solver Settings**.



- If you have provided a `CHEMKIN` file, then **chemkin-import** is chosen as the material from the list of **Mixture** drop-down list. Else a default mixture material **n-heptane-air**, is selected form the Fluent database of materials.

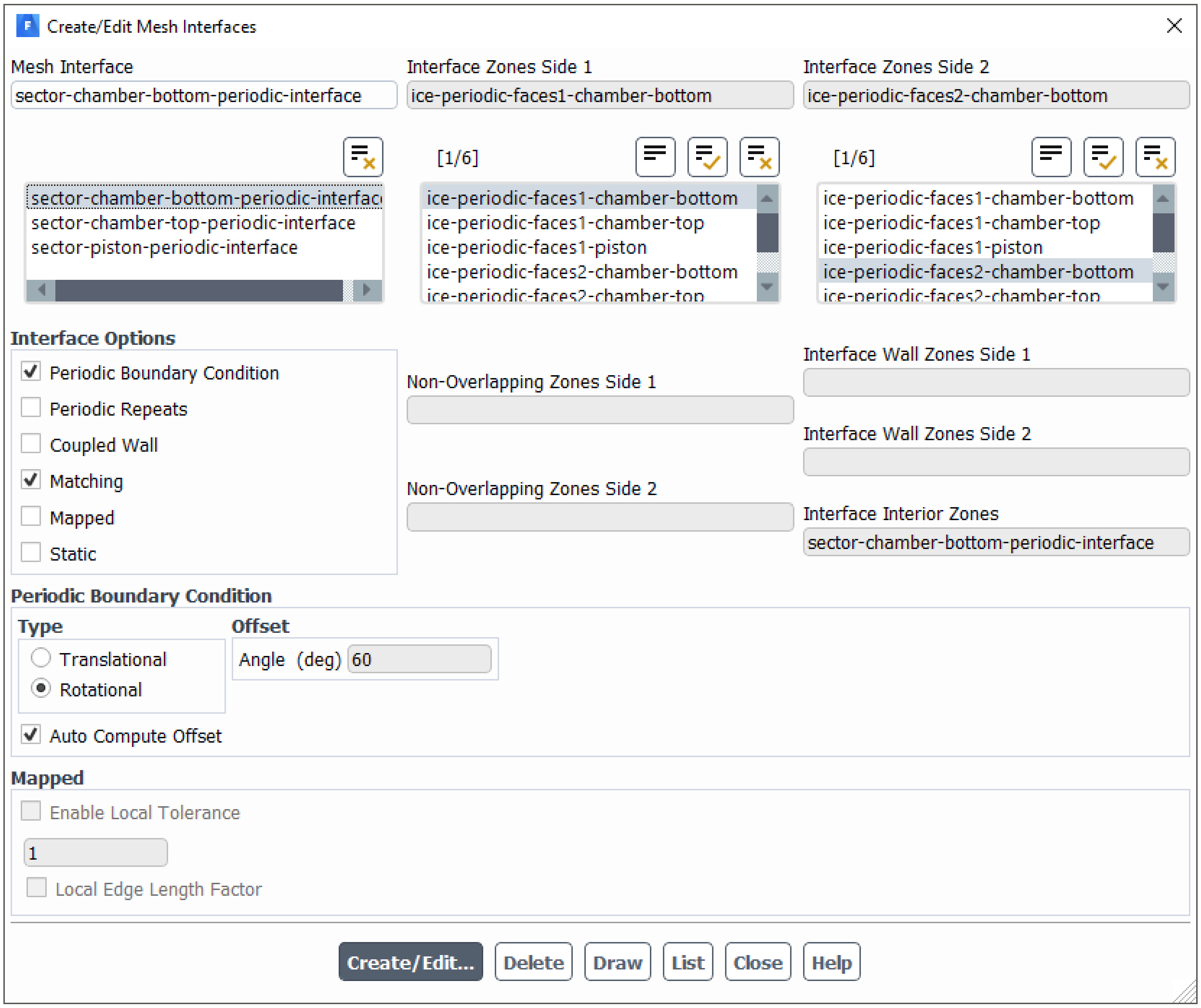
> **Note:**
>
> You can check the properties of the mixture in the **Properties** group box.

- **Droplet Particle** chosen is **diesel-liquid**.

> **Note:**
>
> You can add your material through the `CHEMKIN` file or import a mixture material from the Fluent Database and then change the properties as required in the **Create/Edit Materials** dialog box.

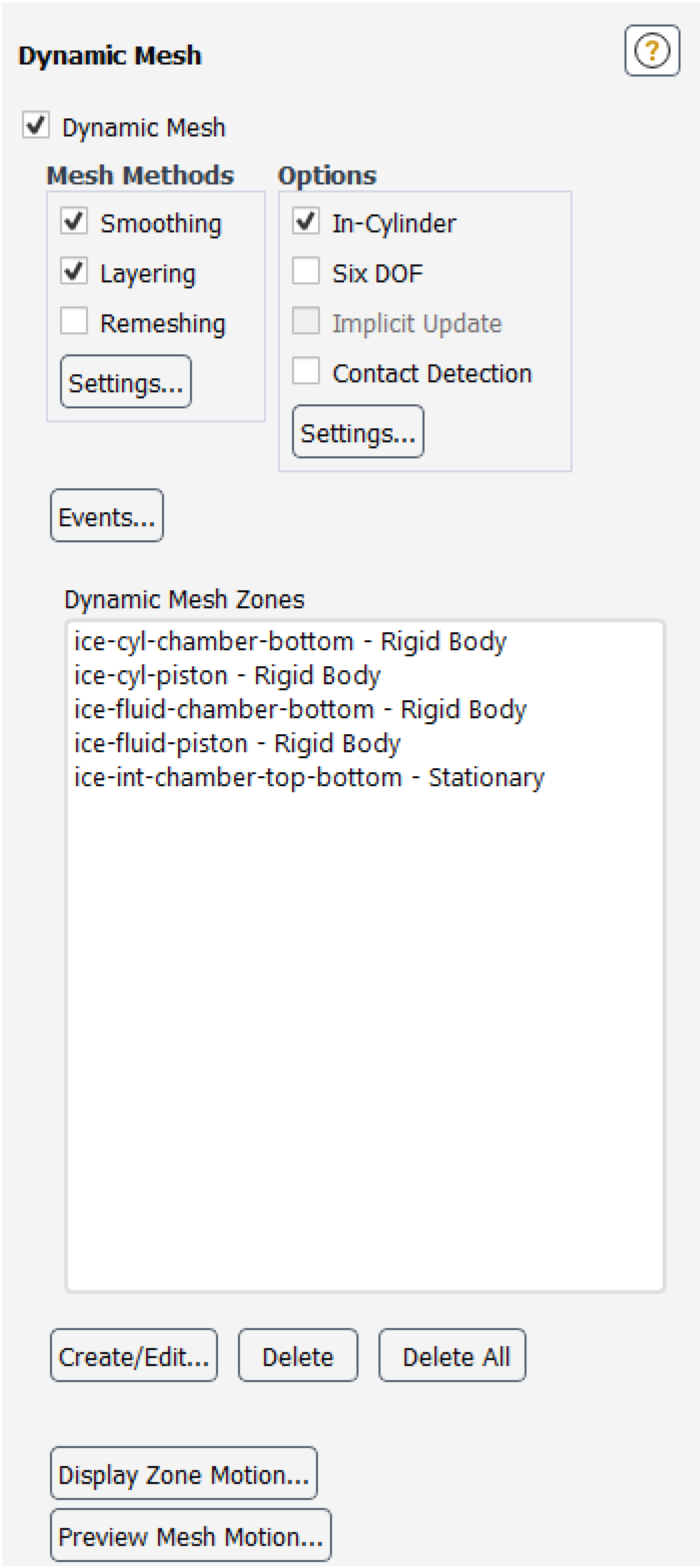
12.2.5. Mesh Interfaces

You can check the three periodic mesh interfaces created in the **Mesh Interfaces** task page. Following is an example for a sector combustion.



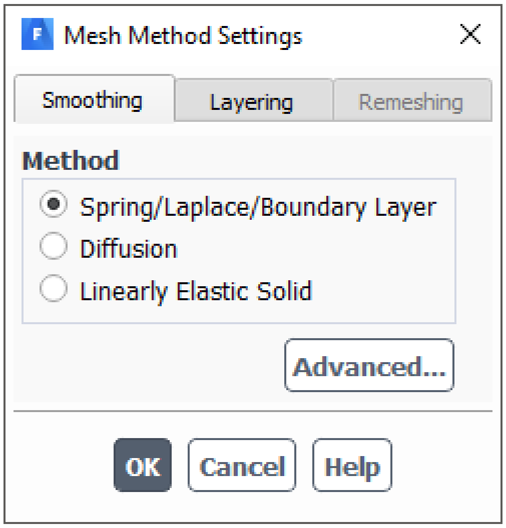
- A periodic interface is created between **ice-periodic-faces1-chamber-bottom** and **ice-periodic-faces2-chamber-bottom**. It is called **sector-chamber-bottom-periodic-interface**.
- The second periodic interface is created between **ice-periodic-faces1-chamber-top** and **ice-periodic-faces2-chamber-top**. It is called **sector-chamber-top-periodic-interface**.
- The third periodic interface is created between **ice-periodic-faces1-piston** and **ice-periodic-faces2-piston**. It is called **sector-piston-periodic-interface**.

12.2.6. Dynamic Mesh

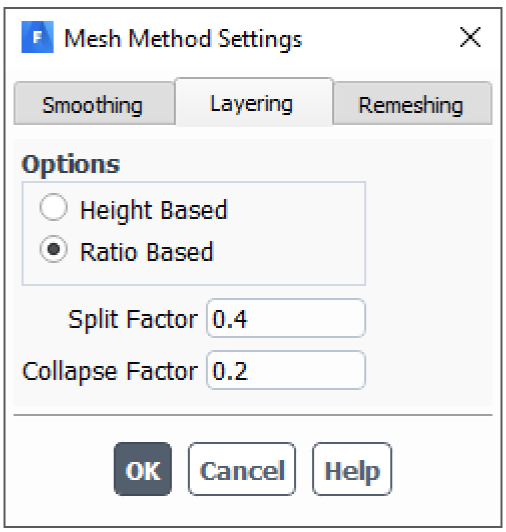


In the **Dynamic Mesh** dialog box you can check the mesh methods and their settings. Click **Settings...** in the **Mesh Methods** group box to open the **Mesh Method Settings** dialog box. Depending upon the type of combustion simulation the settings may vary. Following are examples for sector combustion.

In the **Smoothing** tab the method is set to **Spring/Laplace/Boundary Layer**. The **Parameters** are set by default. For more information, refer to Smoothing Methods in the Fluent User's Guide.



In the **Layering** tab **Ratio Based** is chosen from the **Options** group box. You can control how a cell layer is split by specifying either **Height Based** or **Ratio Based**. The **Split Factor** and **Collapse Factor** are the factors that determine when a layer of cells that is next to a moving boundary is split or merged with the adjacent cell layer, respectively. For more information, refer to Dynamic Layering in the Fluent User's Guide.



You can also see the three dynamic mesh zones created and the type to which each zone is set.

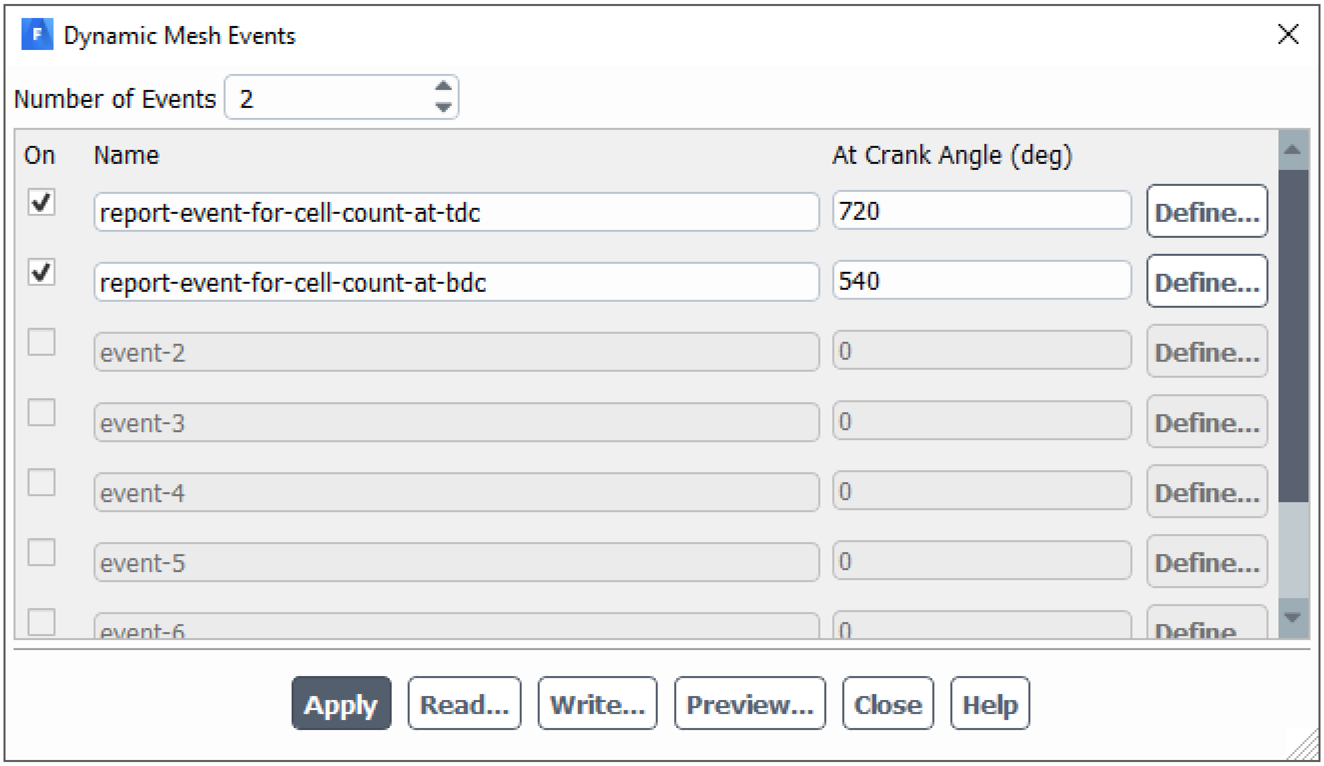
- **ice-fluid-chamber** zone is set to **Rigid Body**.
- **ice-fluid-piston** zone is set to **Rigid Body**.
- **ice-sectortopfaces** zone is set to **Stationary**.

12.2.7. Events

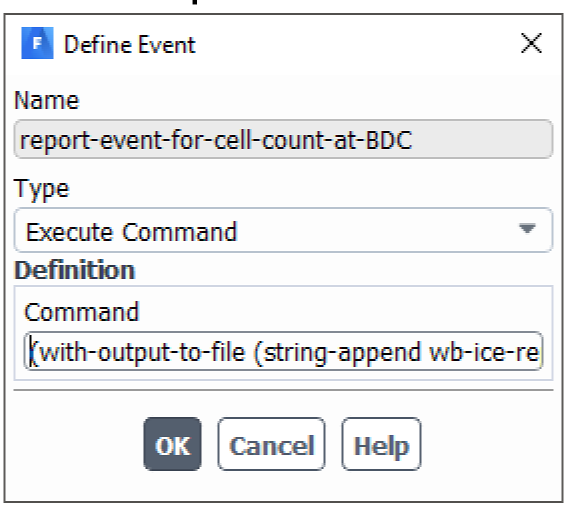
Events are used to reduce the time steps at the beginning of spray injection and increase the time steps after the spray stops. This is done to capture the process details during this period.

The events are specified for one complete engine cycle. Depending upon the type of combustion simulation the settings may vary. Following are examples for sector combustion.

If you click **Events** in the **Dynamic Mesh** task page, you can see these events.

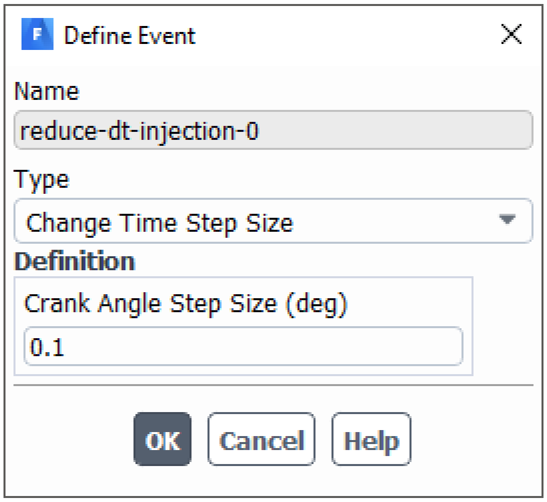


- In the event **report-event-for-cell-count-at-bdc** the cell count at BDC is added to the report.

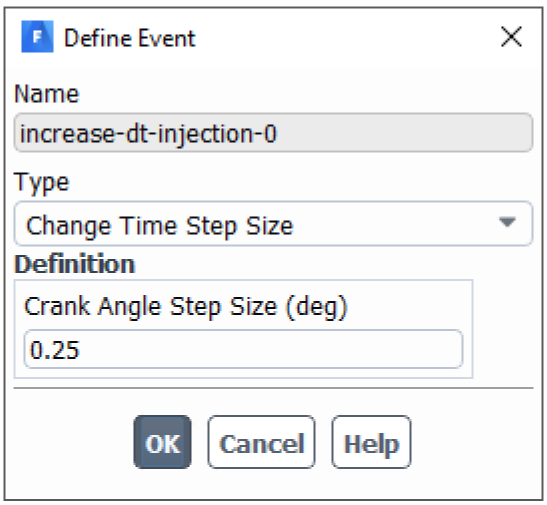


- Similarly, in the event **report-event-for-cell-count-at-tdc** the cell count at TDC is added to the report.

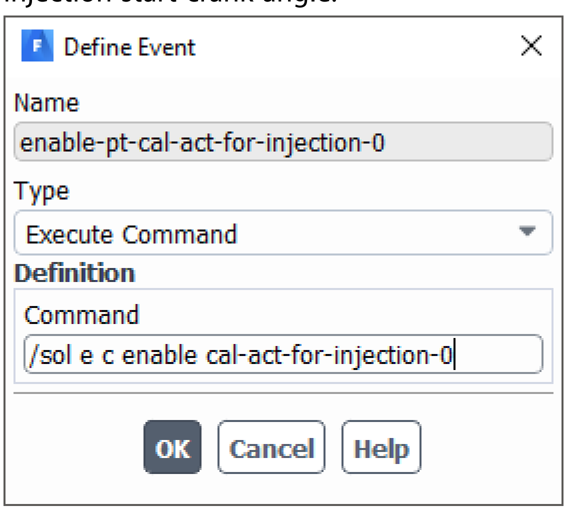
- In the event **reduce-dt-injection-0**, the time step size is reduced to **0.1°** of **Crank Angle Step Size (deg)** at injection start crank angle.



- In the event **increase-dt-injection-0**, the time step size is increased to **0.25°** at injection stop crank angle.

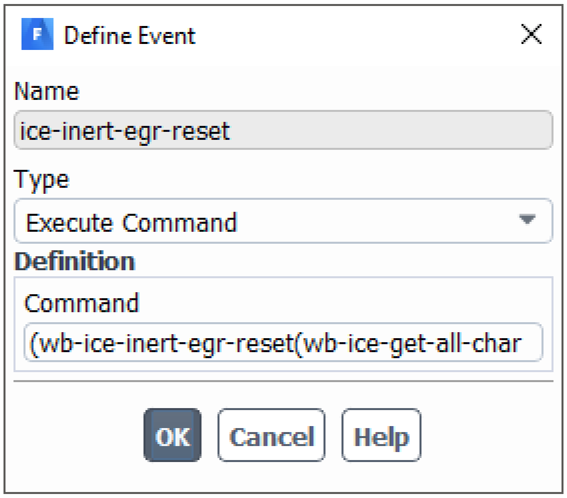


- In the event **enable-pt-cal-act-for-injection-0**, the particle track calculation is enabled at the injection start crank angle.



- Similarly, in the event **disable-pt-cal-act-for-injection-0** the particle track calculation is disabled at the injection stop crank angle.

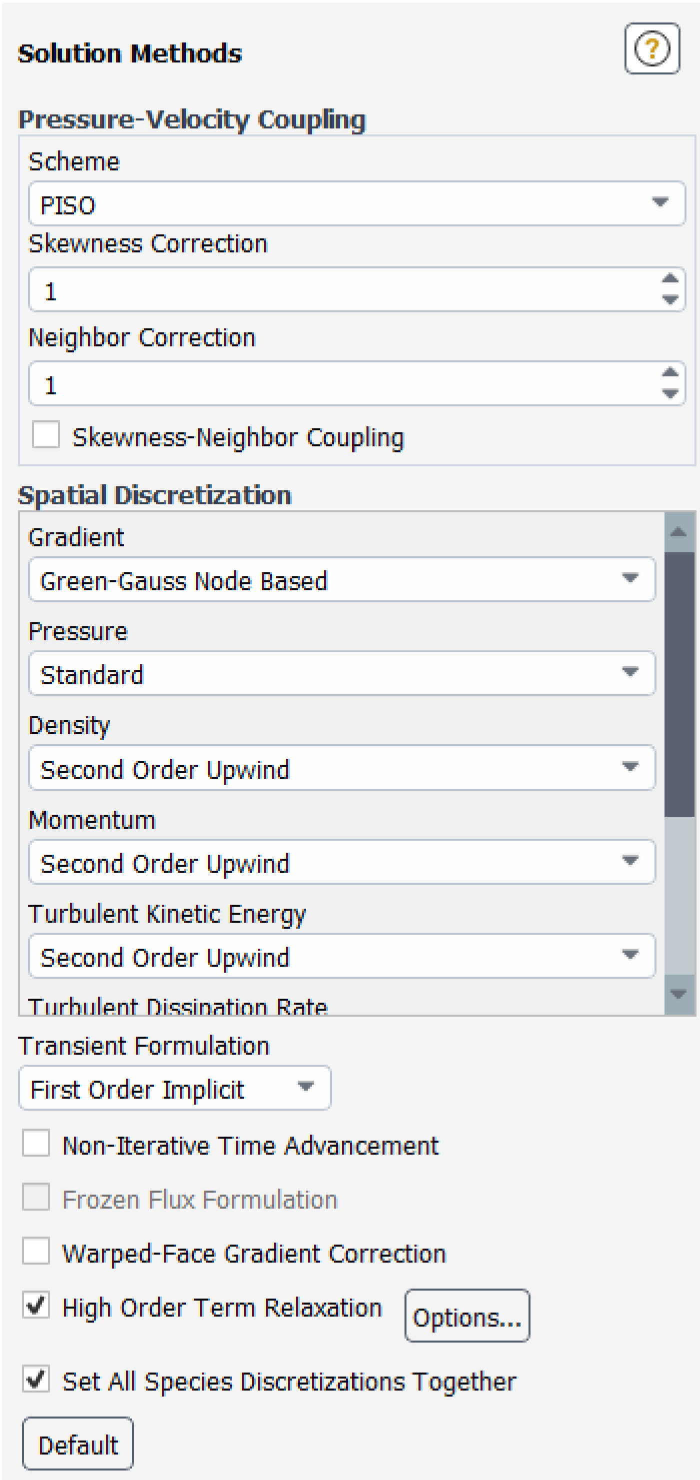
- If you have selected **Inert Model** then an event **ice-inert-egr-reset** is added. This event is before IVO.



You can check the events by clicking **Define**, next to each event name.

12.2.8. Solution Methods

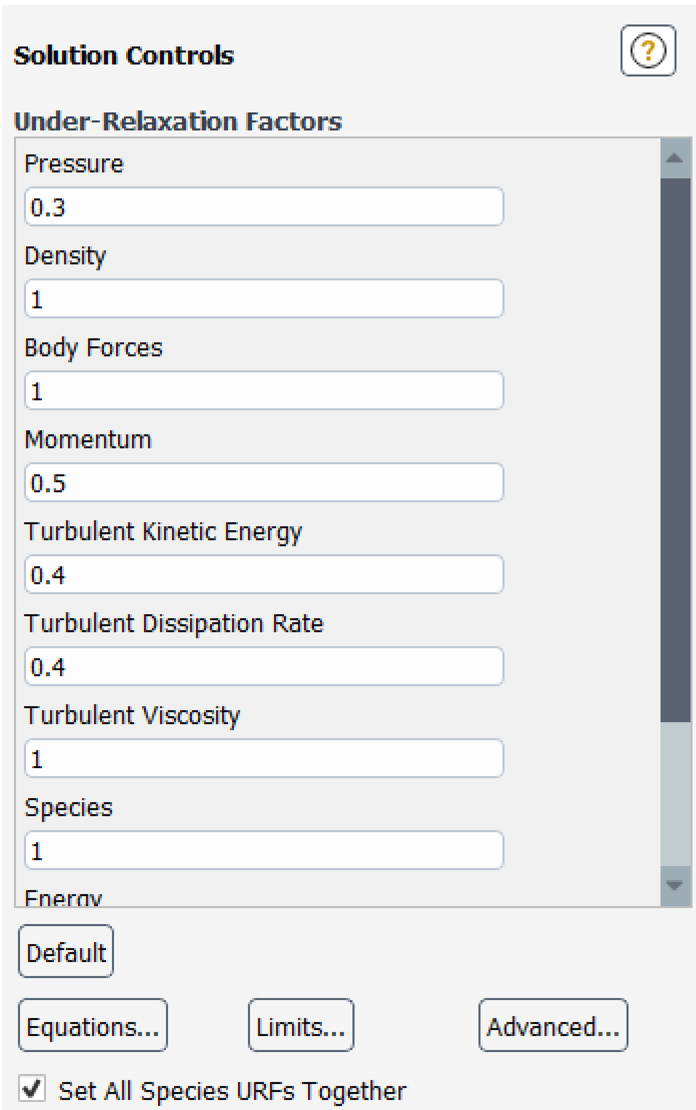
The settings under **Solution Methods** will vary depending upon the type of combustion simulations and the settings in the **Solver Settings** dialog box. Following is an example for sector combustion.



- The **Scheme** for the analysis is set to **PISO** with **Skewness Correction** and **Neighbor Correction** set to **1**, in the **Pressure-Velocity Coupling** group box.
- **Green-Gauss Node Based** is selected from the **Gradient** drop-down list.
- **Standard** is selected from the **Pressure** drop-down list.

- **Density**, **Momentum**, **Turbulent Kinetic Energy**, **Species**, and **Energy** are all set to **Second Order Upwind**.
- **Turbulent Dissipation Rate** is set to **First Order Upwind**
- **High Order Term Relaxation** and **Set All Species Discretization Together** are enabled.

12.2.9. Solution Controls



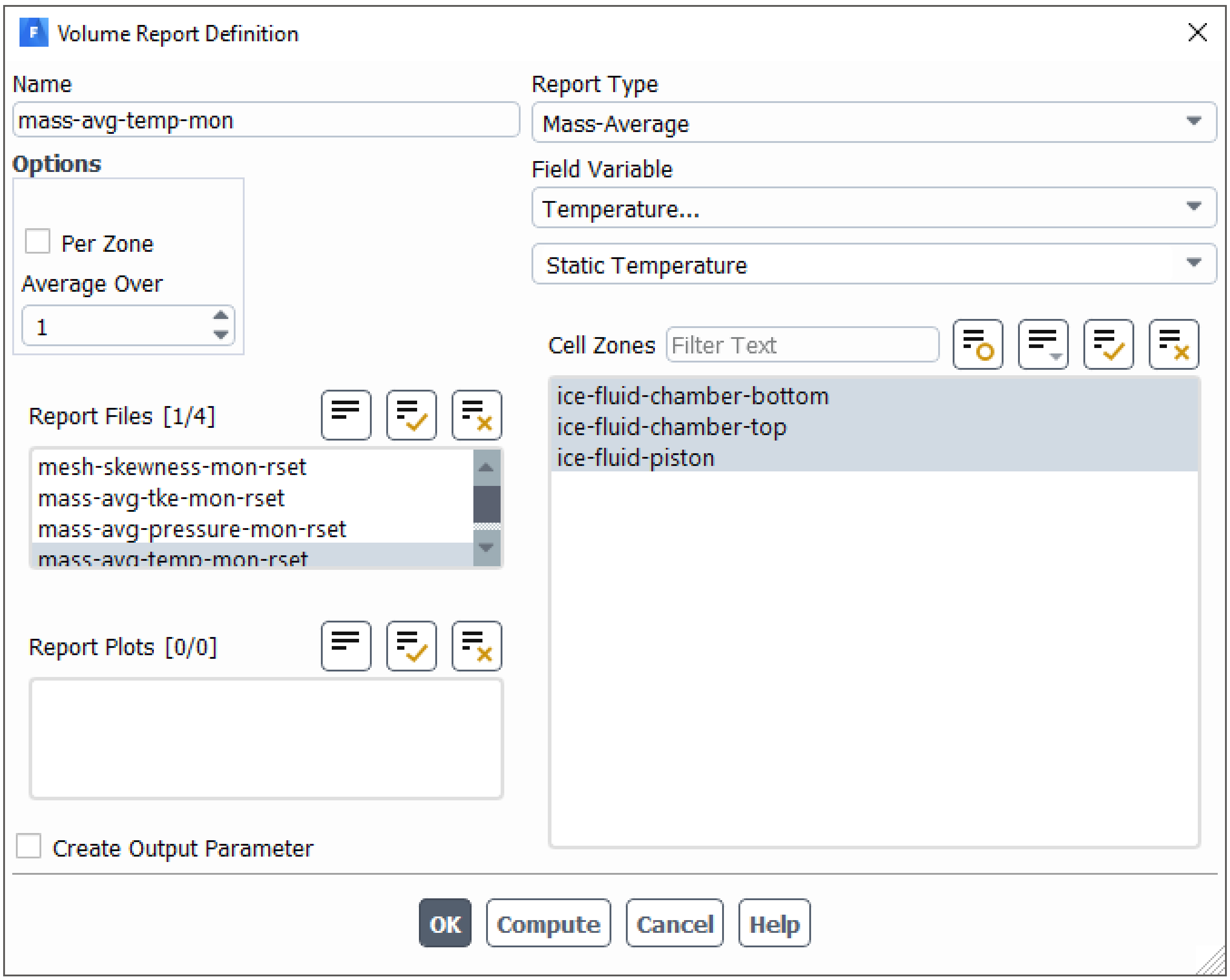
The **Under-Relaxation Factors** will vary depending upon the type of combustion simulations and the settings in the **Solver Settings** dialog box. Following is an example for sector combustion:

- **Pressure**: `0.3`
- **Momentum**: `0.5`
- **Turbulent Kinetic Energy** and **Turbulent Dissipation Rate**: `0.4`
- **Discrete Phase Sources**: `0.5`

- **Density**, **Body Forces**, **Turbulent Viscosity**, **Species**, **Energy**, and **Discrete Phase Sources**, are all set to 1
- **Set all Species URFs Together** is enabled

12.2.10. Monitors

To check the output results of the parameters, some monitors have been created. The monitors set will vary depending upon the type of combustion simulations and the settings in the **Solver Settings** dialog box. Following is just an example.



The monitors defined by default for sector are **Volume Monitors**. The variables they plot are

- **Max** of temperature, pressure, and velocity magnitude.
- **Volume Integral** of density, static temperature, and static pressure.
- **Volume Average** of pressure.
- One monitor plots the **Volume**.

These monitors are defined on the all the listed cell zones.

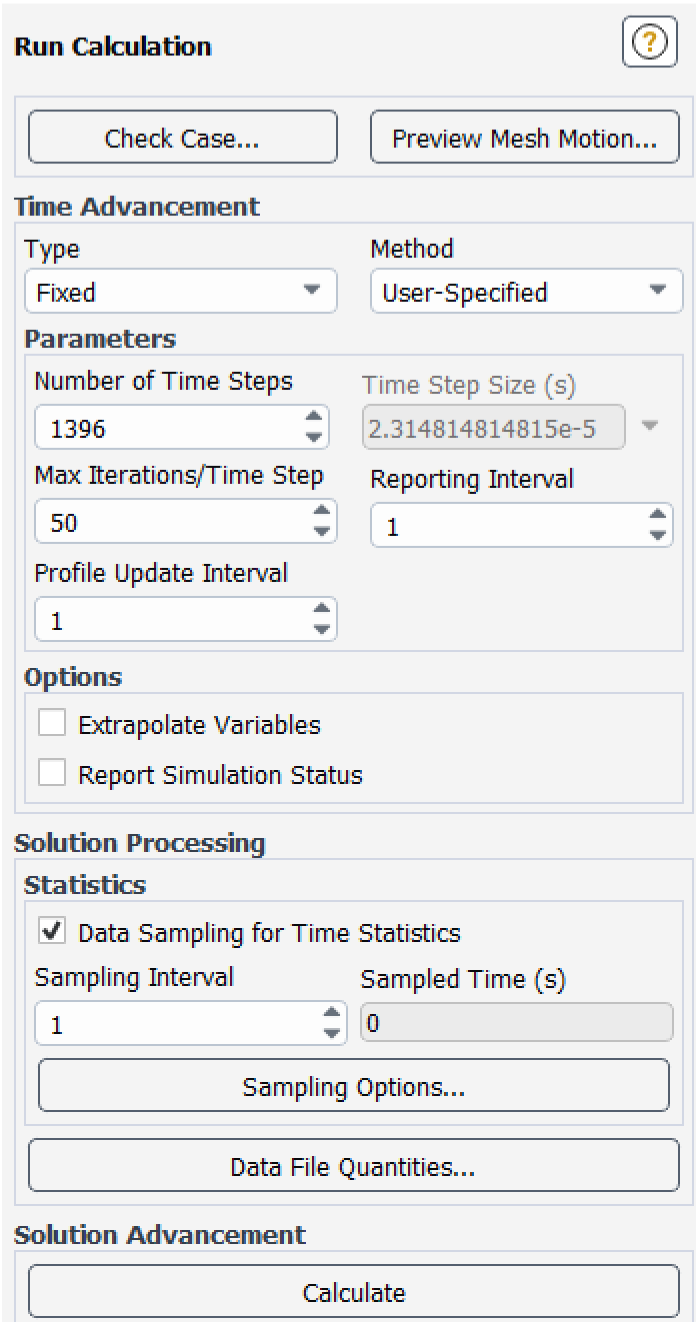
You can create your own **Surface Monitors** or **Volume Monitors** from the **Monitors** task page. Enable **Write** so that an output file is written, which can be loaded and viewed later. The monitor settings which are done by default can be seen in the `icBcSettings.txt` file.

12.2.11. Solution Initialization

Initialization settings will vary depending upon the type of combustion simulations and the settings in the **Solver Settings** dialog box. Following is an example for sector combustion. Standard initialization is used.

- The **Initial Values** of **Turbulent Kinetic Energy** and **Turbulent Dissipation Rate** are set to `1`.
- **Temperature** is set to `300`.
- The **Initial Values** of all the rest of the parameters are set to `0` or as per the values you have set through the **Solver Settings** dialog box.

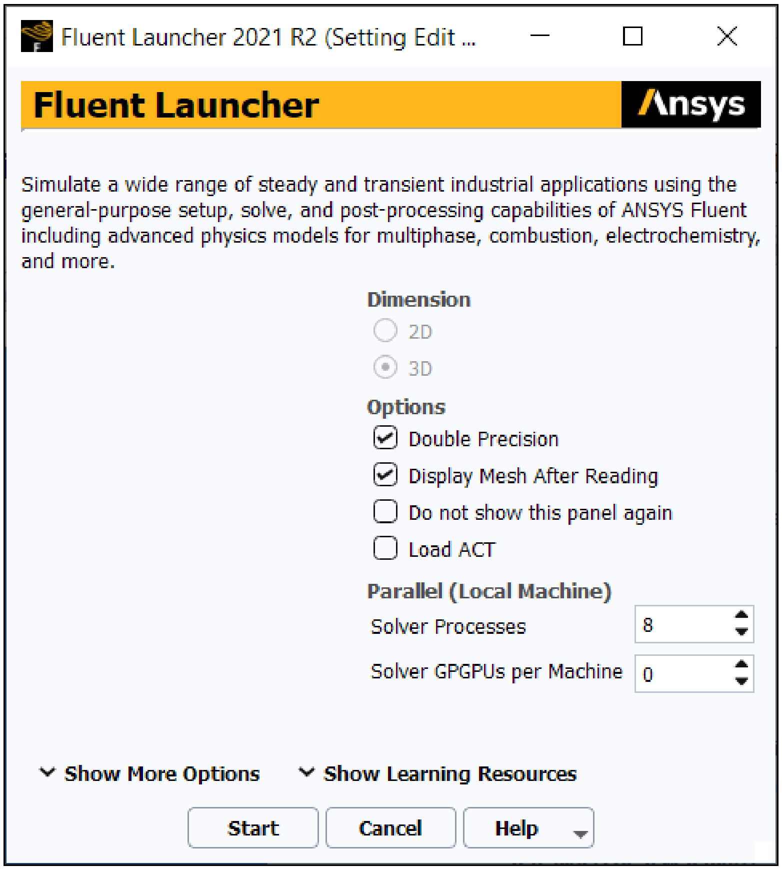
12.2.12. Run Calculation



You can enter the **Number of Time Steps** you want. Click **Calculate** to run the simulation.

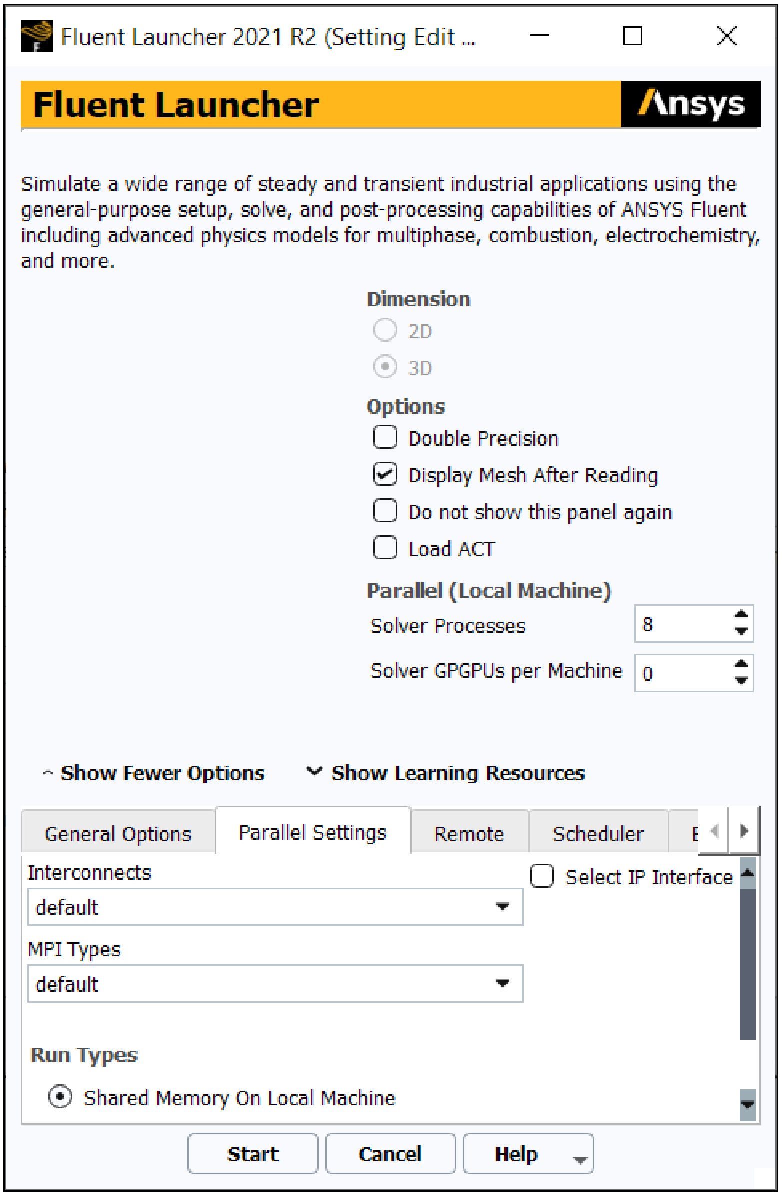
Running the Calculation from Windows on Remote Linux Machines

To run the simulation on Linux, some changes in the setup are required.



1. In the **FLUENT Launcher**, click **Show More Options** to display the tabs.

2. Click the **Parallel Settings** tab.



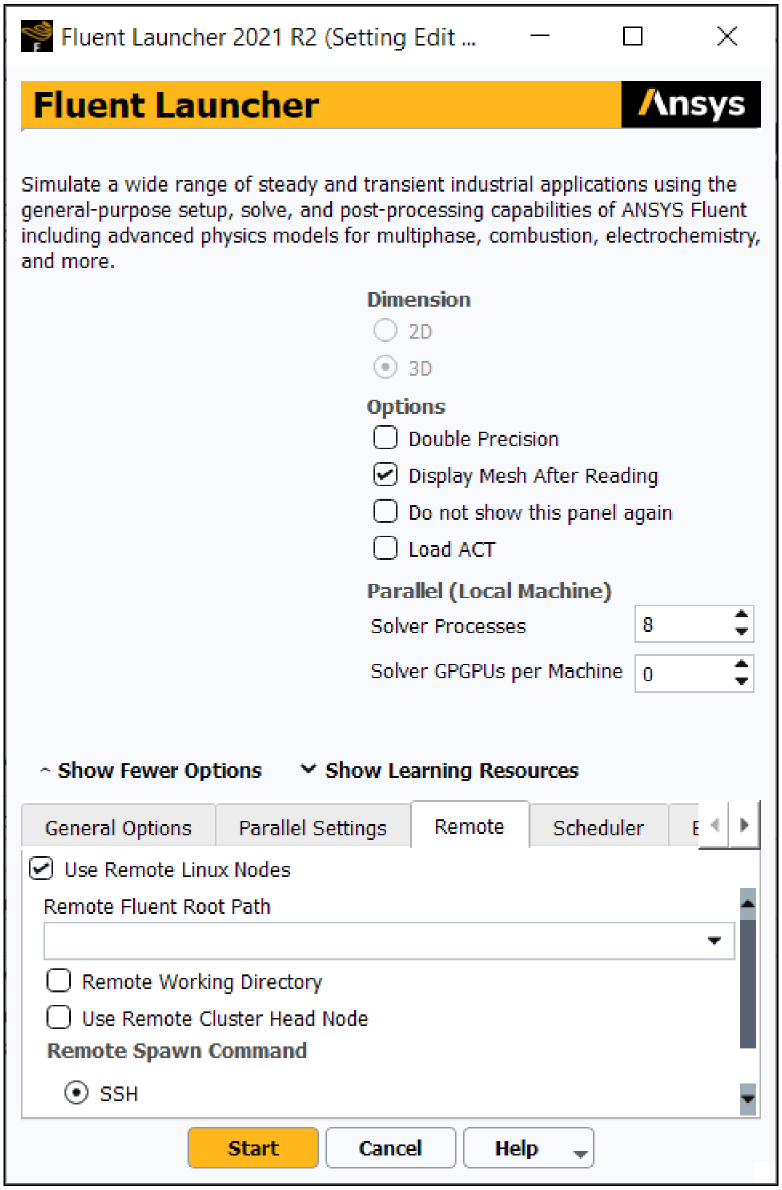
a. Select the **Run Types** from the list.

Note:

If you selected **Distributed Memory on a Cluster**, then you have to provide the **Machine Names**.

---

3. Click the **Remote** tab.



a. Enable **Use Remote Linux Nodes**.

b. Enter the path for **Remote FLUENT Root Path**. It is the root directory of Fluent installation.

c. Select your choice from the list of **Remote Spawn Command**.

Note:

For **SSH**, you need to set up a password-less connection before running the solution remotely.

---

d. Enable **Use Remote Cluster Head Node** and provide the node.

e. Enter the number for **Number of Processes** under **Processing Options**.

> **Note:**
>
> When the node for **Use Remote Cluster Head Node** is specified, then that name will be reflected under **Processing Options**.

When the remote connection is set, it will use the nodes of the remote Linux machine. The details are displayed in the Fluent console.

```
Host spawning Node 0 on machine "pundevrhel64r6" (lnamd64).
/usr/local/fluent/develop/fluent14.0.0/bin/fluent -r14.0.0 3d -pdefault -node -t4 -ssh -mport 10.14.6.198:10.14.2.108:2673:0
Starting /usr/local/fluent/develop/fluent14.0.0/multiport/mpi/lnamd64/pcmpi/bin/mpirun -np 4 /usr/local/fluent/develop/fluent14.0.0/l
Platform-MPI licensed for FLUENT.

------------------------------------------------------------------------------
ID      Comm.   Hostname        O.S.        PID     Mach ID HW ID   Name
------------------------------------------------------------------------------
host    net     punptstxp64t1   Windows-x64 177216  1       5       Fluent Host
n3      pcmpi   pundevrhel64r6. Linux-64    17969   0       3       Fluent Node
n2      pcmpi   pundevrhel64r6. Linux-64    17968   0       2       Fluent Node
n1      pcmpi   pundevrhel64r6. Linux-64    17967   0       1       Fluent Node
n0*     pcmpi   pundevrhel64r6. Linux-64    17966   0       0       Fluent Node

Selected system interconnect: shared-memory
```

All the files will be saved in the working directory. Once the solution is completed, the project is updated and you can proceed with your postprocessing.

For details, see Setting Additional Options When Running on Remote Linux Machines in the Fluent User's Guide.

> **Note:**
>
> You can also submit the run on different Linux machines though RSM.

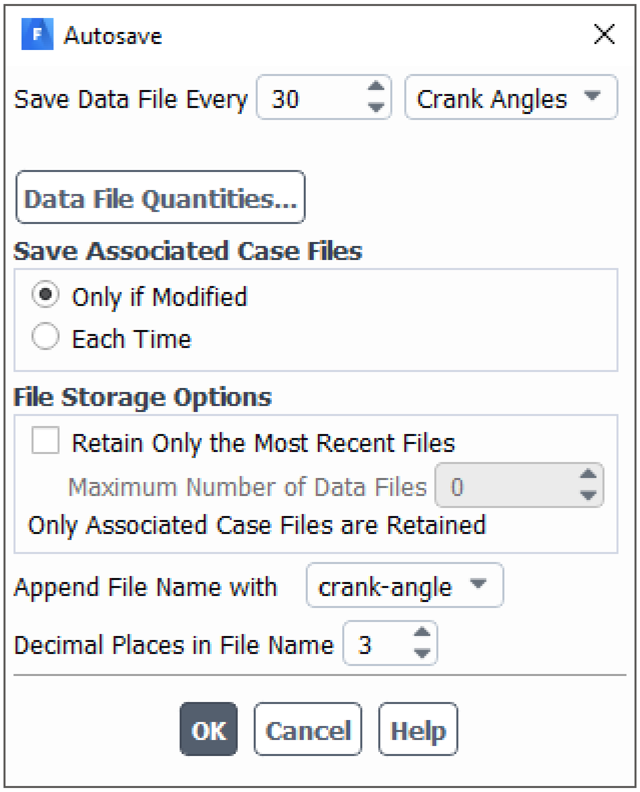
Running on Standalone Fluent

1. Open Fluent and read the case and data file from the `~project-name_files\dp0\ICE\Fluent` directory.

   File → **Read** → **Case & Data...**

2. **Note:**

   In **Calculation Activities** task page click **Edit....** next to **Autosave Every (Time Steps)** and check if that the name and path under **File Name** is correct.

3. In **Run Calculation** task page enter the required **Number of Time Steps** and click **Calculate**.

   Note:

   If the case is setup in the previously released versions, then you might have to read the scheme file, (`WB-ICE-Solver-Setup.scm`) from `~ANSYS Inc\v150\Addins\ICEngine\CustomizationFiles` folder in the mapped directory before running it in the present version.

Note:

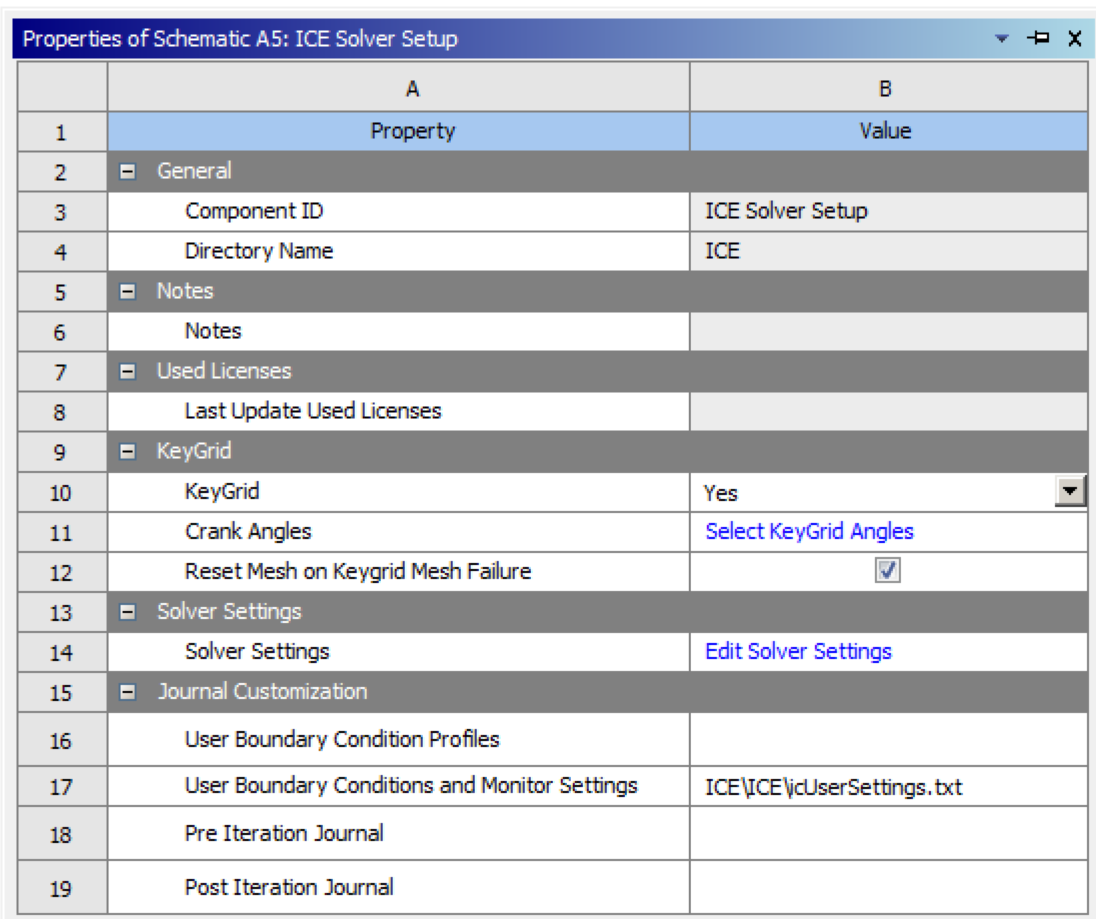
If you want the generate the report from IC Engine System report template, then you need to first make sure that **Solution** cell is updated. This can be done by running the solution for 1 timestep. Then, copy all the files generated by your standalone run into the Fluent directory and update the **Results** cell.

Chapter 13: KeyGrid in IC Engine

This chapter explains what are KeyGrids and how you can set them up. The KeyGrid option is applied to provide multiple meshes during pre-processing for the solver run. The information in this chapter is divided in the following sections:

13.1. KeyGrid Setup in Solver

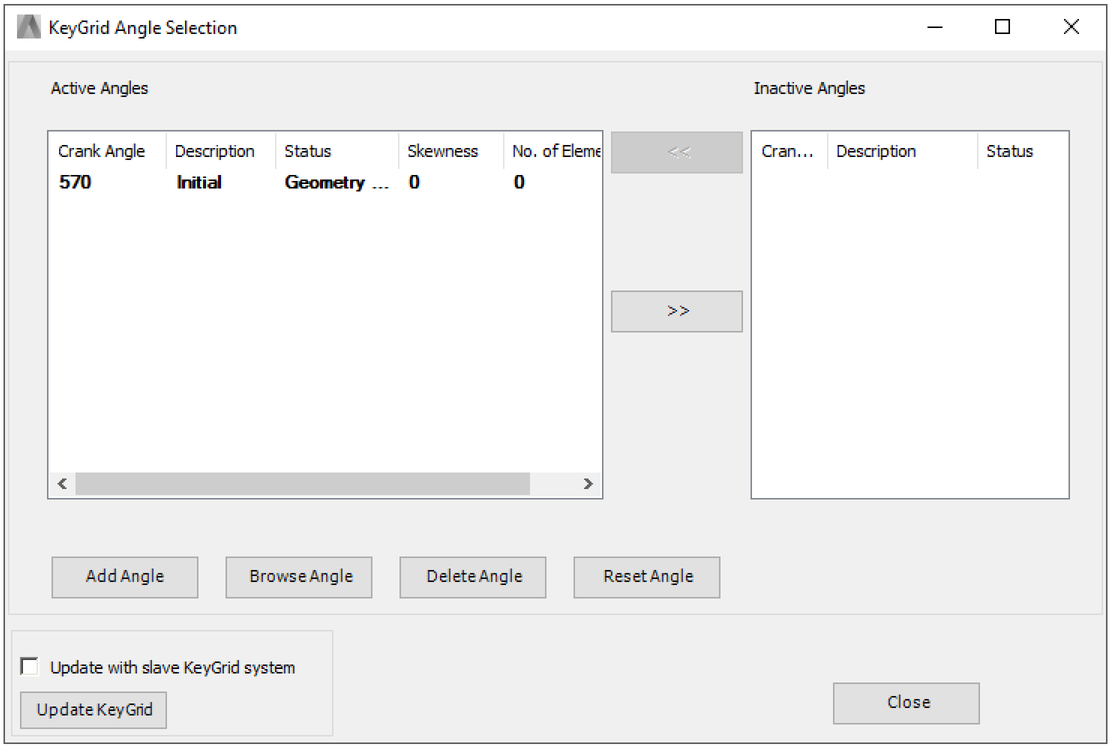
In the **Properties** pane of the **ICE Solver Setup** cell select **Yes** from the **KeyGrid** drop-down list.



Properties of Schematic A5: ICE Solver Setup

	A	B
1	Property	Value
2	General	
3	Component ID	ICE Solver Setup
4	Directory Name	ICE
5	Notes	
6	Notes	
7	Used Licenses	
8	Last Update Used Licenses	
9	KeyGrid	
10	KeyGrid	Yes
11	Crank Angles	Select KeyGrid Angles
12	Reset Mesh on Keygrid Mesh Failure	☑
13	Solver Settings	
14	Solver Settings	Edit Solver Settings
15	Journal Customization	
16	User Boundary Condition Profiles	
17	User Boundary Conditions and Monitor Settings	ICE\ICE\icUserSettings.txt
18	Pre Iteration Journal	
19	Post Iteration Journal	

An additional option is now seen under **Keygrid**, **Reset Mesh on Keygrid Mesh Failure**. As this is enabled by default, it will reset the KeyGrid mesh if it fails and to generate it again. The **Crank Angles** property is now displayed. Click **Select KeyGrid Angles** to open the **Crank Angle Selection** dialog box.



Active Angles

contains a list of crank angles which will be considered for update. These will be the ones you have set for keygrid. The list displays the information about the crank angle:

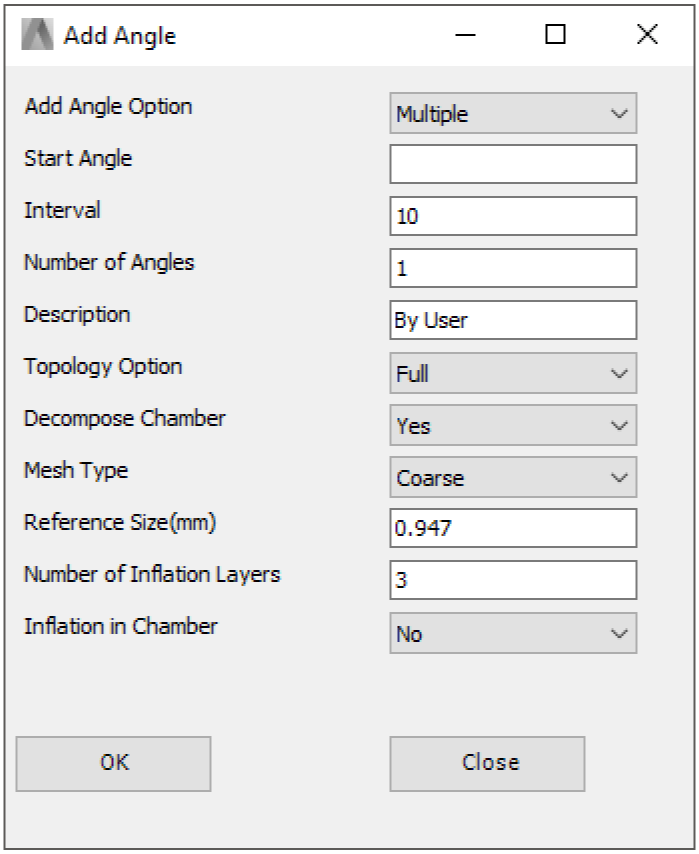
- **Description**: One crank angle will be added by default, **Initial**. You can edit the **Description** of the other crank angles you add.
- **Status**: This column shows the current status of the crank angle. For example, it could be **Mesh Generated**, **Not Generated**, **Geometry Failed**, or **Mesh Failed**.
- **Skewness** displays the skewness of the mesh.
- **No of Elements** displays the number of elements of the generated mesh.

Inactive Angles

contains a list of crank angles which have been removed from the list of **Active Angles** with the help of the arrow buttons.

Add Angle

will open the **Add Angle** dialog box.



Add Angle Option

To add a single angle select **Single** from the **Add Angle Option** drop-down list. Multiple angles can be added by selecting **Multiple** .

Angle

When you select **Single** you can enter the specific angle in the **Angle** text box.

Start Angle, Interval, Number of Angles

When you select **Multiple** from the **Add Angle Option** drop-down list, the multiple angles will be created by entering values for **Start Angle**, **Interval** and **Number of Angles**.

Description

You can edit the **Description**.

Topology Option

You can select **Full** for ICE topology, or **Single** for single zone topology, from the **Topology Option** drop-down list. This will generate meshes with your choice of topology in KeyGrid. Default choice will be the topology of the default angle.

Decompose Chamber

You can select **Yes**, **No**, or **Program Controlled**. For more information see chapter Cold Flow Simulation: Preparing the Geometry (p. 164).

Mesh Type

You can choose **Medium**, **Fine**, or **Coarse** from the **Mesh Type** drop-down list.

Reference Size

You can edit the **Reference Size** value. This will change the other dependent mesh control parameters.

Number of Inflation Layers

You can edit the default value.

Inflation in Chamber

This is set to **No** by default. You can change this if required.

Local Refinement Around Spark

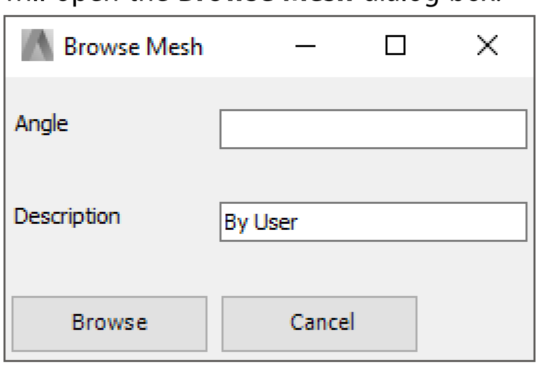
This option will be present only when spark is defined in a combustion simulation. This option will be disabled if **Multiple** is chosen from the **Add Angle Option** drop-down list. The default selection will be **Yes** only if the crank angle is between IVC and (TDC+10). You can select **Yes** or **No** from the drop-down list. Spark refinement for the mesh will only be done if **Yes** is selected for **Local Refinement Around Spark**.

Refinement Radius

By default this value is set to 3.3 X **Reference Size**.

Browse Angle

will open the **Browse Mesh** dialog box.



Here enter the angle for which you would like to read and replace the mesh. Click **Browse** to open the browser window where you can select the mesh. While selecting the mesh, ensure that it meets the minimum zone requirement as required by the IC Engine system. For more information about the IC Engine supported mesh topologies go to Supported Mesh Topologies (p. 477).

> **Note:**
>
> While browsing the mesh ensure that the order of selection of vales while decomposition is the same as for the mesh you are browsing. The valve names and zone names must be consistent with the upstream data.

Delete Angle

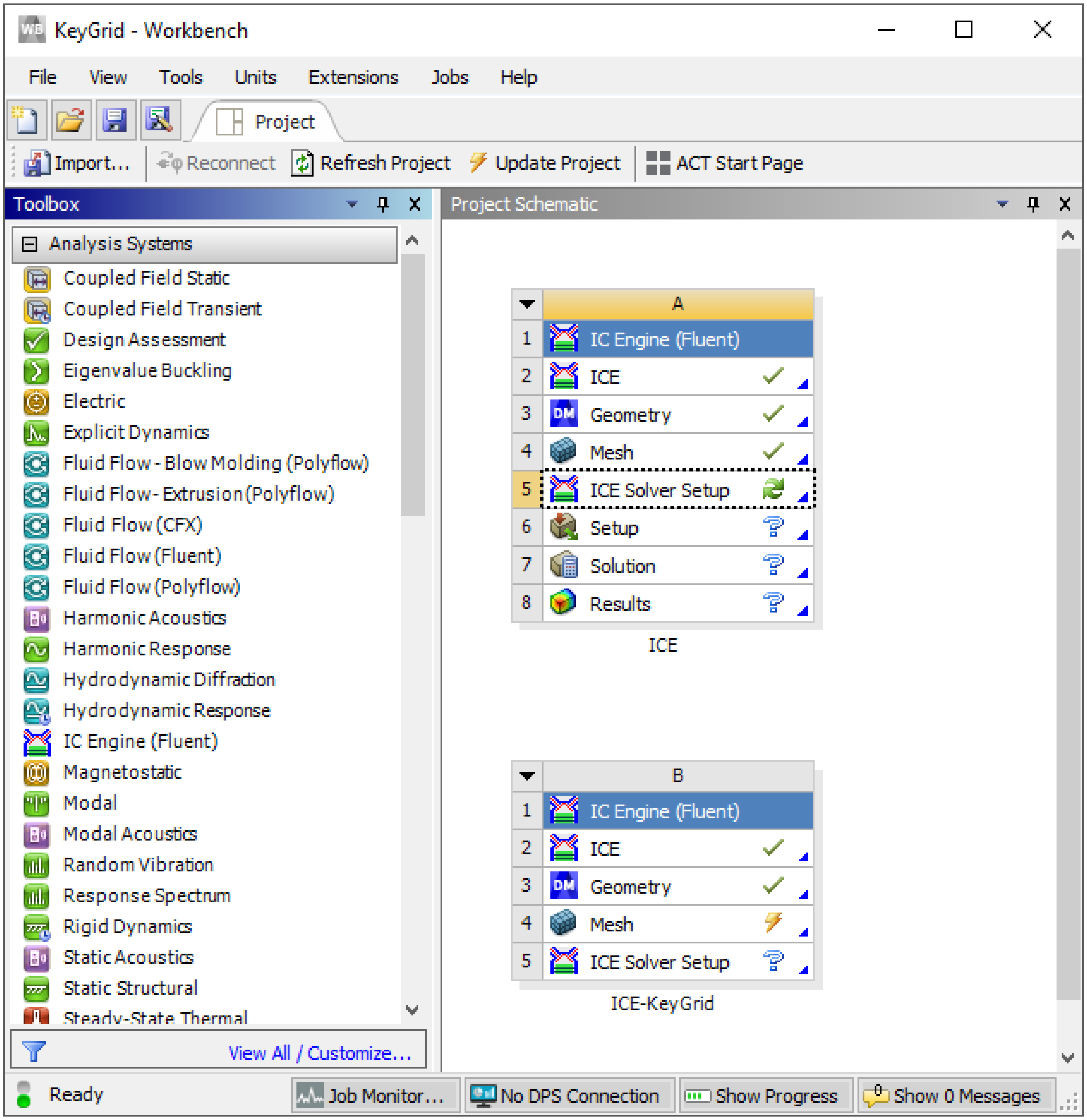
will remove the selected angle from the **Active Angles** list.

Reset Angle

will clear the mesh and other backup data of the selected angle from the **Active Angles** list.

Update with duplicate system

When this option is enabled, and you click **Update Keygrid**, a new system is created in the Workbench **Project Schematic**.



If large number of crank angles are included in KeyGrid, substantial amount of time (over few hours if frequent keygrids are used) is required to complete the meshing before you can start the solver. However, the solver will not need these meshes immediately.

An option to run the solver while meshes are being generated is therefore highly desirable and will greatly expedite the workflow. This feature of creating a new system, will run mesh generation and solution simultaneously to save time.

The new system is a duplicate system of the original **ICE** system and will be named as **ICE-KeyGrid**. This duplicate system will have the cells only till **ICE Solver Setup**. This system will update the KeyGrid meshes. In the meanwhile you can do solver setting changes and run the Fluent solution. You can check the status of mesh generation anytime by clicking on **Select KeyGrid Angles** which opens the **KeyGrid Angle Selection** dialog box.

Once the KeyGrid duplicate system is created, **KeyGrid Angle Selection** dialog box will point to the data in the duplicate system only. Any changes made in the dialog box like add, remove, reset angle, set new default, etc. will be affected in the duplicate system only.

In the solver for each of the KeyGrids defined in the duplicate **ICE-KeyGrid** system, separate mesh-replace event will be created. So even if you insert new KeyGrid in the duplicate system while the solver run is going on, then also dynamic event of mesh replace will be created for that newly inserted KeyGrid. If during the foreground solver run, the KeyGrid mesh file is not found due to mesh failure or the Keygrid being not generated as yet, then the solver will stop at that event for user input. A dialog box will pop up which will have two options —

- **Check for Mesh File** — Clicking this, the solver will search for the mesh file in ../ICE/Keygrids directory and in the current working directory. If it finds the mesh in either place mesh replacement is done. If it cannot find the mesh then the same dialog box will pop up again.

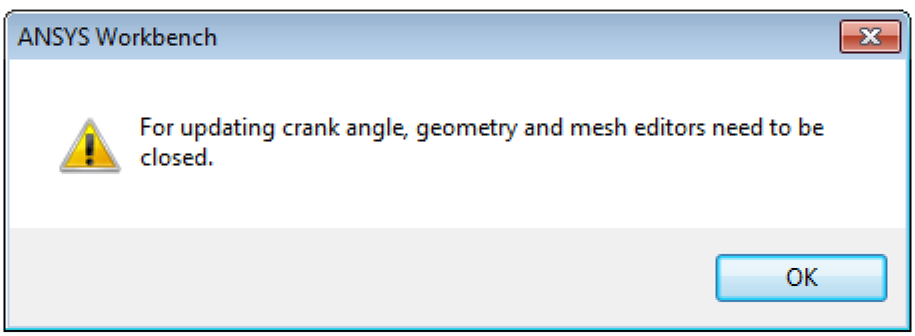
- **Continue with Existing Mesh** — On clicking this option, the solver will proceed without mesh replacement.

If the solver is running in the background mode and if the mesh file is not found then this dialog box will not pop up and the solution will continue without mesh replacement.

Update Keygrid

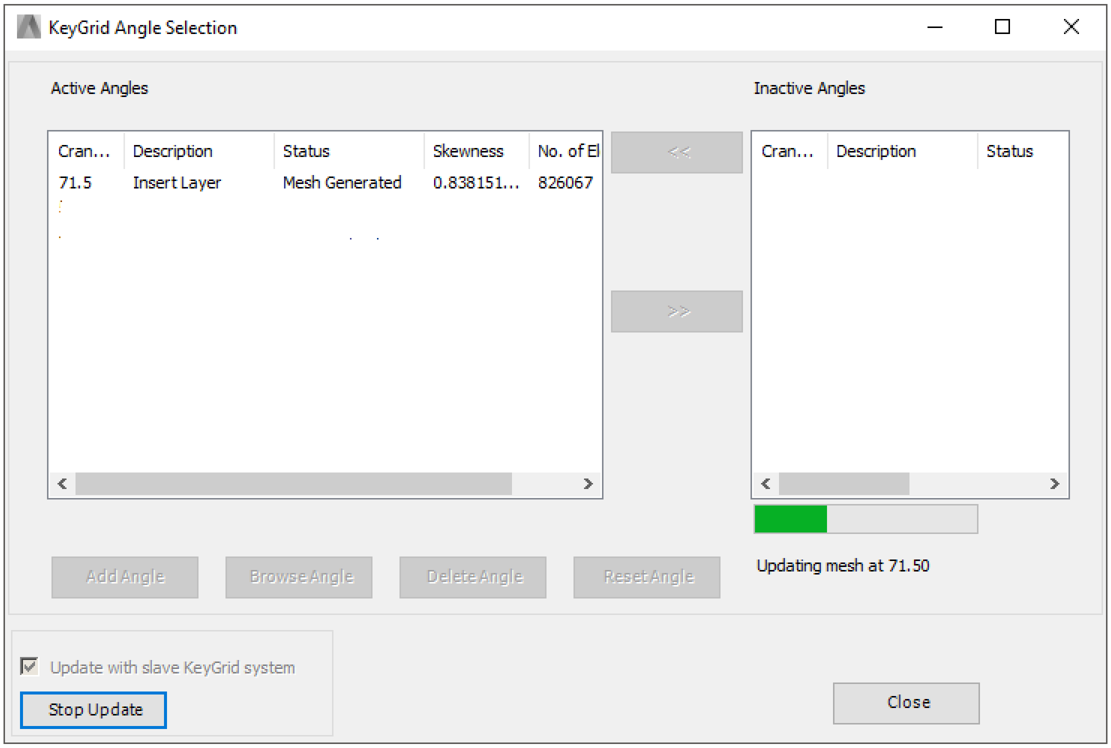
will update the mesh for all those crank angles in the **Active Angles** list, whose status is other than **Mesh Generated**.

After **Update Keygrid** is clicked you will get a message of closing the mesh and geometry editors.



Ensure that all the windows are closed and click **OK**.

The **Update Keygrid** button is now replaced with the **Stop Update** button. Also a progress bar and current update status are displayed.



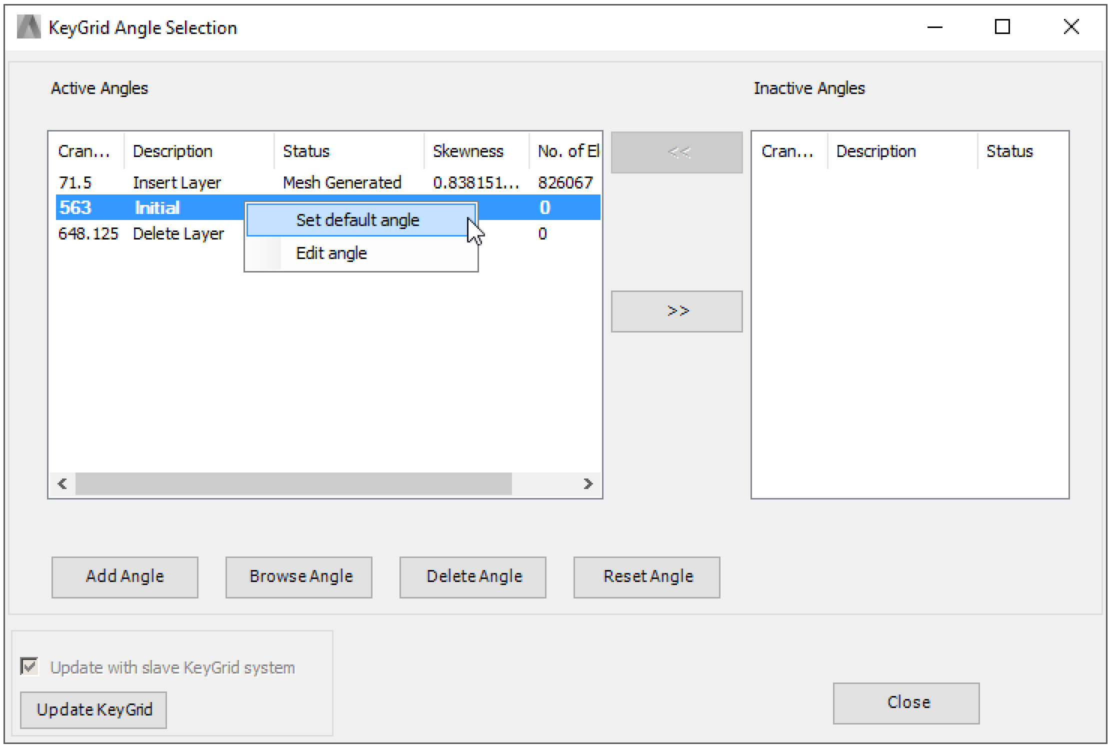
Once update starts, a backup will be taken of the project and mesh files. Mesh files will be saved in the `KeyGrids` folder. The other files will be saved in the `BackupFiles\Crank Angle` folders. If you click **Stop Update** the update process is terminated after completing the current geometry or mesh generation. After update is finished or terminated, `keygrid.txt` file is written which contains a list of the angles for a which a mesh is successfully generated.

The first time the **Crank Angle Selection** dialog box opens the **Initial Angle** will be the default angle. This angle will be highlighted. To change the default angle right-click on the angle of your choice from the list of **Active Angles** and click on **Set default angle**. You will be asked to confirm if you would like to set the selected angle as the default angle. After you click **Yes** the angle chosen to be default will be highlighted.

Important:

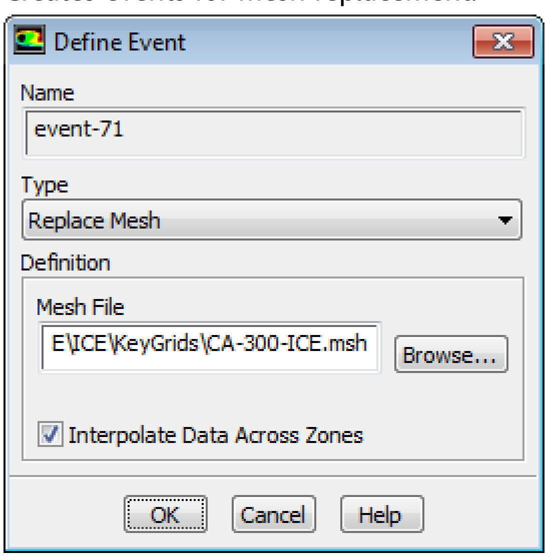
When the option of **Update with duplicate system** is enabled, then in duplicate system if you change the default angle it will not be reflected in the original system.

Before the mesh in generated you can edit the properties of the added angle. Right-click on the angle of your choice from the list of **Active Angles** and click on **Edit angle**. Other than the **Add Angle Option** and **Angle** options others can be edited.



In the list of **Active Angles** you will see the crank angles. If you do not want to update the mesh for any of the crank angles in the list, select it and move it to the **Inactive Angles** list. You can always add these angles later to the list of **Active Angles**. If you would like to update the mesh for an additional angle, click **Add Angle** and enter the angle of interest. You can remove a selected angle by selecting and clicking **Delete Angle**. After you click **Update**, the mesh is updated for all the **Active Angles**. Fluent automatically:

- Replaces mesh.
- Creates required interface.
- Creates events for mesh replacement.



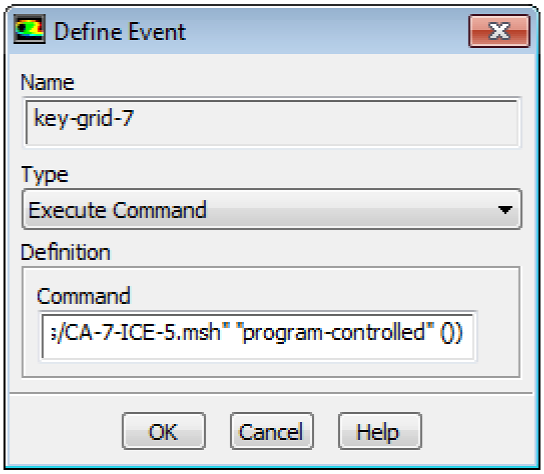
Following is an example of a KeyGrid mesh replace event:

```
(wb-ice-mesh-replace "../ICE/KeyGrids/CA-300-eygrid.msh" "program-controlled" '(fluid-ch fluid-exvalve-1-port fl
```

Here the argument `'(kg-wait-seconds 600)` is set to 600 by default. So the event waits for 600 seconds or 10 minutes to find the mesh. If the mesh file is not found, then a message will be printed for `'(kg-wait-seconds 600)/10` times. So if you are not monitoring the run then a message will popup 60 times, which is every 10 seconds for 10 minutes. You can change the default waiting period.

- Interpolates data.

  By default, data is interpolated between matching zone pairs. That is, between the zones with the same names in both the current mesh and the replacement mesh. A keygrid event of enabling interpolate across zones option is created in Fluent.

  The command in the event will be something similar to :

- `(wb-ice-mesh-replace "../ICE/KeyGrids/CA-7-ICE-5.msh" "program-controlled" (fluid-ch fluid-piston fluid-crevice))`

  In the command you can enable or disable the interpolate across zones option.

  – When this option is set to `yes` then the solution data will be interpolated across the zones.

  – When it is set to `no` then the solution data will not be interpolated across the zones but will be interpolated from each zone to the corresponding zone in the replaced/new mesh.

  – When this option is set to `program-controlled`, then number of fluid zones in keygrid mesh will be extracted from `keygrid.txt` file present in /ICE/Keygrids directory. So, if you are running the solution in stand alone Fluent (instead of Workbench) with different directory structure, then you should copy `keygrid.txt` in present- working directory. If all the fluid zones in the existing mesh are not present in the new replacement mesh then the option will be set to `yes`. If the fluid zones in both meshes match then the option will be set to `no`.

  You can change the `program-controlled` option in the **Define Event** dialog box.

  Note:

  Global conservation of data is not enforced when the Interpolate Data Across Zones option is enabled, so it should only be used when absolutely necessary. For best data interpolation, the zone boundaries of the replacement mesh should be coincident with those of the current mesh.

For more information see Replacing the Mesh.

- Continues solution for the requested number of time steps.

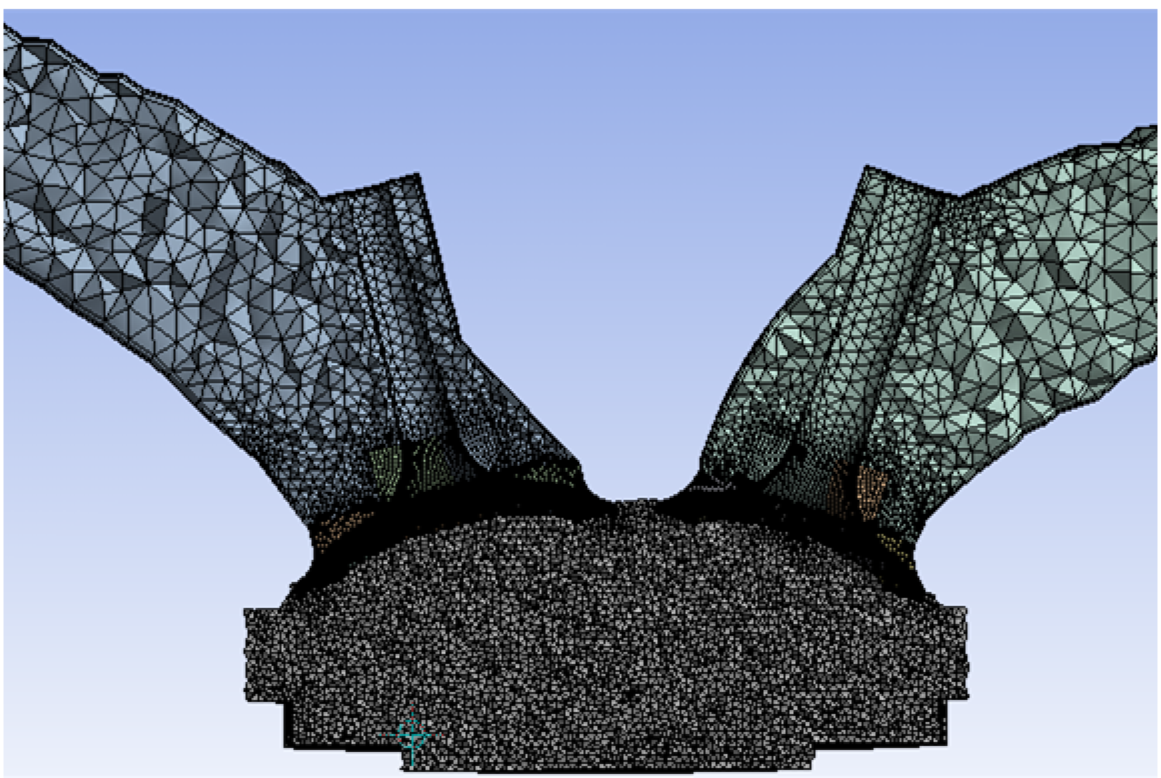
This is how you can set up the simulation using KeyGrids.

13.2. Importance of KeyGrid

During a complete engine cycle, the mesh changes with the crank angle. If required, you can change the crank angle, and observe the changes in the decomposed geometry and the mesh, at the different crank angles.

When crank angle is set to **0** (TDC position), the chamber is not decomposed. The chamber mesh consists of single tetrahedron mesh zone. See Figure 13.1: Mesh at 0 Degrees Decomposition Angle (p. 471).

Figure 13.1: Mesh at 0 Degrees Decomposition Angle



When the crank angle is increased beyond the piston cutoff limit, then the planes created automatically in decomposition are used to decompose the chamber into three bodies: fluid-ch, fluid-layer-cylinder, and fluid-piston. New named selections are created for these bodies and the mesh controls are updated. The valves are moved to their position defined by the valve lift curves.

Figure 13.2: Piston Cutoff

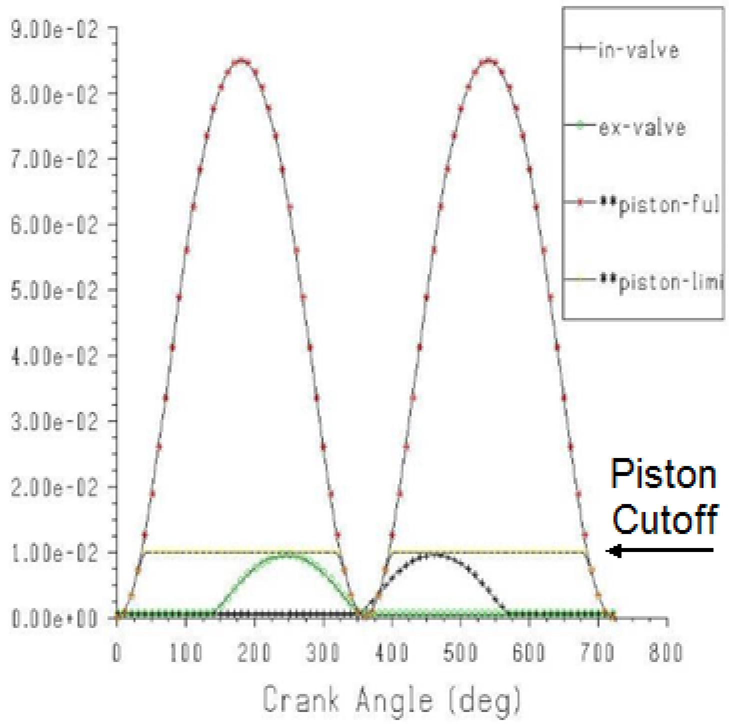


Figure 13.3: Decomposition at Different Decomposition Crank Angles

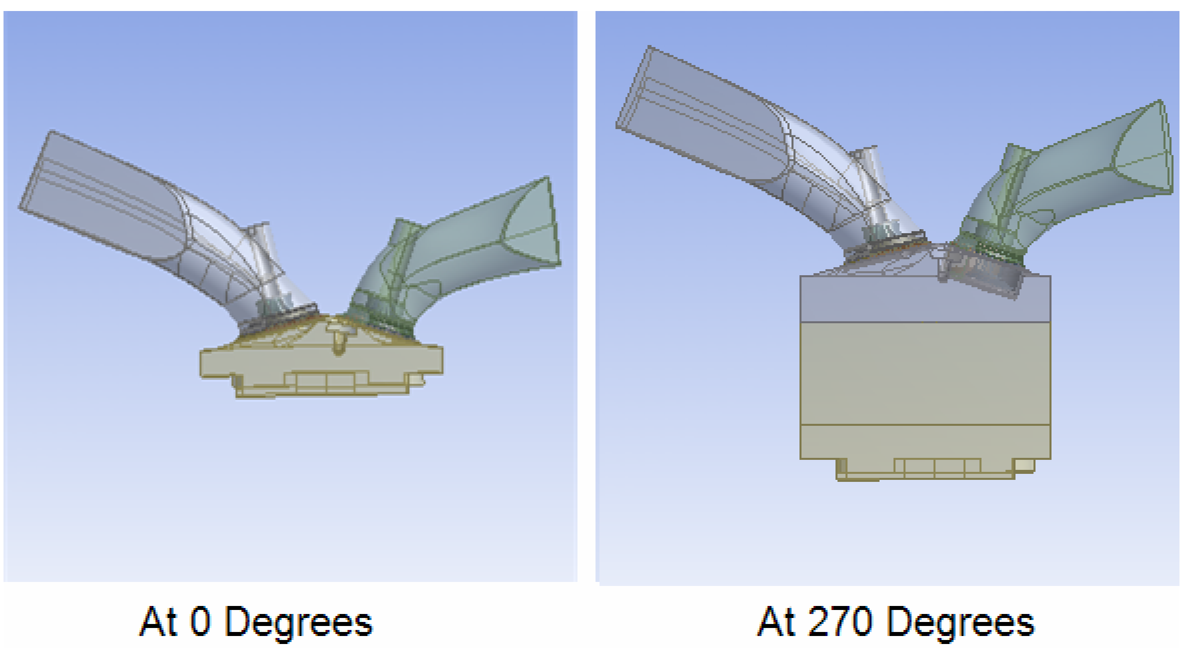
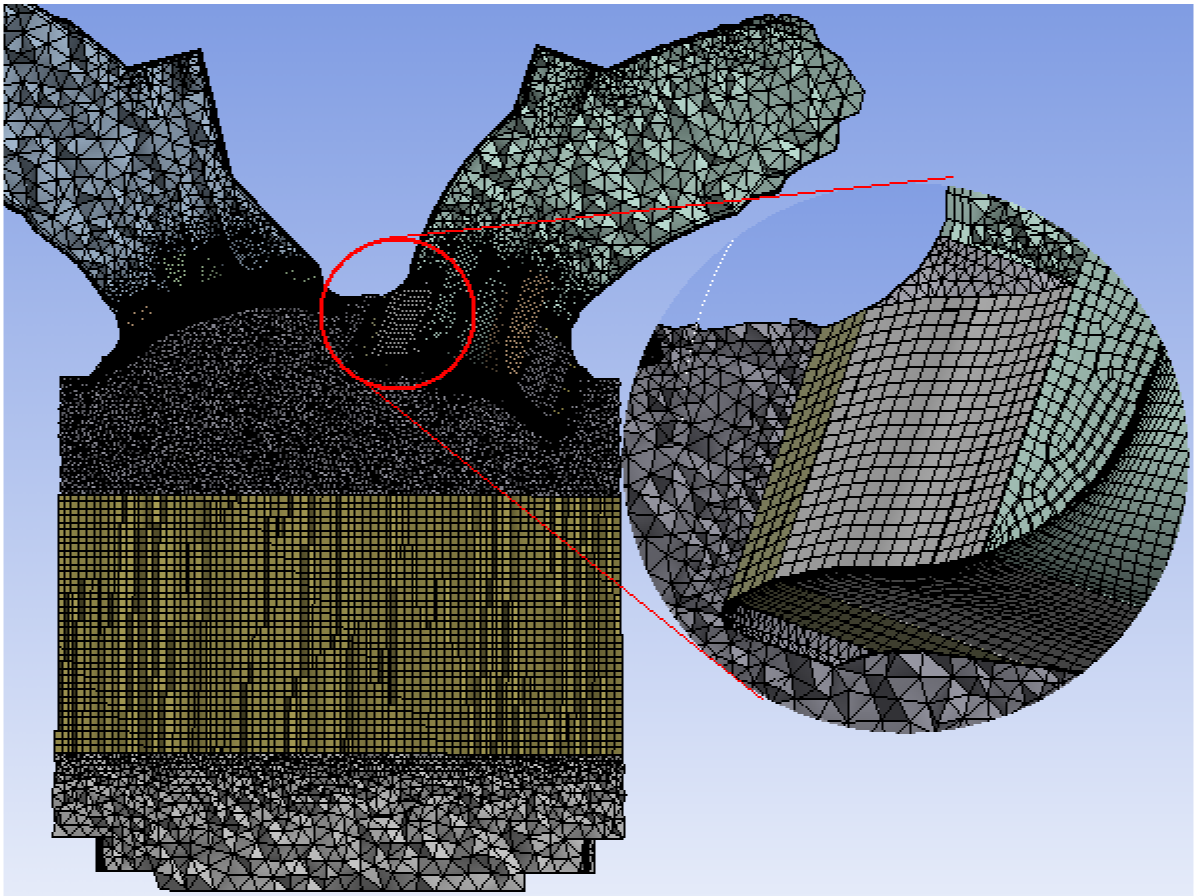


Figure 13.4: Mesh at 270 Degrees Decomposition Crank Angle



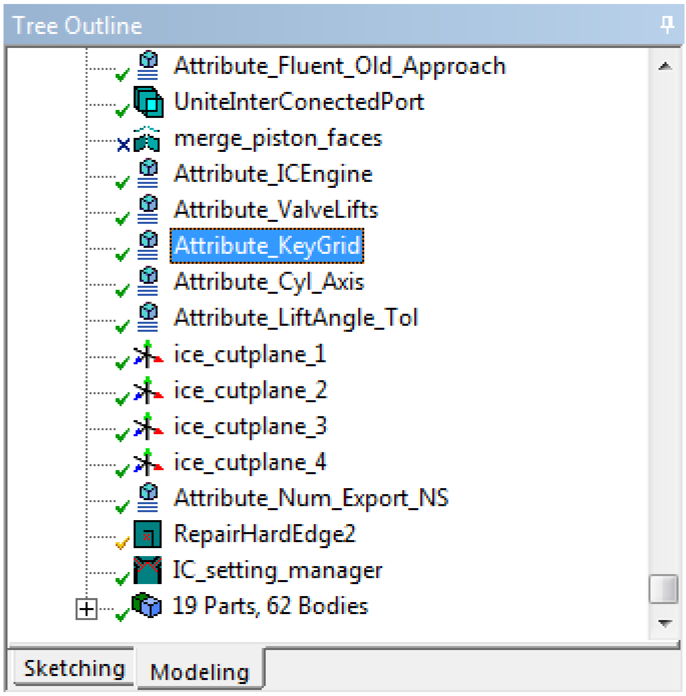
After meshing you will be able to see three different mesh zones. The center one is the **layer-cylinder** and contains the sweep mesh. As the valves open, the mesh will be updated depending upon the valve lift. Thus using parameters you can observe the effect of change of crank angle on the decomposition and the mesh.

Chamber Inflation in KeyGrids

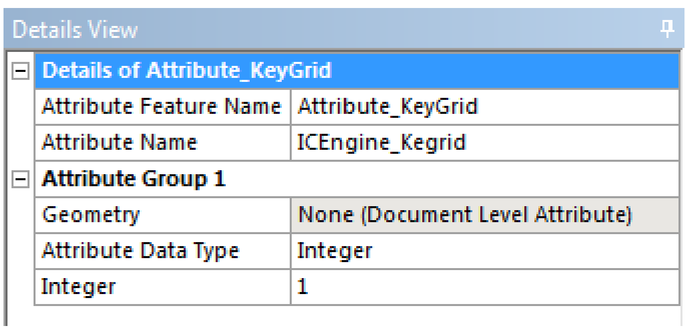
In KeyGrid, Fluent meshes are replaced by better quality Ansys Meshing meshes. It is beneficial to have a prism layer or inflation in chamber region, so that the boundary layer flows can be correctly captured.

When **Decompose Chamber** is set to **No**, chamber inflation is created if the following conditions are satisfied:

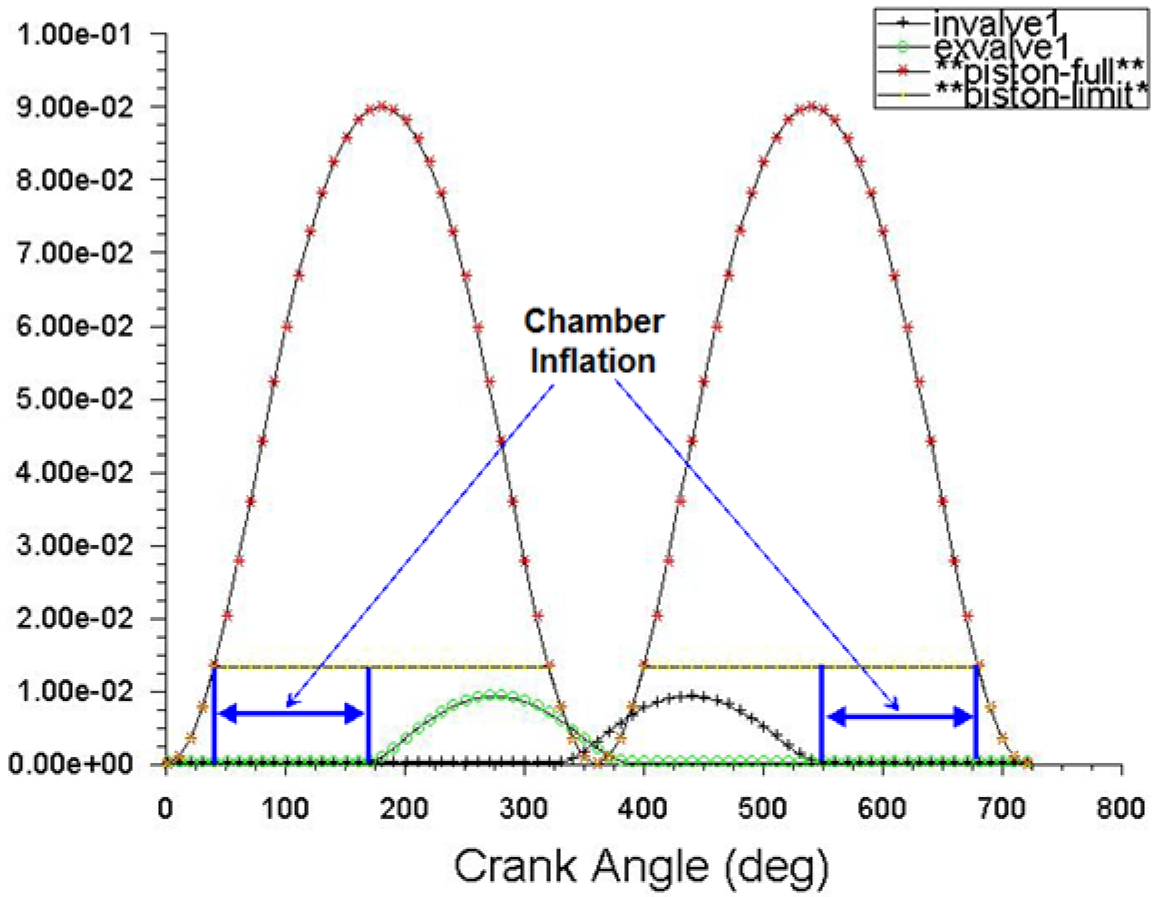
- The piston has moved more than the piston stroke cutoff distance.
- All valves are closed. A tolerance of two degrees before valve opening is considered.
- The KeyGrid attribute is set. This attribute is present in the **Tree Outline** of the DesignModeler.



Under the **Details of Attribute_Keygrid** set the **Integer** value to 1.

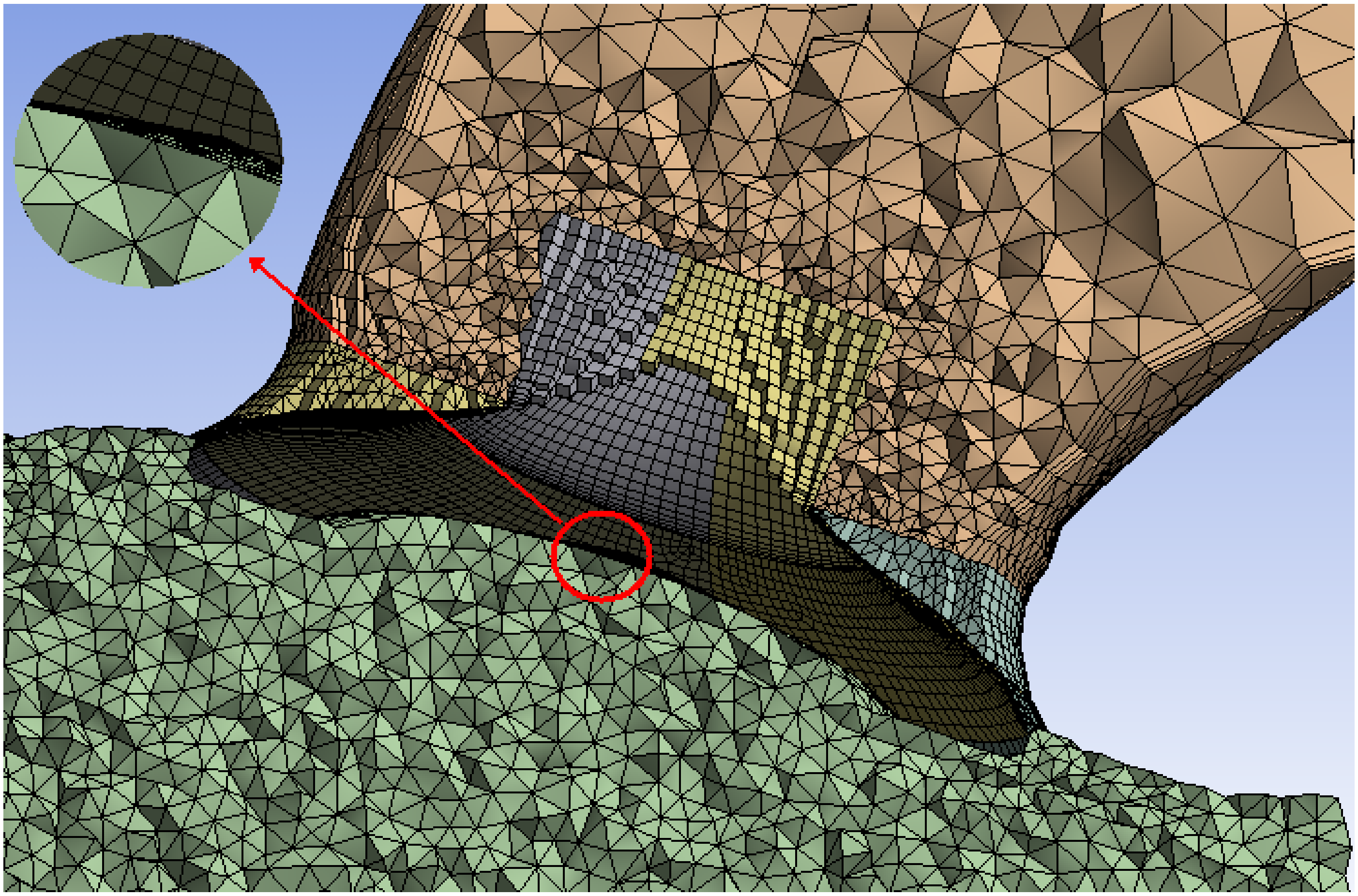


In the following valve-piston motion profile, the angles at which chamber inflation is possible are marked.



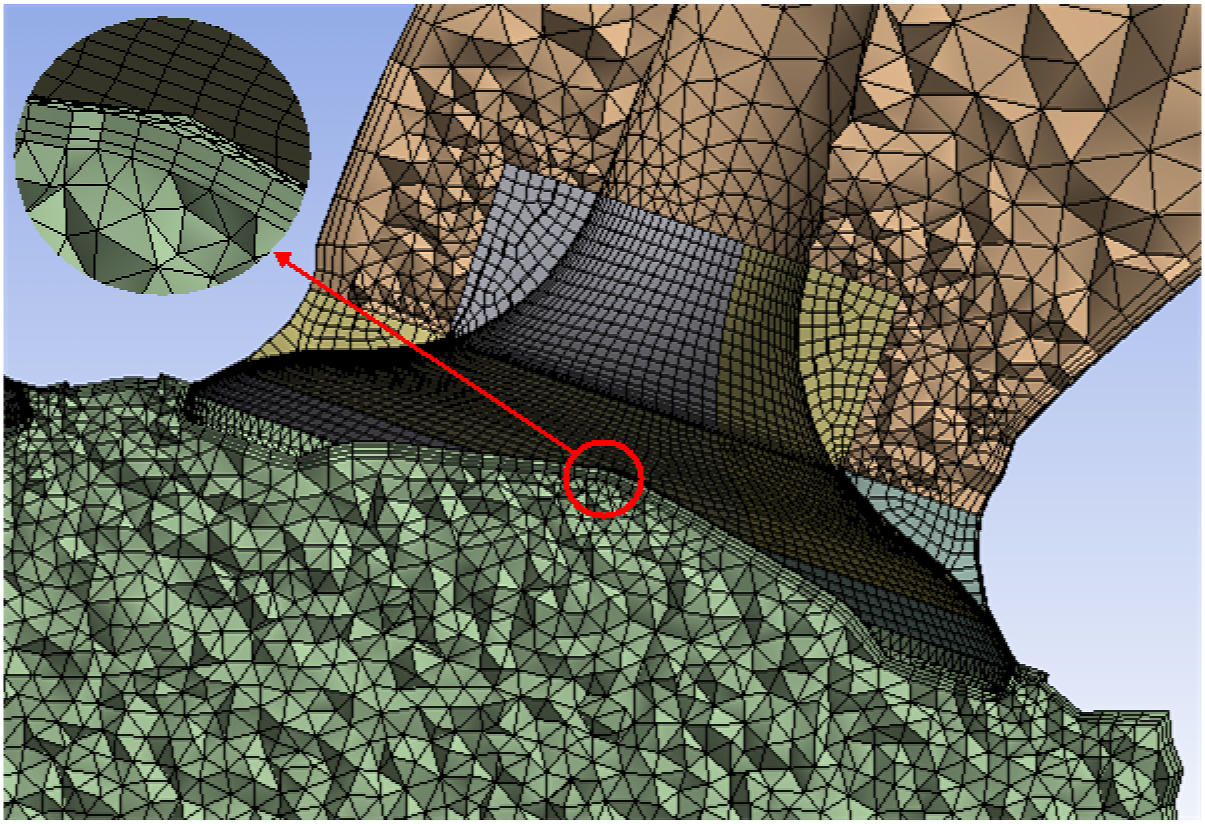
For example, at 0 degree crank angle even though both the valves are closed, piston has not moved more than piston limit (piston stroke cutoff). Hence inflation will not be created.

Figure 13.5: Mesh at 0 degree



For crank angle of about 40–140 degrees inflation will be created as both of the conditions are satisfied.

Figure 13.6: Mesh When Prism Layer is Created



The concept of KeyGrid is replacing the existing mesh in the simulation with better and improved quality mesh. With the help of design points you can generate meshes at different crank angles. For a simulation, all the meshes generated through design points update will be saved in `~\project_name\dp0\ICE\ICE\KeyGrids\` directory. If you want to modify/improve these meshes individually you can do so. After generating the meshes, start the solver in the main project file. Events are written in Fluent for importing the generated/modified meshes at the specified crank angles, so while running the simulation the modified meshes will be swapped into the simulation at the specified crank angles.

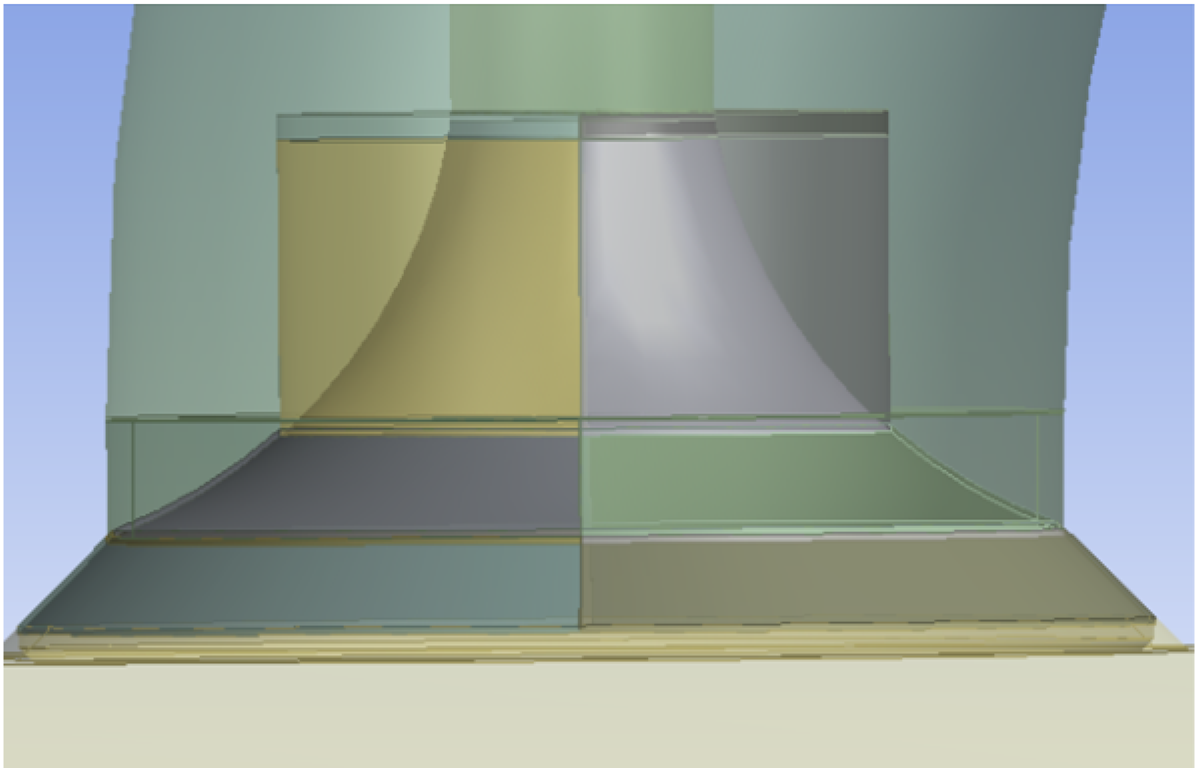
13.3. Supported Mesh Topologies

If the mesh topology is different from the IC Engine generated topology, ensure that the mesh meets the requirement of minimum zones, and that the zones and interfaces are properly named. Mesh topologies are categorized based on number of bodies and the type of mesh the bodies have (hex, layered, tet mesh). The types of mesh topologies supported in the IC Engine system are discussed below. If you follow the minimum required conventions you can also use engine mesh created outside Workbench.

13.3.1. ICE Topology

In this type of mesh, vlayer, ib and ports are present. This kind of mesh is the result of ICE default decomposition and meshing where, in the port region and near the dynamic zones maximum possible hex cells are generated. As a result, this topology has maximum number of hex cells as compared to the other mesh topologies. In the ib and vlayer, hex layering occurs when the valves move.

Figure 13.7: ICE Toplogy



For more information about the minimum zone requirements check Nomenclature of Decomposed Geometry (p. 167). Ensure that the named selection is similar to the IC Engine system convention.

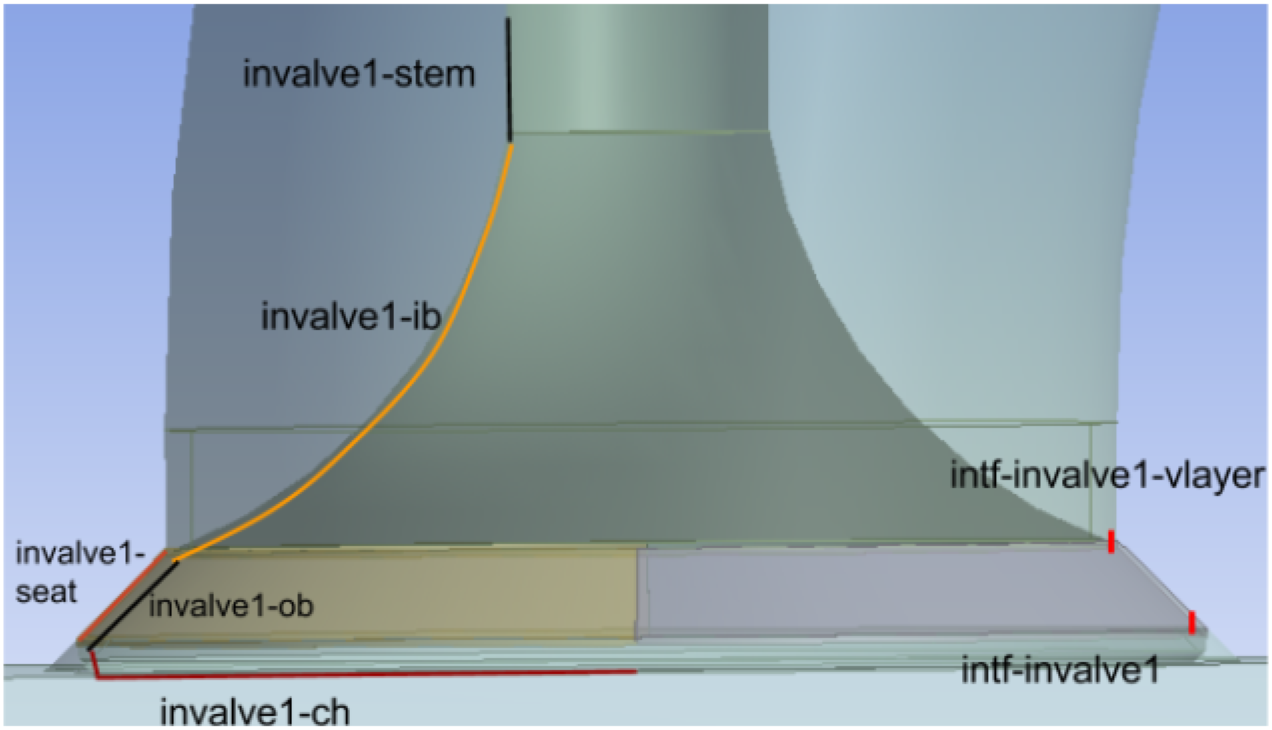
13.3.2. Vlayer and Port

In this type of mesh only the vlayer and port zones of the mesh are present. The fluid zone related to the ib region is not present. You can use this topology when you want to study only the region of velocity jet with hex mesh. In this type of mesh topology, the valyer fluid zone has layered mesh, and layering will happen in that region when the valve moves. The vlayer should have non-conformal interface between port and chamber.

Vlayer is only at seat region

In this case the vlayer is smaller. It is present only in the seat region.

Figure 13.8: Vlayer at Seat Region



Minimum zones required

The minimum interface boundary zones required for this type of mesh are:

- intf-valveID-vlayer
  - intf-valveID-vlayer-fluid-port

    This zone id defined as **deforming** under **Dynamic Zones**.
  - intf-valveID-vlayer-fluid-vlayer
- valveID
  - intf-valveID-ob-fluid-vlayer
  - intf-valveID-ob-fluid-ch

    This zone id defined as **deforming** under **Dynamic Zones**.

The minimum fluid zones required for this type of mesh are:

- fluid-ch
- fluid-valveID-vlayer
- fluid-valveID-port

Dynamic zones

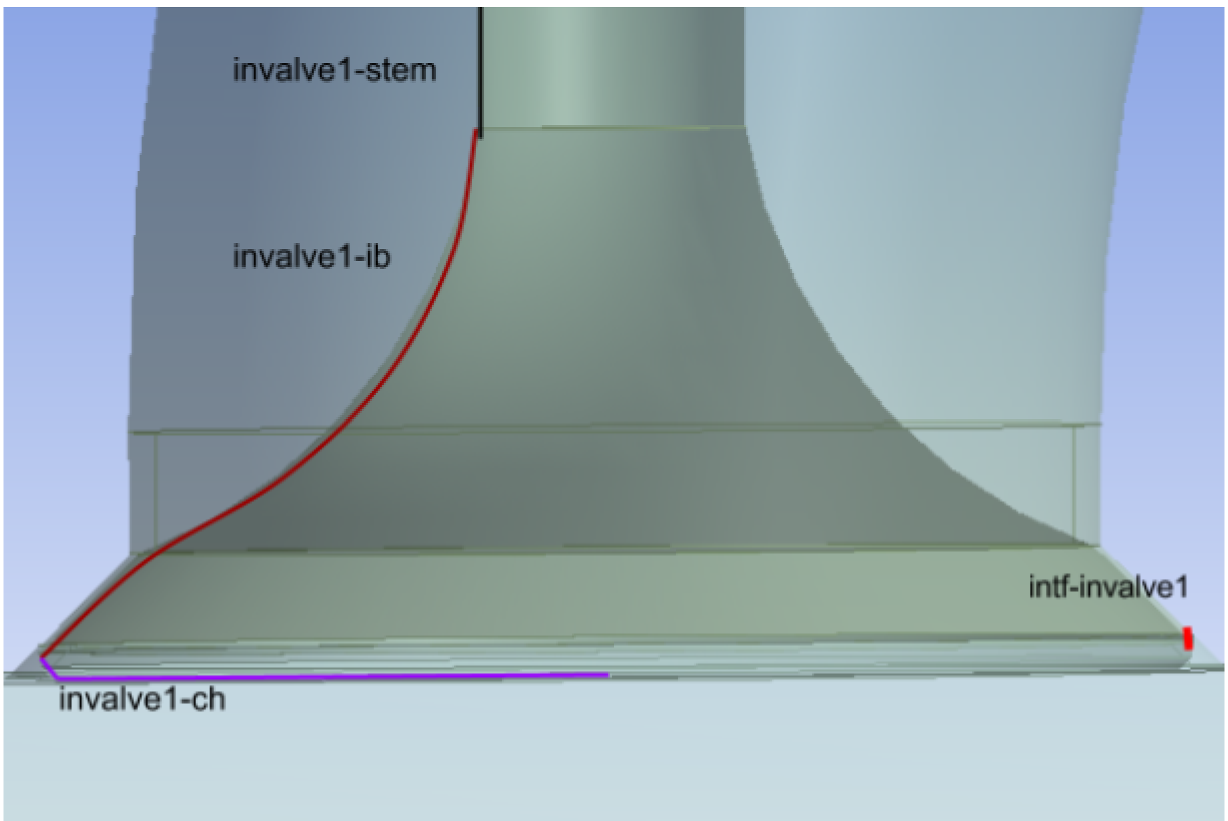
The following dynamic zones will be defined for this kind of mesh:

- valveID-ib: rigid body
- valveID-ob: rigid body
- valveID-ch: rigid body
- valveID-seat: stationary with layer height

13.3.3. Only Port is Present

In this mesh topology only the port is present. Vlayer and ib are absent. In this type of mesh topology there should be non-conformal meshes between port and chamber bodies. Thus there will be an interface between the port and the chamber. The zones are as seen in the Figure 13.9: Only Port with Interface Between Port and Chamber (p. 480).

Figure 13.9: Only Port with Interface Between Port and Chamber



Minimum zones required

The minimum interface boundary zones required for this type of mesh are:

- valveID
  - intf-valveID-ch-fluid-port

    This zone id defined as **deforming** under **Dynamic Zones**.
  - intf-valveID-port-fluid-ch

This zone id defined as **deforming** under **Dynamic Zones**.

The minimum fluid zones required for this type of mesh are:

- fluid-ch
- fluid-valveID-port

Dynamic zones

The following dynamic zones will be defined for this kind of mesh:

- valveID-stem: deforming
- valveID-ib: rigid body
- valveID-ch: rigid body

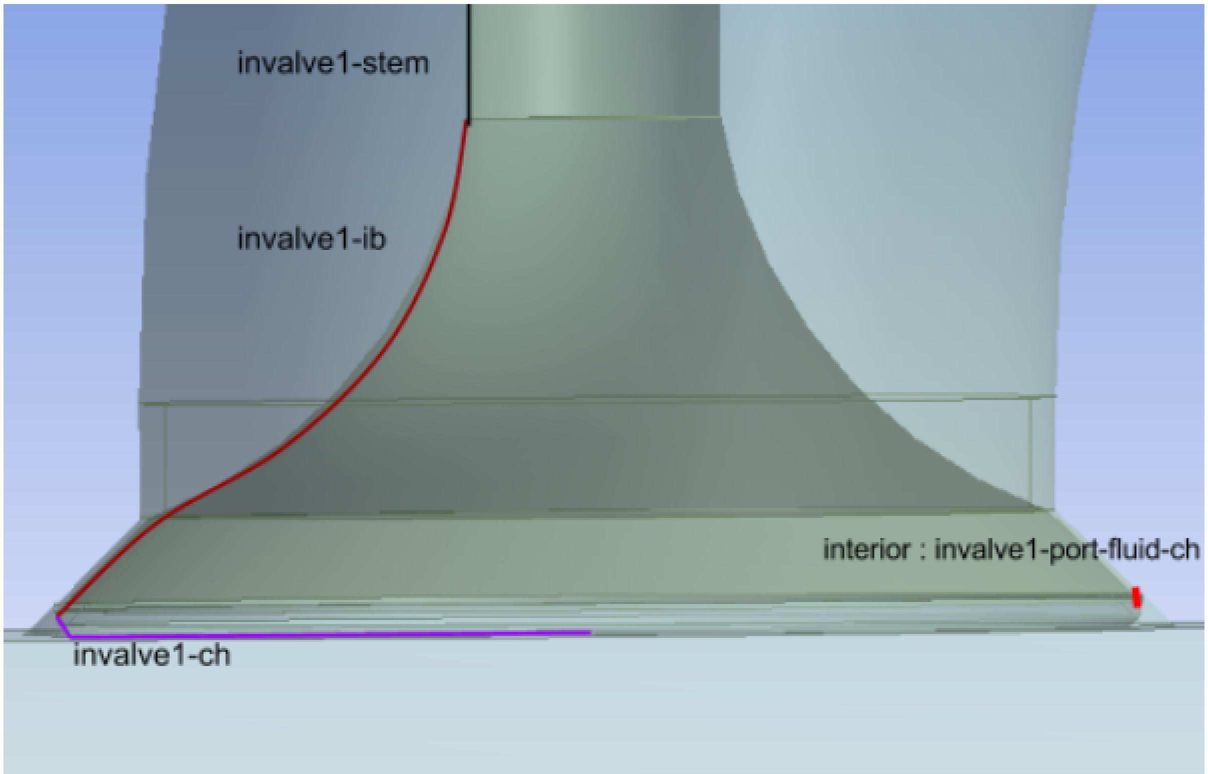
13.3.4. Single body

In this of mesh topology only a single body is present. This kind of topology is seen in the CFX meshes. There are no separate port, ib, or vlayer bodies. A single mesh topology can have two different cases.

Port and Chamber Have Different Bodies

In this case you can have different bodies for ports and chamber but with conformal meshes between them. There will be an interior zone between the port and the chamber. Since the ports are in different bodies they can be deactivated independently. Also zone wise patching can be easily done.

Figure 13.10: Port and Chamber Have Different Bodies



Minimum zones required

The minimum interface boundary zones required for this type of mesh are:

- None

The minimum fluid zones required for this type of mesh are:

- fluid-ch
- fluid-valveID-port

Dynamic zones

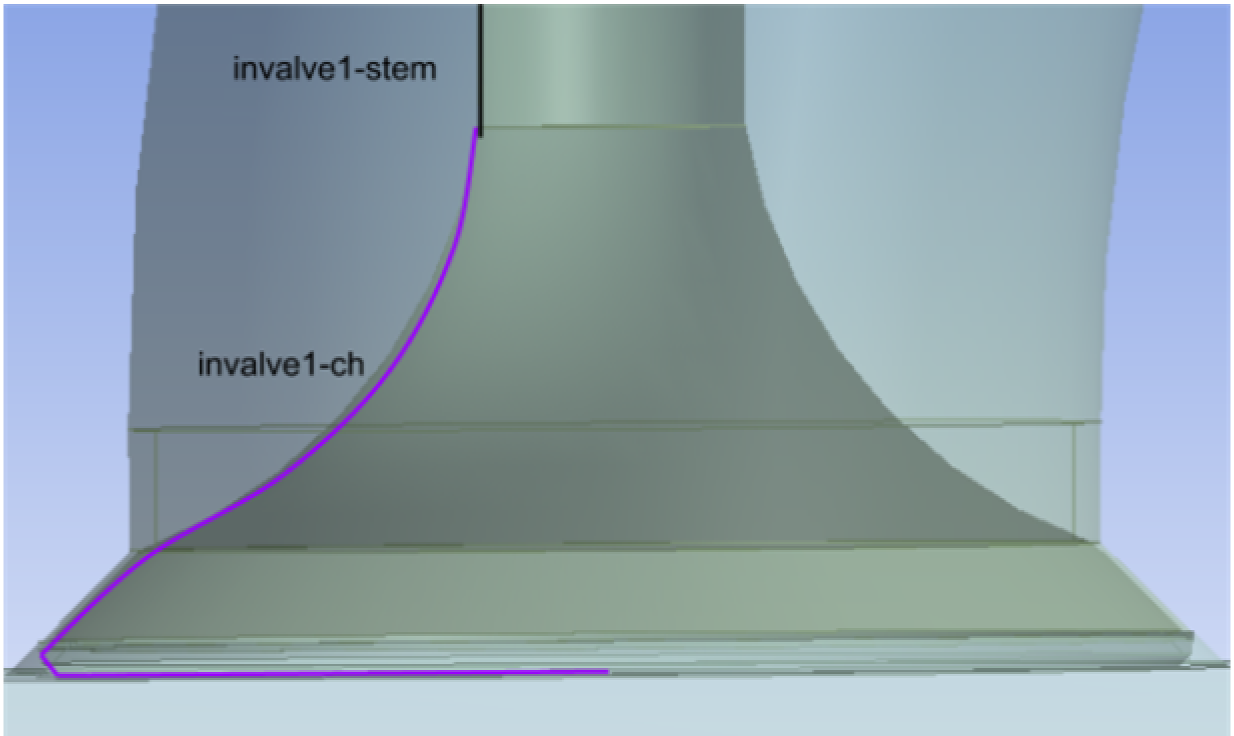
The following dynamic zones will be defined for this kind of mesh:

- valveID-stem: deforming
- valveID-ib: rigid body
- valveID-ch: rigid body

Port and Chamber are in a Single Body

In this case the ports and the chamber are in a single body.

Figure 13.11: Port and Chamber in a Single Body



Minimum zones required

The minimum interface boundary zones required for this type of mesh are:

- None

The minimum fluid zones required for this type of mesh are:

- fluid-ch

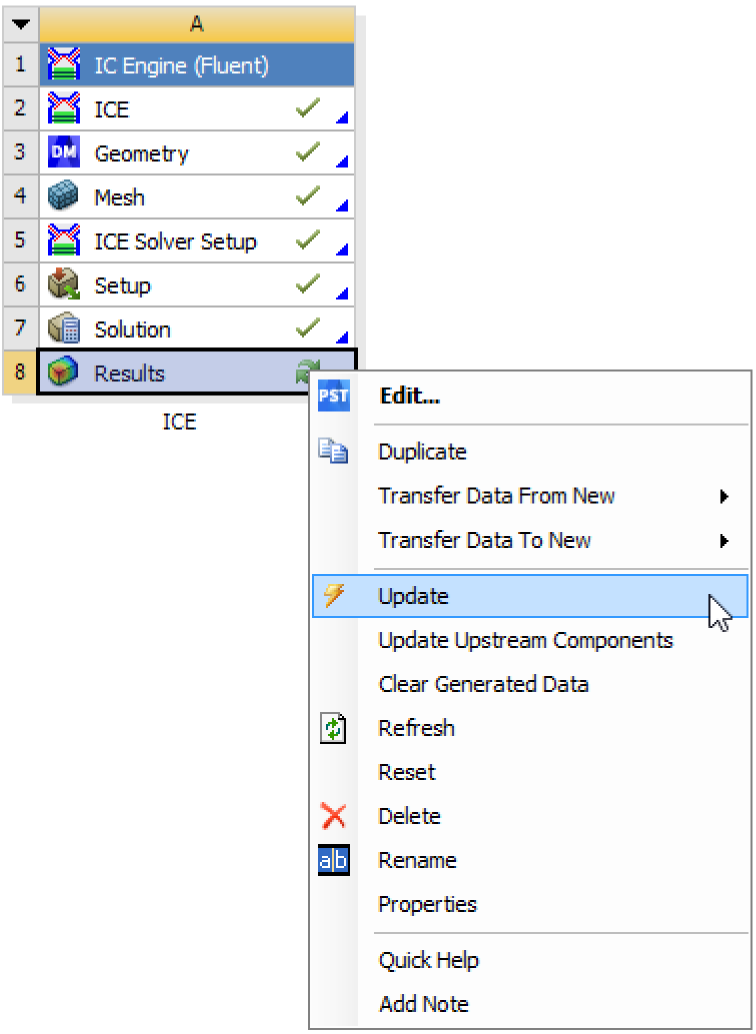
Dynamic zones

The following dynamic zones will be defined for this kind of mesh:

- valveID-stem: deforming
- valveID-ch: rigid body

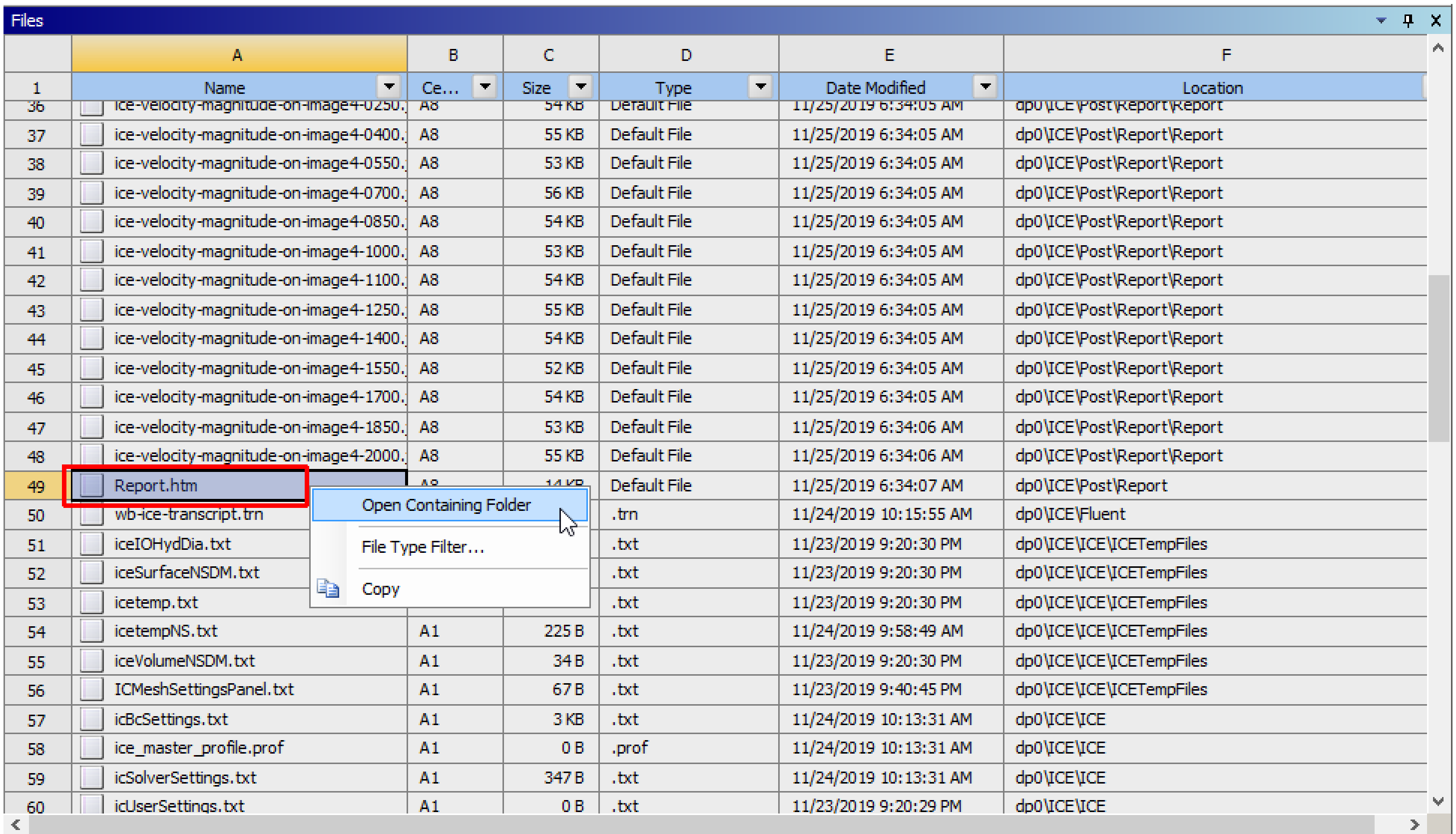
Chapter 14: Working with the Simulation Results

After Ansys Fluent finishes running the calculation for the entered number of steps, you can close it. In Ansys Workbench the **Solution** cell changes its state to up-to-date. Right-click on the **Results** cell and click on **Update** from the context menu.



After the **Results** cell is updated, you can check the result in the report. An HTML file is generated, having the details of setup and results of the simulation. You can open the report file by clicking on **View** from the menu bar.

View → **Files**



In the **Files** pane, right-click on **Report.html** and click on **Open Containing Folder** from the context menu. In the **Report** folder double click on **Report.html** to open the report. The details about the post-processing are described in the following sections.

For Forte you can find the report at:

```
..\dp0\ICE\Post\Nominal
```

for first design point/Default solution. For parameters you can find the report at:

```
..\dp0\ICE\Post\Run_001
```

Where Run_001 is for the DP1

14.1. Report

This is a sample report.

ANSYS

Title

IC Engine Cold Flow Simulation Report

Contents

1. File Report

Table 1. File Information for ICE

Case	ICE
File Path	E:\ICETutorials16\New folder\demo_tut_files\dp0\ICE\Fluent\ICE-4-02961.dat.gz
File Date	
File Time	11:38:18 AM
File Type	FLUENT
File Version	

2. Mesh Report

Table 2. Mesh Information for Cold_flow_demo_tut

Domain	Nodes	Elements
fluid ch	44314	219131
fluid exvalve 1 port	35099	122576
fluid exvalve1 ib	6612	4988
fluid exvalve1 vlayer	72000	64600
fluid invalve 1 port	64615	227128
fluid invalve1 ib	4482	3402
fluid invalve1 vlayer	25000	19200
fluid layer cylinder	21228	38052
fluid piston	3816	17008
All Domains	277166	716085

The **File Report** displays information about the Fluent data file location, date and time of file creation, and version.

The **Mesh Report** shows mesh count and node count of the cell zones.

Chart 1. Monitor: Max Cell Equivolume Skew (fluid-piston fluid-layer-cylinder fluid-ch)

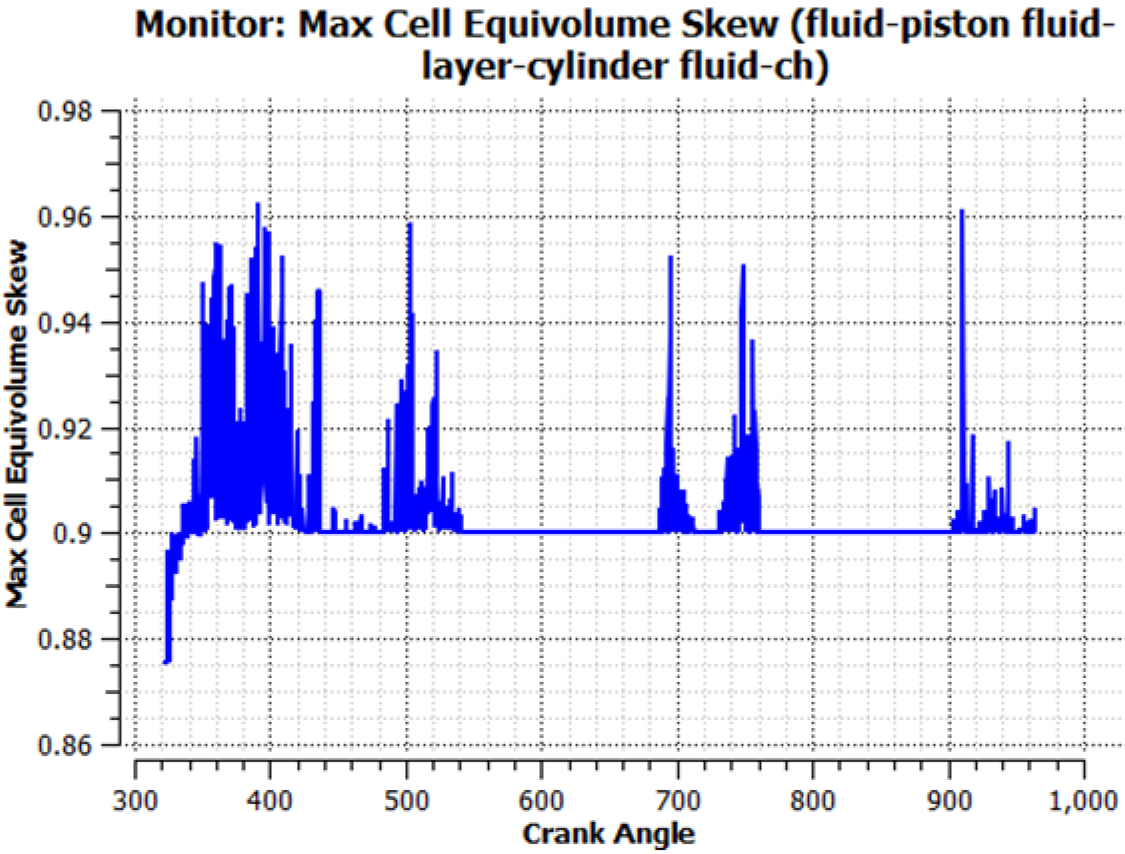


Table 3. Cell count at crank angles

Crank Angle	Cell Count
0.000e+00	5.809e+05
1.800e+02	6.592e+05

Monitor: Volume-Average Cell Equivolume Skew (fluid-piston fluid-layer-cylinder fluid-ch) is a monitor chart, which shows the volume-average cell skewness at different crank angles of fluid zones.

The table **Cell count at crank angles**, shows the mesh count at 0 and 180 degree crank angles.

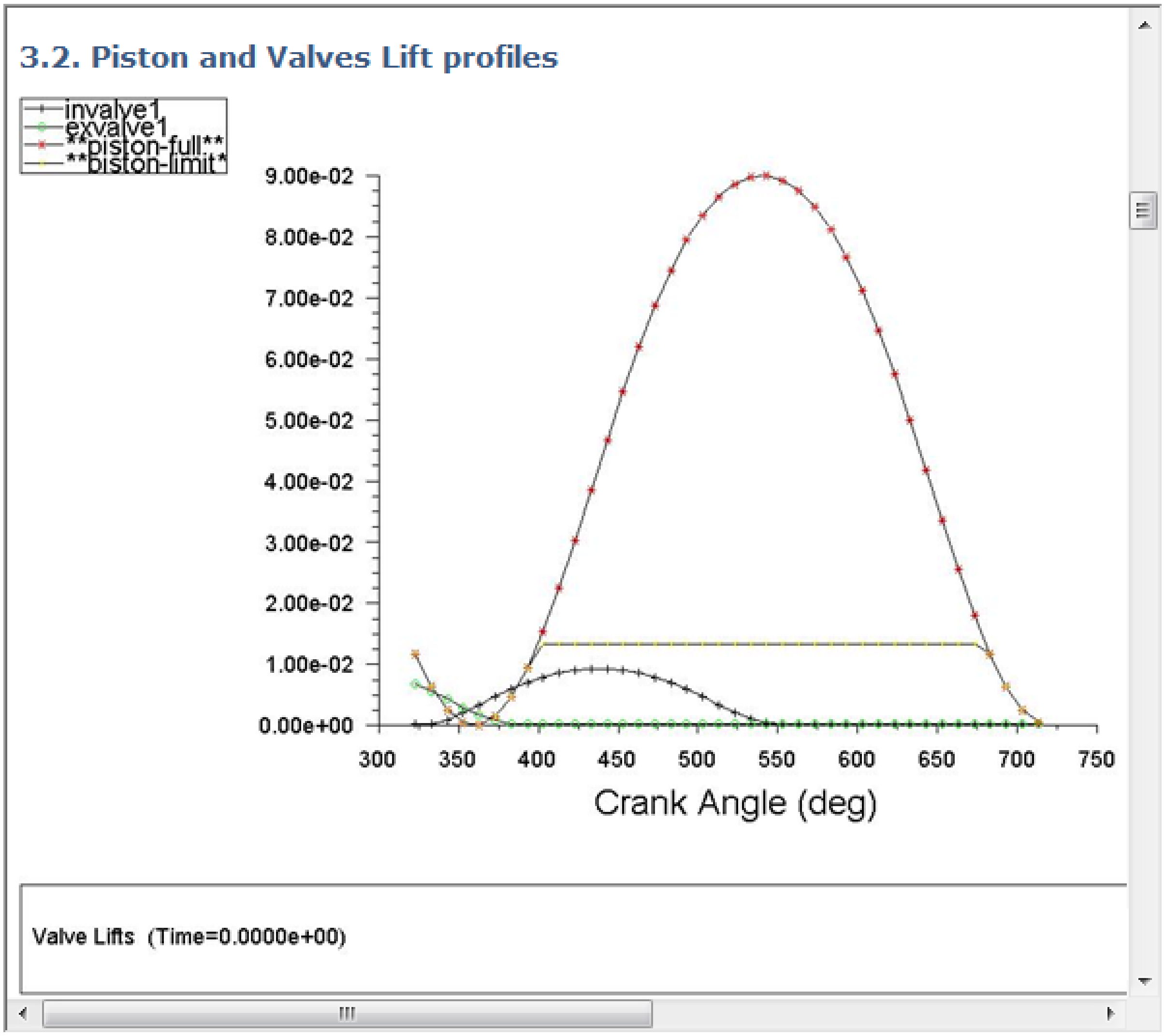
3. Setup

3.1. Physics

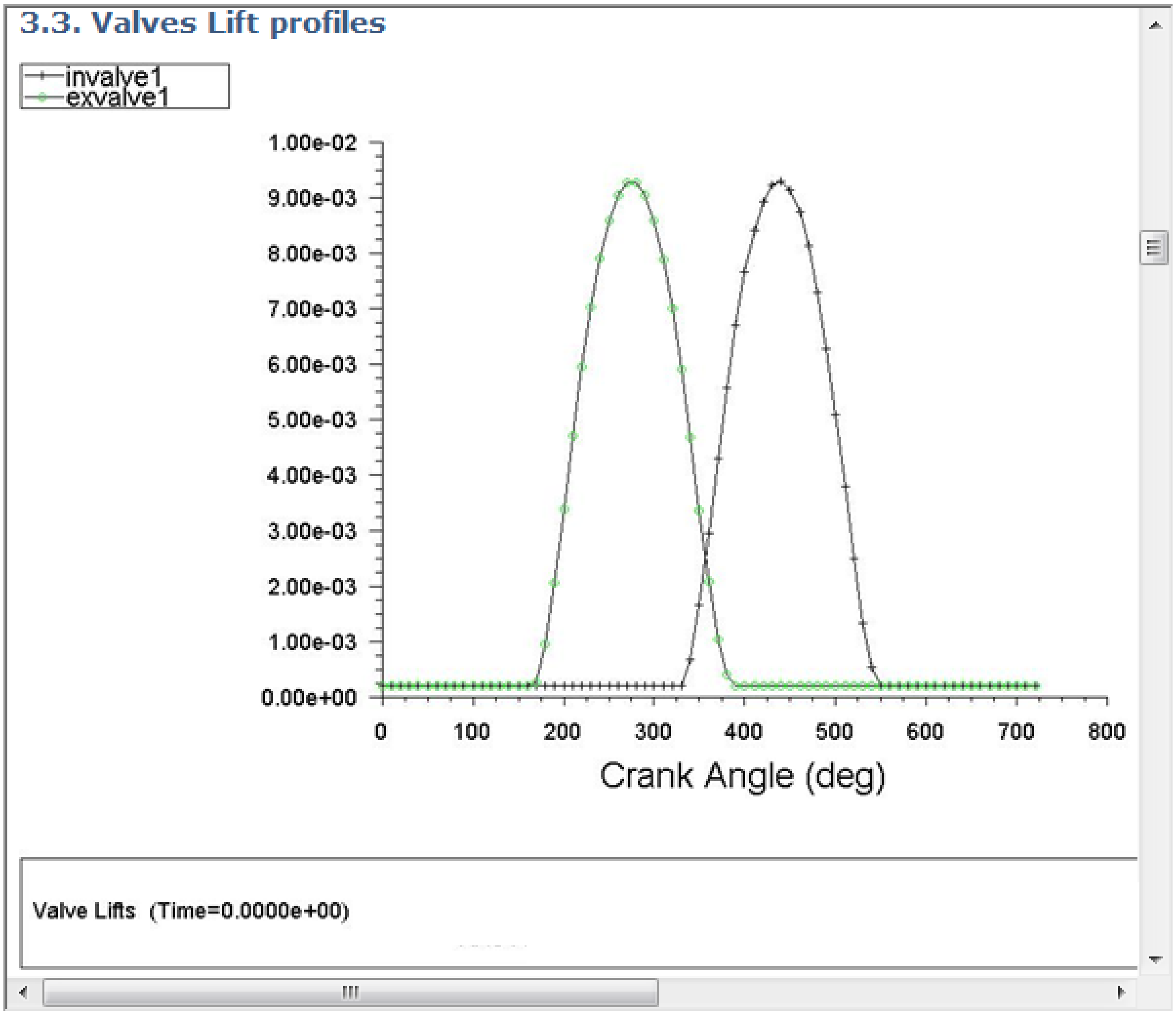
Table 4. Boundary Conditions

Type	Zones	Values
wall (invalve1)	invalve1-stem, invalve1-ob, invalve1-ch, invalve1-ib	Temperature (k) 300
wall (exvalve1)	exvalve1-stem, exvalve1-ob, exvalve1-ch, exvalve1-ib	Temperature (k) 300
wall (invalve-port)	invalve-1-port	Temperature (k) 300
wall (exvalve-port)	exvalve-1-port	Temperature (k) 300
pressure-inlet	ice-inlet-invalve-1-port	Gauge Total Pressure (pascal) -21325
		Supersonic/Initial Gauge Pressure (pascal) 0
		Total Temperature (k) 313
pressure-outlet	ice-outlet-exvalve-1-port	Gauge Pressure (pascal) -1325
		Backflow Total Temperature (k) 333
wall	invalve1-seat	Temperature (k) 300
wall	exvalve1-seat	Temperature (k) 300
wall	cyl-head	Temperature (k) 348
wall	cyl-piston	Temperature (k) 318
wall	cyl-quad	Temperature (k) 318
wall	cyl-tri	Temperature (k) 318
wall	piston	Temperature (k) 318

The table **Boundary Conditions**, shows the boundary conditions of all inlets, outlets, and walls.



The section **Piston and Valves Lift profiles** displays the profiles of valve lifts and piston.



The section **Lift profiles** displays the profiles of valve lifts.

3.4. Relaxations

Table 5. Relaxation changes through events

Crank Angle	Pressure	Density	Body Forces	Momentum	Turbulent Kinetic Energy	Turbulent Dissipation Rate	Turbulent Viscosity	Energy
0.000	0.300	1.000	1.000	0.500	0.400	0.400	1.000	1.000
166.400	0.500	,,	,,	0.500	0.800	0.800	,,	1.000
168.400	0.300	,,	,,	0.500	0.400	0.400	,,	1.000
329.600	0.500	,,	,,	0.500	0.800	0.800	,,	1.000
331.600	0.300	,,	,,	0.500	0.400	0.400	,,	1.000

The table **Relaxations** shows the under-relaxation factors at different crank angles.

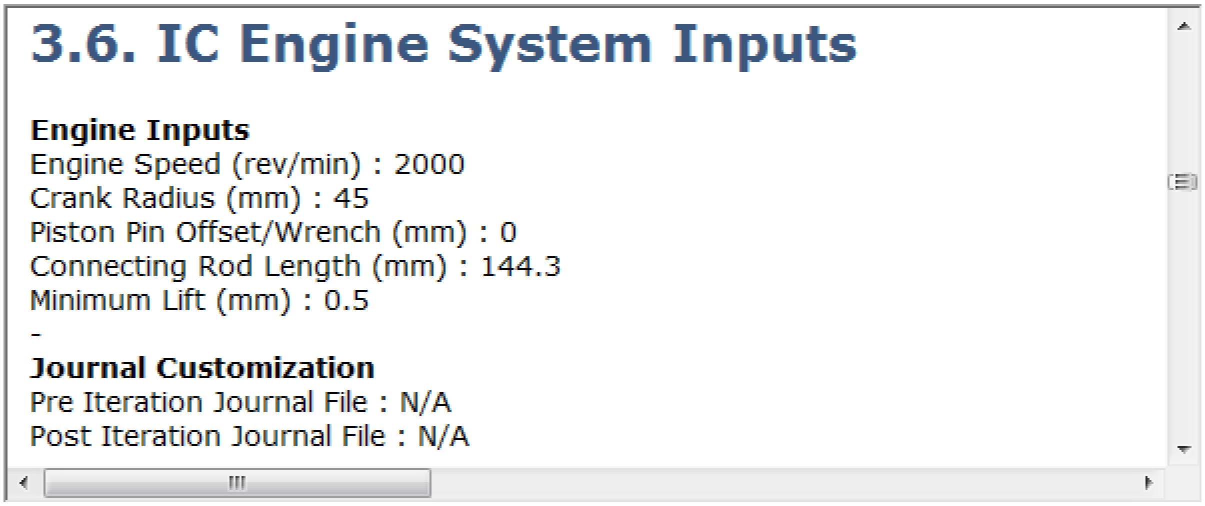
3.5. Dynamic Mesh Setup

Table 6. Dynamic Mesh Events

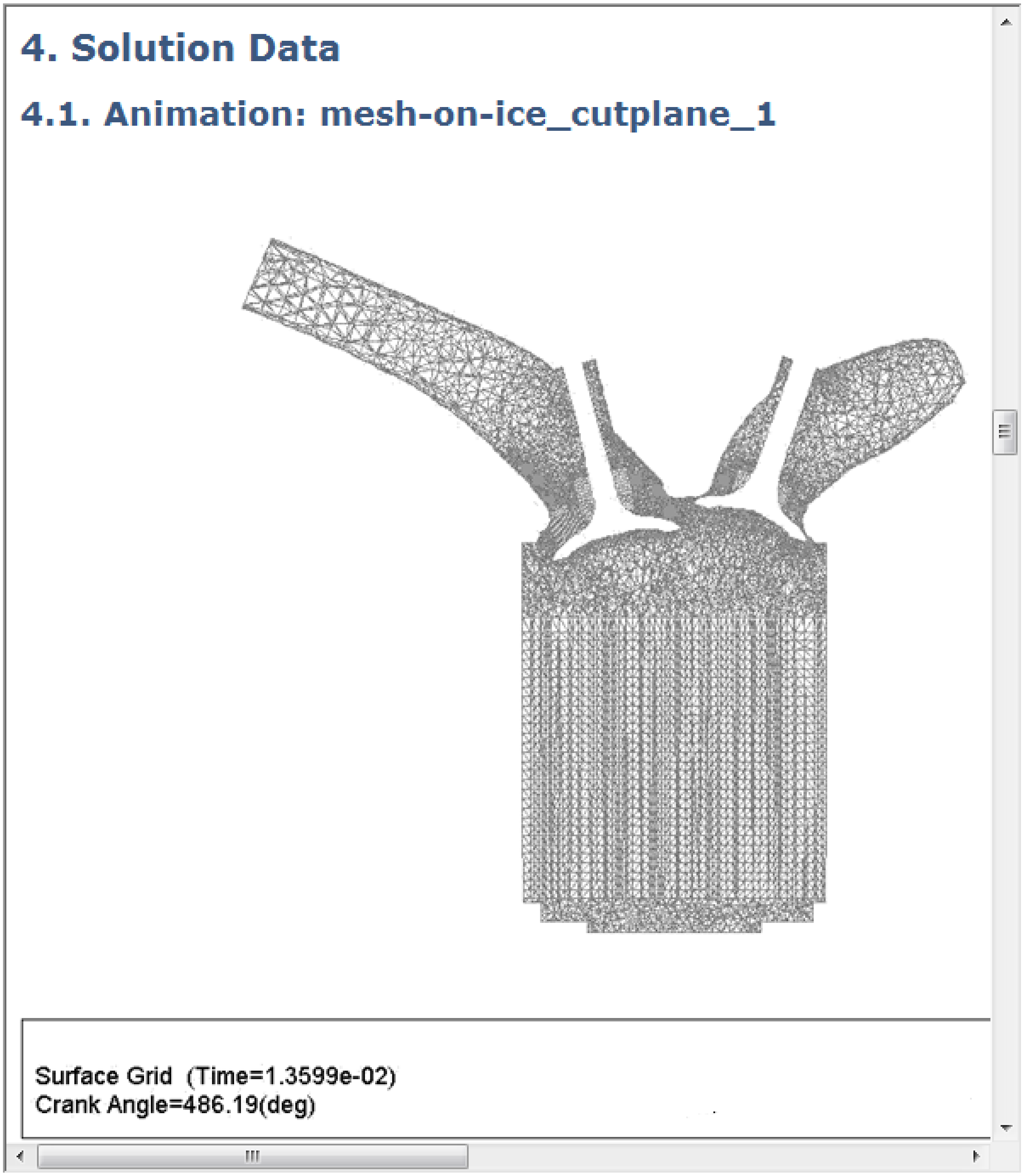
At Crank Angle (deg)	Name	Description
0.000	dt-event-at-0(0.25)	Changing the Time step size in terms of crank angle.
160.200	write-solution-point-at-ca-160.200	Saves solution files at this point.
166.400	reduce-urf-due-to-open-exvalve1, change-positivity-at-valve-open, open-exvalve1, start-smoothing-at-exvalve1-open, dt-event-at-166.4(0.125)	epsilon=0.8, k=0.8, mom=0.5, pressure=0.5, temperature=1. Reducing URFs 1deg before valve opening for solution stability. Changing the Time step size in terms of crank angle.
168.400	increase-urf-due-to-open-exvalve1, change-positivity-after-valve-open	epsilon=0.4, k=0.4, mom=0.5, pressure=0.3, temperature=1. Increasing URFs for accelerating the solution.
169.400	change-positivity-after-valve-open	
171.400	dt-event-at-171.4(0.25)	Changing the Time step size in terms of crank angle.
180.000	save-residual-plot-180	Saves the residual plot image from last saved iteration to the current iteration.
185.300	stop-smoothing-after-exvalve1-open	Stops smoothing in vlayer and starts layering.
323.300	write-solution-point-at-ca-323.300	Saves solution files at this point.
329.600	reduce-urf-due-to-open-invalve1, change-positivity-at-valve-open, open-invalve1, start-smoothing-at-invalve1-open, dt-event-at-329.6(0.125)	epsilon=0.8, k=0.8, mom=0.5, pressure=0.5, temperature=1. Reducing URFs 1deg before valve opening for solution stability. Changing the Time step size in terms of crank angle.
331.600	increase-urf-due-to-open-invalve1, change-positivity-after-valve-open	epsilon=0.4, k=0.4, mom=0.5, pressure=0.3, temperature=1. Increasing URFs for accelerating the solution.
332.600	change-positivity-after-valve-open	

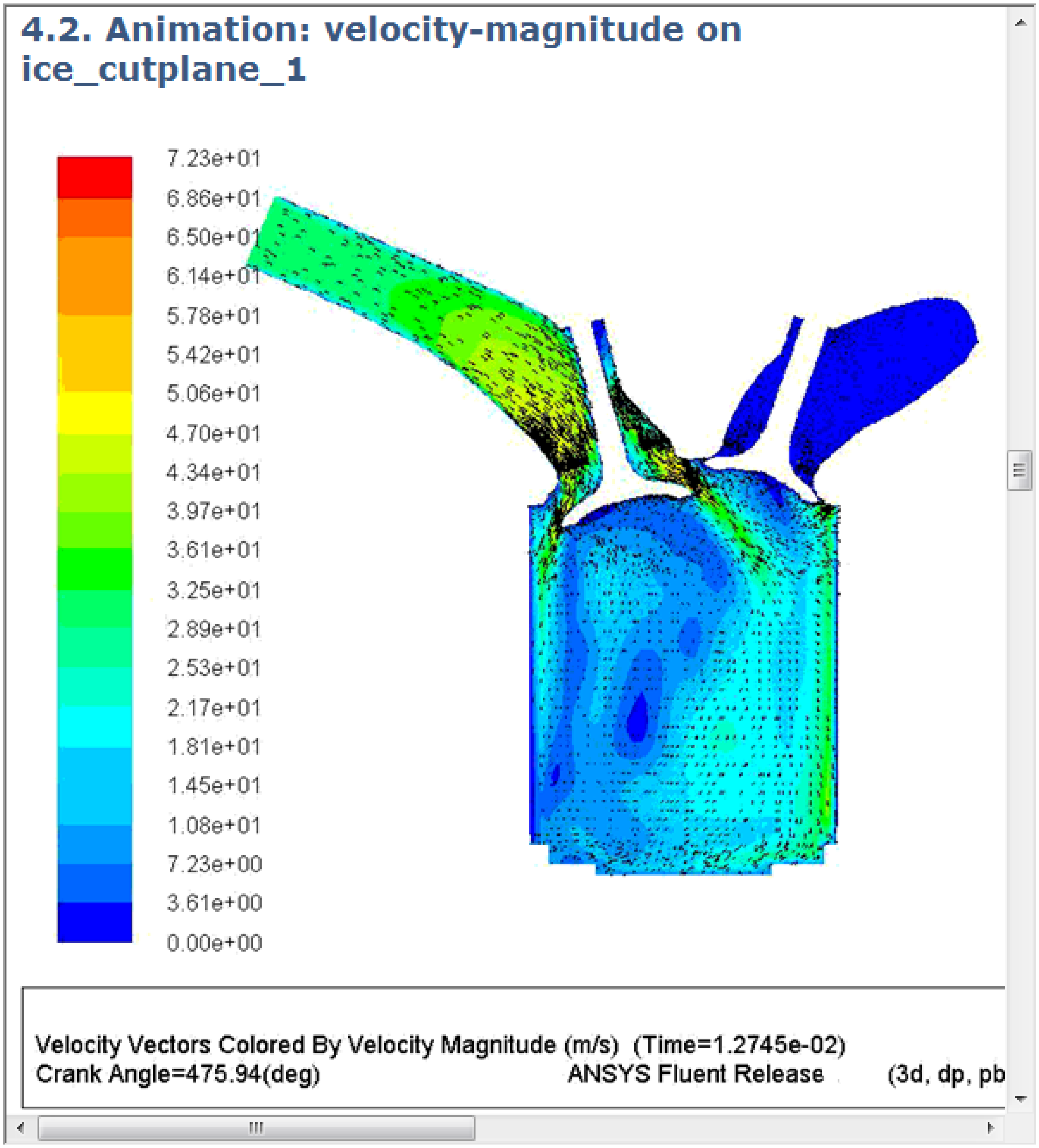
334.600	dt-event-at-334.6(0.25)	Changing the Time step size in terms of crank angle.
348.400	stop-smoothing-after-invalve1-open	Stops smoothing in vlayer and starts layering.
360.000	save-residual-plot-360	Saves the residual plot image from last saved iteration to the current iteration.
365.400	start-smoothing-before-exvalve1-close	Stops layering in vlayer and starts smoothing.
385.400	change-positivity-at-valve-close, dt-event-at-385.4(0.125)	Changing the Time step size in terms of crank angle.
390.400	close-exvalve1, dt-event-at-390.4(0.25)	Deleting interface between vlayer and chamber for stopping flow. Changing the Time step size in terms of crank angle.
391.400	change-positivity-at-valve-close	
392.400	change-positivity-at-valve-close	
402.300	write-solution-point-at-ca-402.300	Saves solution files at this point.
528.700	start-smoothing-before-invalve1-close	Stops layering in vlayer and starts smoothing.
540.000	save-residual-plot-540	Saves the residual plot image from last saved iteration to the current iteration.
547.200	change-positivity-at-valve-close, dt-event-at-547.2000000000001 (0.125)	Changing the Time step size in terms of crank angle.
552.200	close-invalve1, dt-event-at-552.2000000000001(0.25)	Deleting interface between vlayer and chamber for stopping flow. Changing the Time step size in terms of crank angle.
553.200	change-positivity-at-valve-close	
554.200	change-positivity-at-valve-close	
563.000	write-solution-point-at-ca-563.000	Saves solution files at this point.
720.000	save-residual-plot-720, dt-event-at-720(0.25), write-solution-point-at-ca-720.000	Saves the residual plot image from last saved iteration to the current iteration. Changing the Time step size in terms of crank angle. Saves solution files at this point.

The table **Dynamic Mesh Setup** list the events taking place at specific crank angles.

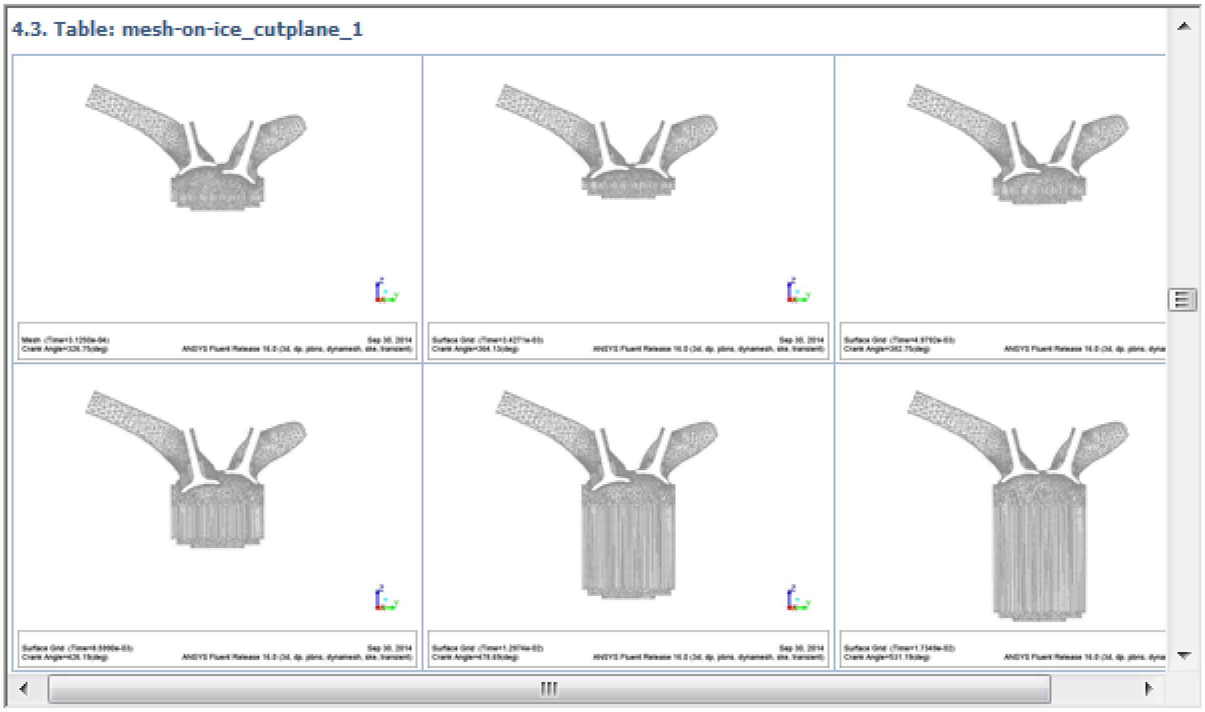


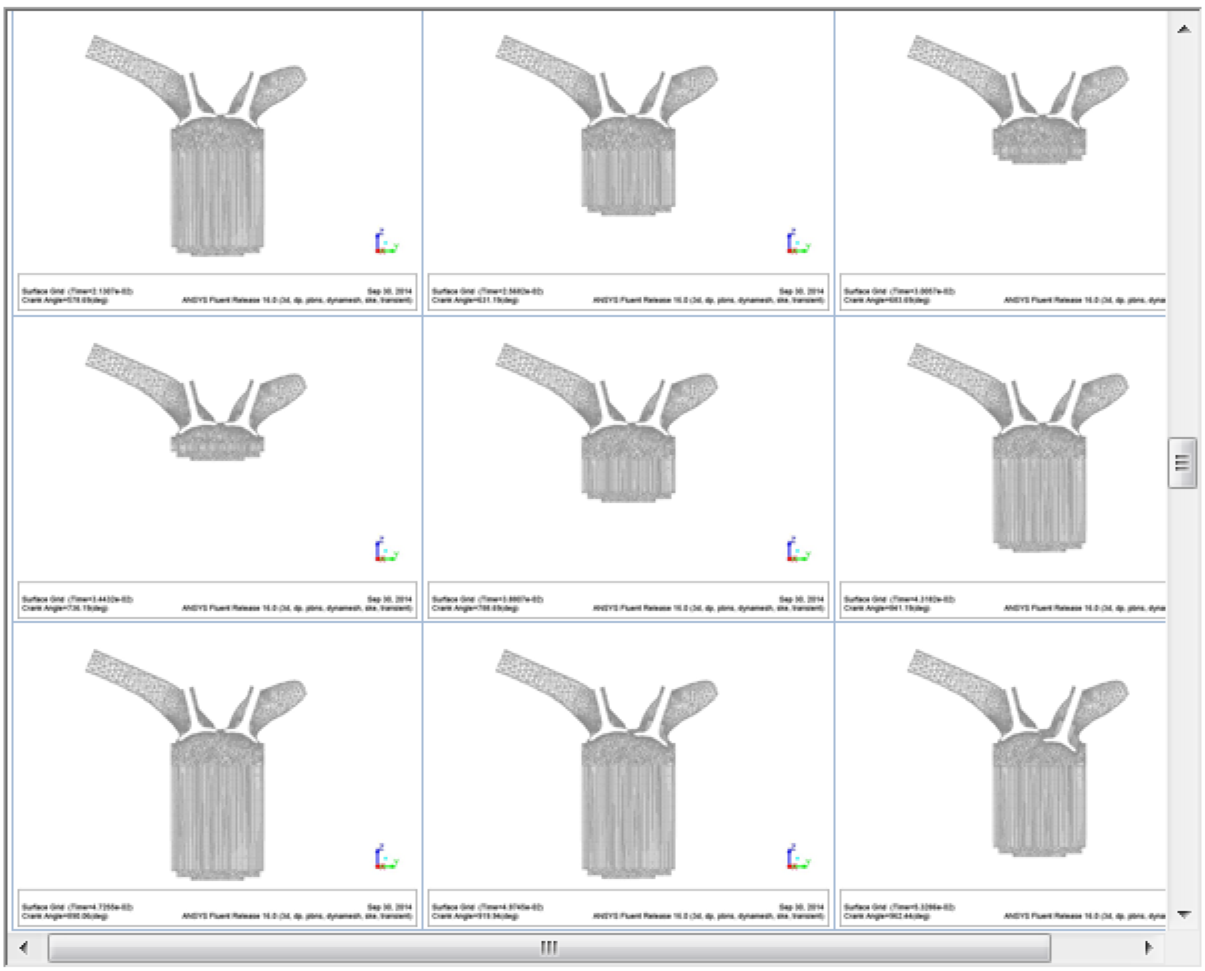
In the section **IC Engine System Inputs** you can check the engine inputs you have entered in the Properties dialog box. It also lists the **Journal Customization** files if they are provided.


4. Solution Data
4.1. Animation: mesh-on-ice_cutplane_1
Surface Grid (Time=1.3599e-02)
Crank Angle=486.19(deg)

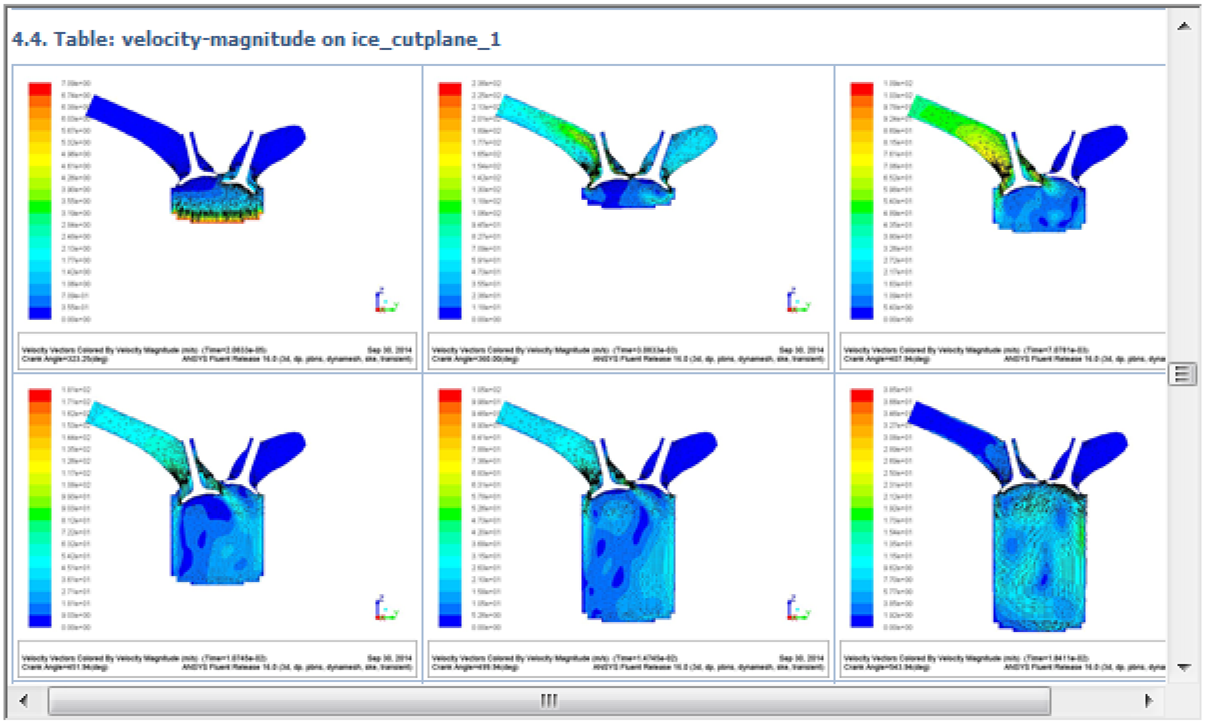


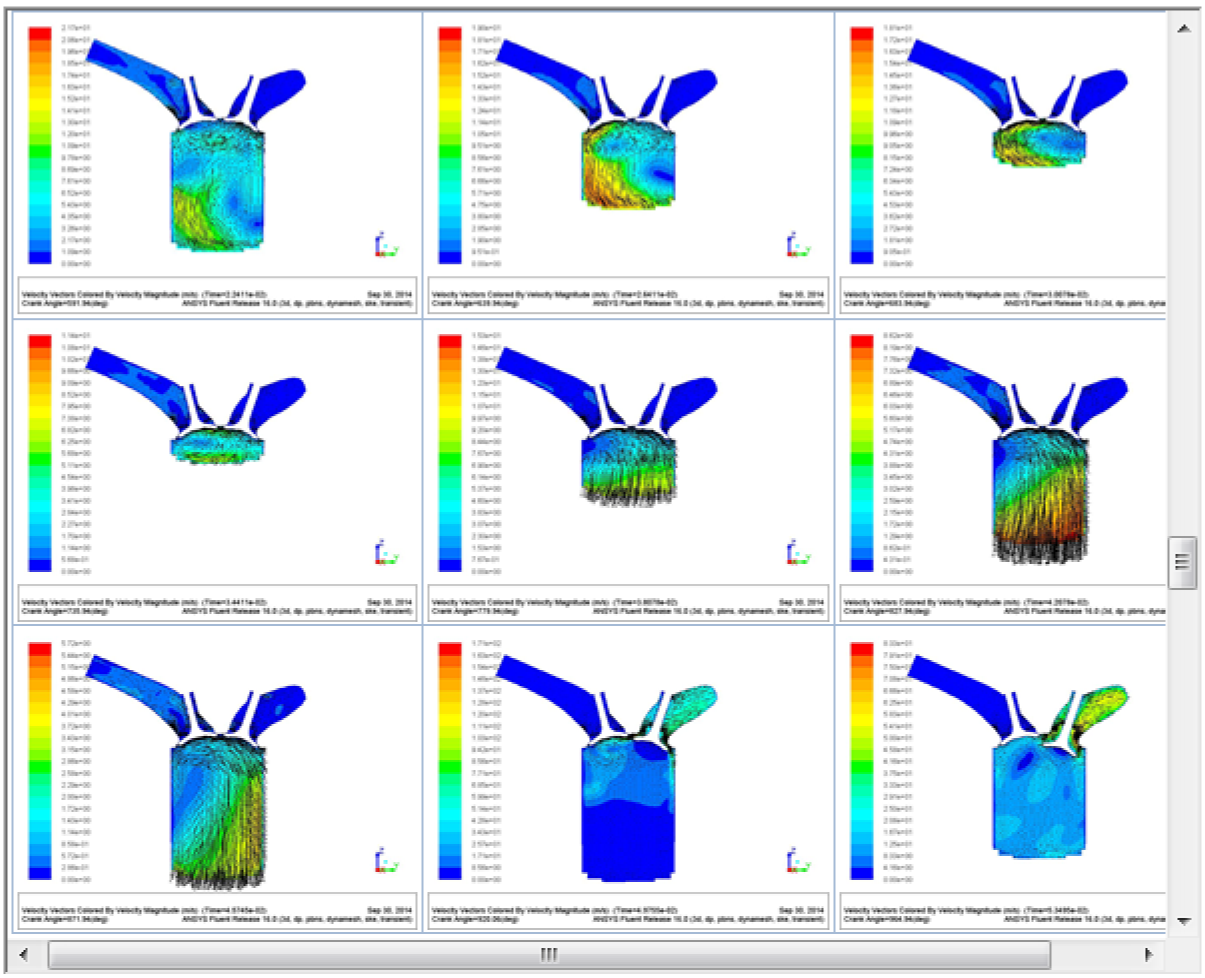
In the section **Solution Data**, you can see the animation of mesh and velocity contours under the **Animation** subsection.



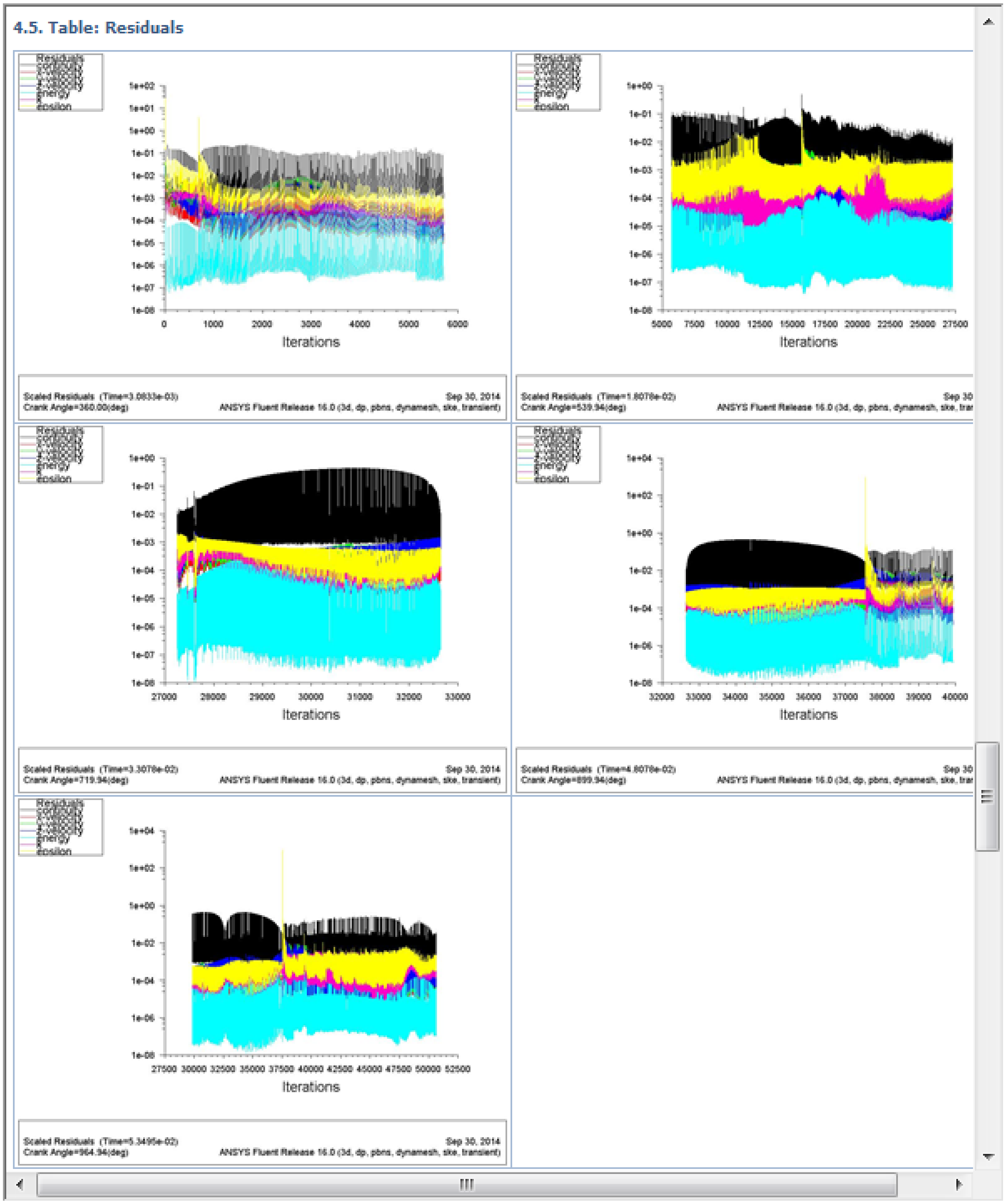


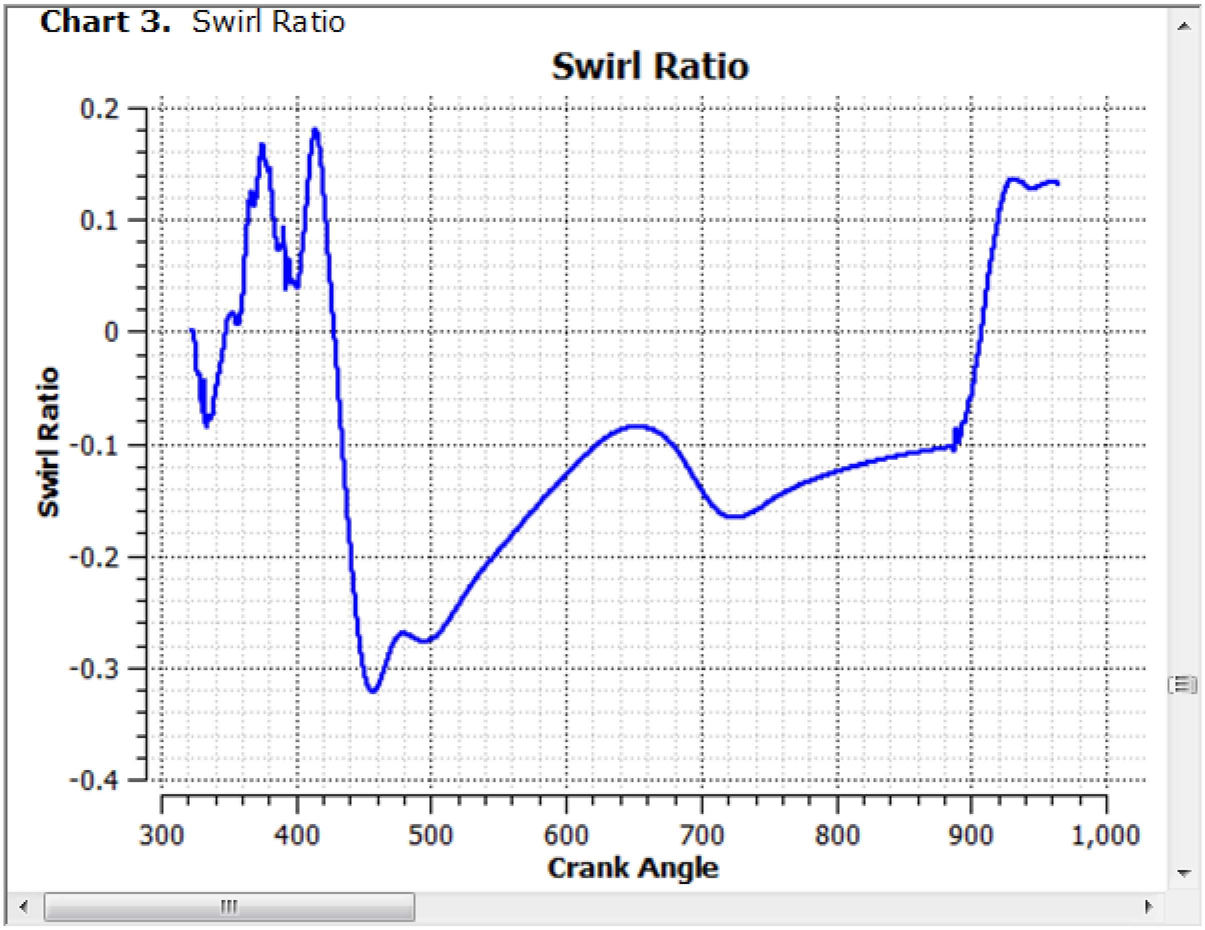
You can also check the images of the mesh at the cut-plane in the table **mesh-on-ice_cutplane_1**.





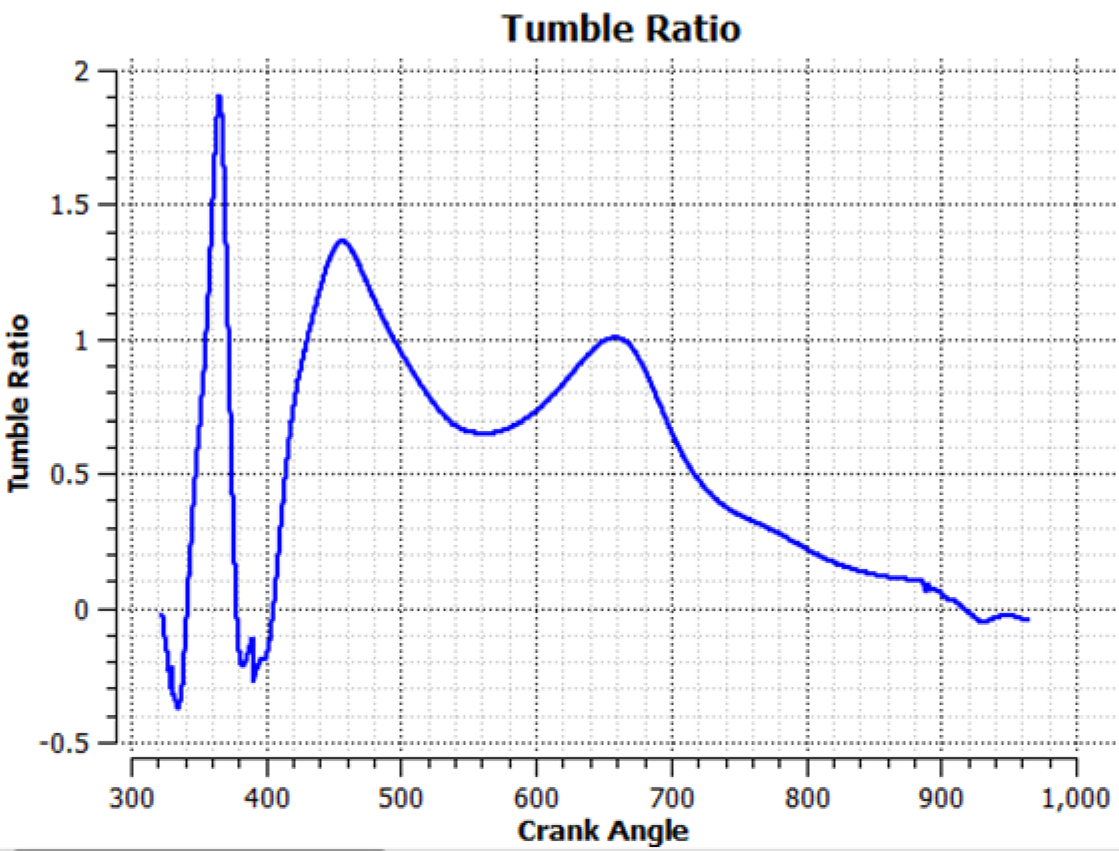
The table **velocity-magnitude on "ice_cutplane_1"** displays the saved images of the velocity contours on the cut-plane.


4.5. Table: Residuals
Residuals
Iterations
Scaled Residuals (Time=3.0833e-03)
Crank Angle=360.00(deg)
Sep 30, 2014
ANSYS Fluent Release 16.0 (3d, dp, pbns, dynamesh, ske, transient)
Scaled Residuals (Time=1.8078e-02)
Crank Angle=539.94(deg)
Scaled Residuals (Time=3.3078e-02)
Crank Angle=719.94(deg)
Scaled Residuals (Time=4.8078e-02)
Crank Angle=899.94(deg)
Scaled Residuals (Time=5.3495e-02)
Crank Angle=964.94(deg)



Under subsection **Charts**, plots of residuals, swirl data, tumble, and charts of other defined monitors are displayed.

Chart 4. Tumble Ratio



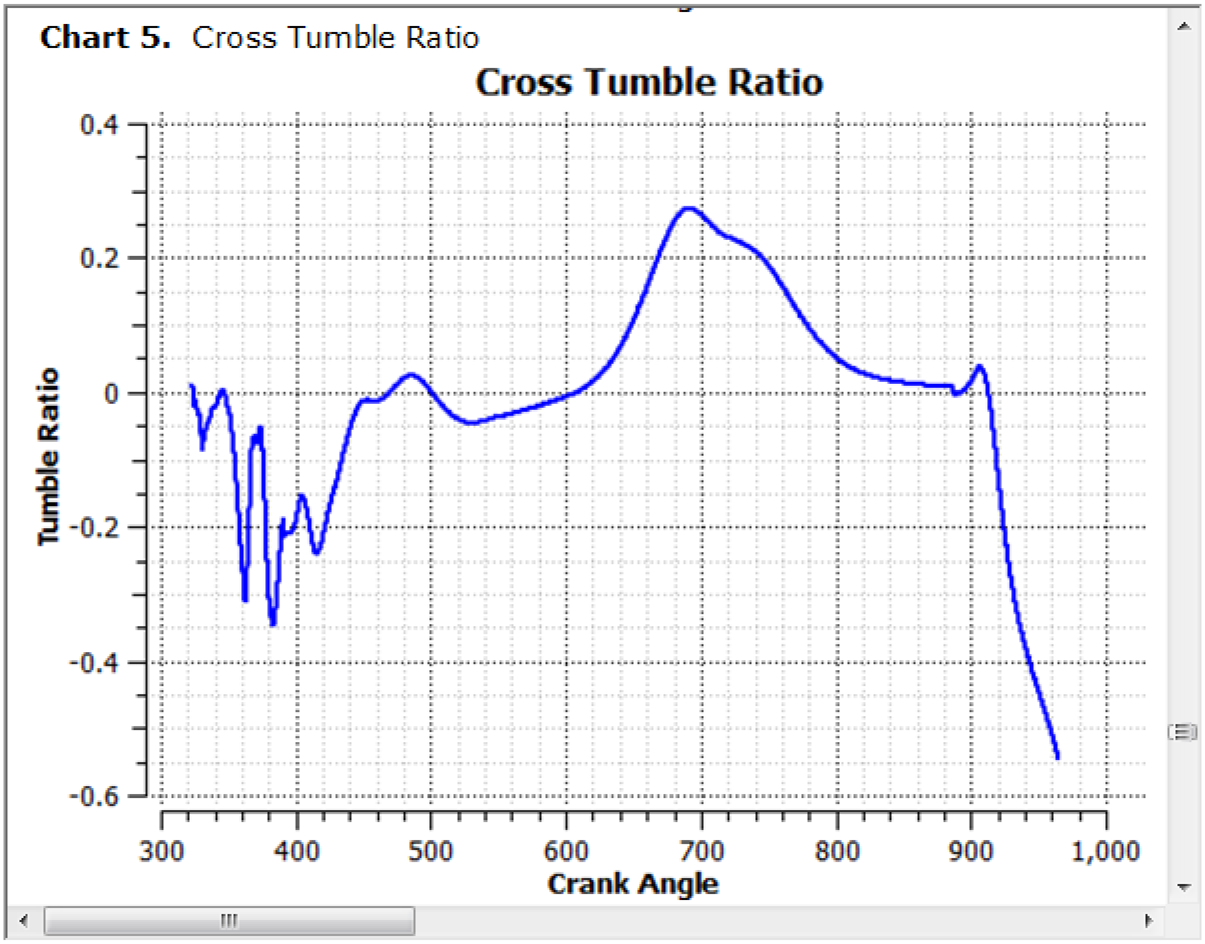


Chart 5. Cross Tumble Ratio
Cross Tumble Ratio
0.4
0.2
0
-0.2
-0.4
-0.6
Tumble Ratio
300
400
500
600
700
800
900
1,000
Crank Angle

Chart 9. Monitor: Mass-Average Static Temperature (fluid-piston fluid-layer-cylinder fluid-ch)

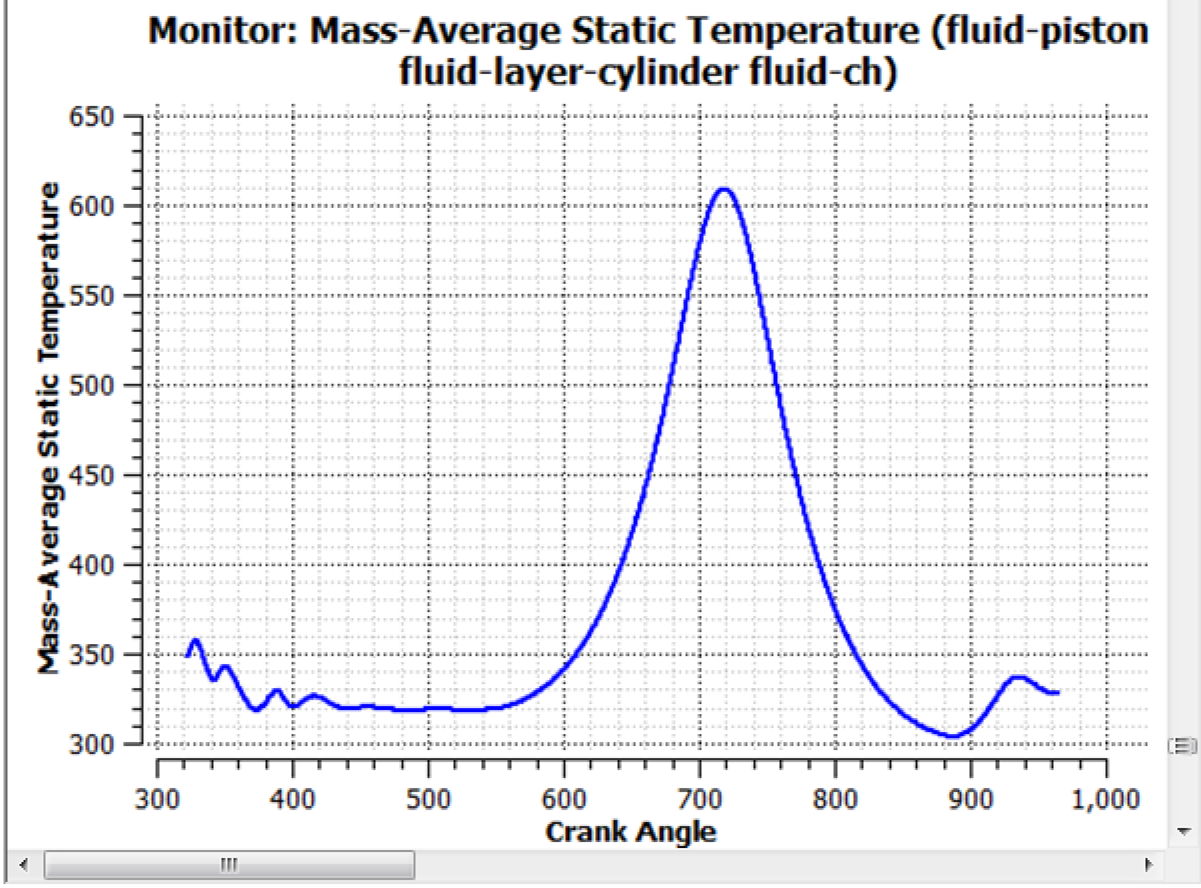
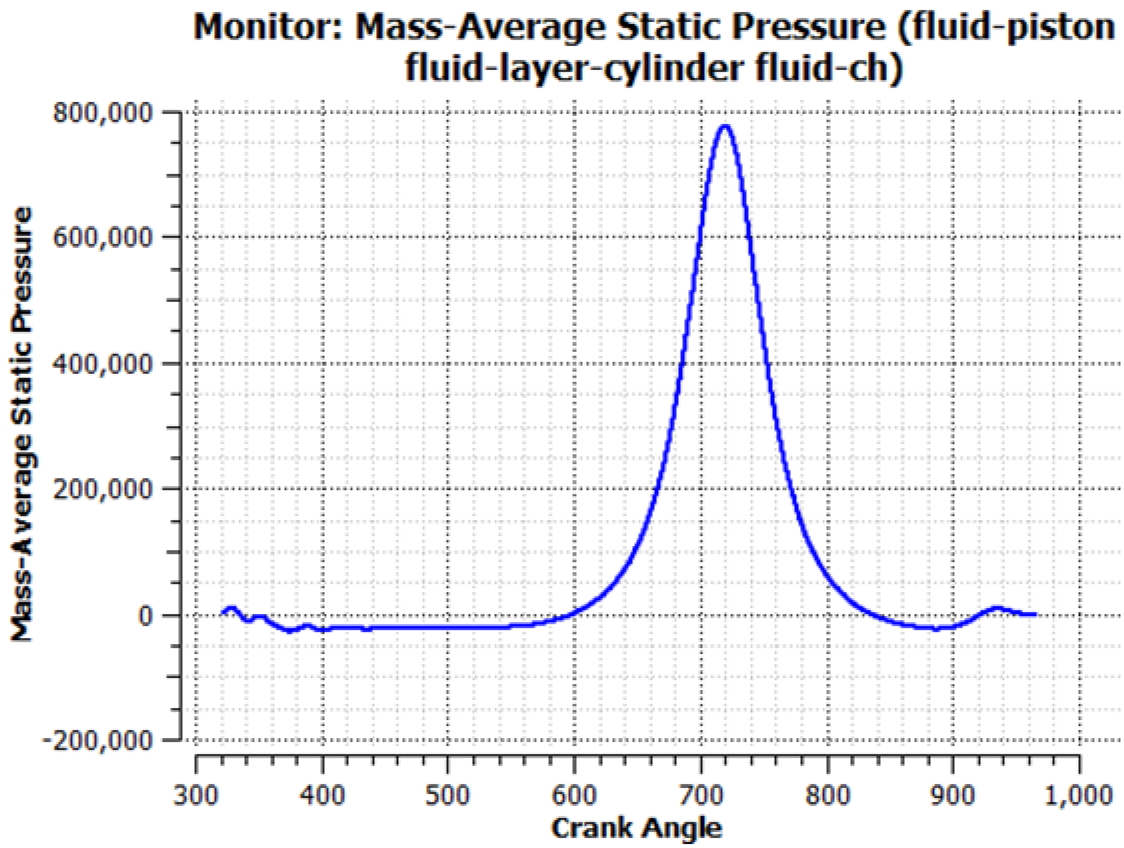
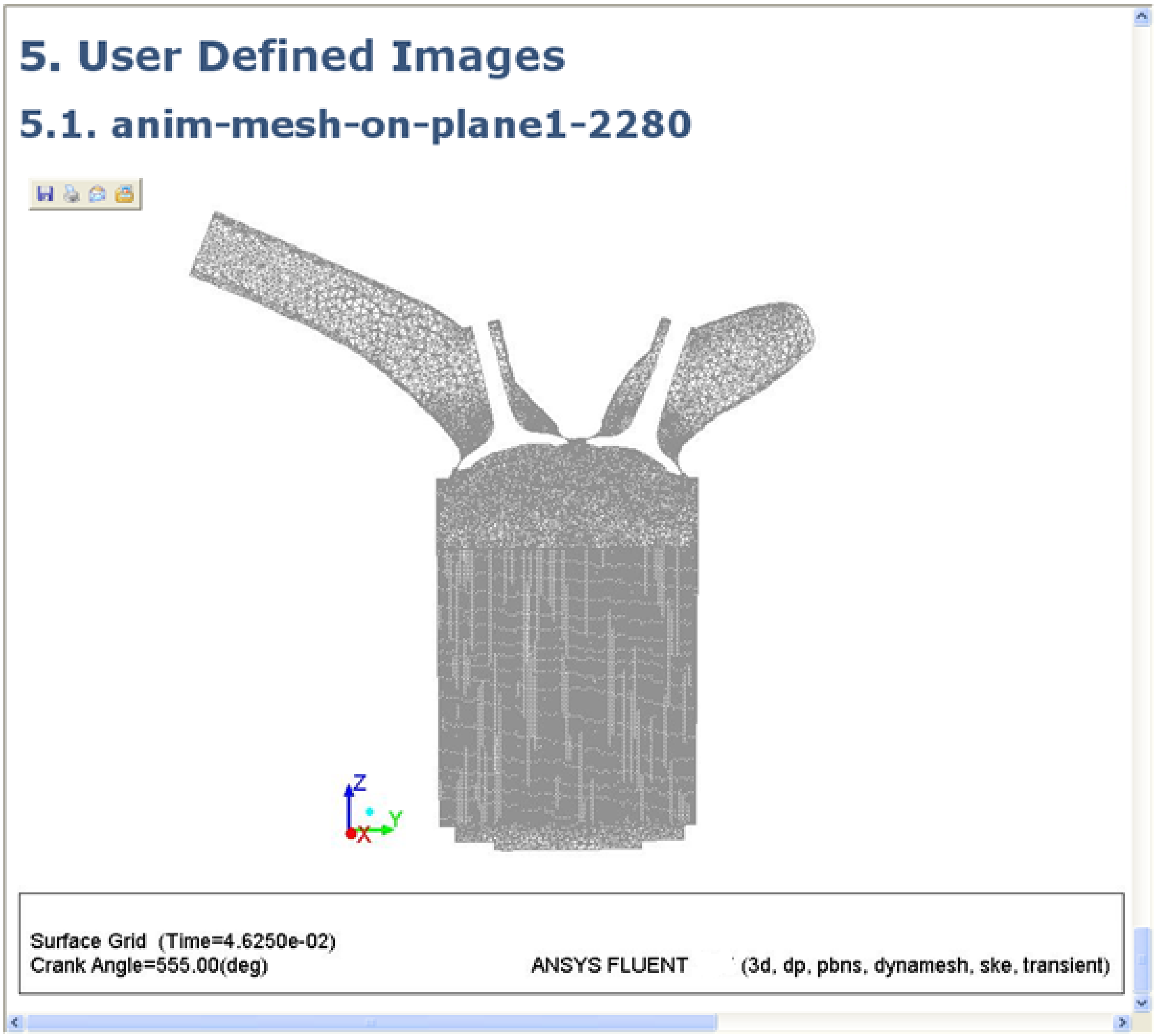


Chart 8. Monitor: Mass-Average Static Pressure (fluid-piston fluid-layer-cylinder fluid-ch)



If you would like to add some images or animations to the report you can do so by placing the images or animations in the **Report** folder under **ICE**.

- Ensure that the names of the images you would like to add in the report start with `"ice-image-"`. All such images will be displayed in the report under the section **User Defined Images**.

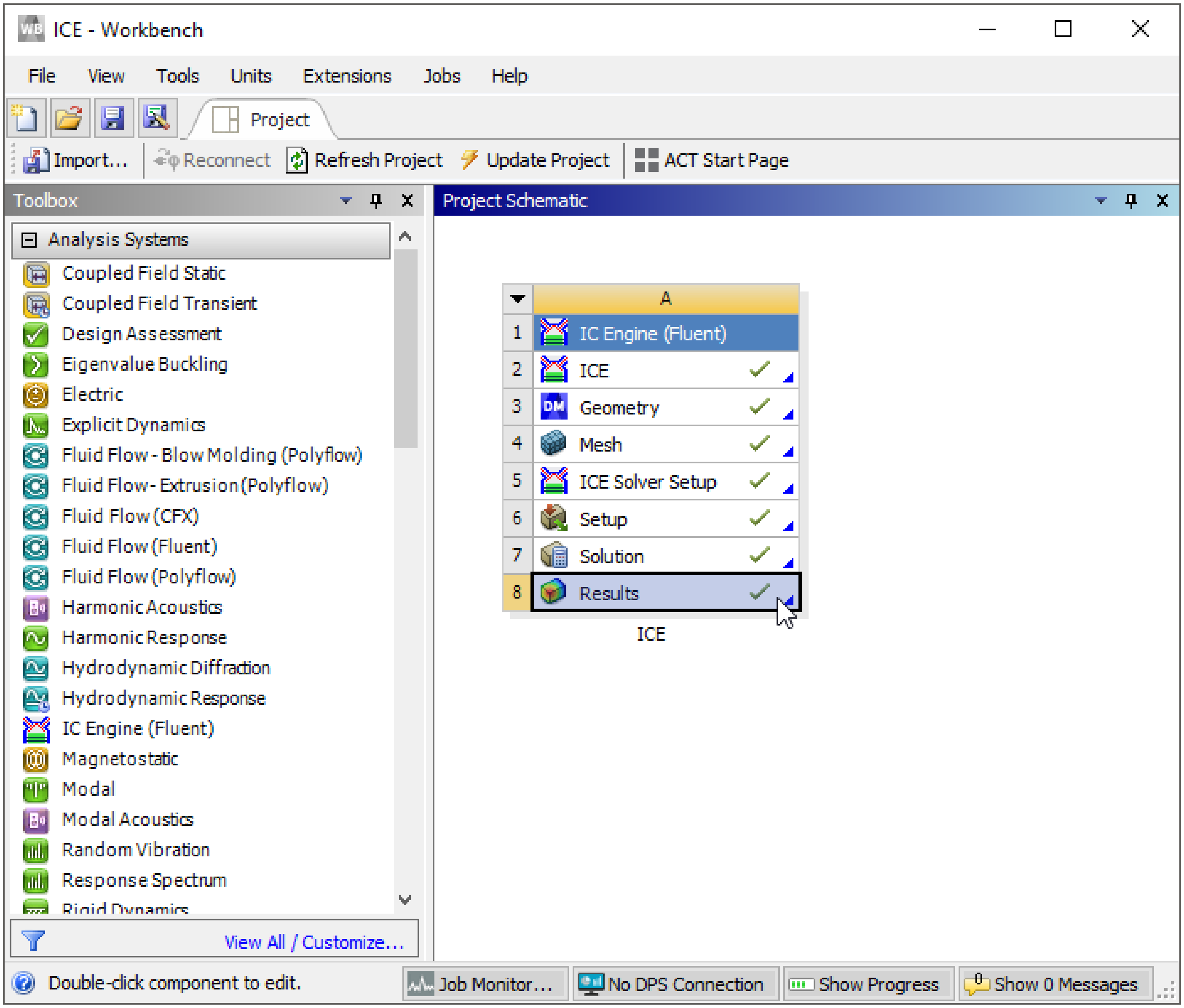


- Ensure that the names of the images in the animation start with `"ice-anim-"`. The animation of the images will then be added to the **Solution Data** section along with other animations.

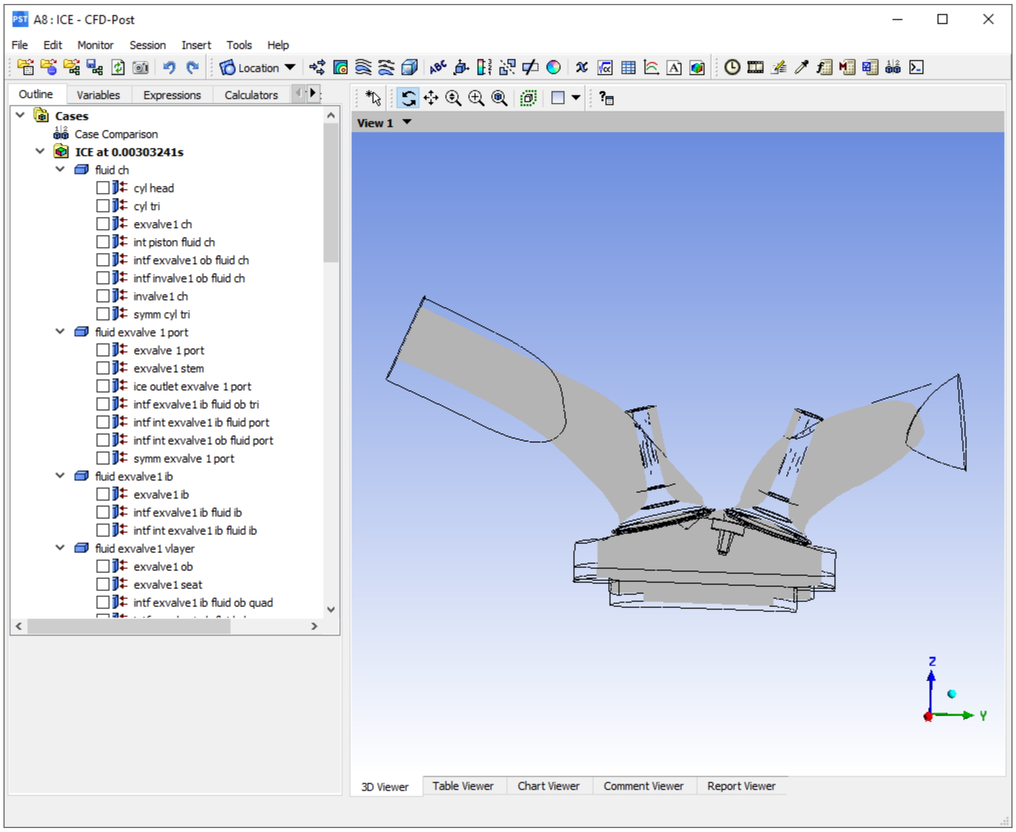
14.2. Postprocessing in CFD-Post

Once the report is generated you can also open it in CFD-Post. You can do further postprocessing here.

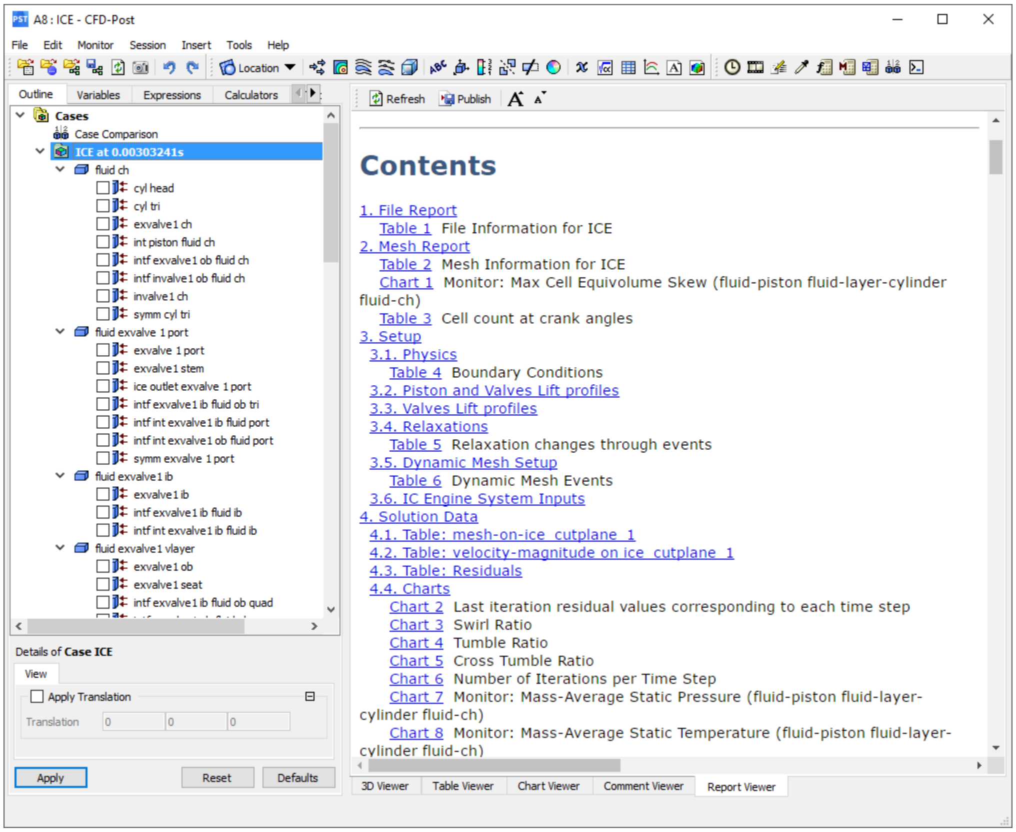
To open the report in CFD-Post, double click the **Results** cell.



The CFD-Post window opens. It loads the geometry.



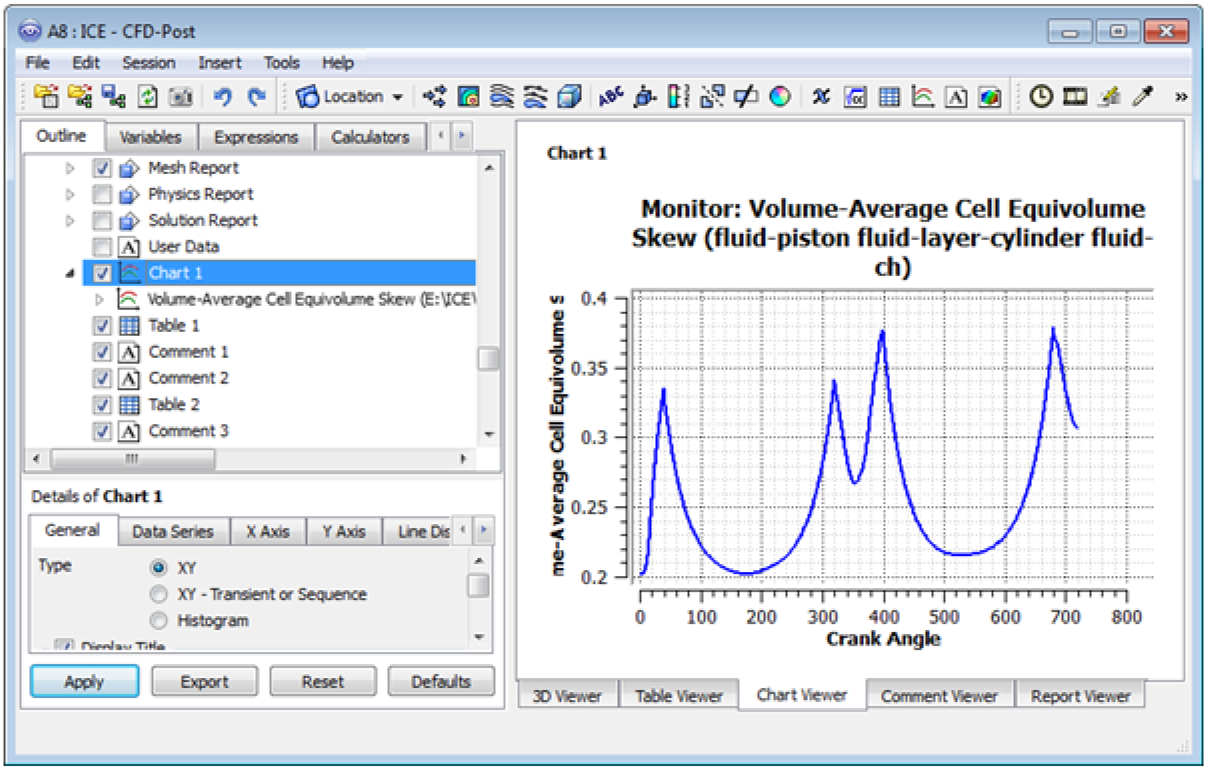
The **Report** is loaded in **CFD-Post**. You can see the plane created in the geometry in the **3D Viewer** tab. In the **Report Viewer** tab you can see the report as it was seen in **Report.html**.



Note:

Instead of the animation, an image is displayed.

You can open the charts, tables, and reports listed under **Report** in the **Outline** tree.

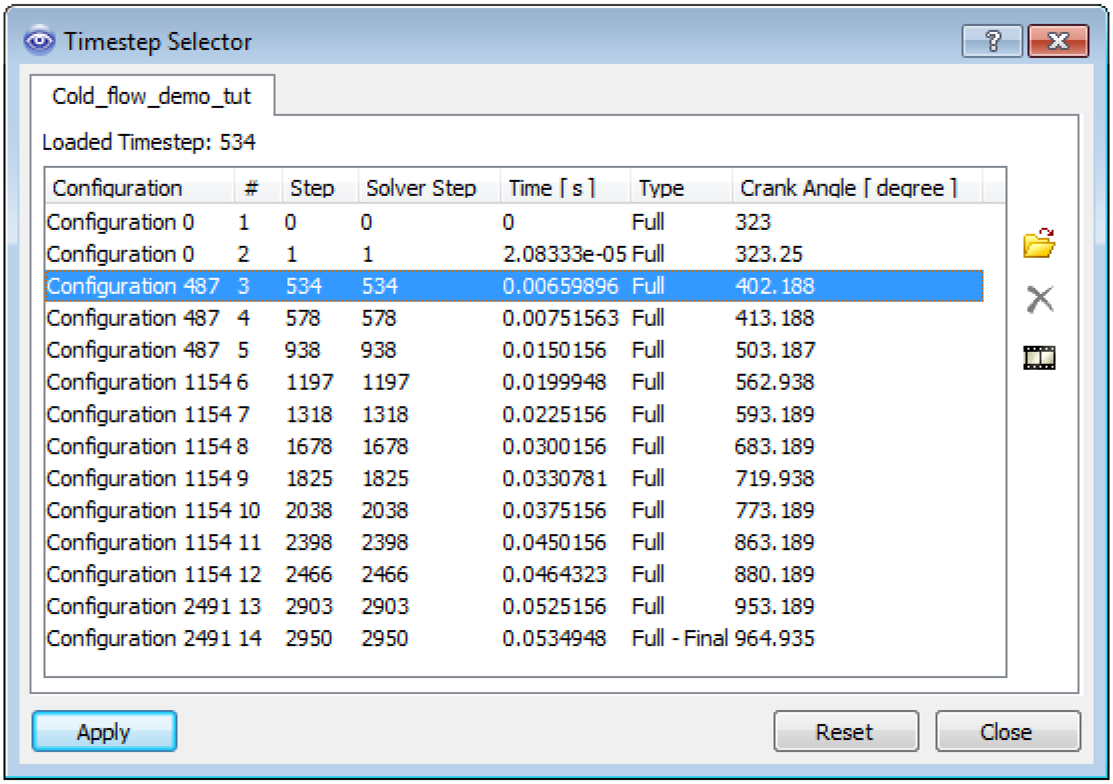


Creating Streamlines in CFD-Post

1. You can choose the time step at which you want to display the streamlines.

   Open the **Timestep Selector** dialog box by selecting **Timestep Selector** () from the **Tools** menu.

   Tools → **Timestep Selector**



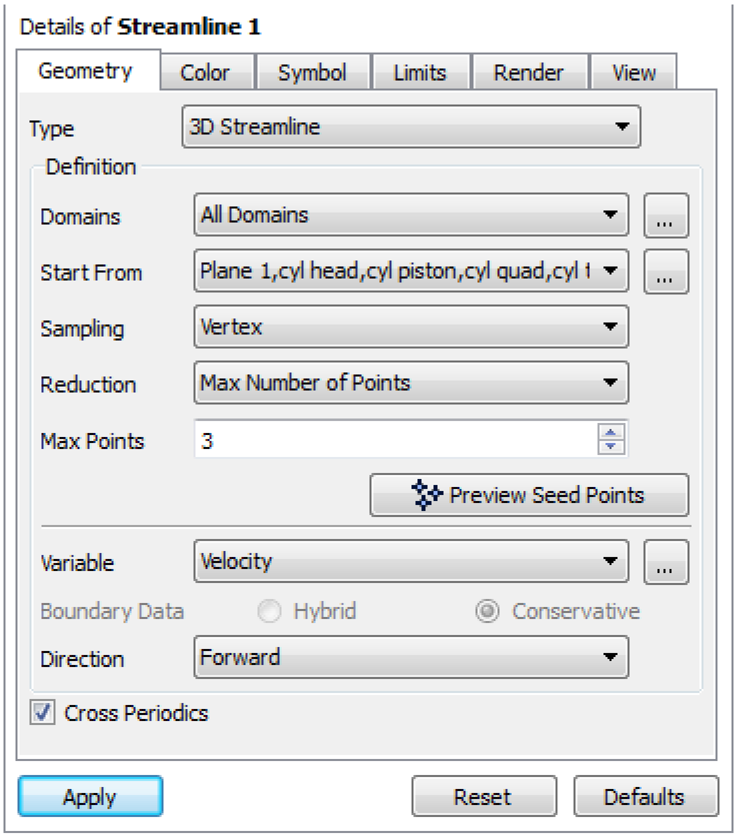
a. Select the **Step** of your choice from the list and click **Apply**.

b. Close the **Timestep Selector** dialog box.

2. Select **Streamline** () from the **Insert** menu.

Insert → Stramline

a. You can change the name here and then click **OK** in the **Insert Streamline** dialog box.

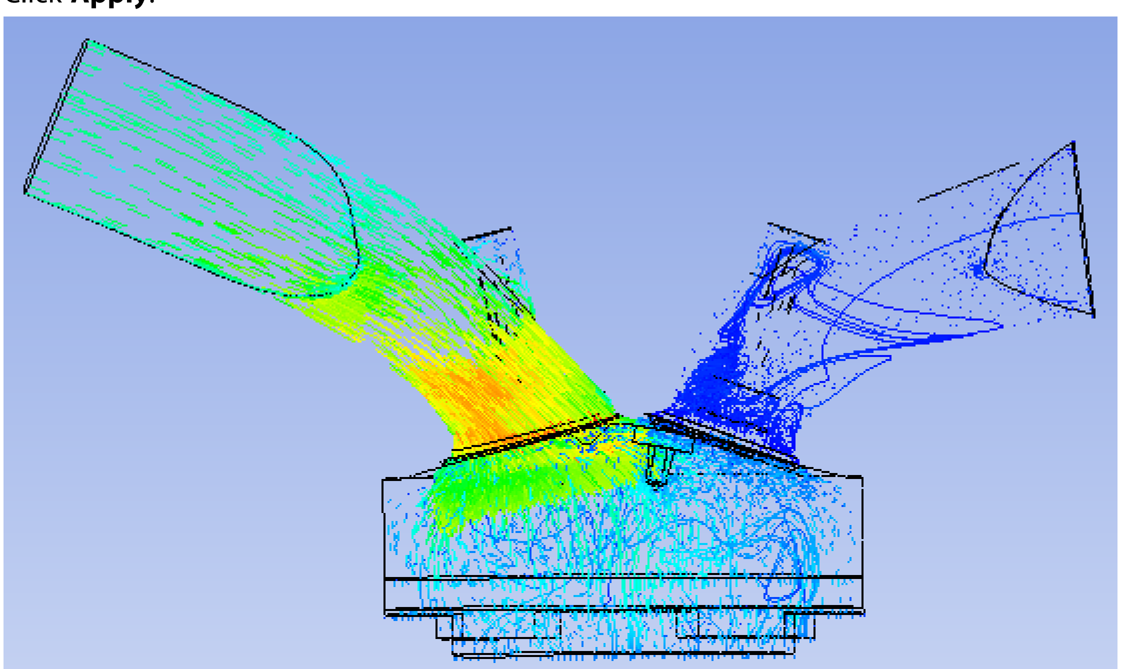


b. Click **Location Editor** (...) next to **Locations**, in the **Geometry** tab.

c. Select all items under **ICE** in the **Location Selector** dialog box and click **OK**.

d. You can change the name, color, symbol, and other factors of the vectors.

e. Click **Apply**.

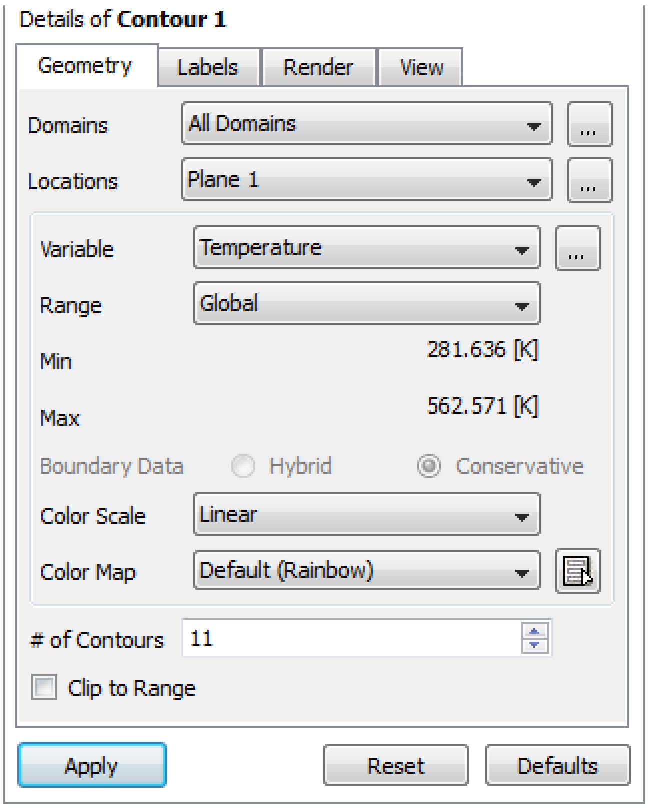


Creating Contours in CFD-Post

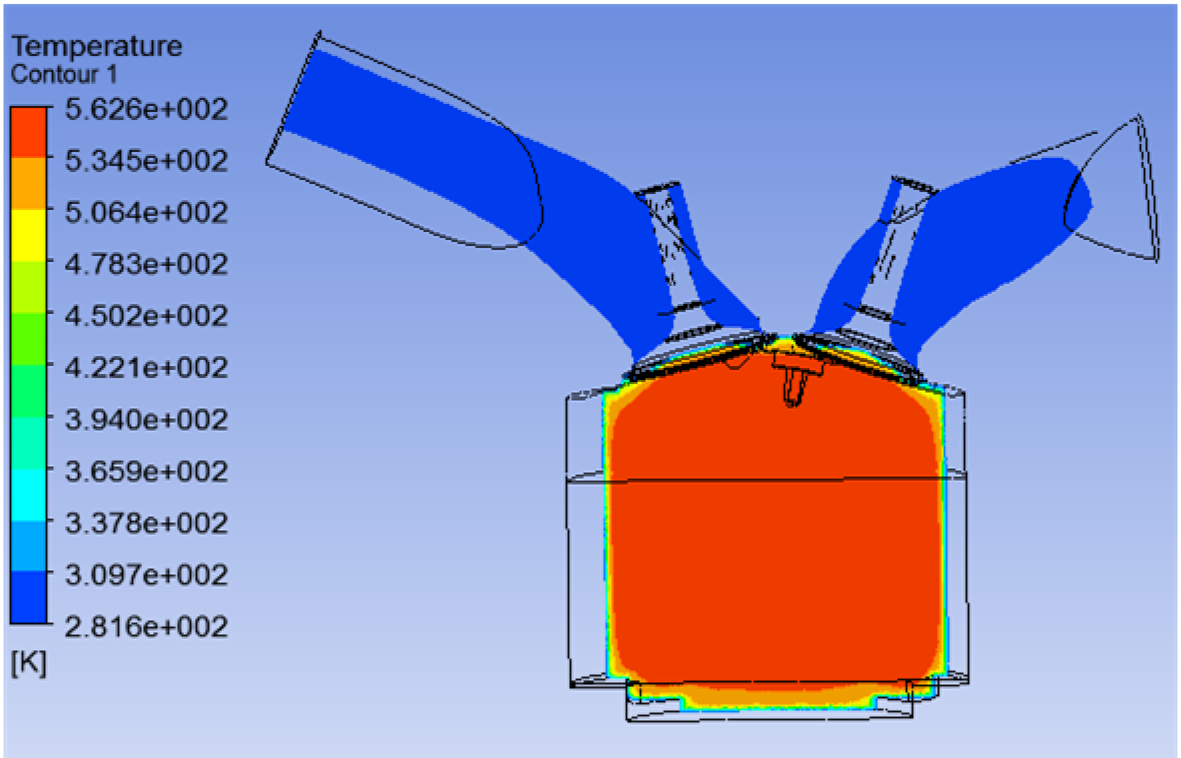
1. After selecting the time step at which you want the contours to be displayed, select **Contour** () from the **Insert** menu.

   Insert → Contour

   a. You can change the name here and click **OK** in the **Insert Contour** dialog box.

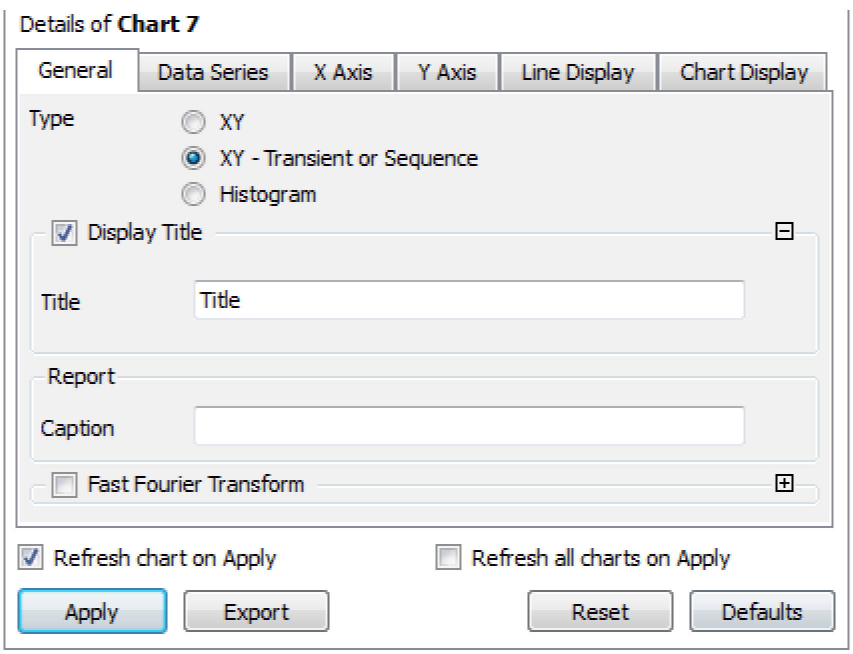
   

   b. In the **Geometry** tab you can select the location and the variable whose contour you would like to display and click **Apply**.



Creating Plots in CFD-Post

1. Select **Chart** () from the **Insert** menu.



2. You can select **XY - Transient or Sequence** from the **Type** list in the **General** tab.

3. You can select **Time** from the **Expression** drop-down list in the **X Axis** tab.

4. Select the variable you would like to plot from the **Variable** drop-down list in the **Y Axis** tab and click **Apply**.

You can create additional planes, contour images, reports, etc. For more information on CFD-Post, see the CFD-Post User's Guide.

Chapter 15: Troubleshooting the Simulation

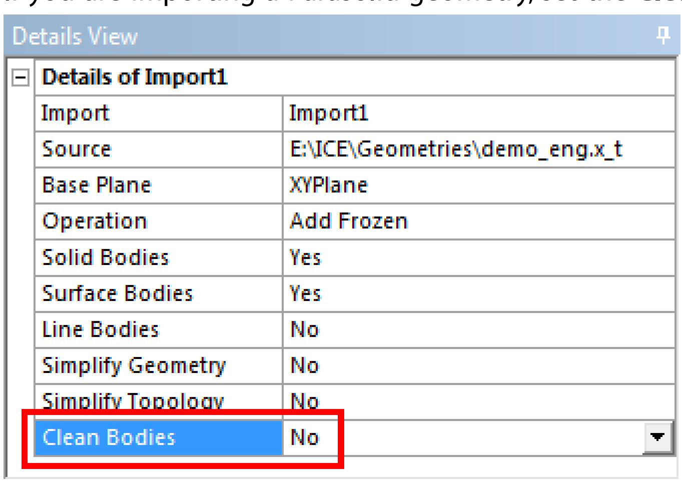
This section is intended to provide you with tips and strategies for avoiding and handling problems that may occur. It has been divided into the following sections.

15.1. Geometry Check

It is a good practice to check the geometry before decomposing. There are a few things that should be verified so that the problems occurring later on can be reduced.

- Piston should be at TDC position.

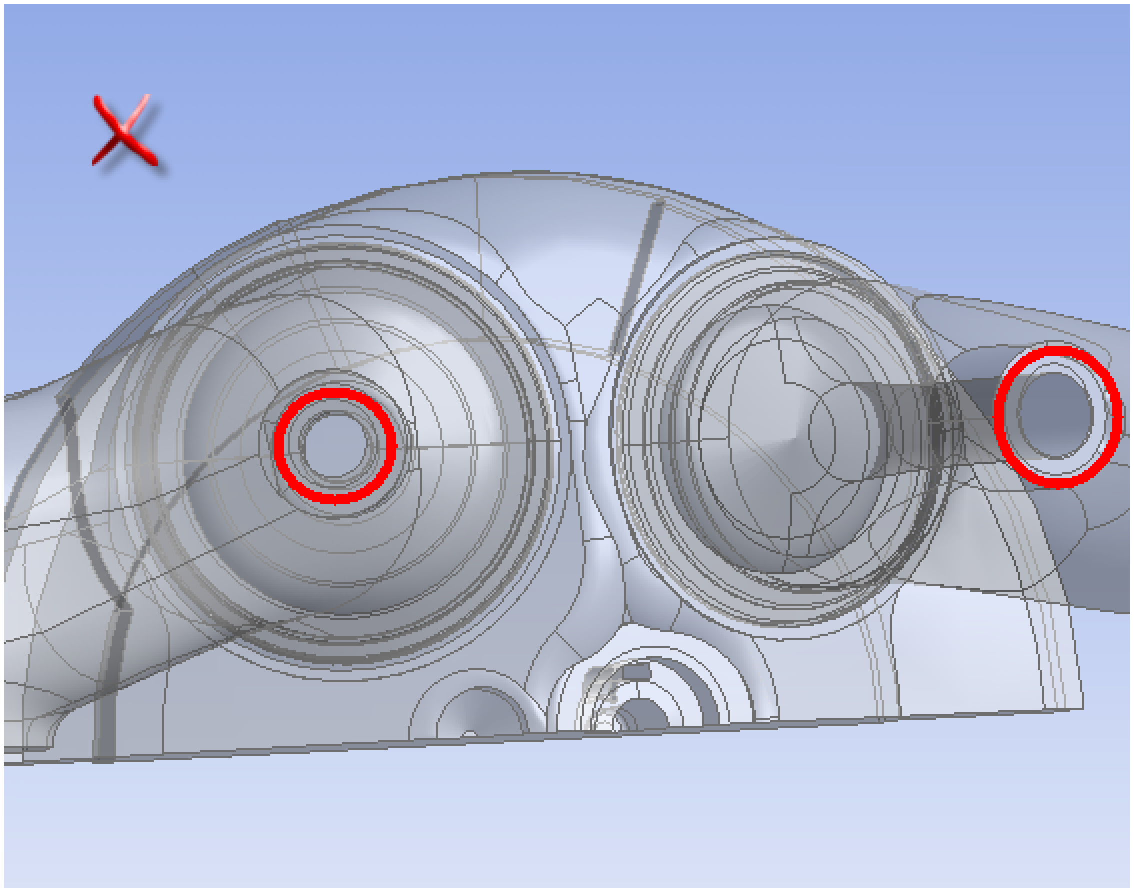
- If you are importing a Parasolid geometry, set the **Clean Bodies** option to **No**.



- The imported geometry should have only one flow volume with solid valves. If no flow volume is present see Creating Flow Volume (p. 566) for the procedure to fix it.

- Ensure that the valves are not extracted from the port volume in the initial geometry as shown in Figure 15.1: Valves Extracted from Port Volume (p. 518).

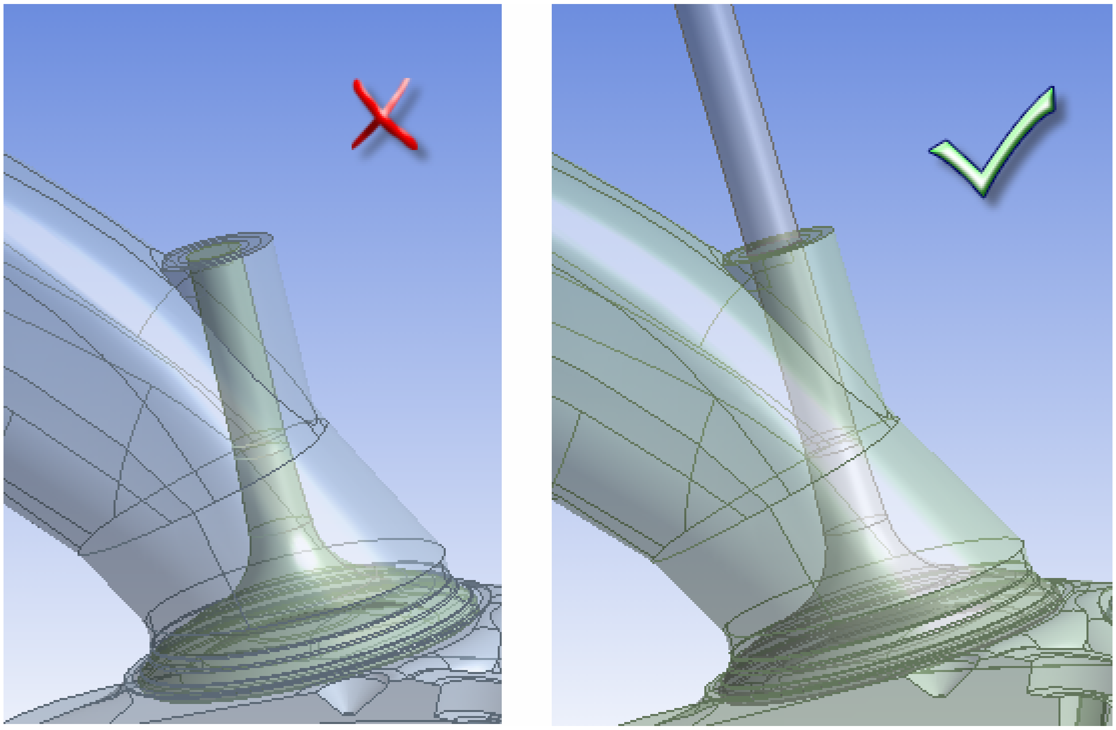
Figure 15.1: Valves Extracted from Port Volume



To fix this use the **Pre Manager** (p. 149).

- Ensure that the valve stem protrudes out of the port body. If the stem is fully enclosed in the port body, then you will have to modify the geometry by increasing the stem length. See Figure 15.2: Check Stem Size (p. 519).

Figure 15.2: Check Stem Size



- Ensure that the valve is in correct position. The valve should be located below the valve seat. It should not be present above the valve seat. The valve body should not overlap the valve seat, as shown in Figure 15.3: Valve Body Overlapping Valve Seat (p. 520). It should not cut the port body as shown in Figure 15.4: Incorrect Valve Position (p. 520). If the criteria are not met you will have to cleanup the geometry before the decomposition. There should be some minimal gap between the valve body and valve seat as shown in Figure 15.5: Correct Valve Position (p. 521).

Figure 15.3: Valve Body Overlapping Valve Seat

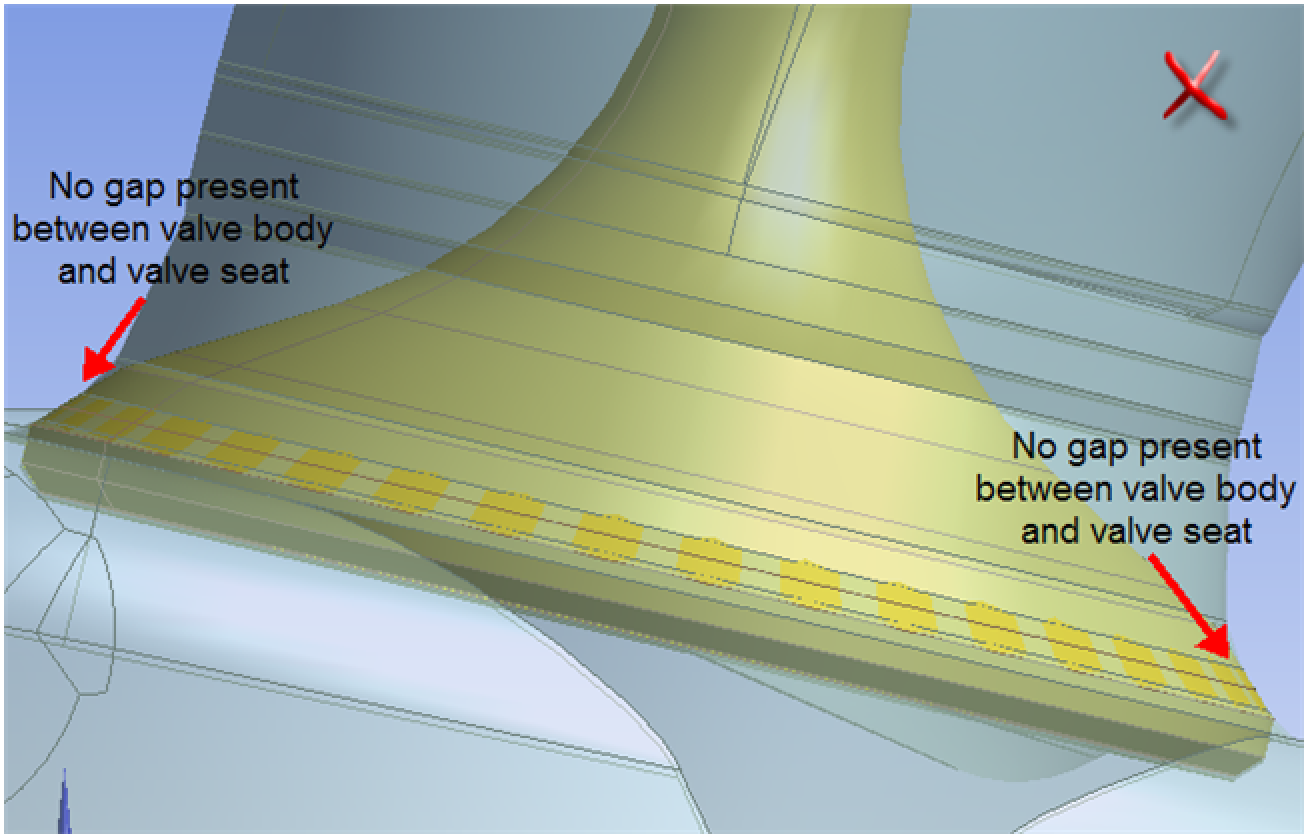


Figure 15.4: Incorrect Valve Position

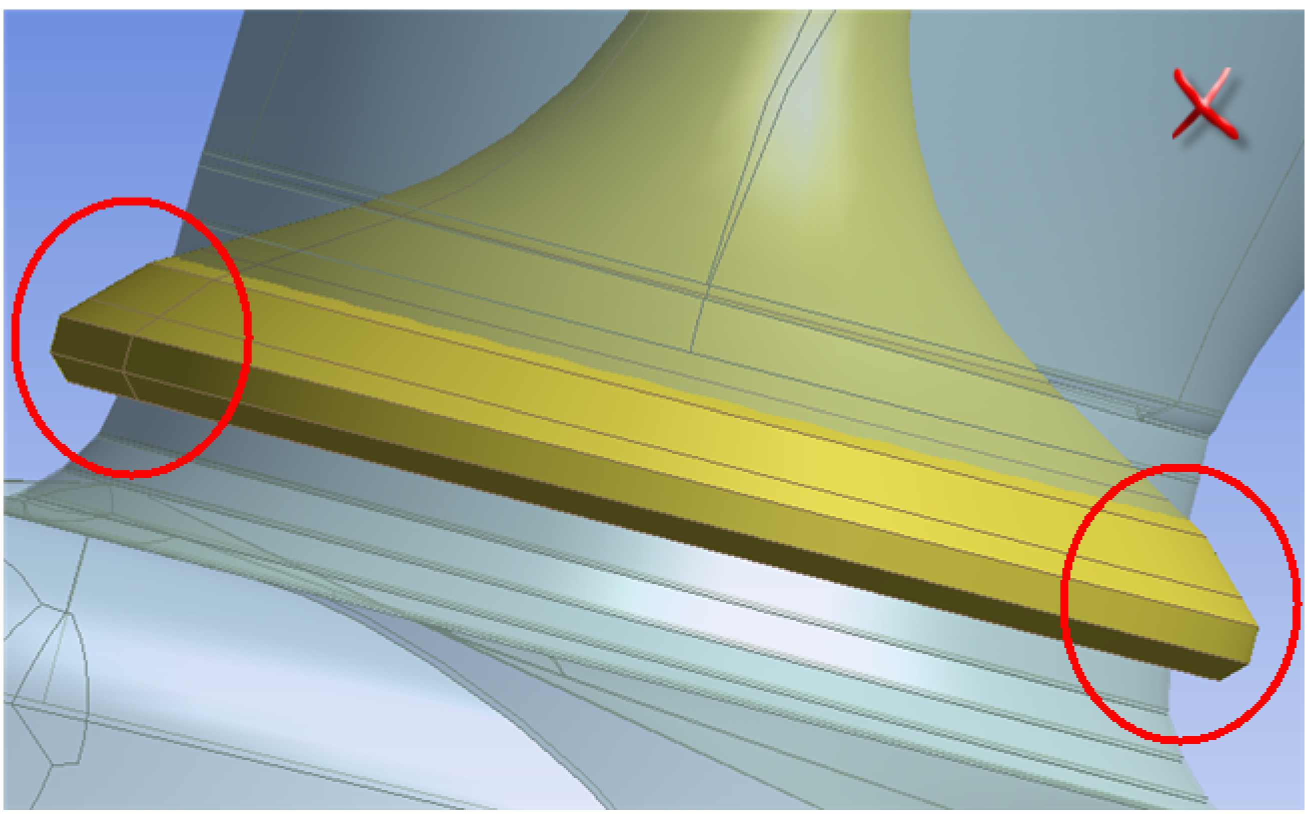
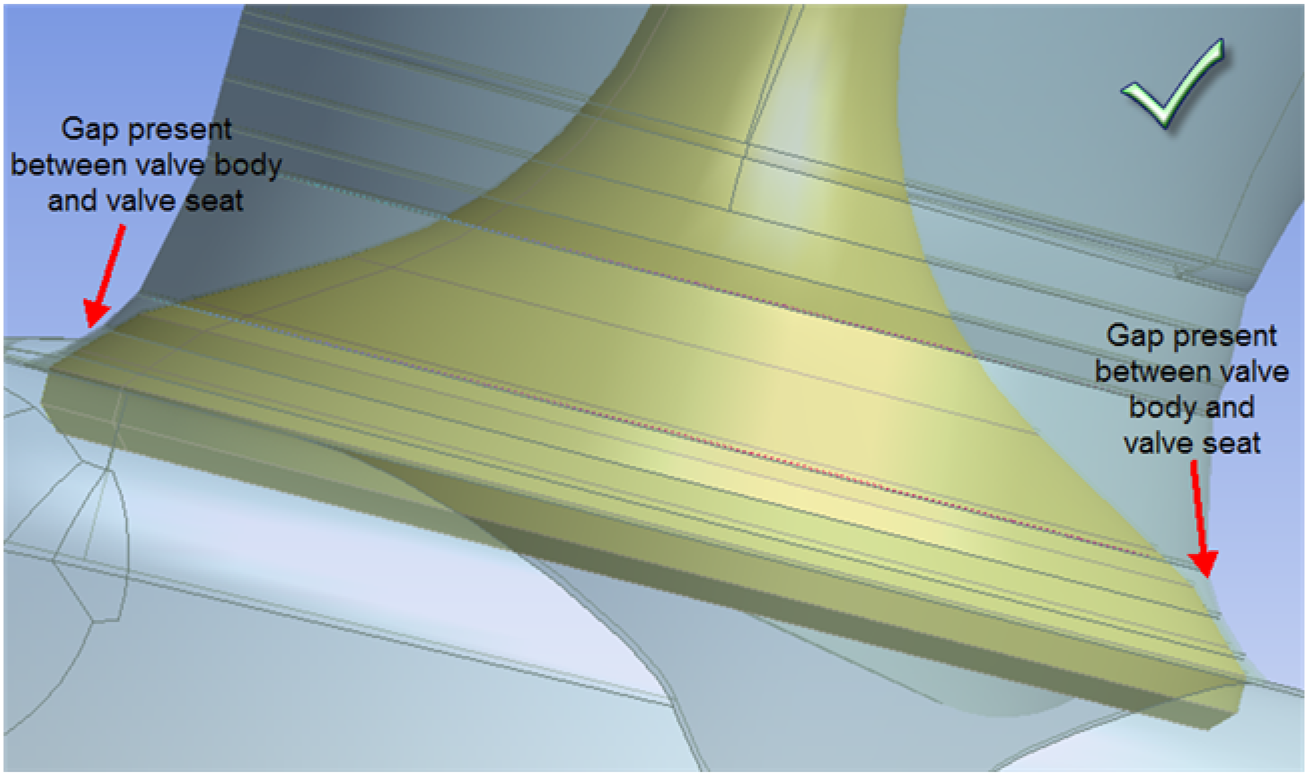
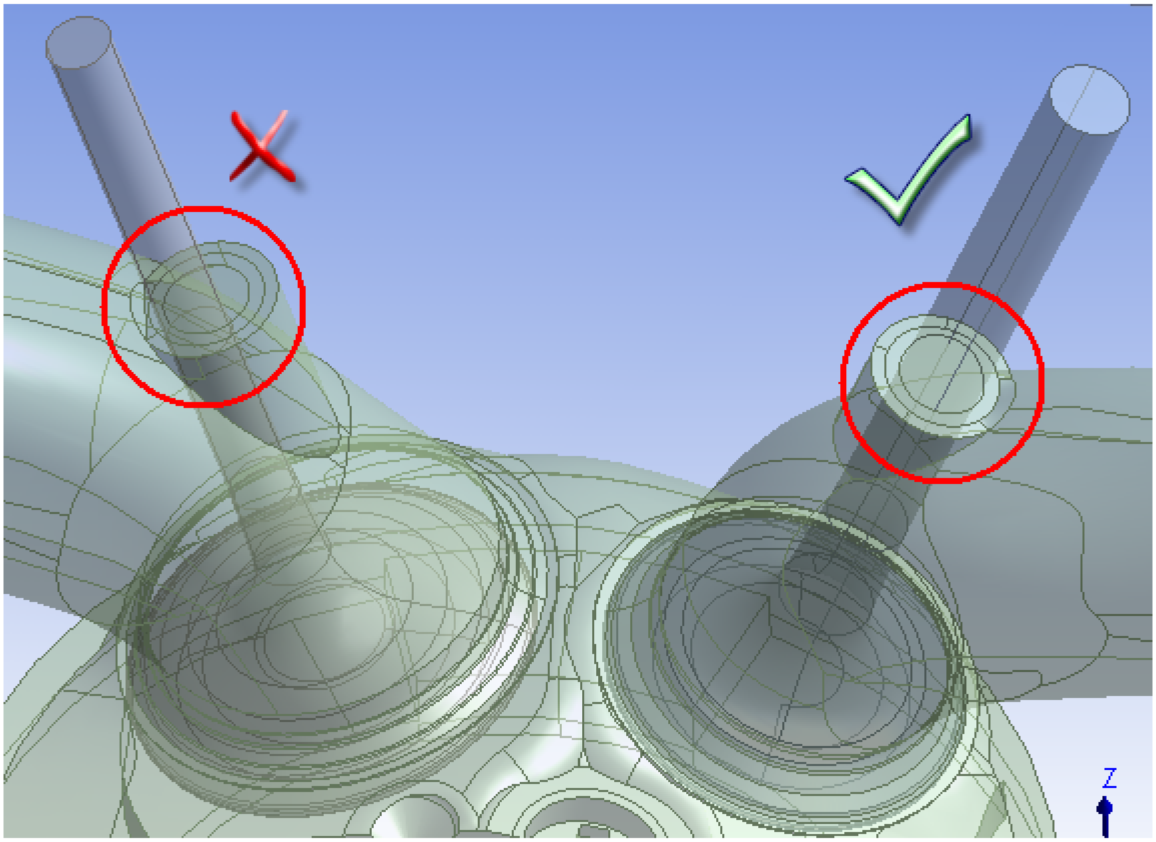


Figure 15.5: Correct Valve Position



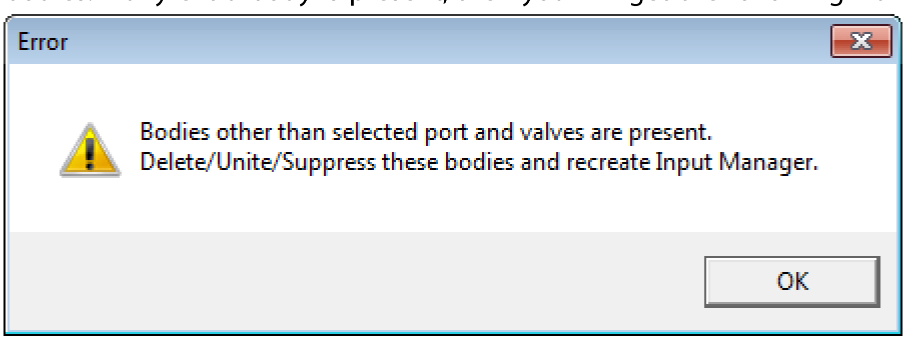
- Ensure that the valve is centrally aligned to the valve guide. An off-centered valve can result into failures and wrong results. See Figure 15.6: Check Valve Alignment (p. 522).

Figure 15.6: Check Valve Alignment



To correct the alignment follow the steps listed in the section Aligning the Valves With the Seat Faces (p. 525).

- Ensure that there are no extra bodies in the geometry, other than one port body and the valve bodies. If any extra body is present, then you will get the following warning while decomposing:



You will have to delete the body or unite it with the appropriate body so that other than port and valves no other body is present. Suppressing the body can also work.

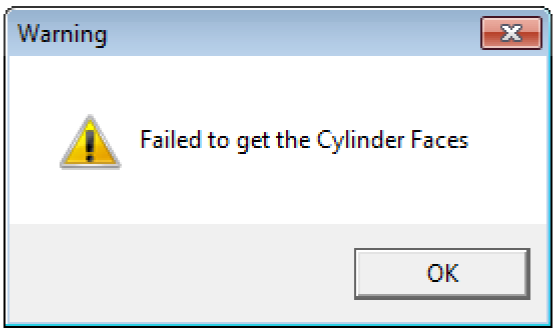
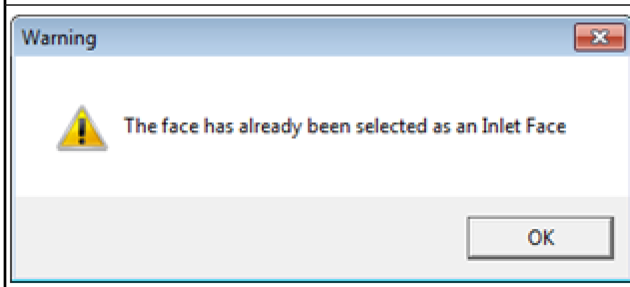
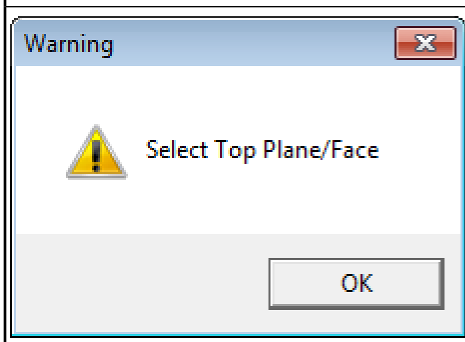
- If you want to decompose the chamber for a straight (vertical) valve engine with valve pockets you need to do some manual operations as explained in Decomposing a Straight Valve Pocket Engine (p. 552).

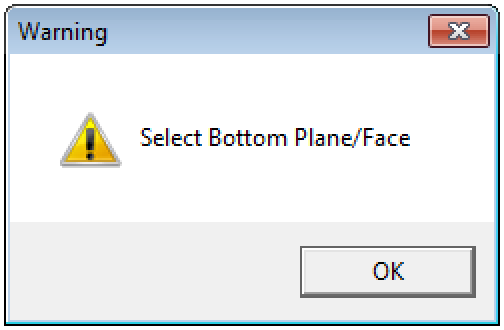
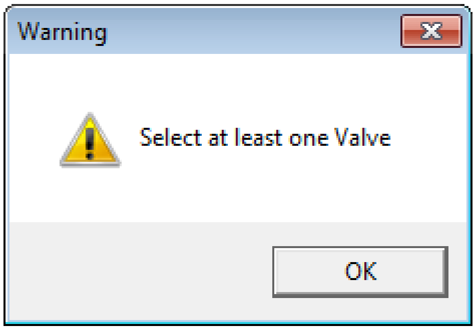
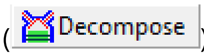
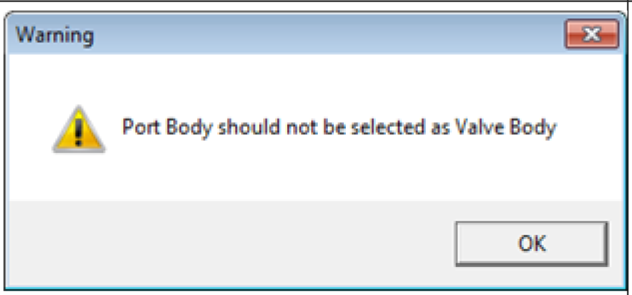
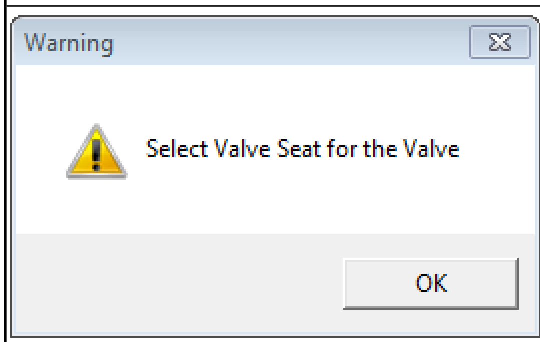
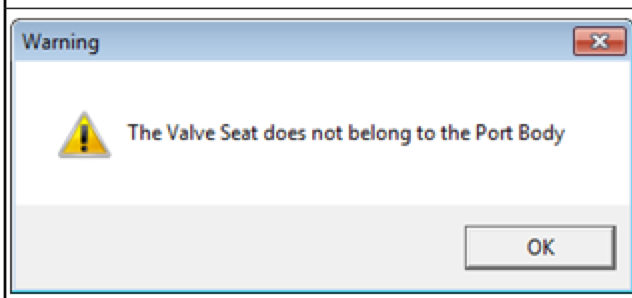
Important:

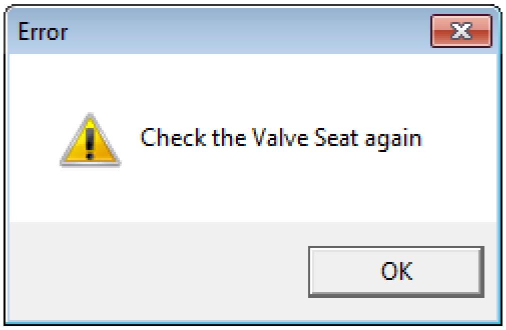
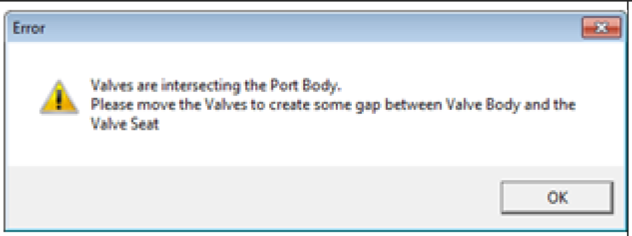
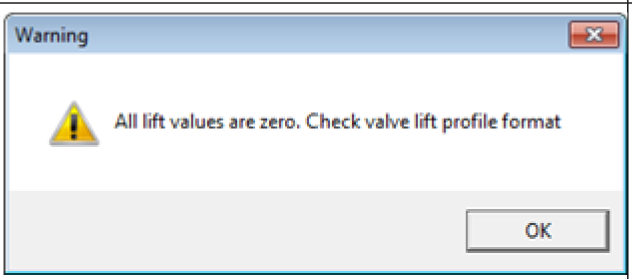
If there are any hard edges in the geometry they will be automatically removed to avoid further errors/failures in the simulation.

15.2. Geometry Preparation

Following are some warning and error messages you might get, while preparing the geometry for decomposition.

Warning/Error Message	Troubleshooting
	Selecting face(s) for **Cylinder Faces** in the **Input Manager** dialog box is mandatory. You cannot proceed with decomposition unless this step is completed. If you fail to select any face for the **Cylinder Faces** option, then the warning is displayed.
	If while selecting faces for inlet, exhaust, cylinder, or symmetry, you select a face which had already been selected for another parameter, then you get the warning that, the face has already been selected; you will have to change the selections if you want to continue.
	Selecting a plane or face for **Top Plane/Face** in the **Input Manager** dialog box is mandatory if **Decompose Chamber** is set to **Yes**. You cannot proceed with decomposition unless this step is completed. If you fail to select any face/plane then the warning is displayed.

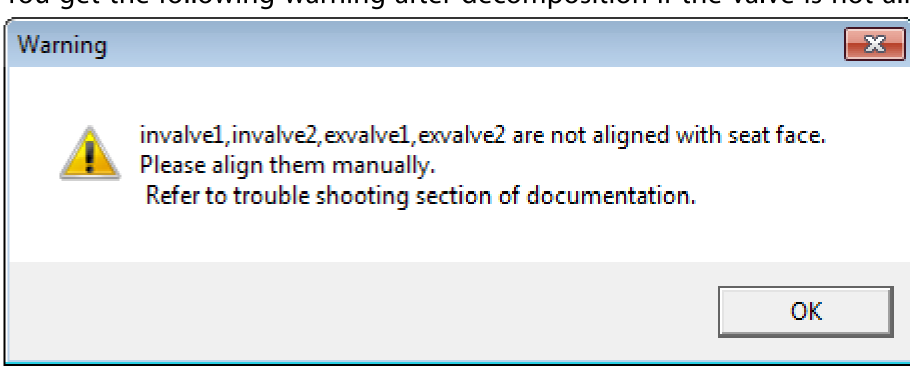
	Selecting a plane or face for **Bottom Plane/Face** in the **Input Manager** dialog box is mandatory if **Decompose Chamber** is set to **Yes**. You cannot proceed with decomposition unless this step is completed. If you fail to select any face/plane then the warning is displayed.
	Selecting a body for **Valve** is mandatory. If you click **Decompose** () without selecting a body for the **Valve** option in the **Input Manager** then the warning message is displayed.
	When prompted to select a body for a valve, if you select a body which can be a part of the port body, then the warning is displayed. Even though the warning is displayed the body part you selected will be accepted. If during decomposition the program recognizes that the selected valve body is a part of the port body, then the decomposition will not be completed and another warning is displayed. If the selected body part is accepted again as the valve body then decomposition will take place.
	Selecting a face for **Valve Seat** in the **Input Manager** is mandatory. If you fail to select a face for the valve seat then the warning is displayed.
	When prompted to select a face for the **Valve Seat** in the **Input Manager**, and you select a face which can be a part of valve body, then the warning is displayed. Even though the warning is displayed, the face you selected will be accepted and decomposition will take place. But if the

	program determines that the chosen face is wrong another warning is displayed. You will have to then select the right face and execute the decomposition again.
	If the program determines that the chosen face for the **Valve Seat** is wrong the error is displayed. You will have to select the right face and execute the decomposition again. In some cases this warning is repeatedly displayed even after you have ensured that you have selected the right faces. In such a case add a Body Operation, set the Type to Clean Bodies, and select all the bodies. Clean Bodies should be set to High. After this recreate **Input Manager**.
	If the distance between the valve body and the **Valve Seat** is less than the minimum required then the error is displayed while decomposing. You should move the valves to create some gap and execute the decomposition again.
	The format of the profile file selected for **Valve Lift and Piston Motion Profile** should be correct. Also not all values of the lifts in the profile file should be zero.

- If you change any of the decomposition parameters like **Decomposition Crank Angle**, and you get errors while updating the model, then you should delete all the features under **Input Manager** in the tree and decompose the geometry again.

Aligning the Valves With the Seat Faces

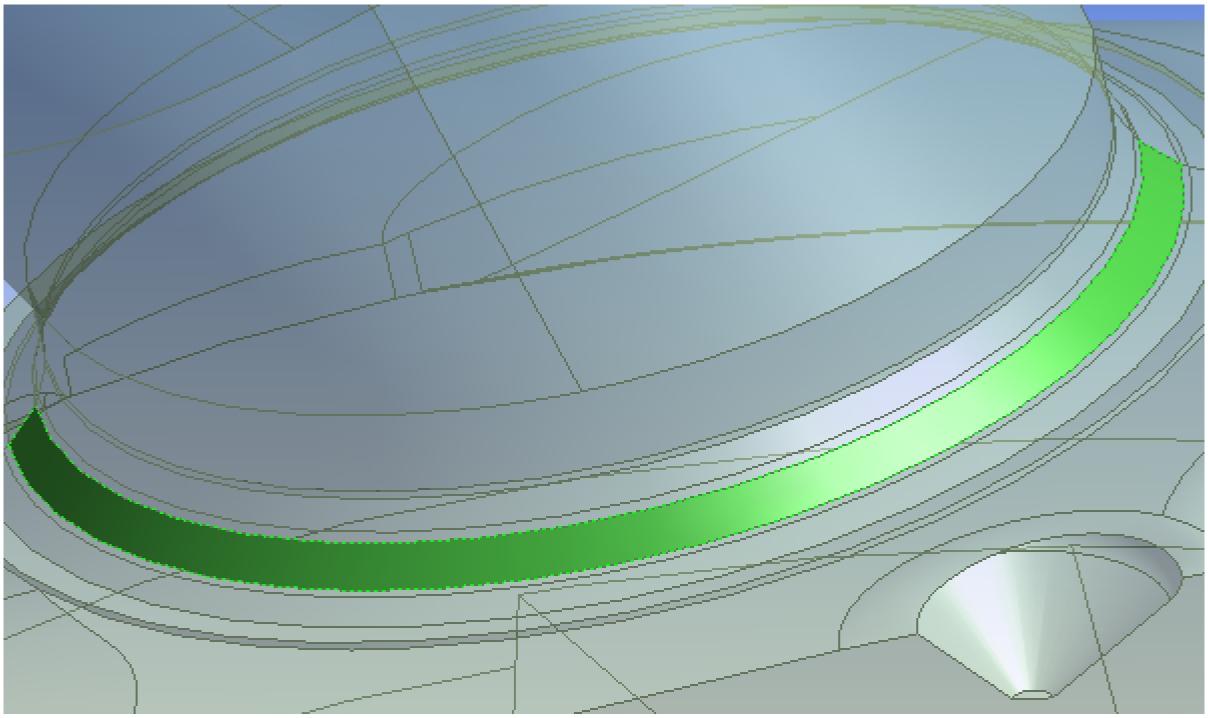
You get the following warning after decomposition if the valve is not aligned to the seat.



The warning will mention which valve(s) are not aligned to their respective seat face(s). To correct the problem perform the following steps:

1. First you need to create a plane using the valve seat. For creating a plane three non-colinear points/vertices are required. Two points on the valve seat can be selected, but the third point should be at the center.

   a. Select the valve seat face. It is easy to manipulate the geometry if the other faces are hidden. You can hide the other faces by first selecting the face and then selecting **Hide All Other Faces** from the context menu.

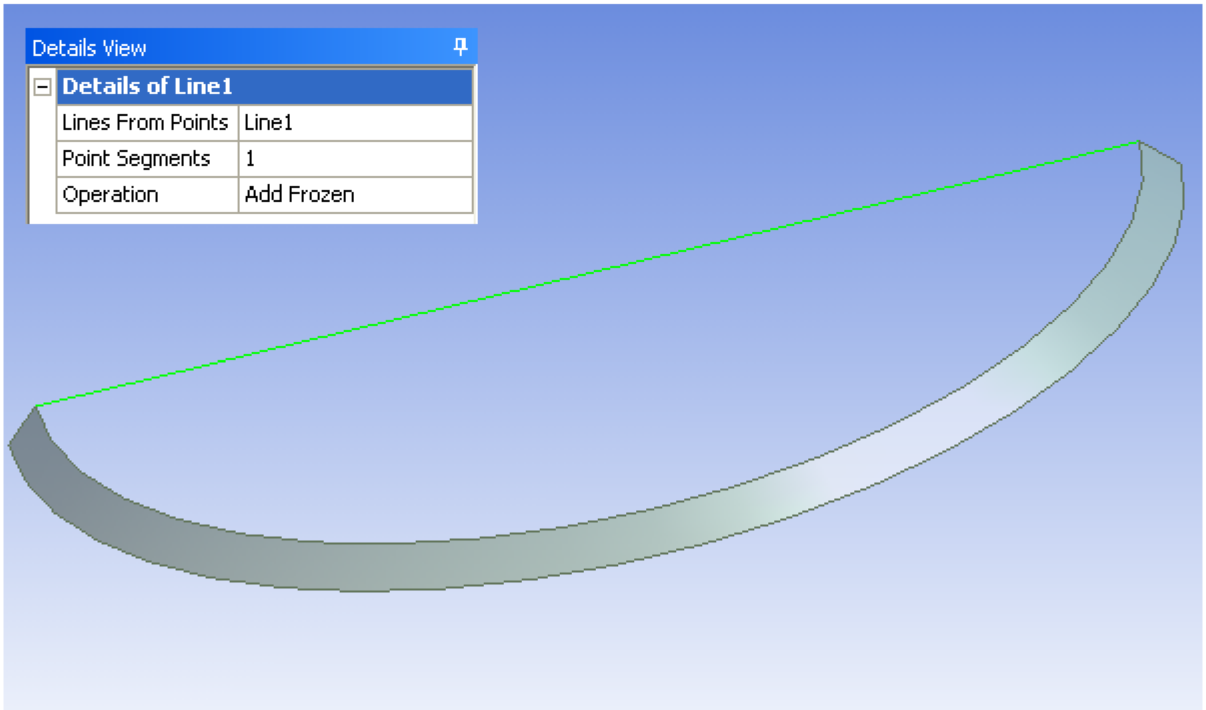
   

   b. Create a line.

   Concept → **Lines From Points**

      i. Select the two edge points of he valve seat for **Point Segments** in the **Details of Line** dialog box and click **Apply**.

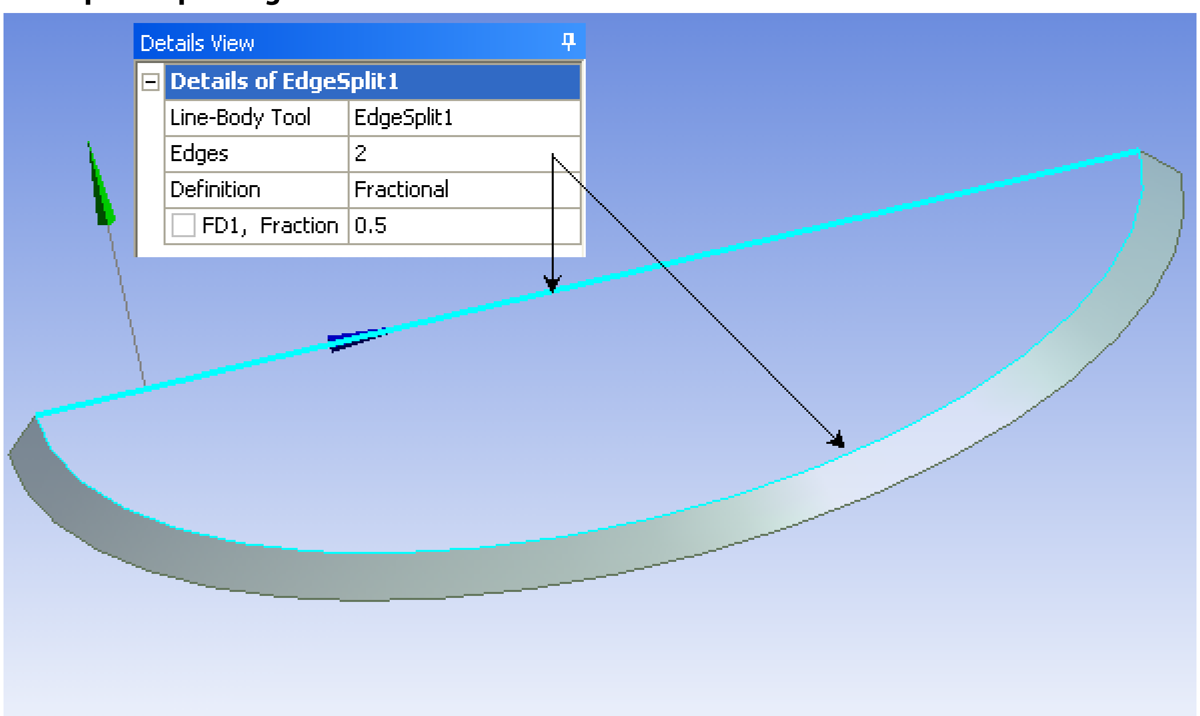
      ii. Select **Add Frozen** from the **Operation** drop-down list.



iii. Click **Generate** Generate.

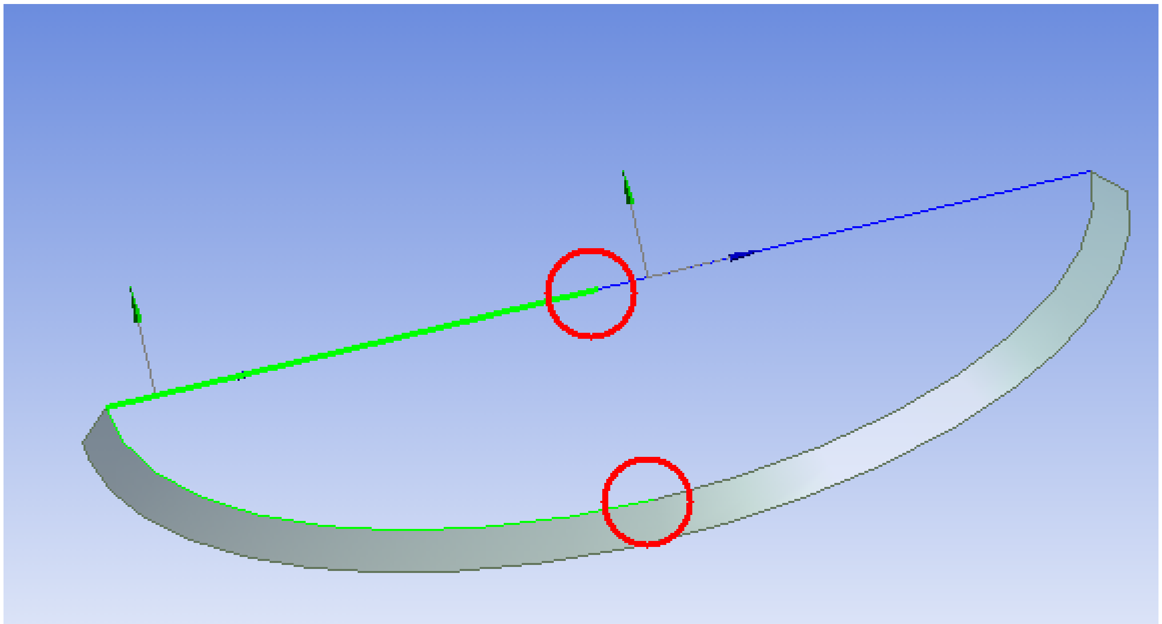
c. Create a center point of the lines.

Concept → **Split Edges**



Select the line created and the edge of the valve seat for **Edges** in the **Details of EdgeSplit** dialog box and click **Generate** Generate.

This will split the created line and the valve seat edge into two equal halves. Thus you now have the center points of both the lines.

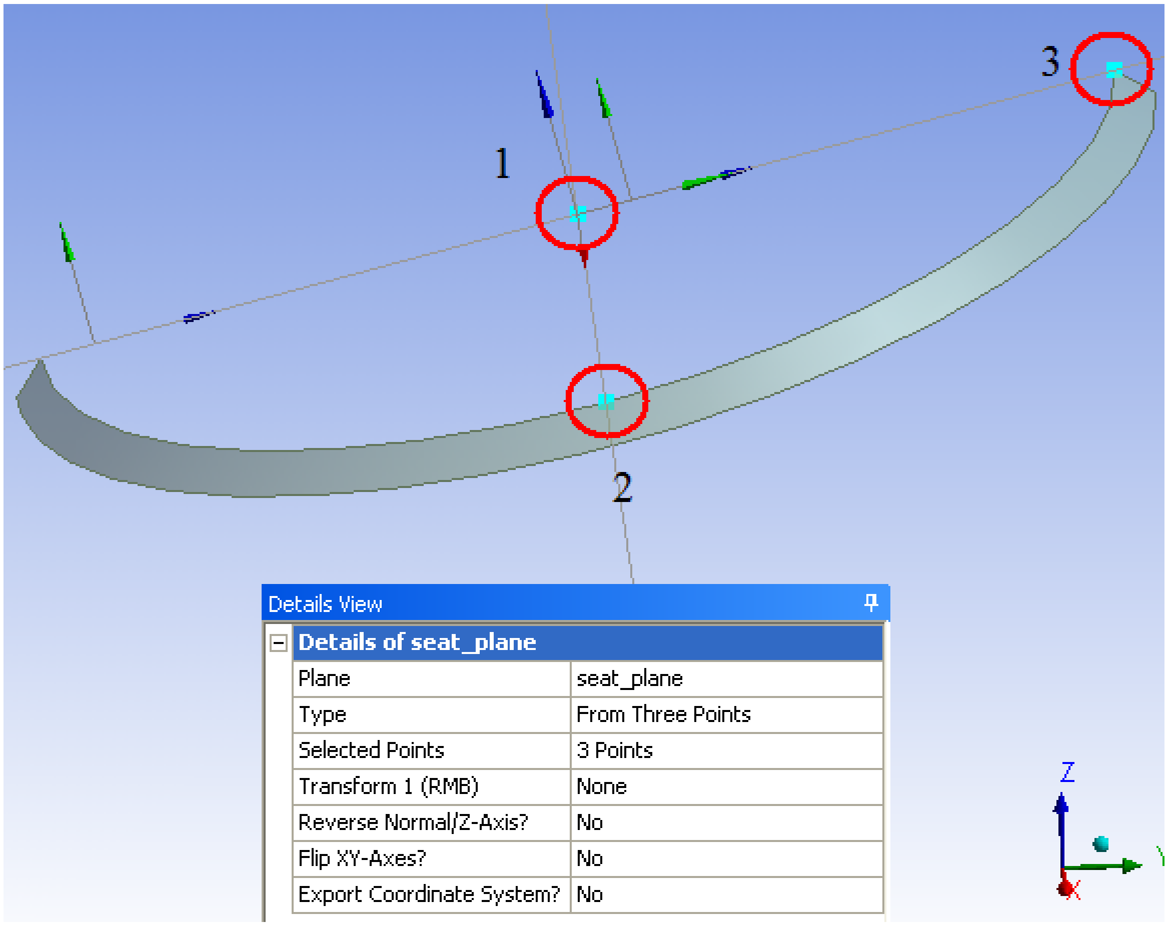


d. Click on **New Plane** .

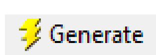
   i. Select **From Three Points** from the **Type** drop-down list.

   ii. Select the three points as shown in the Figure 15.7: Selecting 3 Points For Creating Plane (p. 529). Ensure that while selecting the points, you first select the center point of the line created. This sequence is important. Also all the three points should not be colinear.

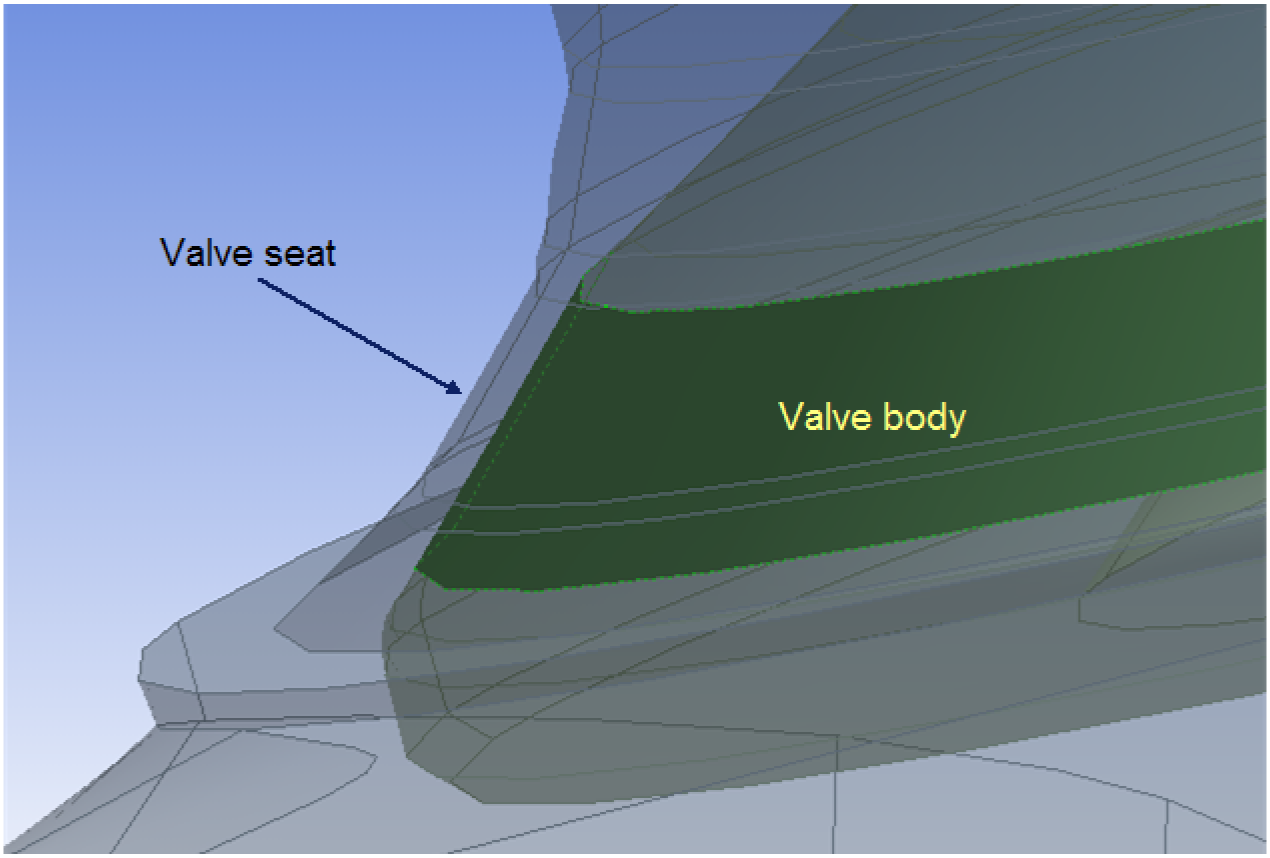
Figure 15.7: Selecting 3 Points For Creating Plane



iii. Ensure that the Normal or Z-Axis points upwards else you can select **Yes** from the **Reverse Normal/Z-Axis** drop-down list

iv. You can name the plane as `seat_plane` and then click **Generate** 
.

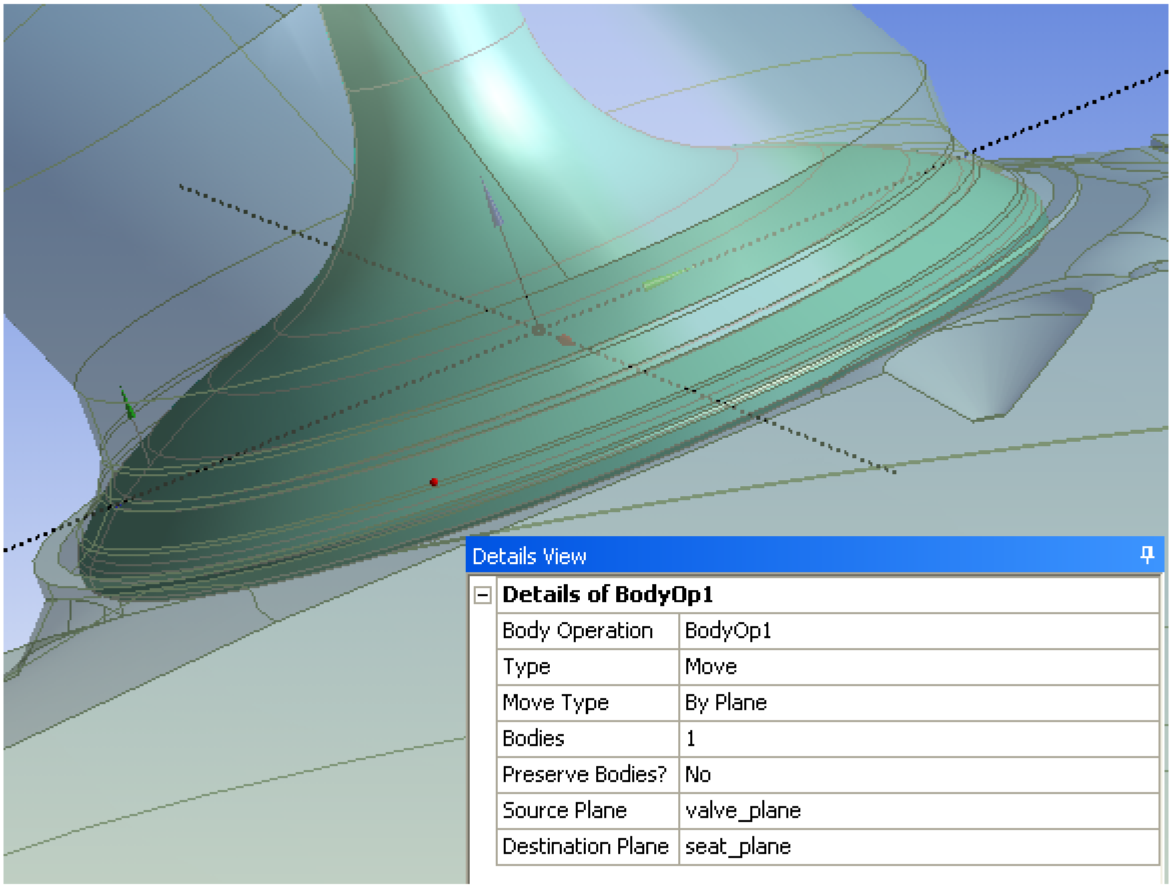
2. Similarly create a plane for the valve face. This is the face of the valve body which comes into contact with the valve seat.



You can name the plane created as `valve_plane`.

3. Move the valve body to align it with the seat.

   Create → Body Transformation → Move



a. Select **Move** from the **Type** drop-down list in the **Details of BodyOp** dialog box.

b. Retain **By Plane** for **Move Type**.

c. Select the valve body for **Bodies**.

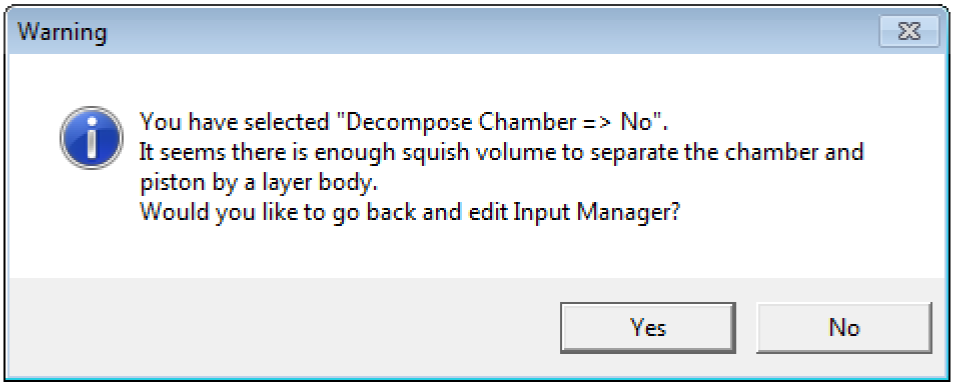
d. Select **valve_plane** for **Source Planes**.

e. Select **seat_plane** for **Destination Planes**.

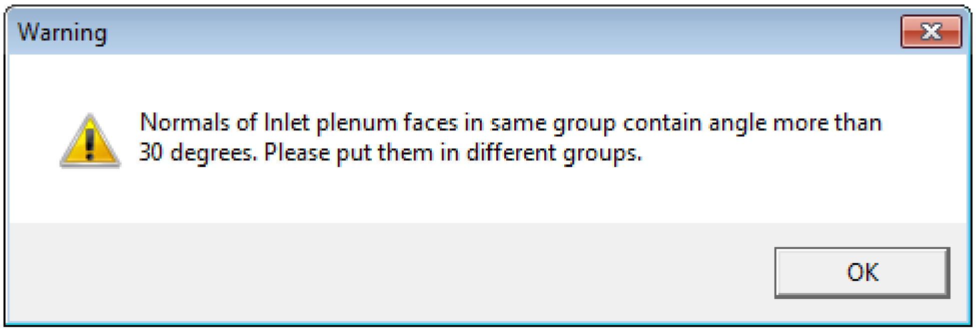
f. Click **Generate** Generate. to complete the operation.

4. If the valve moves in the upward or downward direction while aligning, you can use the **Translate** operation to bring it to proper position. Ensure that the valve is correctly positioned. Refer to the section on valve positioning (p. 519) for more information.

- If you have enough squish volume to separate the chamber and piston by a layer body and still you select **No** for **Decompose Chamber**, then the following warning is displayed. The squish volume is checked internally and you are given a chance to go back to the **Input Manager** and change the option of **Decompose Chamber** to **Yes**.



- If you select multiple faces for **IC Inlet Plenum** in the port flow simulation, then you should ensure that the angle between the normals of the faces is less than 30°. Else you will get the following warning.

  You should create different groups for the faces.

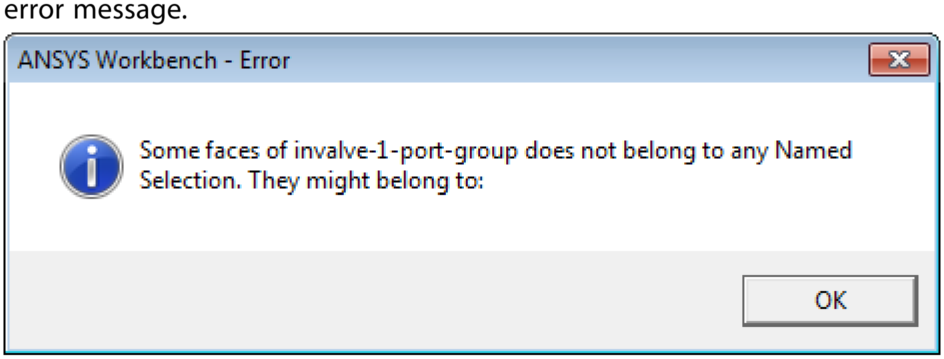
- If the geometry decomposes successfully only at TDC and at no other angle, then add a Body Operation, set the Type to Clean Bodies, and select all the bodies. Clean Bodies should be set to High. After this recreate **Input Manager**.

15.3. Mesh Generation

Following are some troubleshooting techniques for meshing.

- If the chamber meshing fails, then you should delete its pinch controls and execute the meshing again.

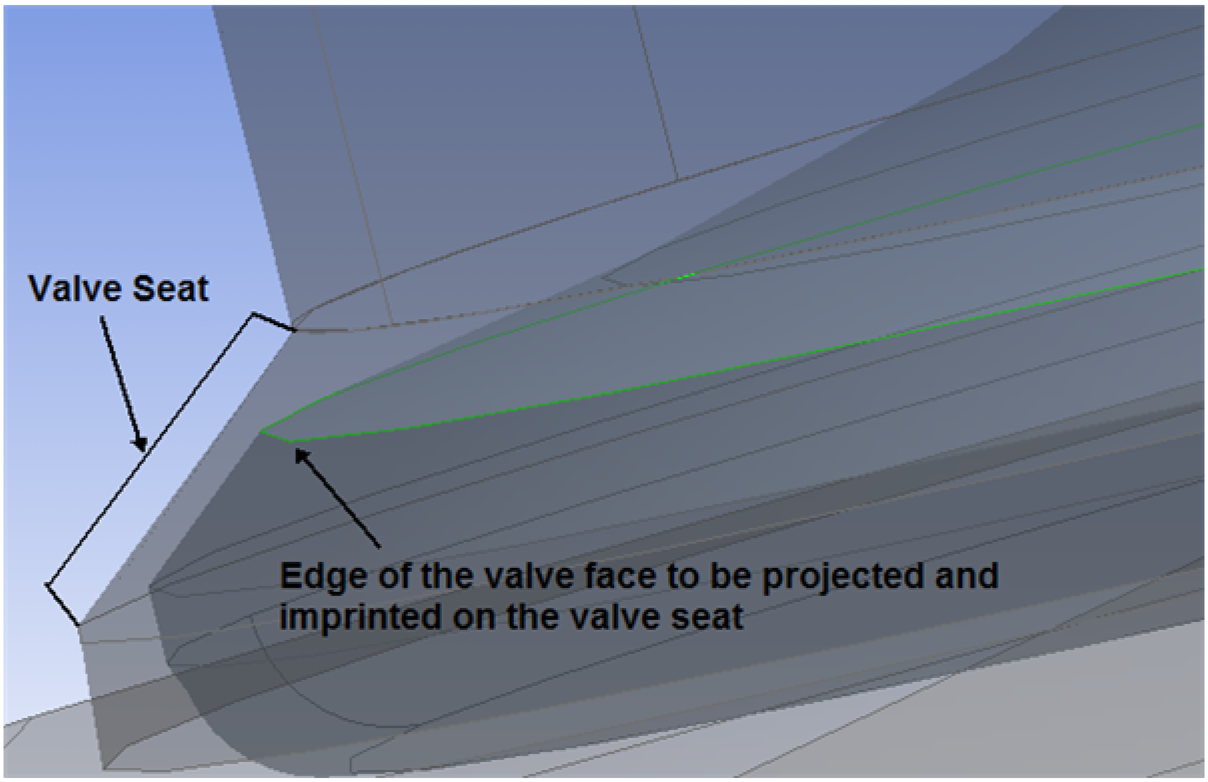
- If after mesh setup you have created some virtual faces then you may sometime get following error message.

You must check and add these faces to the appropriate **Named Selection** it belongs to. If some faces belonging to a named selection group are not selected for the **Geometry** option they belong to, then the warning is displayed.

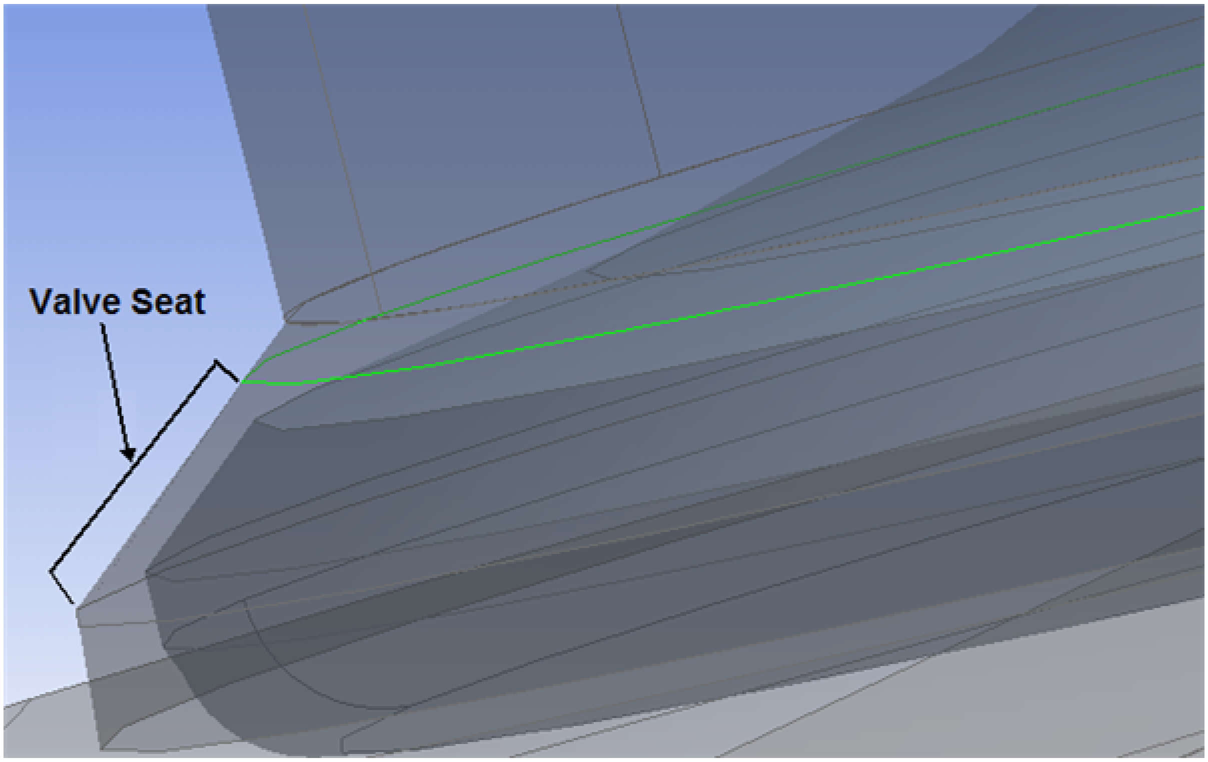
- If there are any small faces causing a meshing failure, then these faces should be merged with their adjacent faces using Virtual Topology. This helps in successful mesh generation.
- If vlayer meshing fails, then you should try to project and imprint the edge of the valve face on the valve seat, in the direction of the valve.

Figure 15.8: Imprint and Project the Edge of the Valve Face

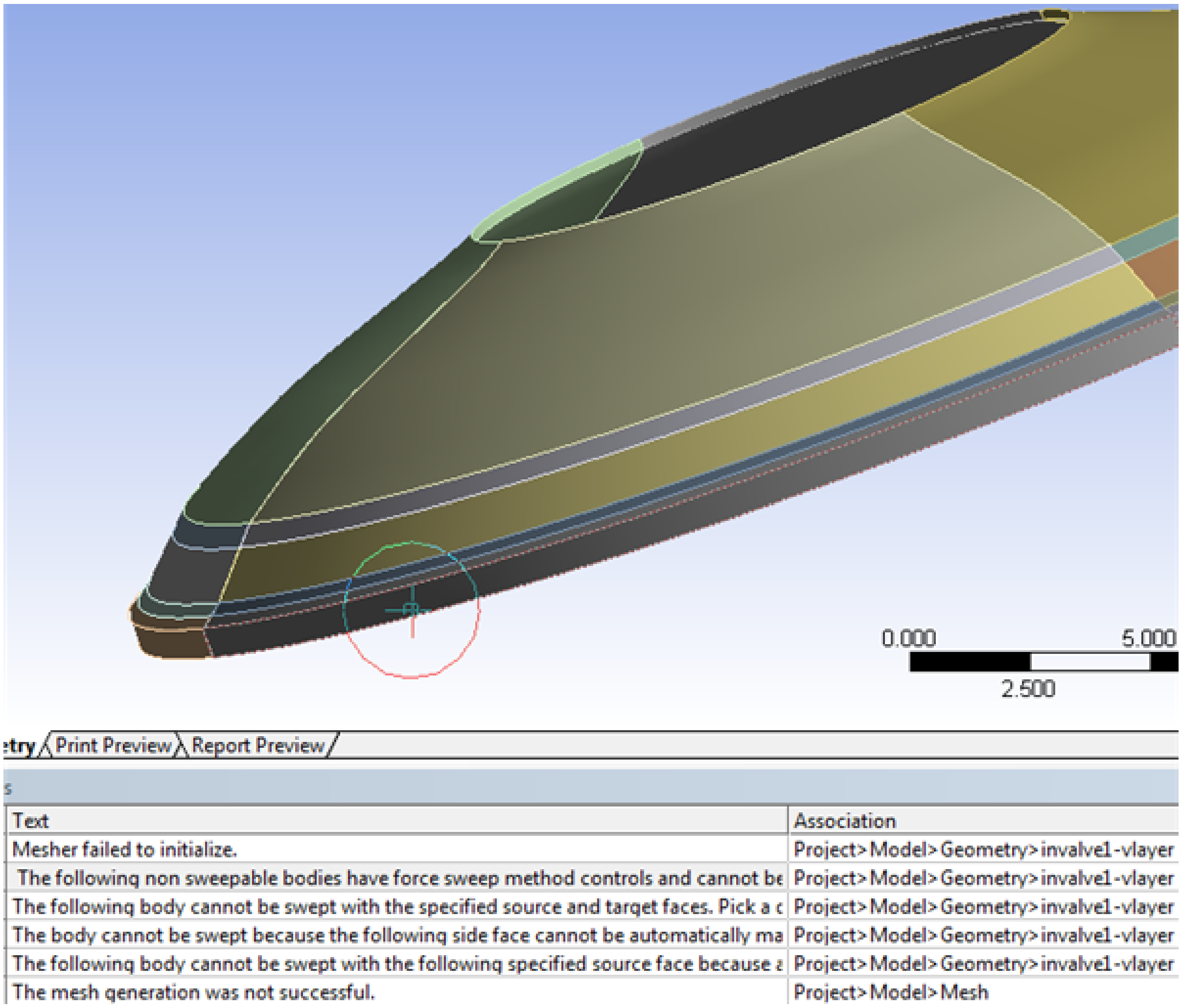


This will split the valve seat. You will have to decompose the geometry again. This procedure will create a proper sweep mesh in the vlayer.

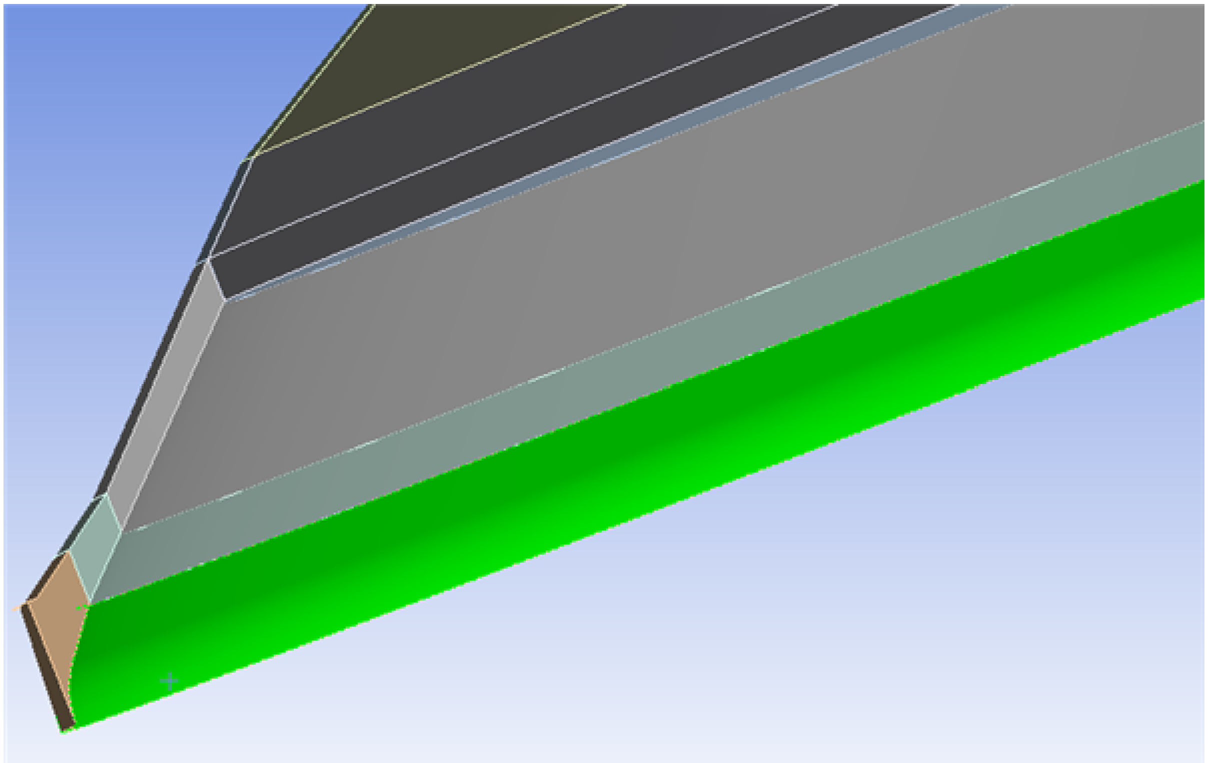
Figure 15.9: Valve Seat Split After Projection and Imprint



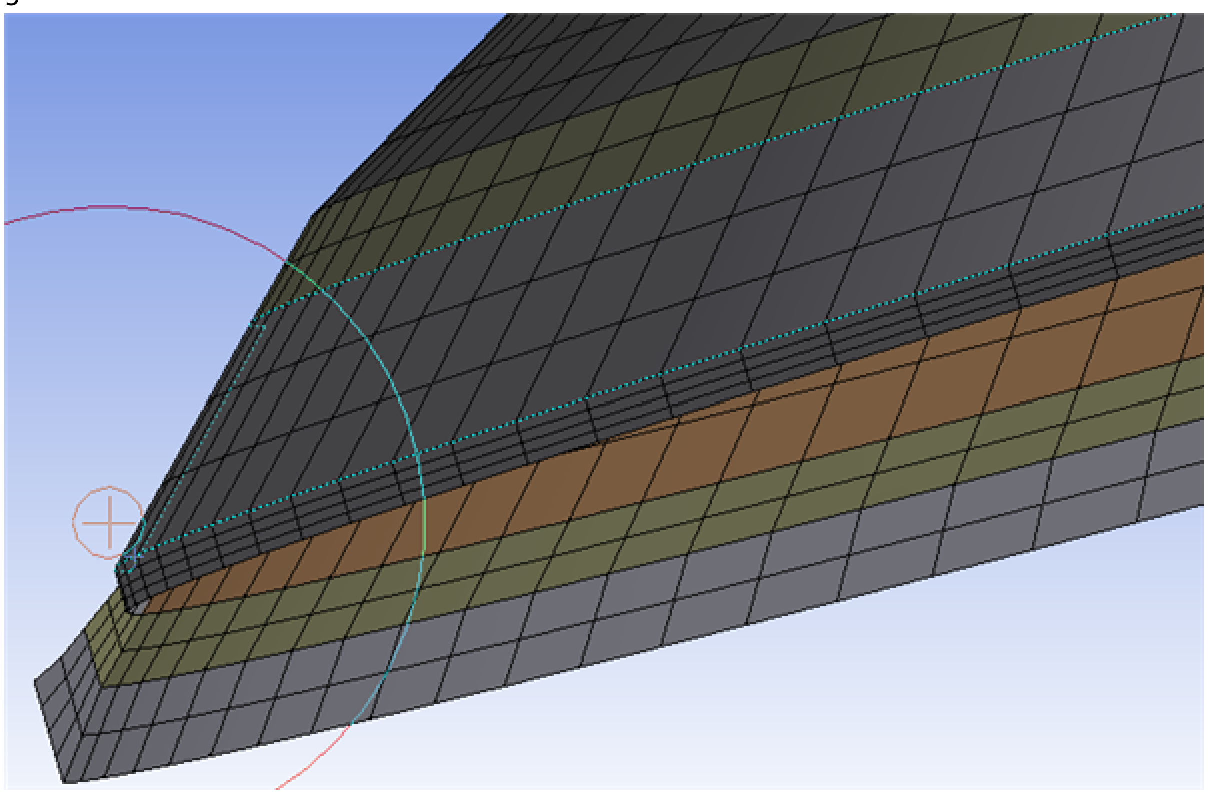
- Sometimes sweep in vlayer meshing fails due to high valve curvature. You might get the error message as shown in the following figure.



The following figure shows the high curvature region where the sweep fails in vlayer.



In such case enable the beta option. This problem will be resolved automatically. The mesh generates without errors.



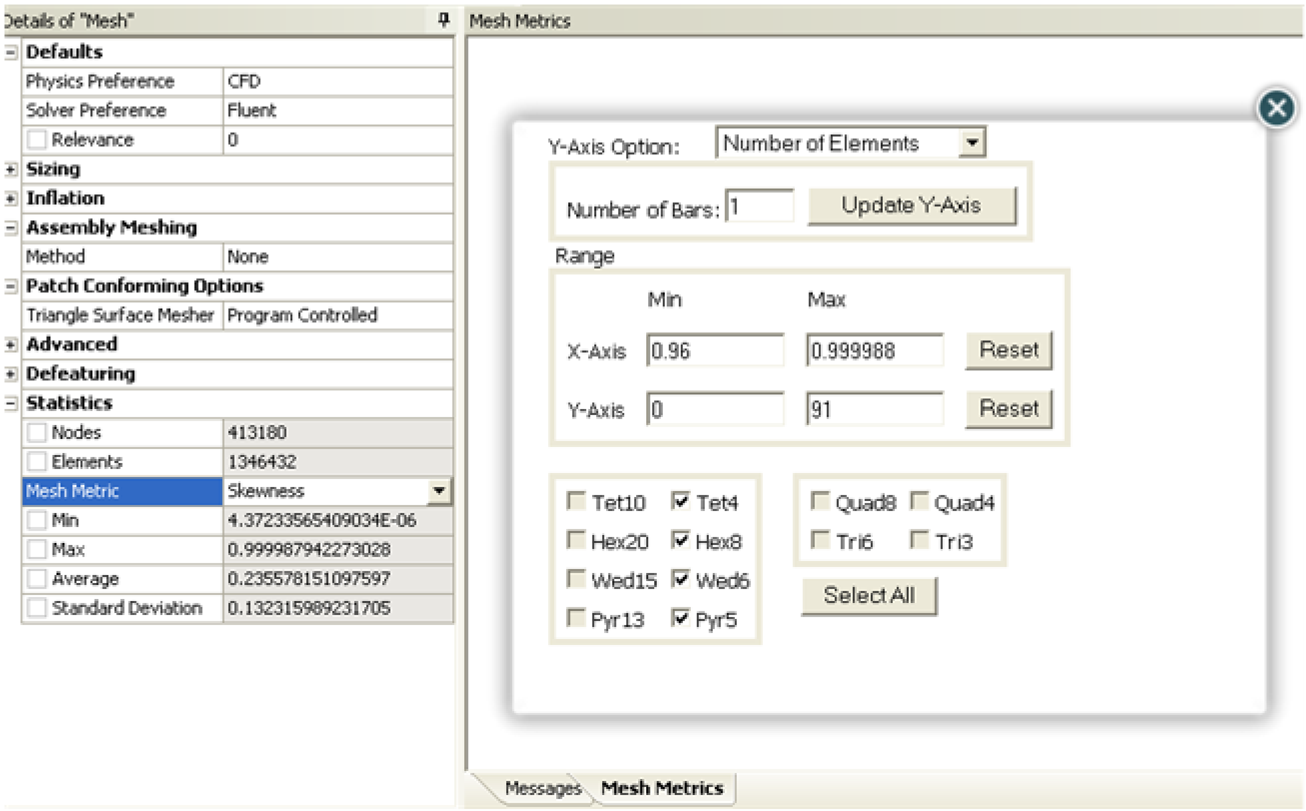
- If the mesh skewness is high, i.e. close to 1, then you can try to improve the mesh by performing the following steps.

  1. Display the area where highly skewed elements are present.

a. Select **Mesh** in the tree **Outline**.

b. In the **Details of "Mesh"** ensure that **Skewness** is selected from the **Mesh Metric** drop-down list under **Statistics**.

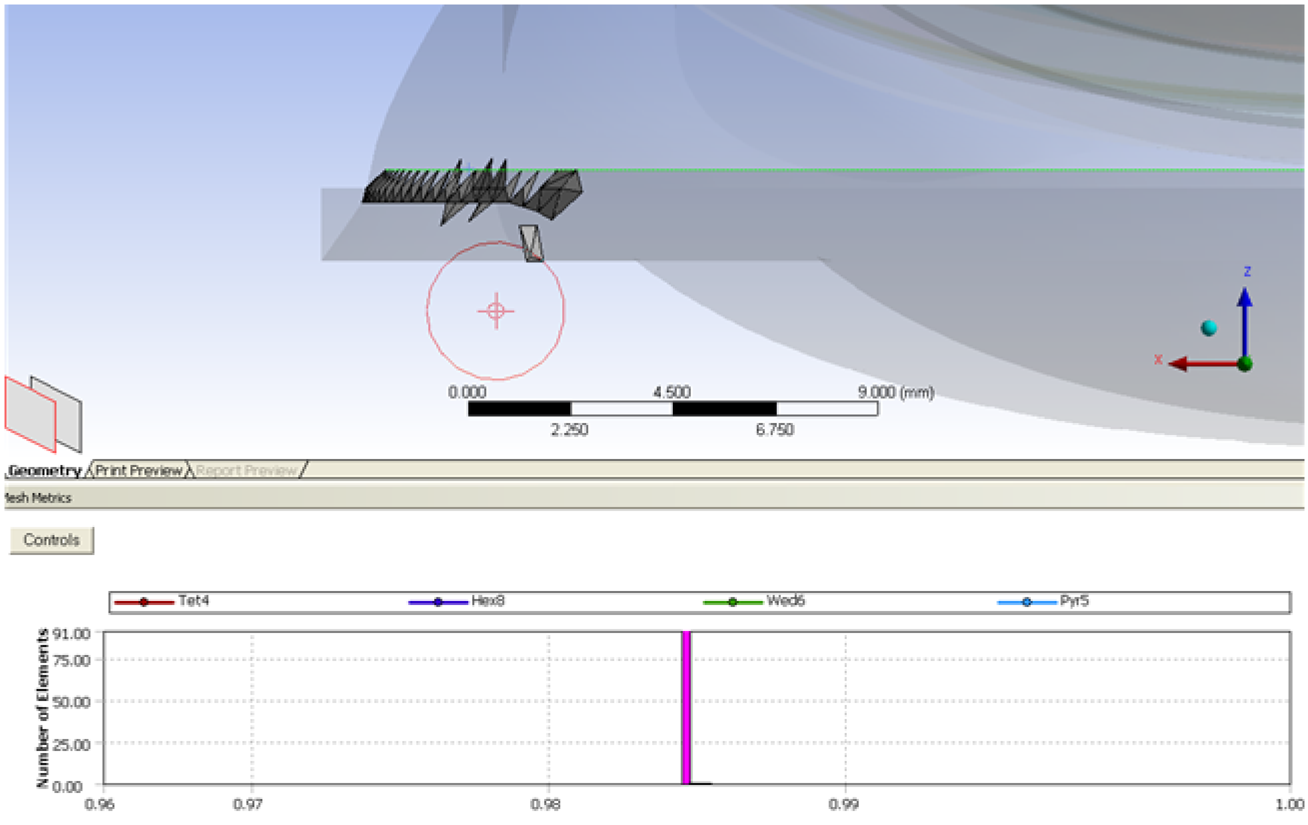
c. In the **Mesh Metrics** dialog box click **Controls**.



d. Enter the **Range** of skewness and close the **Mesh Metrics** dialog box.

e. Select the bars from the bar chart displayed.

All the elements having skewness in the specified range are highlighted in the graphics window in the meshed geometry.



2. To improve the skewed mesh any of the following methods can be used depending upon the geometry.

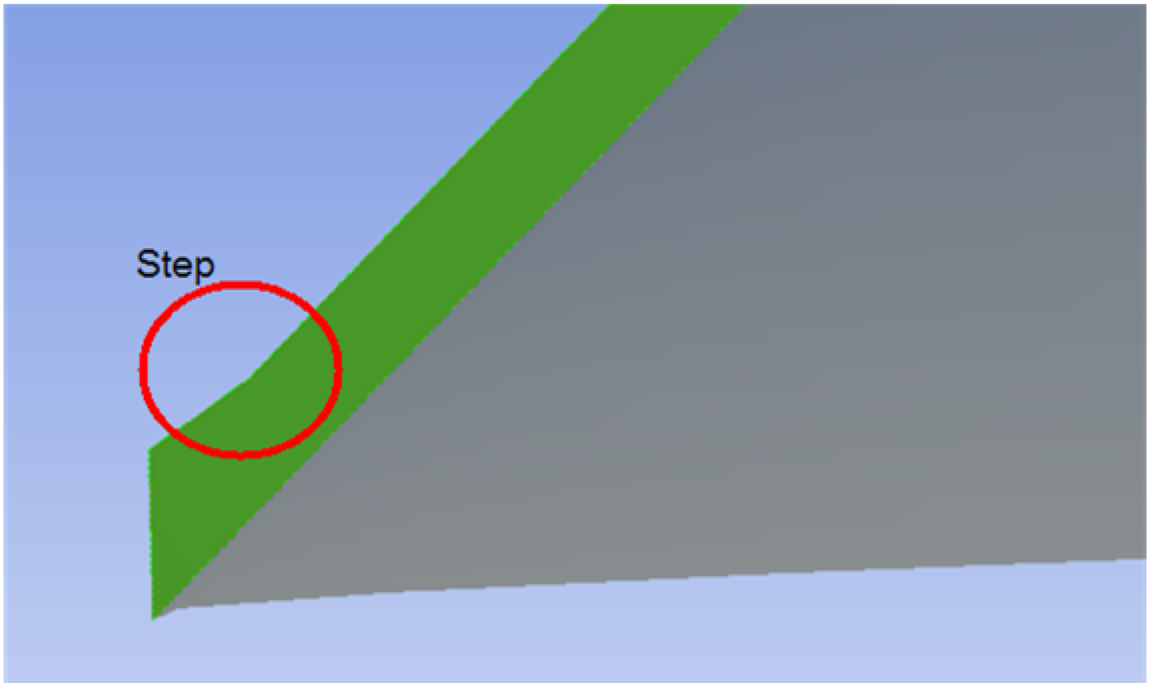
   – If edges/lines are present in the close proximity then you can create new pinch controls or increase the default tolerance for the existing pinch controls.

   – If small faces are present then you can merge them with adjacent faces using Virtual Topology.

   – Sometimes in the chamber body, very small faces, faces at small angles, or faces in close proximity are present, which create skewed elements. To improve this, specify additional hard face sizing controls, or reduce the body sizing control of the chamber, or reduce the curvature angle related to the body sizing control of the chamber mesh control. Reducing body sizing and curvature angle will increase the mesh size. On the other hand, specifying local hard face sizing controls will result in an improved mesh without a substantial increase in mesh size.

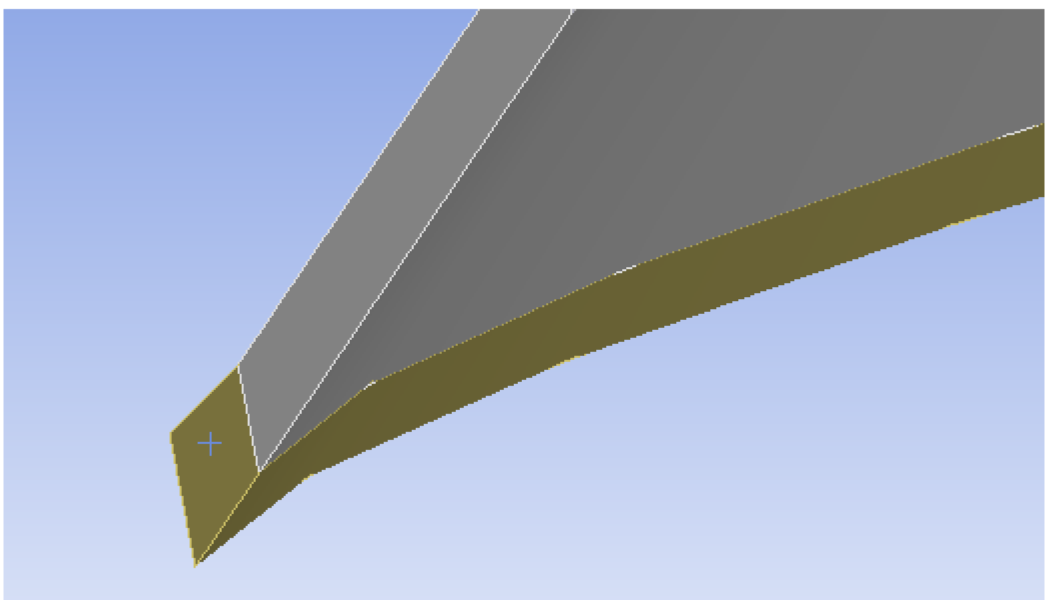
• V-layer meshing can fail in some cases where the face has a step. In such a case select **Show the Problematic Geometry** from the context menu of the error message in the **Messages** window. This will point to the face which has the step. See Figure 15.10: Face Having a Step (p. 539).

Figure 15.10: Face Having a Step

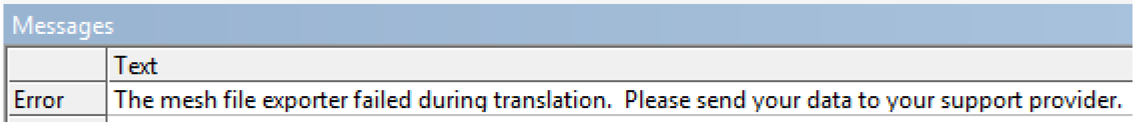


The problem can be corrected by reducing the **V Layer Slice Angle** parameter in the **Input Manager**, such that the face is split into two as shown in Figure 15.11: Face Split Into Two (p. 539). You should then reset the Mesh cell and follow the meshing procedure to re-mesh the geometry.

Figure 15.11: Face Split Into Two

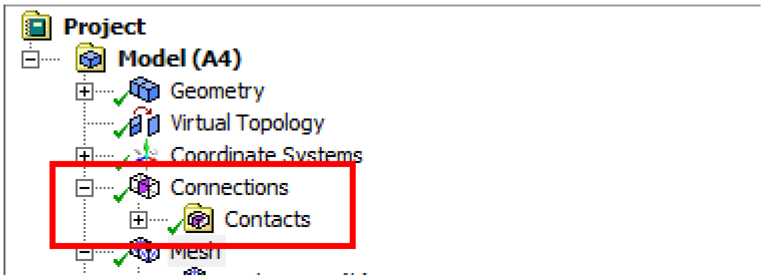


- If an IC Engine project of older version is opened in the latest version then, during Mesh cell update it may give the following error.

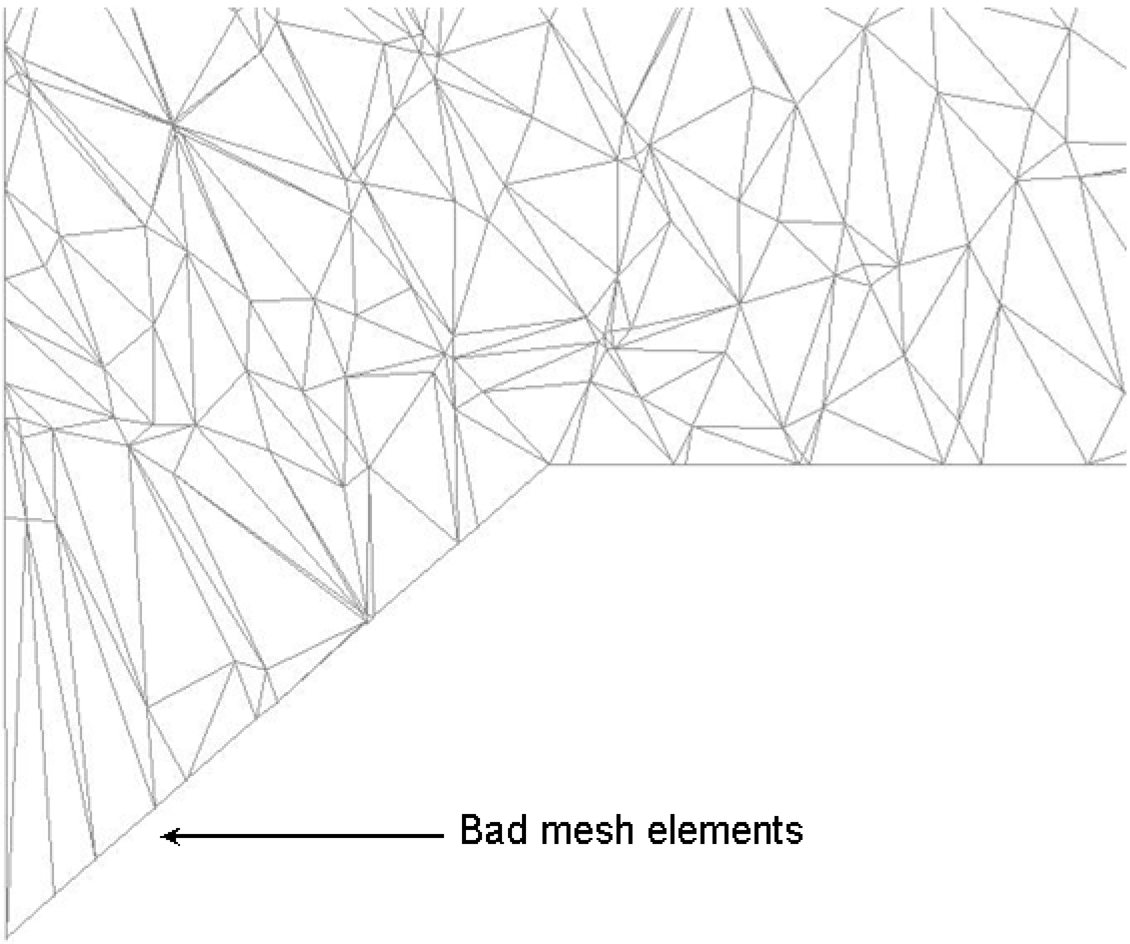


Messages	
	Text
Error	The mesh file exporter failed during translation. Please send your data to your support provider.

To avoid this error you should delete the **Contacts** group under **Connections** in the tree of Meshing application.



- If there are sharp corners between piston and liner or in valve pockets of the piston, then these can cause bad mesh elements during mesh motion. To prevent such issues you can create partial crevices.



15.4. KeyGrid Troubleshooting in IC Engine

- During Keygrid generation if you get an **Attention Required** ( ) cell state for **ICE Solver Setup** cell then mesh for one of the given crank angles has failed. In such a case you can correct the mesh as given in the point below (p. 541). If you want to continue with further computation you can also move the failed angle(s) to the **Inactive Angles** list.

- During Keygrid generation if mesh generation fails for any angle, set that angle as the default angle by selecting **Set Default Angle** from the context menu, which you get by right mouse click. Check the error message.
  - If the message says that the geometry failed, then open DesignModeler.
    1. Decompose the geometry once more.
    2. **Reset** the **Mesh** cell in the **Project Schematic**.
    3. Open Ansys Meshing by double-clicking the **Mesh** cell.
    4. Set the mesh parameters.
    5. Generate the mesh.
  - If the message says that mesh failed, then try the following steps.
    1. Open Ansys Meshing and check the meshing errors. See if you can rectify them. If you cannot close Ansys Meshing.
    2. **Reset** the **Mesh** cell in the **Project Schematic**.
    3. Open Ansys Meshing by double-clicking the **Mesh** cell.
    4. Set the mesh parameters.
    5. Generate the mesh.
- While browsing mesh in KeyGrid, valve names and zone names must be consistent with upstream data. Else after mesh replace valve-ch is swapped leading to negative volume error. So always remember the order of selection of the valve bodies.

15.5. Solver Troubleshooting in IC Engine

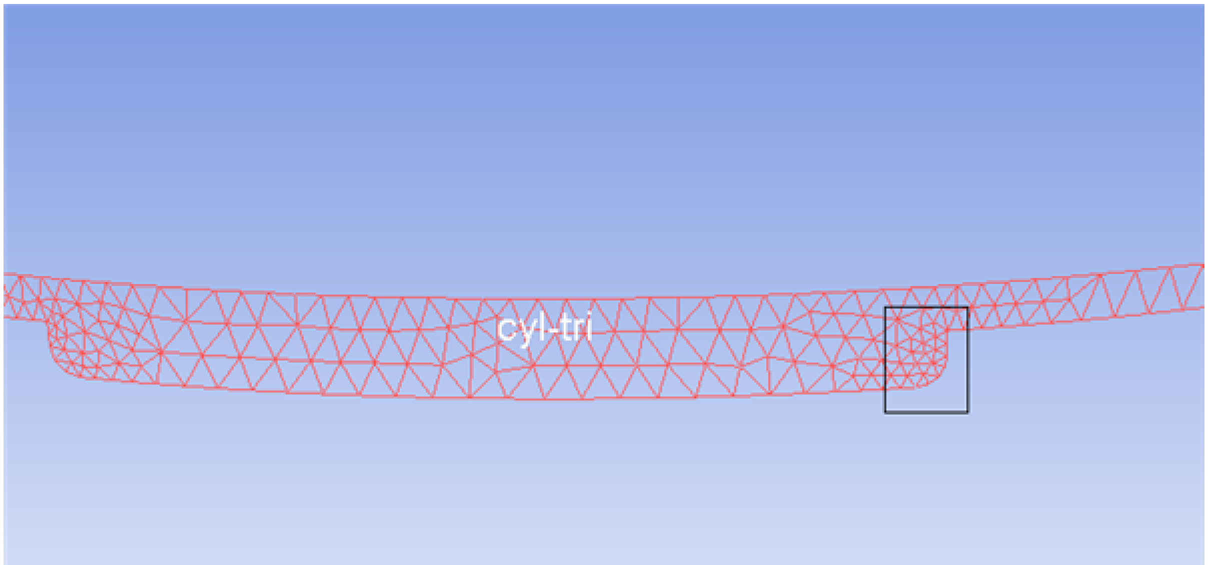
- If you want to use the case file generated by IC Engine system and run the simulation in standalone Fluent, you should read the scheme file (`WB-ICE-Solver-Setup.scm`) after you read the case file in Fluent. This is mandatory. Else you will get some errors while running.

  Important:

  The report (**Report.html**) may not be properly generated if you run the simulation in standalone Fluent.

- In the cold flow simulation case, if the `cyl-tri` or `symm-cyl-tri` face zone contains a curvature then it might result in smaller elements at the curve. See Figure 15.12: Curvature in cyl-tri (p. 542).

Figure 15.12: Curvature in cyl-tri

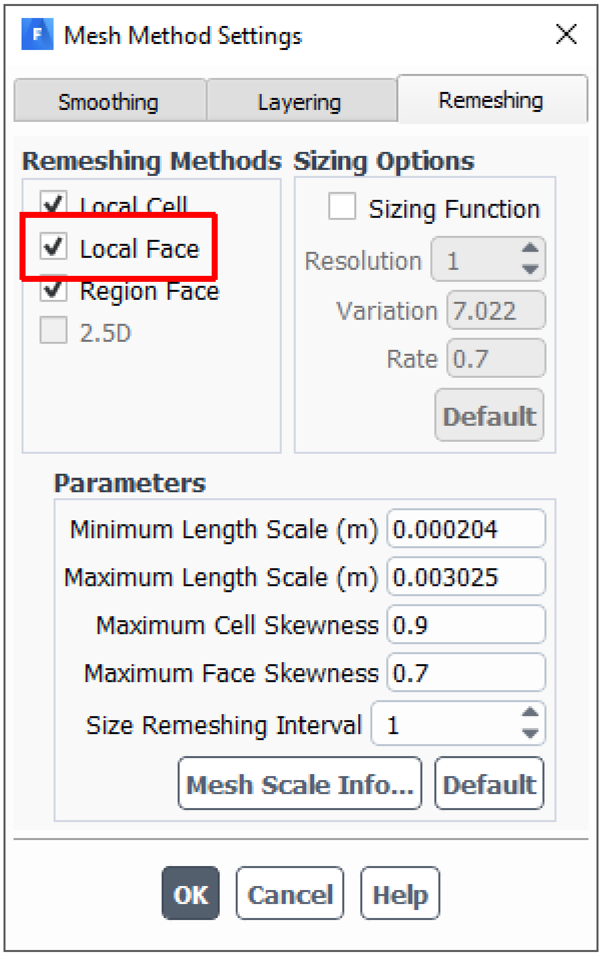


In such cases if you are facing “Negative Volume Cell" errors while running the mesh motion or computation, then enabling **Local Face** remeshing option both globally and locally for those face zones will improve the solver stability. To enable local face remeshing in Fluent:

1. Go to **Dynamic Mesh** in the navigation pane.

   a. Click **Settings....** under **Mesh Methods**.

   b. In the **Mesh Method Settings** dialog box click the **Remeshing** tab and enable **Local Face**.

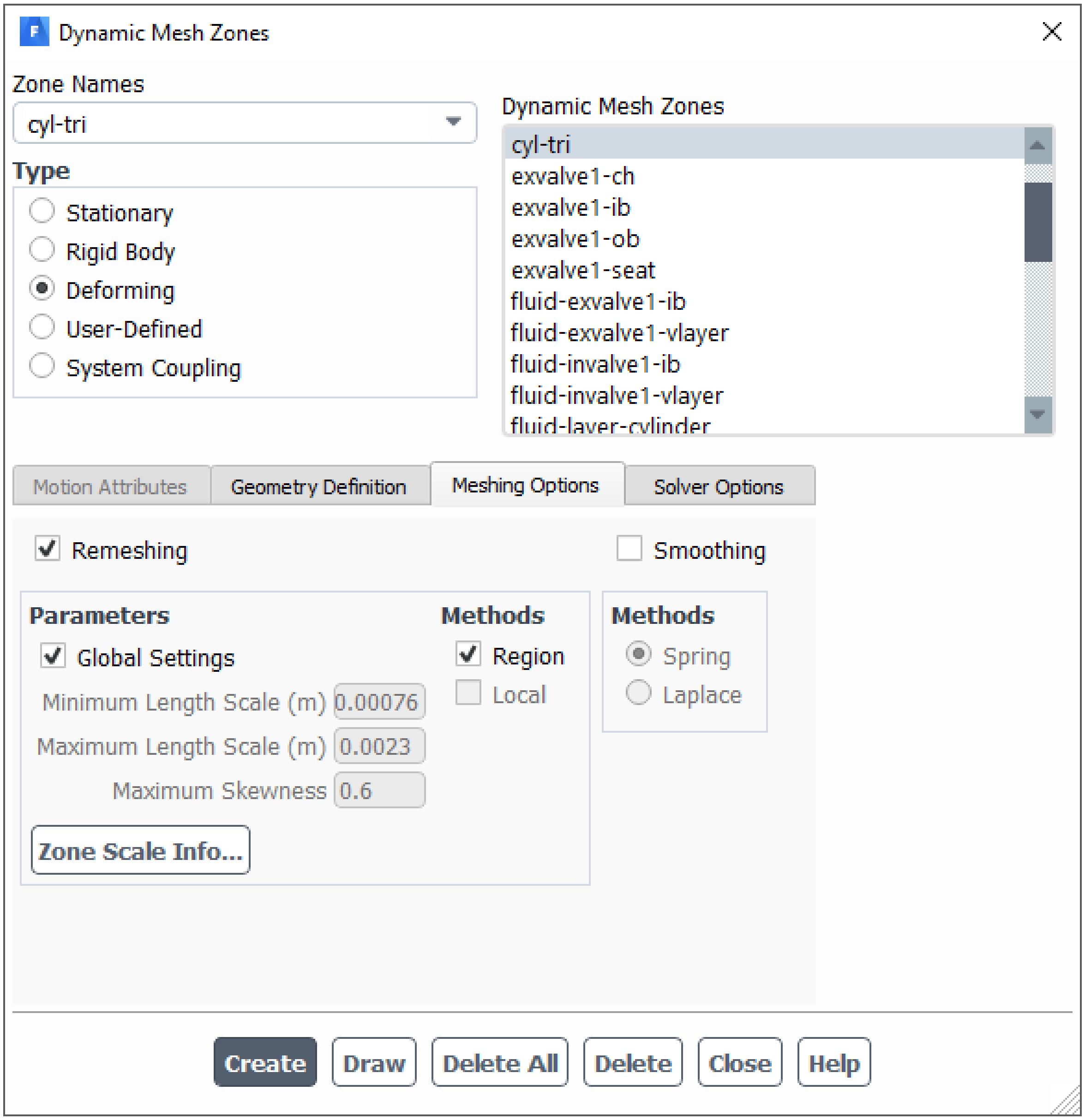


c. Click **OK** and close the dialog box.

2. Select the `cyl-tri` and `symm-cyl-tri` (in case of symmetry) face zone from the **Dynamic Mesh Zones** list and click **Create/Edit...**.

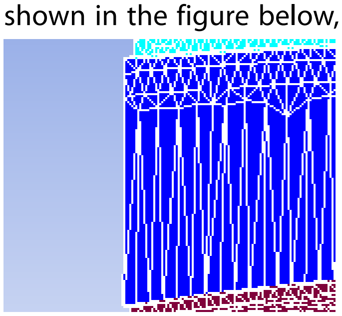
   a. In the **Dynamic Mesh Zones** dialog box click the **Meshing Options** tab.

   b. Enable **Local** in the **Remeshing Methods** group box.



c. Click **Create** and close the **Dynamic Mesh Zones** dialog box.

- In some cold flow or combustion cases cyl-tri and symm-cyl-tri mesh might be stretched as shown in the figure below, due to failure of remeshing in cyl-tri and symm-cyl-tri.

  This is mainly because of the small kinks in the piston faces. To avoid this problem, you need to enable the following options manually:

- Enable the "Local Face" remeshing both globally as well as locally for cyl-tri and symm-cyl-tri zones as shown in troubleshooting point above (p. 541).

  Note:

  You cannot enable Local Face remeshing in all cases because it affects the cycle to cycle mesh repeatability.

- Enable zone-remeshing through following TUI:

  ```
  /define/dynamic-mesh/controls/remeshing-parameters/zone-remeshing
  yes
  ```

  Note:

  You cannot enable zone remeshing for all cases — In parallel, the zone remeshing will automatically migrate all elements to be remeshed to a single CPU. Consequently, the machine memory will limit the size of the zone that can be remeshed.

- In case you get a negative volume error during computation —
  - Check the mesh. If you observe that the zone motion is not happening properly, then in the DesignModeler check that the named selections are having proper supports, especially near the chamber and piston areas.
  - If you have used Kegrids, check that the replaced mesh has the same order of selection of valve bodies as per the originally decomposed mesh. Else, after mesh replace valve-ch is swapped, leading to negative volume.
- When you select **Crank Angle** from the **Auto Save Type** drop-down list in the **Solver Settings** dialog box, then **Auto Save Frequency** is set to some value, say "X". In Fluent **Autosave Every (Time Steps)** in the **Calculation Activities** task page is set to **1**, but the case and data are saved at the frequency "X" which you have set.

  If you take the ICEngine system generated case file to standalone Fluent and run the simulation without reading the ICEngine scheme file (`WB-ICE-Solver-Setup.scm`) then the case and data will be saved at every time step. To avoid this problem and to make sure that case and data get saved at requested frequency "X" (**Auto Save Frequency**), you will have to read ICEngine scheme file `WB-ICE-Solver-Setup.scm` before reading the case file in standalone Fluent. If this is not done then the case and data files will be saved on the frequency set in **Autosave Every (Time Steps)**, which is 1 by default.
- During remeshing if you observe denser cells near piston regions while the piston is moving upwards or skewness is increasing, then you need to increase the **Number of Iterations** in the **Smoothing** tab of the **Mesh Method Settings** dialog box in Fluent. This will help in improving the mesh, but it might increase the mesh motion time. It has been observed that values between 50 to 150 for **Number of Iterations** are more effective.

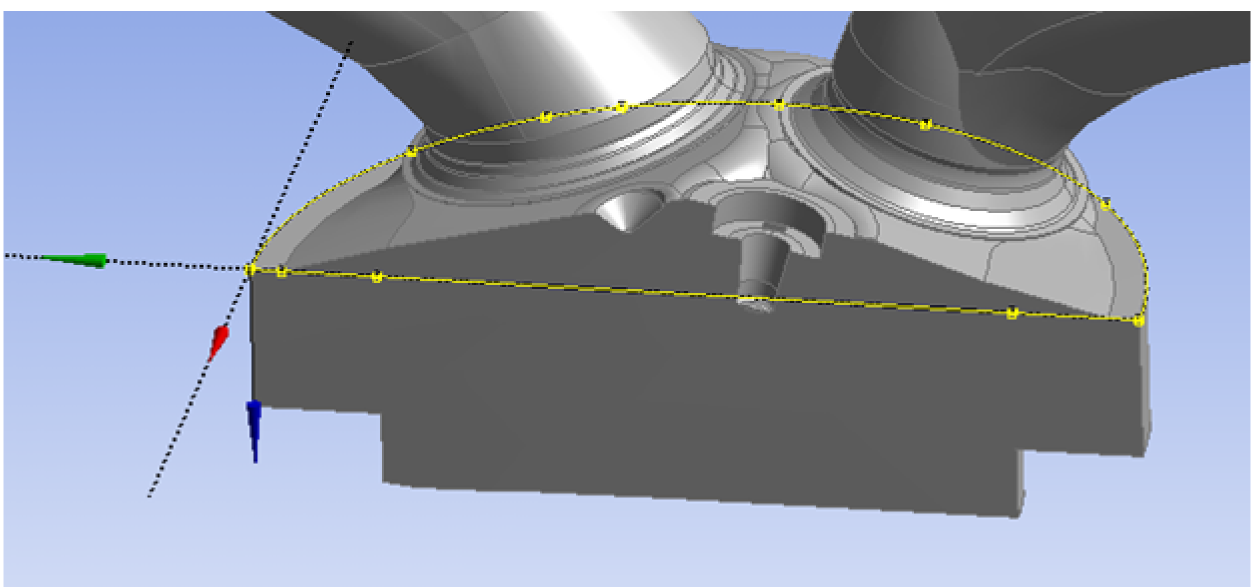
Chapter 16: Customization and Improvements

This chapter has some topics on customizing and improving the simulation.

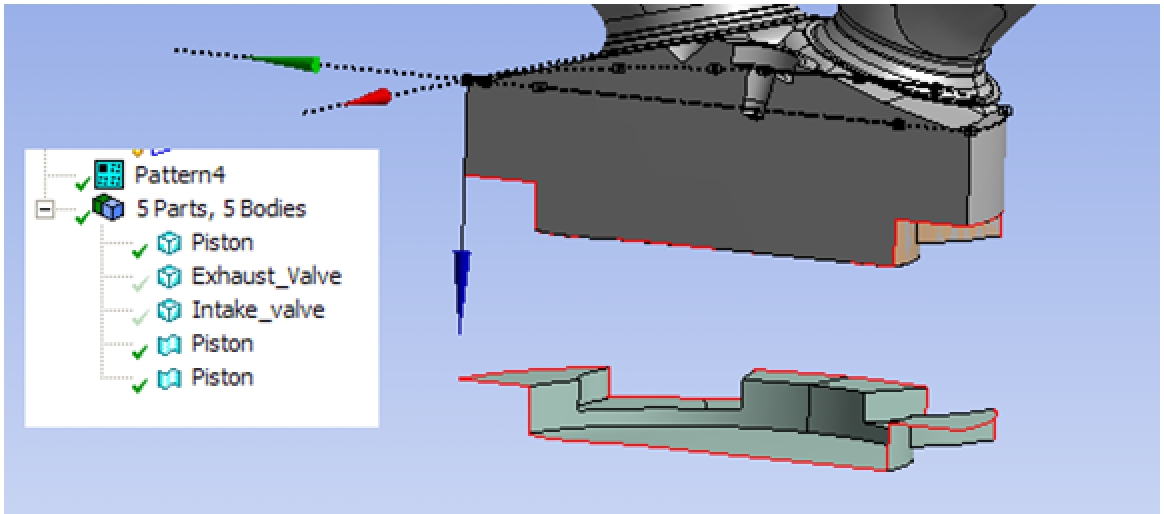
16.1. How IC Engine System Moves the Piston to the Specified Crank Angle

When you enter a value for the **Decomposition Crank Angle** in the **Input Manager**, the IC Engine System moves the piston to that specified angle. The actions performed internally are as follows:

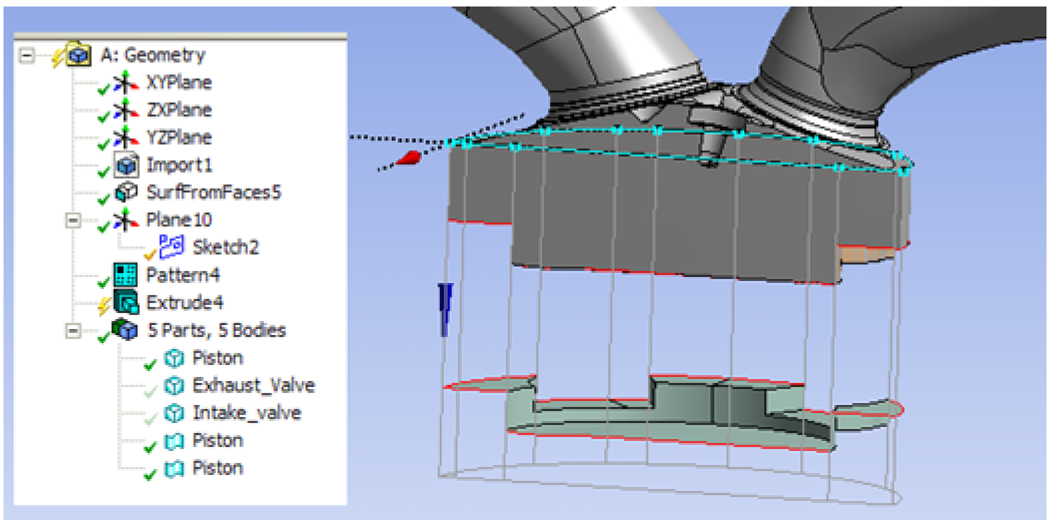
- A plane is created above the piston along the line of cylinder axis. By inserting **Sketch Projection** and selecting the border edges of the piston surface a projected profile is created.



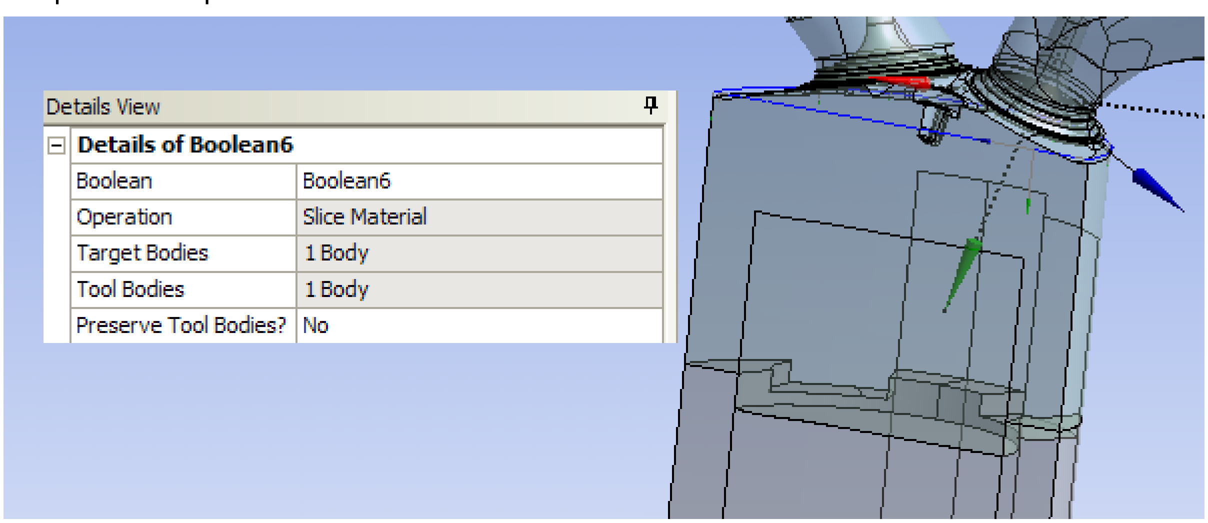
The **Pattern** feature is used to translate the piston surface.



- Extrude the named selection by a distance more than that of the piston bottom position.



The extruded body is then sliced with the piston surface. Deleting the top body and uniting the bottom body with the port body and then, slicing the united body with the bottom piston surface, completes the operation.

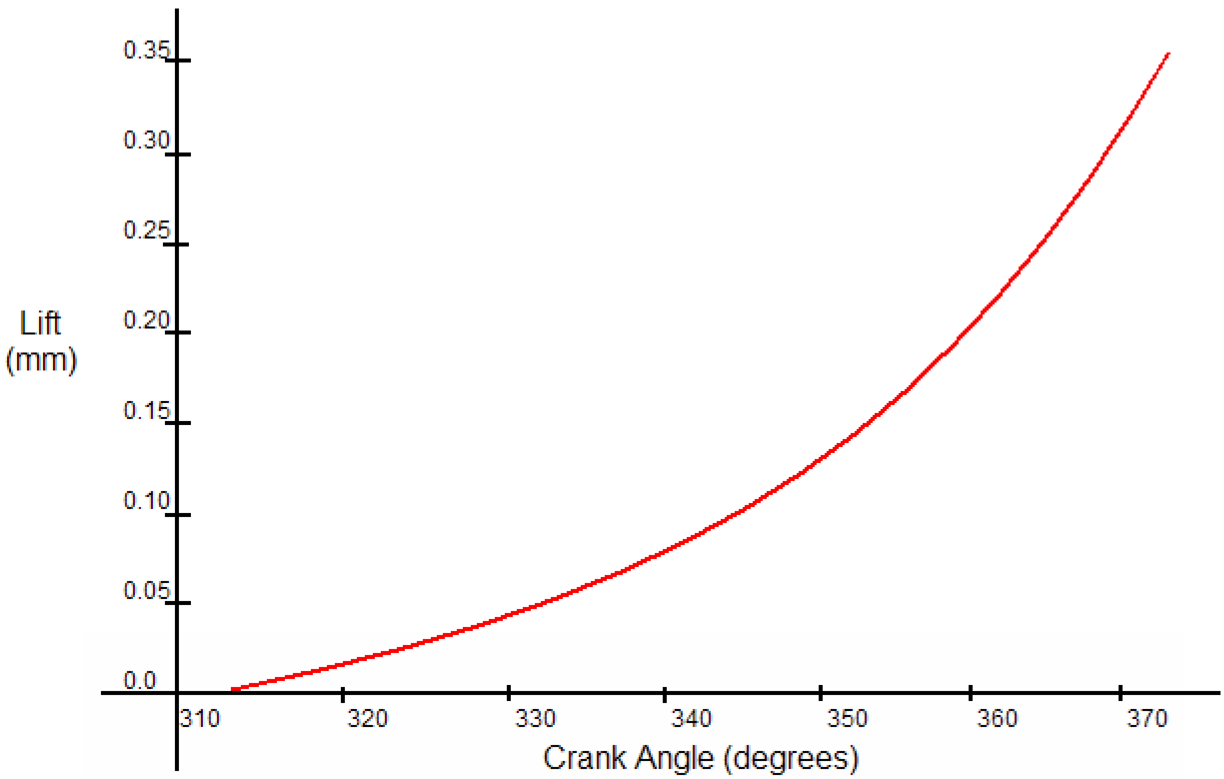


16.2. How IC Engine System Calculates Valve Opening and Closing Angles

The valve opening and closing angles are internally calculated in such a way that the flow to the chamber is neither under-predicted nor over-predicted. As there is a minimum lift for the valve, the actual valve opening will allow too much of flow into the chamber.

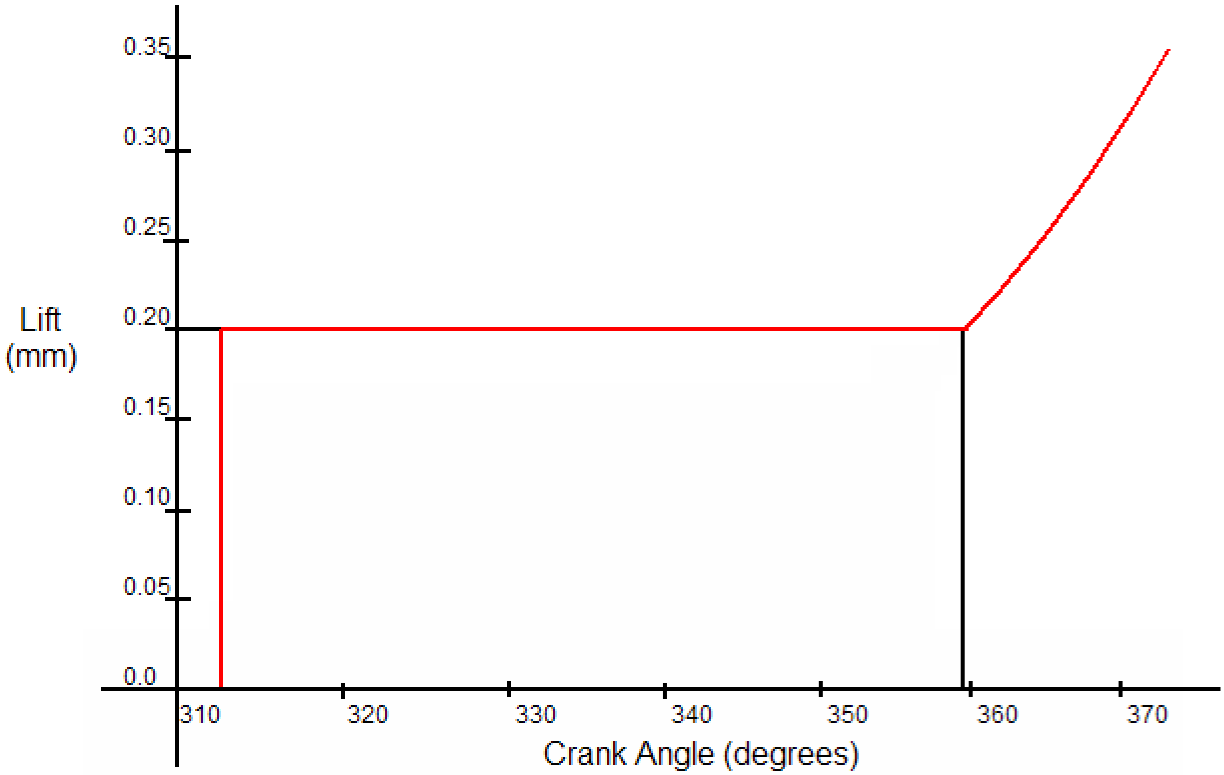
According to the profile and in practice the flow will vary according to the lift profile. For example in the following figure the valve opens at 312° crank angle (CA) according to the profile. At about 360° CA the valve lift will reach 0.20 mm.

Figure 16.1: Actual Valve Opening Profile



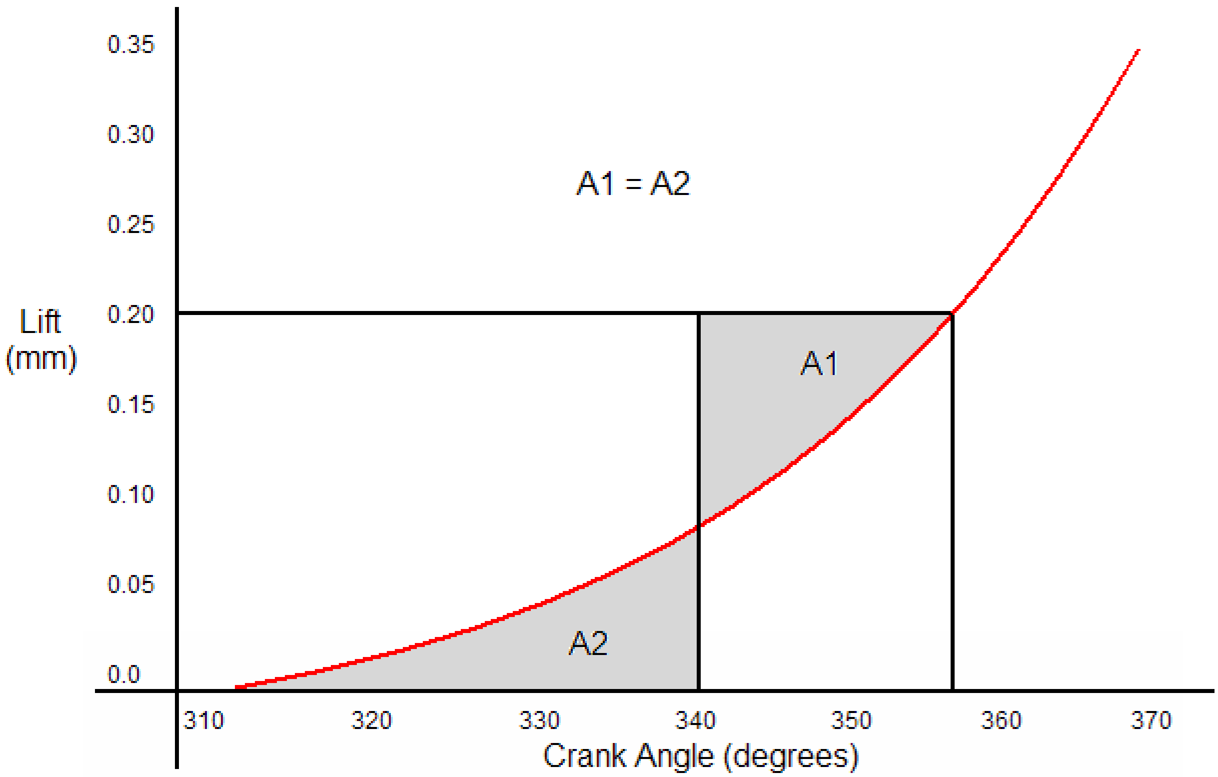
If you specify **Minimum Lift** as `0.20` then the valve will open at 312°, and the simulated valve profile will be as shown in Figure 16.2: Valve Opened at CA 312° (p. 550), and the flow will be over-predicted. Hence the valve opening and closing angles are modified.

Figure 16.2: Valve Opened at CA 312°



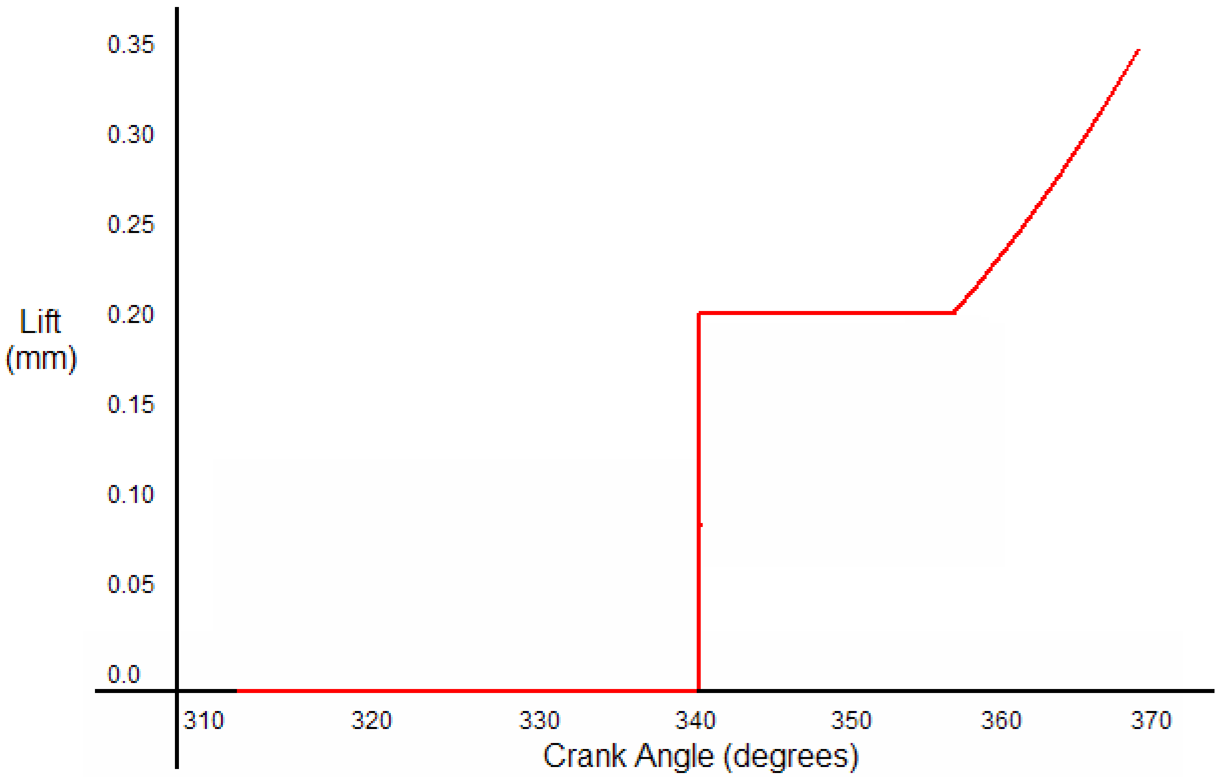
The valve opening and closing angles are calculated using the trapezoidal method of numerical integration. As shown in Figure 16.3: Mass Flow Value Equivalent to Variation of the Curve (p. 551), an average value of CA is calculated when the mass flow value of A1 and A2 are equivalent. Here it comes as 340°.

Figure 16.3: Mass Flow Value Equivalent to Variation of the Curve



With the calculated value, the simulated valve lift profile is as shown in Figure 16.4: Simulated Valve Lift Profile (p. 552).

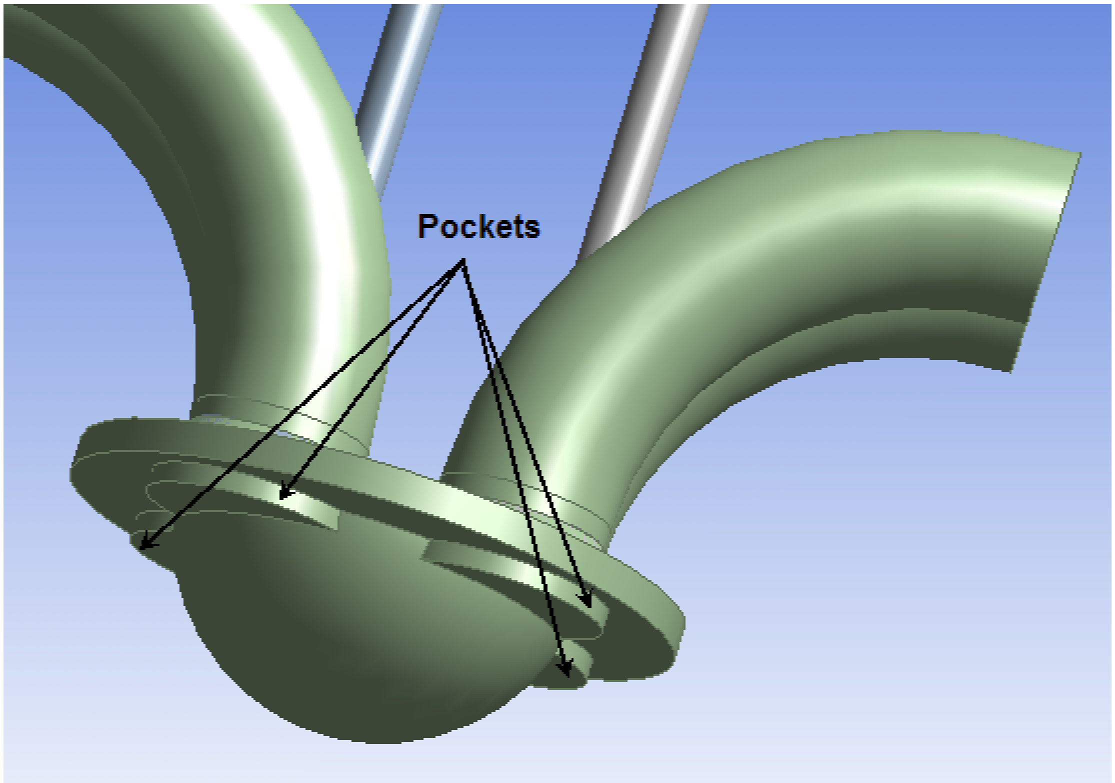
Figure 16.4: Simulated Valve Lift Profile



16.3. Decomposing a Straight Valve Pocket Engine

Figure 16.5: Straight Valve Engine with Pockets (p. 553) shows a straight valve engine with valve pockets. For decomposition of a straight valve pocket engine you will have to do some changes manually. Refer to Straight Valve Geometry With Chamber Decomposition for **IC Engine** (p. 167) for the nomenclature of the decomposed geometry.

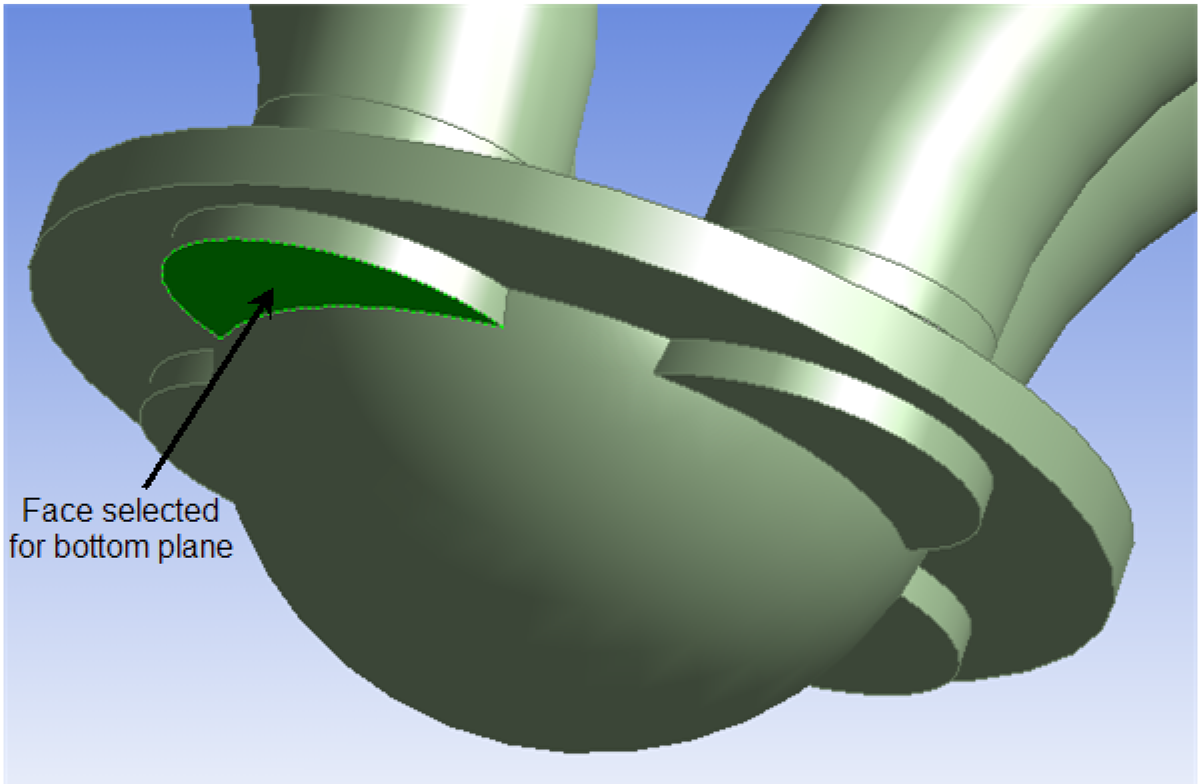
Figure 16.5: Straight Valve Engine with Pockets



Perform the following steps:

1. In the **Input Manager** ensure that **Yes** is selected for the **Decompose Chamber** option.
2. Select **Manual** from the **IC Plane Insert Option**.
3. Select the bottom plane such that the pockets will come under the **ch-lower** body as shown in the Figure 16.6: Bottom Plane Face (p. 554).

Figure 16.6: Bottom Plane Face

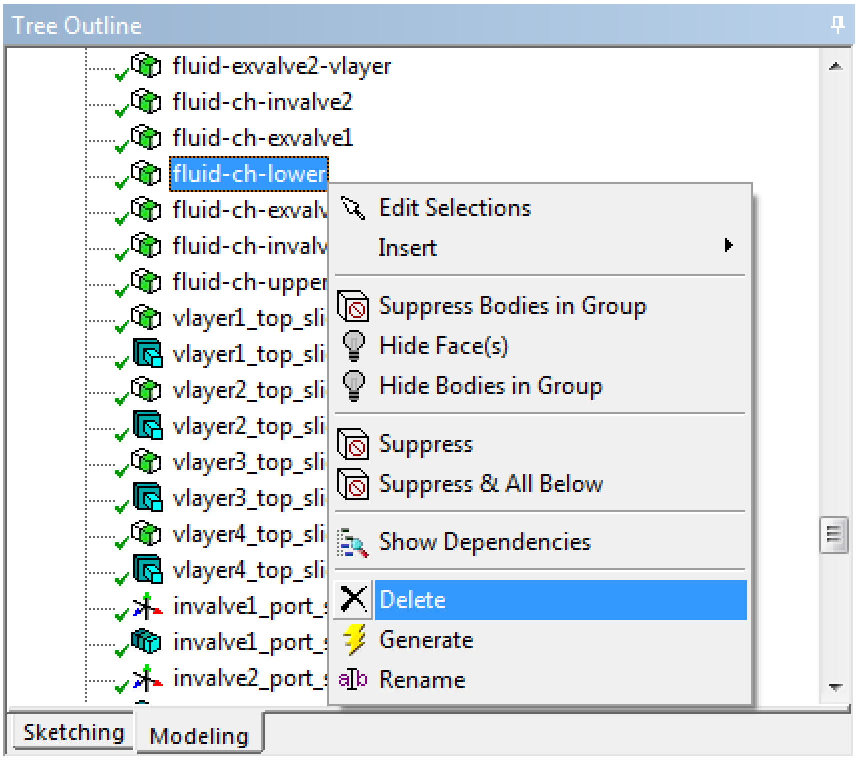


4. Ensure that you have entered all the required data in the **Input Manager** and then decompose the geometry.

5. Disable **Share Topology**.

6. After decomposition you will have to slice the pockets from the **ch-lower** body and unite it with the piston body. For slicing perform the following steps:

    a. Delete the named selection **fluid-ch-lower** from the tree.



b. Click **Slice** 

.

c. Select **Slice by Surface** from the **Slice Type** drop-down list.

Note:

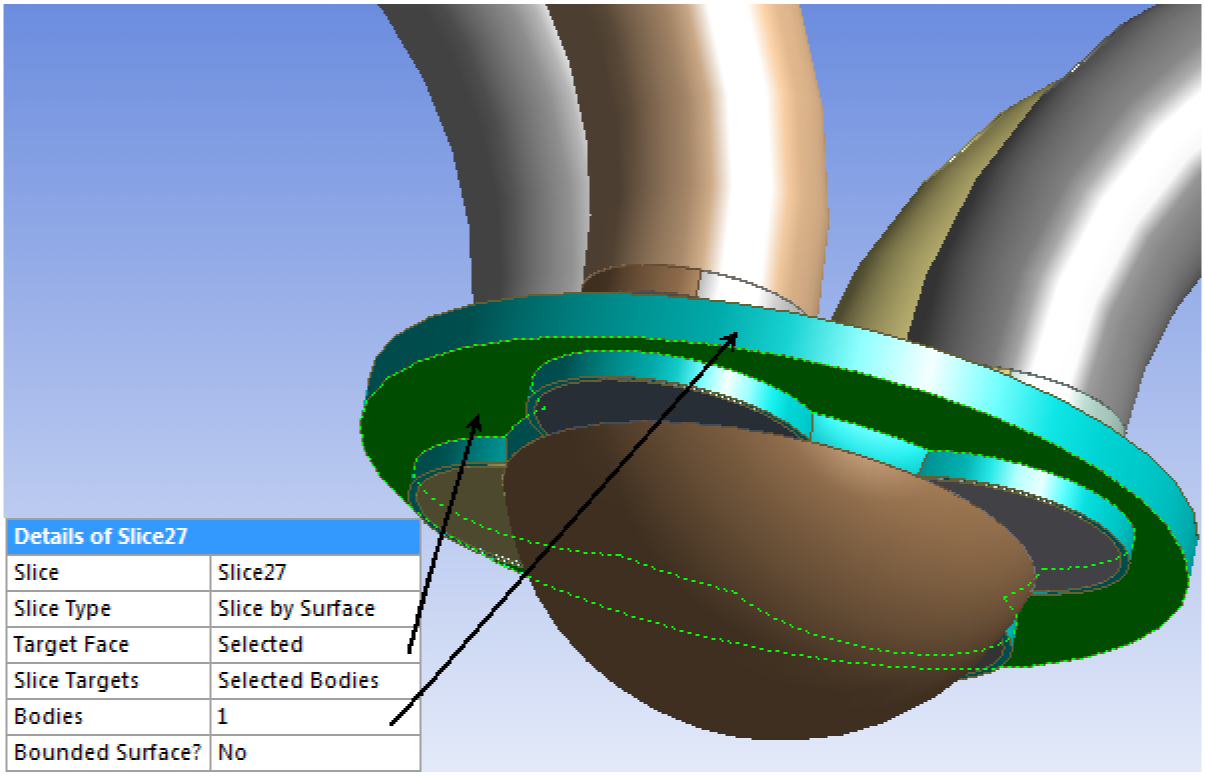
In some cases sharp corners are generated in the piston body by using **Slice by Surface**. To avoid this create a plane slightly above the surface highlighted in Figure 16.7: Slicing (p. 556). Then select **Slice by Plane** from the **Slice Type** drop-down list.

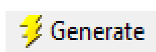
---

d. Select **Selected Bodies** from the **Slice Targets** drop-down list.

e. Select the **ch-lower** body for **Bodies**.

Figure 16.7: Slicing



f. Click **Generate** 

.

7. For uniting perform the following steps:

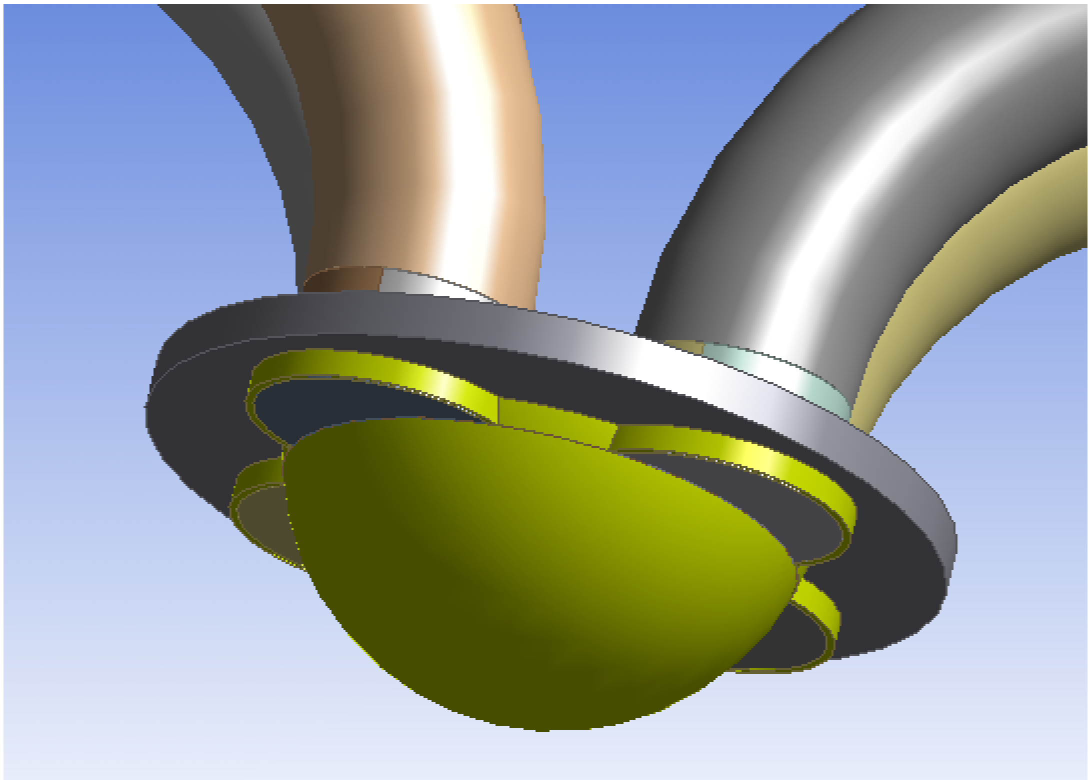
   a. Select **Boolean** from the **Create** menu.

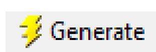
      Create → **Boolean**

   b. Ensure that **Unite** is selected from the **Operation** drop-down list.

   c. For **Total Bodies** select the piston body and the body formed by the pockets.

Figure 16.8: Piston Body and Body Formed by Pockets

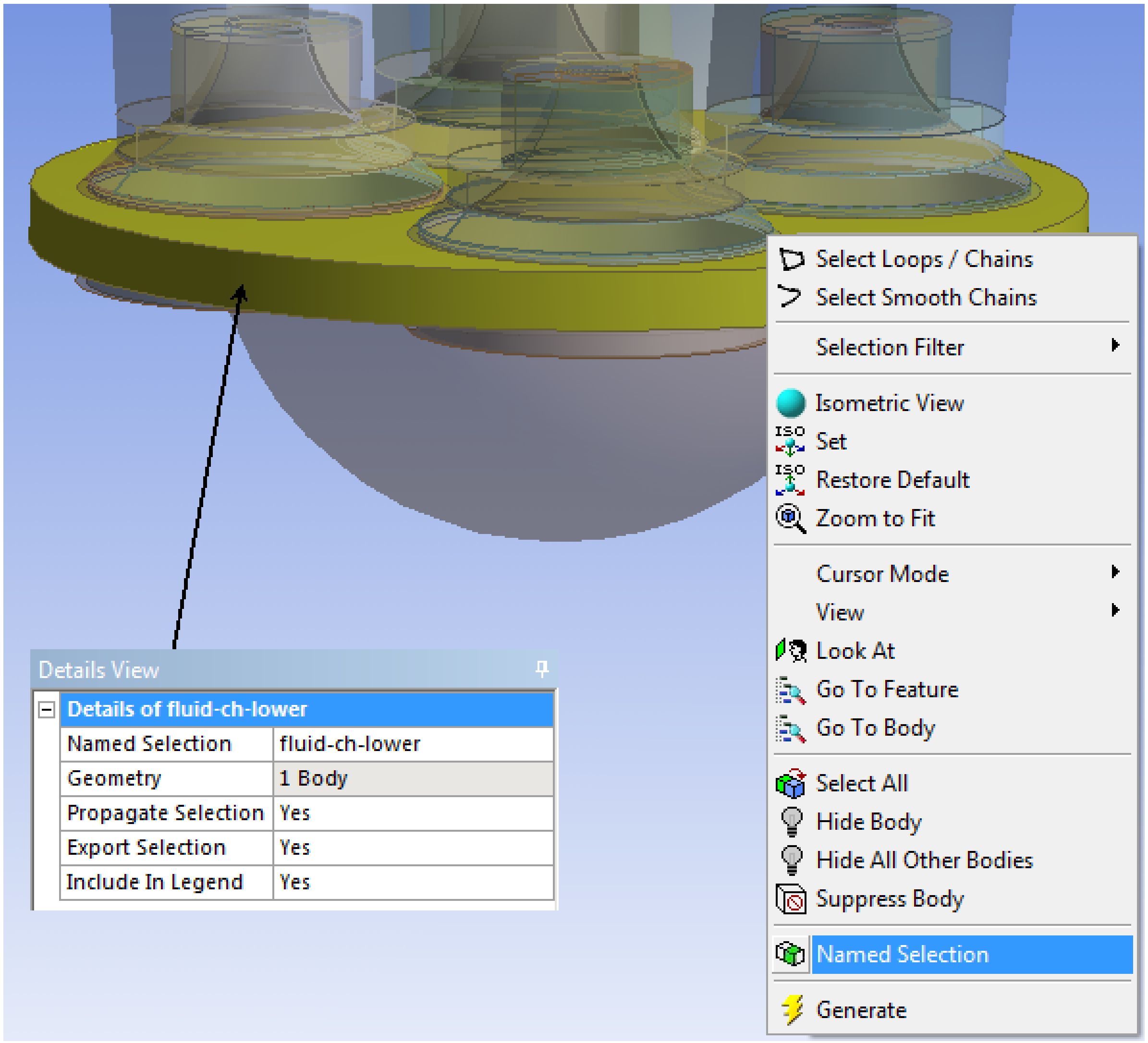


d. Click **Generate** 

.

e. Create the named selection **fluid-ch-lower** by selecting the body.

Figure 16.9: Create named selection fluid-ch-lower

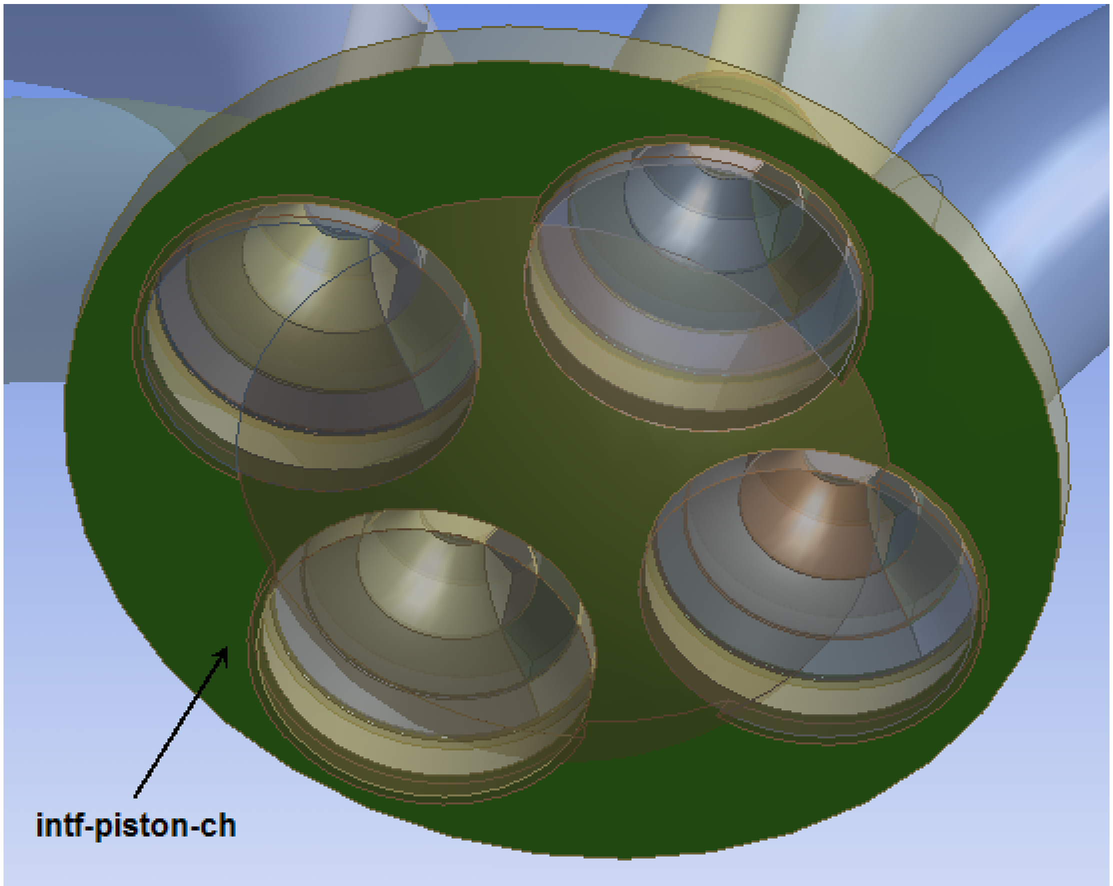


8. Redefine the **intf-piston-ch** interface with proper faces, which are all the faces of **ch-lower** body and interface of piston body:

   a. Delete the named selection **intf-piston-ch** from the tree.

   b. Select the proper face as shown in the Figure 16.10: Create Named Selection intf-piston-ch (p. 559) and then create the named selection with the name **intf-piston-ch**.

Figure 16.10: Create Named Selection intf-piston-ch

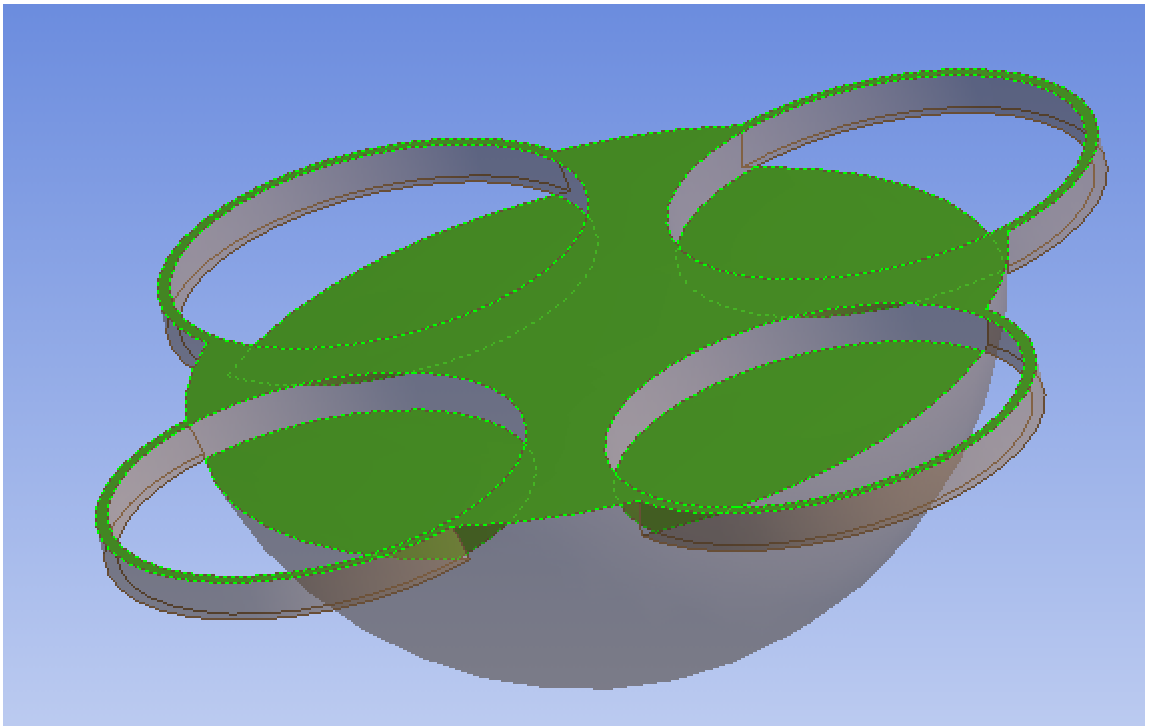


9. Redefine the **intf-piston-bowl** interface with proper faces, which are all the faces of piston body and interface of **ch-lower** body and **ch-valveID** body.

    a. Delete the named selection **intf-piston-bowl** from the tree.

    b. Select the proper faces as shown in the Figure 16.11: Create Named Selection intf-piston-bowl (p. 560) and then create the named selection with the name **intf-piston-bowl**.

Figure 16.11: Create Named Selection intf-piston-bowl

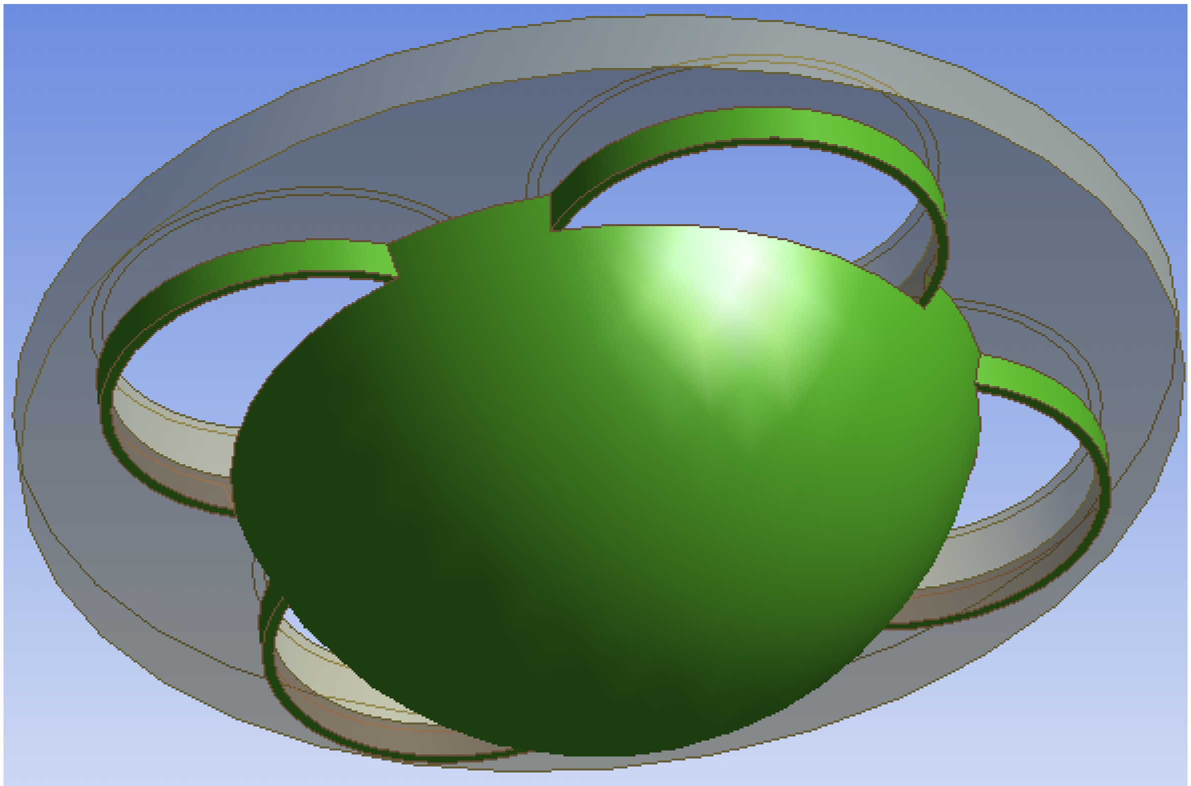


10. Redefine all piston faces, which are supposed to be given rigid body motion in Fluent.

    a. Delete the named selection **piston** from the tree.

    b. Select the proper faces as shown in the Figure 16.12: Create Named Selection piston (p. 561) and then create the named selection with the name **piston**.

Figure 16.12: Create Named Selection piston

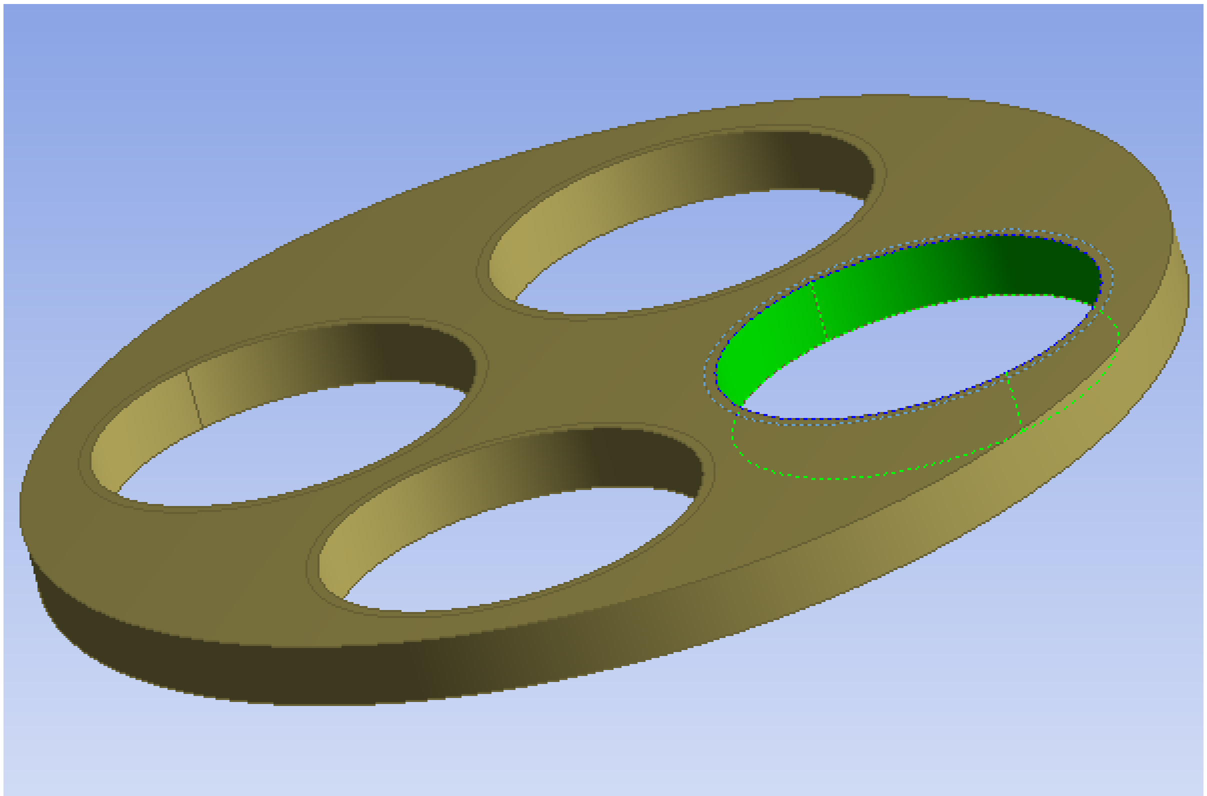


11. Redefine the **intf-valveID-ob-fluid-ch-lower** with faces belonging to **ch-lower** body and interface of **ch-valveID** body.

    a. Delete the named selection **intf-valveID-ob-fluid-ch-lower** for an individual valve from the tree.

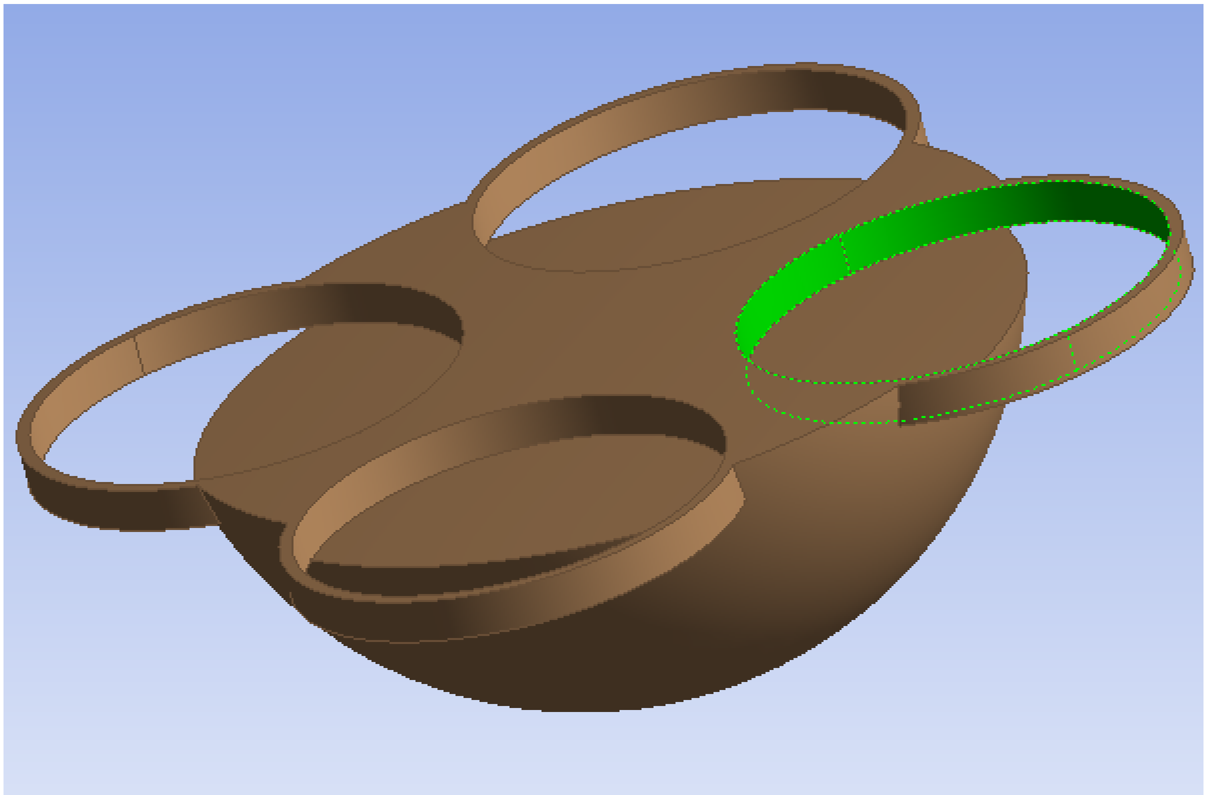
    b. Select all the proper faces as shown in Figure 16.13: Create Named Selection intf-valveID-ob-fluid-ch-lower (p. 562) and then create the named selection **intf-valveID-ob-fluid-ch-lower**.

Figure 16.13: Create Named Selection intf-valveID-ob-fluid-ch-lower



c. Select all the faces as shown in Figure 16.14: Create New Named Selection (intf-valveID-ob-fluid-piston-lower) (p. 563) and create a new named selection for it (for example **intf-valveID-ob-fluid-piston-lower**). This belongs to the piston body and the interface of **ch-valveID** body.

Figure 16.14: Create New Named Selection (intf-valveID-ob-fluid-piston-lower)



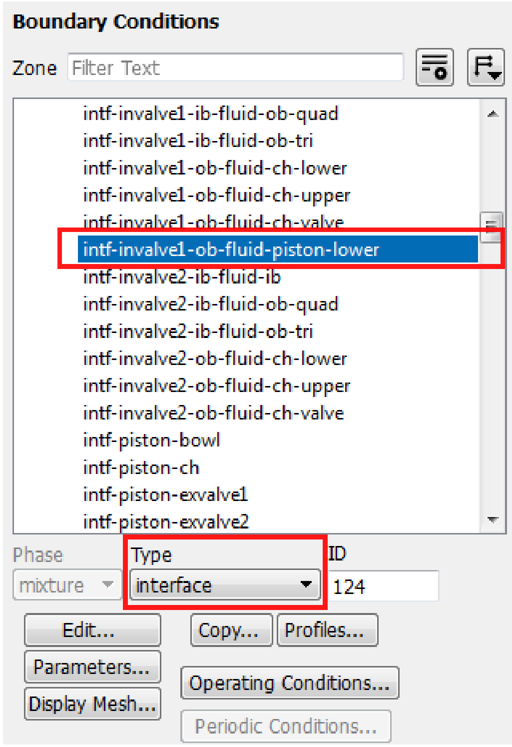
d. Similarly redefine the **intf-valveID-ob-fluid-ch-lower** for all the valves present.

12. Mesh the geometry.

13. After meshing the geometry, click **Setup** in the Project Schematic to open Fluent.

    a. The new interfaces you have created (here you have named them as **intf-invalve1-ob-fluid-piston-lower**) will be of type wall. Change the type to interface.

        i. Double-click to open **Boundary Conditions** in the Fluent tree.



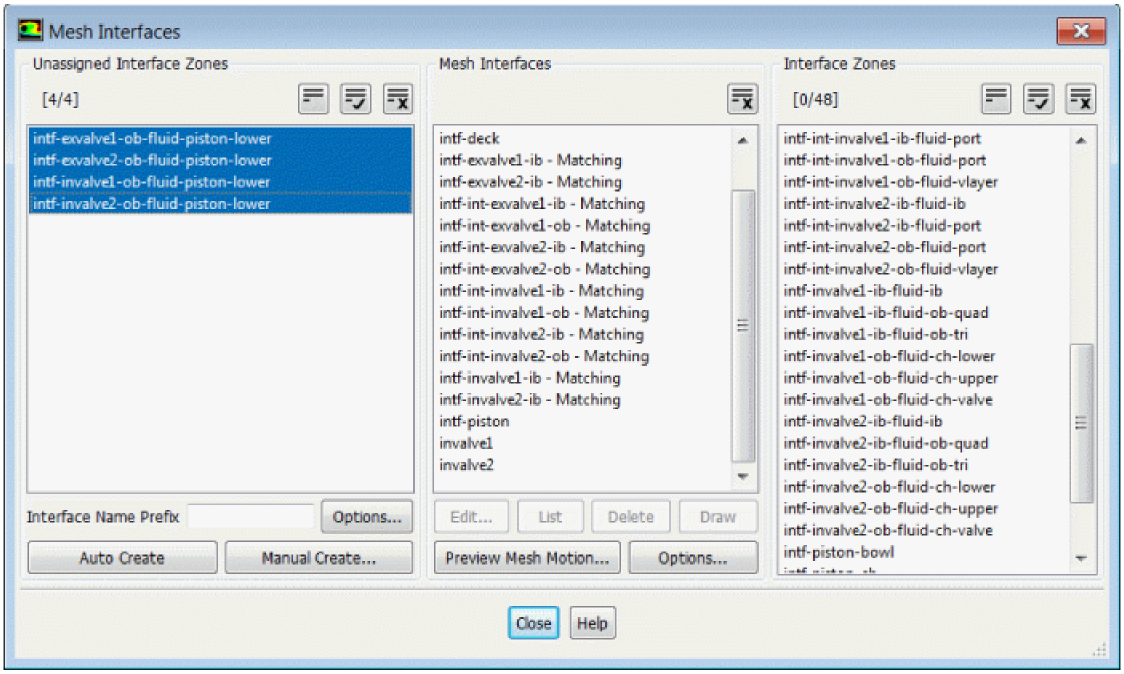
ii. From the list of **Zone**, filter for ***piston-lower** and, for each one, select it and change the **Type** to **interface**. A dialog pops up for renaming; click **OK**.

b. Create the interface zones for the valve mesh interface.

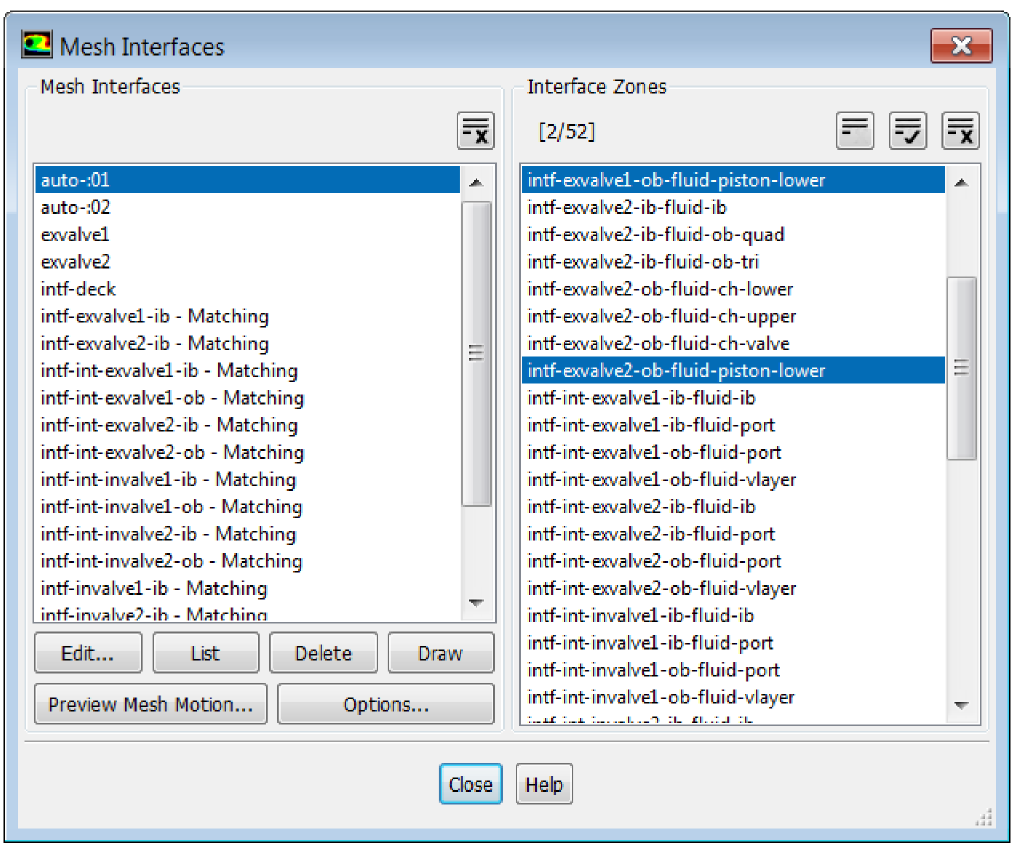
i. On the Fluent tree, double-click **Mesh Interfaces**.

ii. In the list of **Unassigned Mesh Interface Zones** select the four interfaces whose type you just changed, assign an **Interface Name Prefix** (here, "auto-" is used), and click **Auto Create**.

Figure 16.15: Select and Name Interface Zones (intf-invalveN-ob-fluid-piston-lower)



iii. The **Mesh Interfaces** dialog box shows the new interfaces in the **Mesh Interfaces** list (here, **auto-:01** and **auto-:02**).



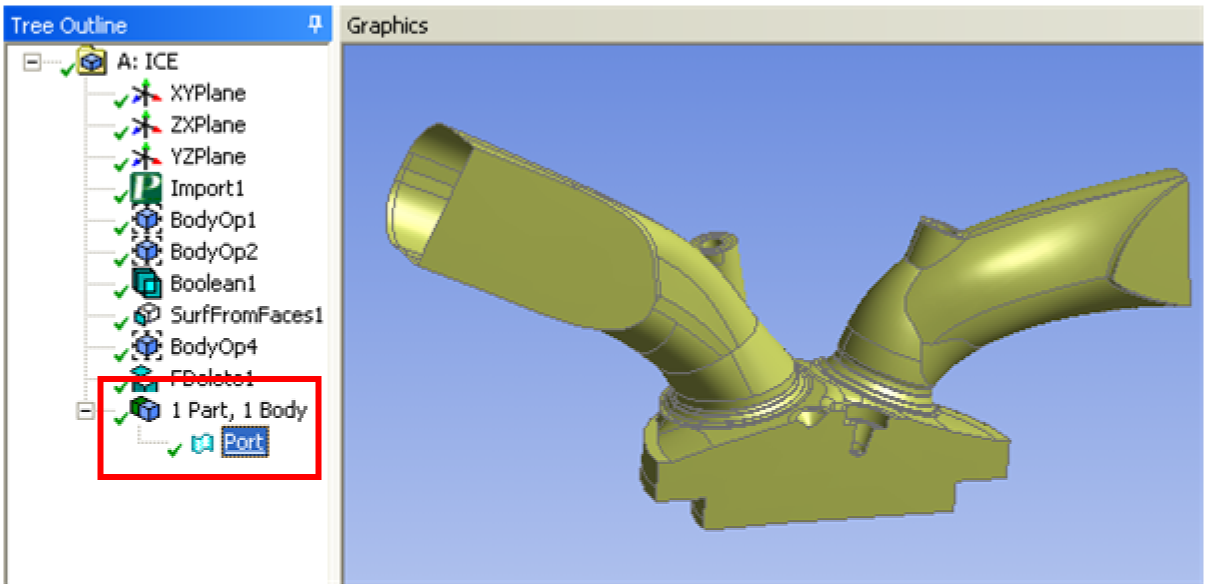
iv. Select one of the two automatically created mesh interfaces and click **Edit...** to rename it, based on its Interface Zones. (Here, **auto-:01** is renamed to **ex-valve** and **auto-:02** becomes **invalve**.) Repeat the renaming for your second new mesh interface.

16.4. Creating Flow Volume

Sometimes the imported geometry has no flow volume. It may consist of only the surface body. In such a case you will have to create a flow volume before proceeding with decomposition.

Figure 16.16: Geometry with No Flow Volume (p. 566) shows a geometry with no enclosed flow volume and no solid valves.

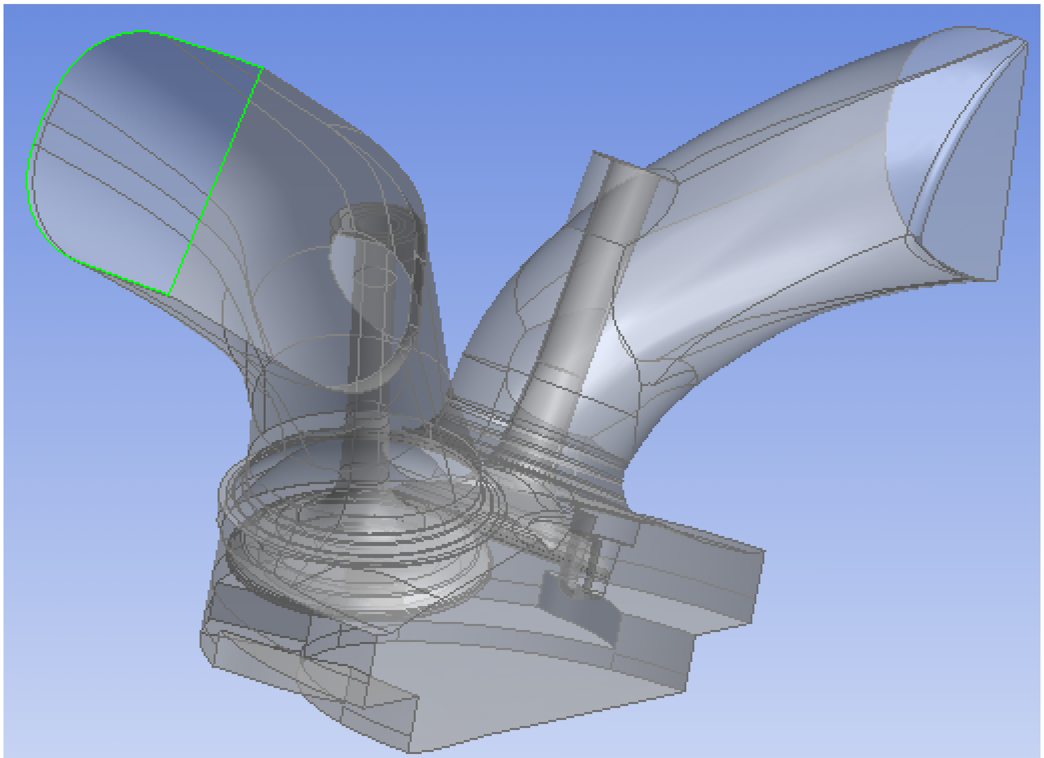
Figure 16.16: Geometry with No Flow Volume



For creating a flow volume:

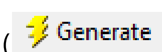
1. Select all the edges at any one of the open ends. See Figure 16.17: Edges Selected of the Open End (p. 567).

Figure 16.17: Edges Selected of the Open End



2. From the **Tools** menu select **Surface Patch**.

   a. All the edges at the open end are selected for **Patch Edges**.

   b. Click **Generate** ().

3. Repeat the procedure for all the open ends.

4. After all the open edges are patched select **Fill** from the **Tools** menu.

   a. In the **Details View** of **Fill** select **By Caps** from the **Extraction Type** drop-down list.

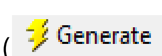
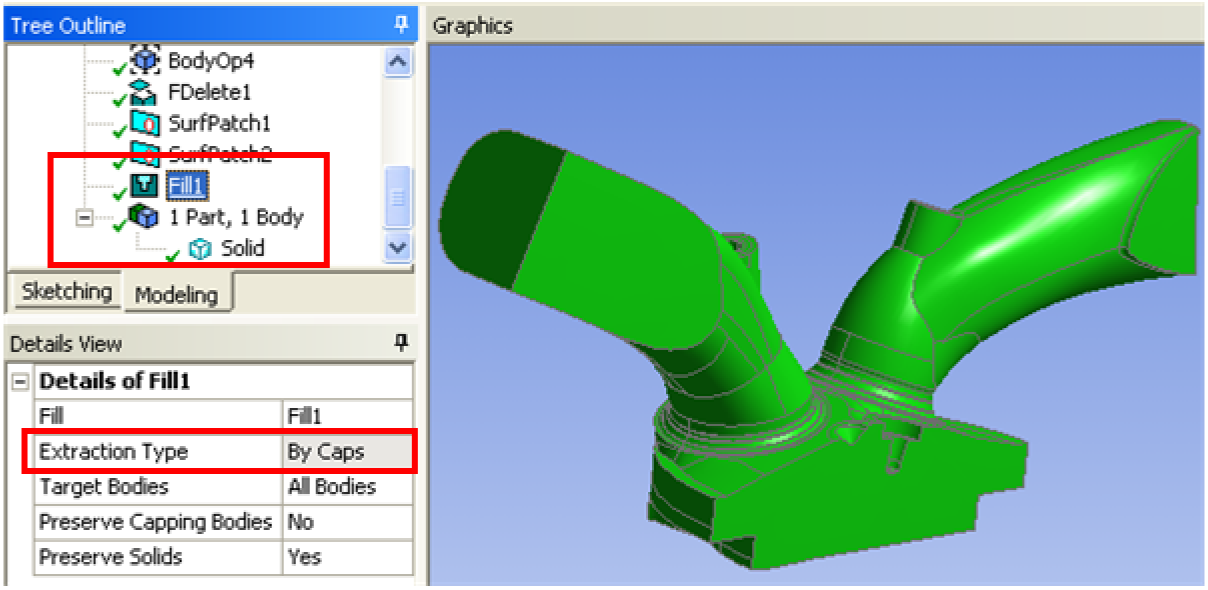
   b. Click **Generate** () to complete the operation. See Figure 16.18: Creating a Flow Volume (p. 568). A solid body is created in the **Tree Outline**.

Figure 16.18: Creating a Flow Volume

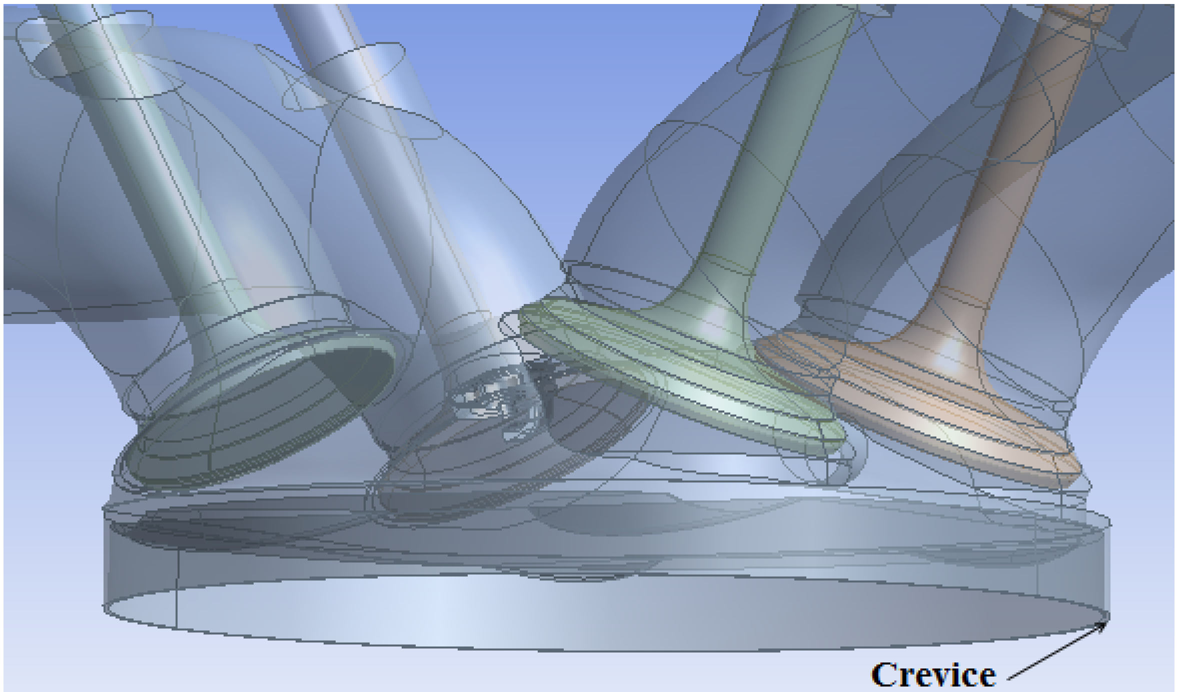


Now the geometry has a flow volume but the solid valves are extracted. You can use **Pre Manager** (p. 149) to create the valves.

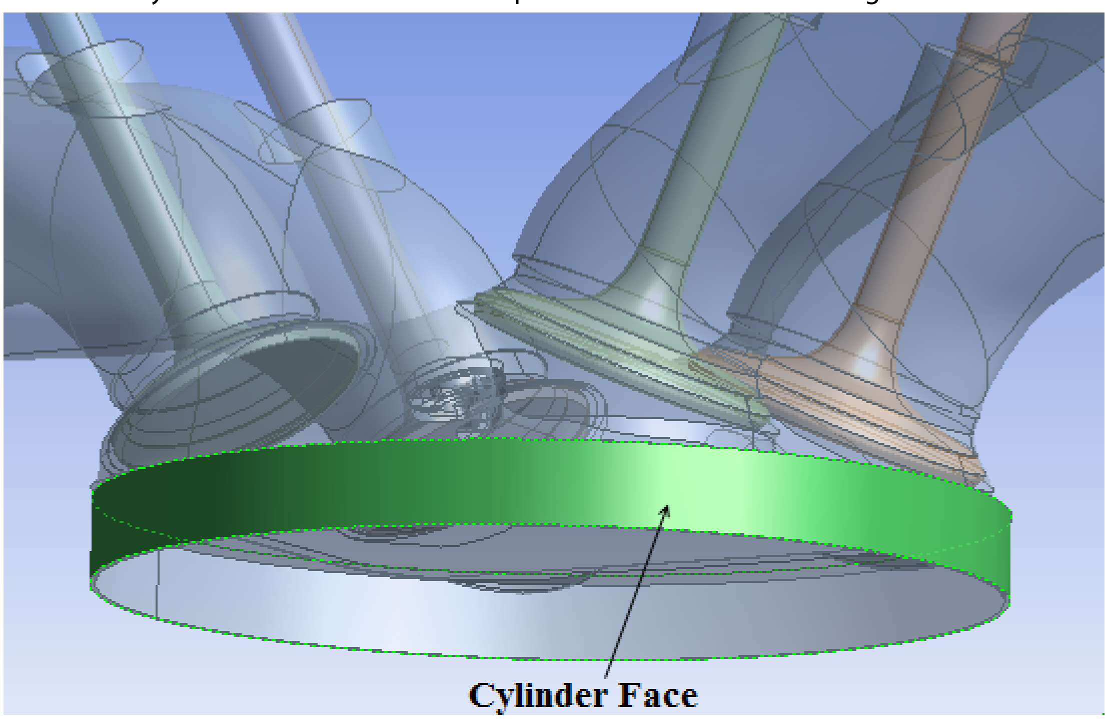
16.5. Separating the Crevice Body

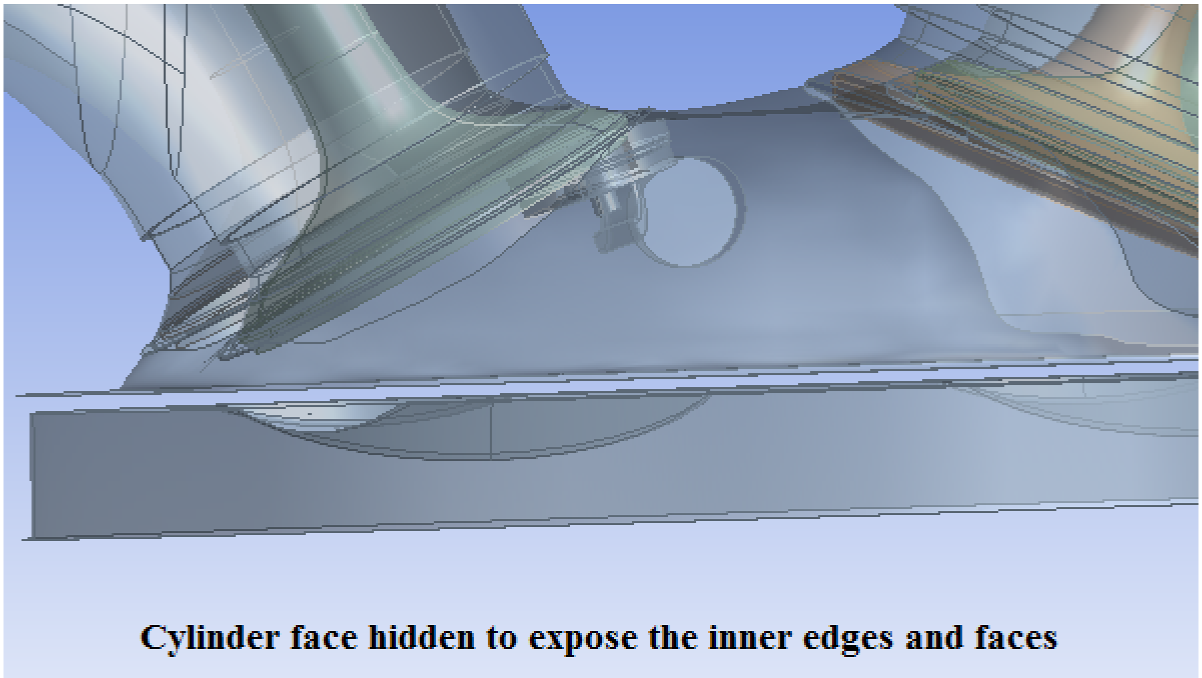
If the imported geometry has a crevice, then the crevice body should be separated before decomposition. The crevice body will be set as a rigid body during simulation and the neighboring bodies will be set as deforming bodies. So the separation of the crevice body from the rest of the geometry is important. The following steps have to be followed:

1. Read the geometry with the crevice.



2. Select the cylinder face and hide it to expose the inner faces and edges.

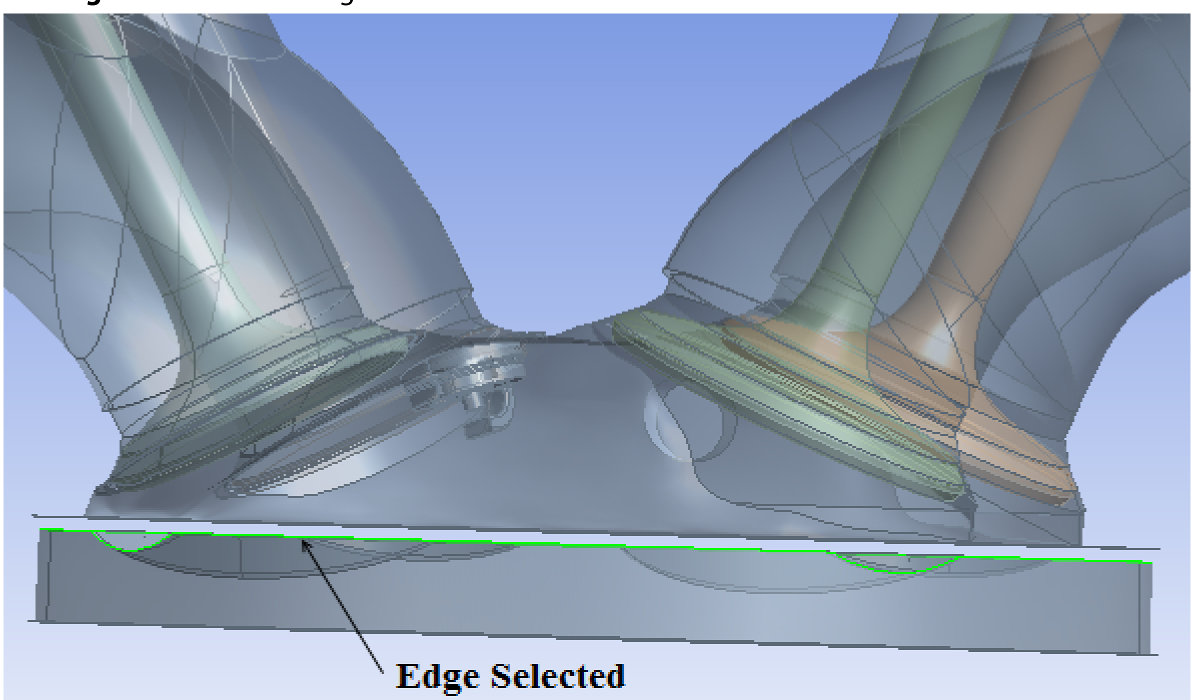




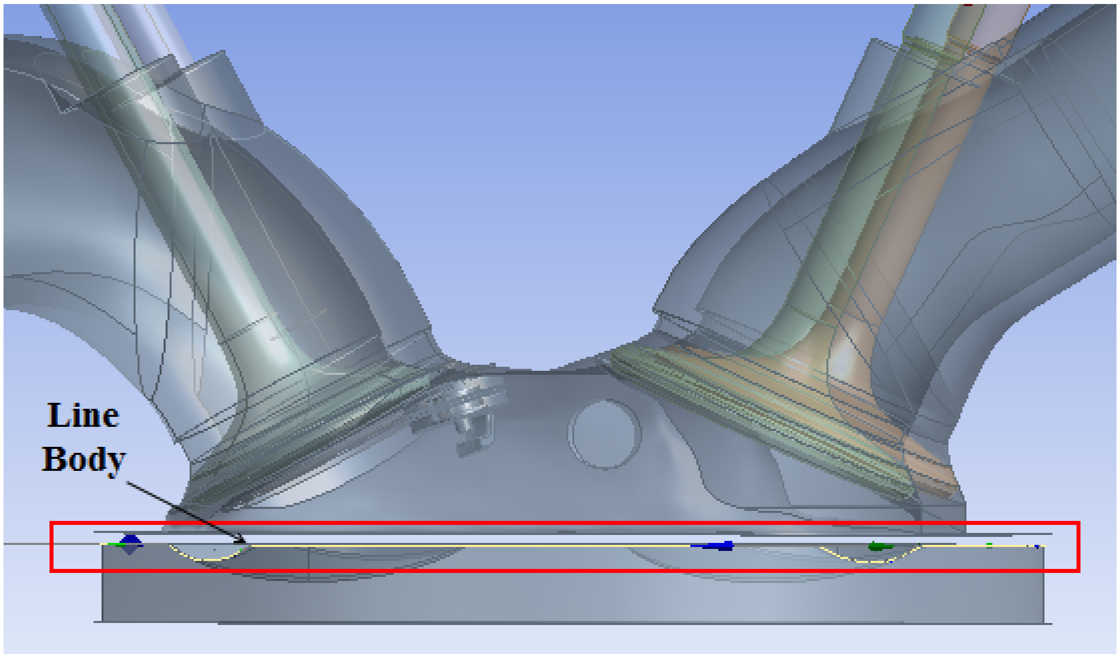
3. Create a line body of the edge of the crevice.

   Concept → **Lines From Edges**

   For **Edges** select all the edges of the crevice.



When you click **Generate** a **Line Body** is formed under **Parts, Bodies** in the tree.



4. **Tools** → **Freeze**

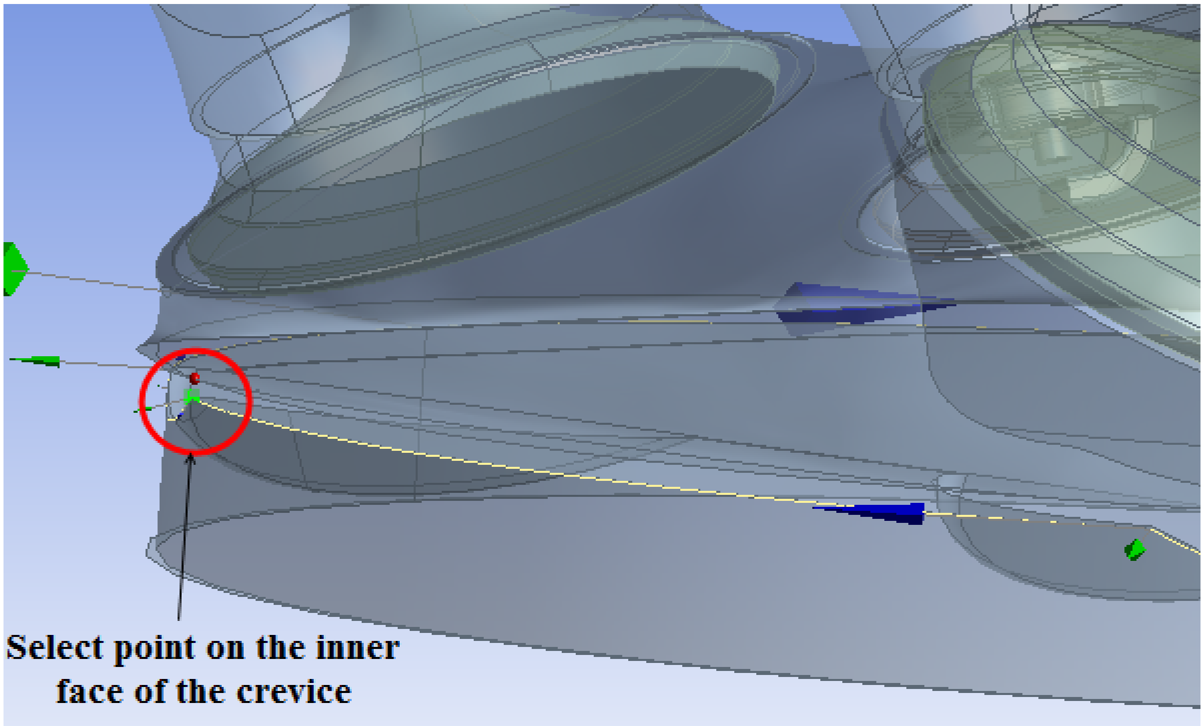
5. Project a point from the inner edge of the crevice onto the cylinder surface.

> **Note:**
>
> If inside edge is single and circular, then you may need to create a point on the edge, which you can use to split the edge.

Tools → **Projection**

a. Select **Points on Face** from the **Type** drop-down list.

b. Select a point on the edge for **Points**.



c. Unhide the previously hidden cylinder face and select it for **Target**.

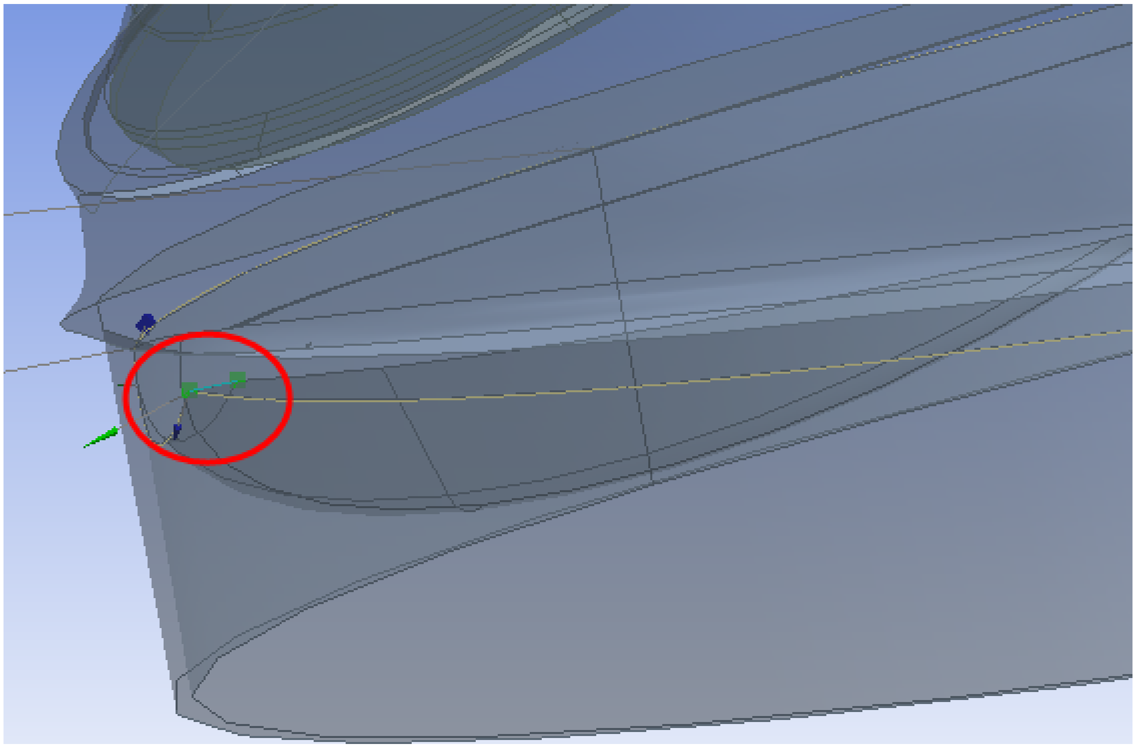
d. Click **Generate**.

6. Create a line from the points.

Concept → **Line From Points**

a. For **Point Segments** select both the point and the projected point.

b. Click **Generate**. A small line body is created between the two faces of the crevice.

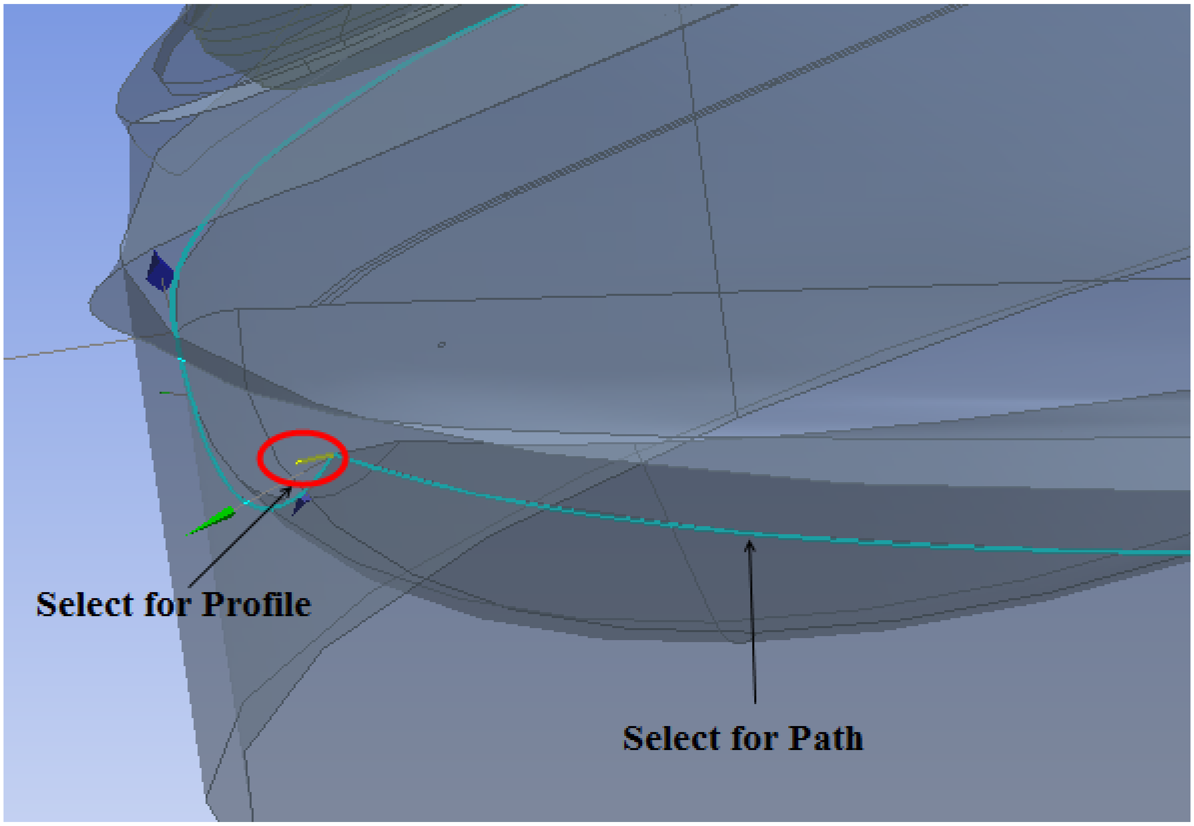


7. Generate a surface body by sweeping the line body over the edge of the crevice.

   Create → **Sweep**

   a. For **Profile** select the **Line Body** under **Parts, Bodies**, which is the line segment created between the inner and outer faces of the crevice.

   b. For **Path** select the **Line Body** under **Parts, Bodies** which is the edge of the crevice.



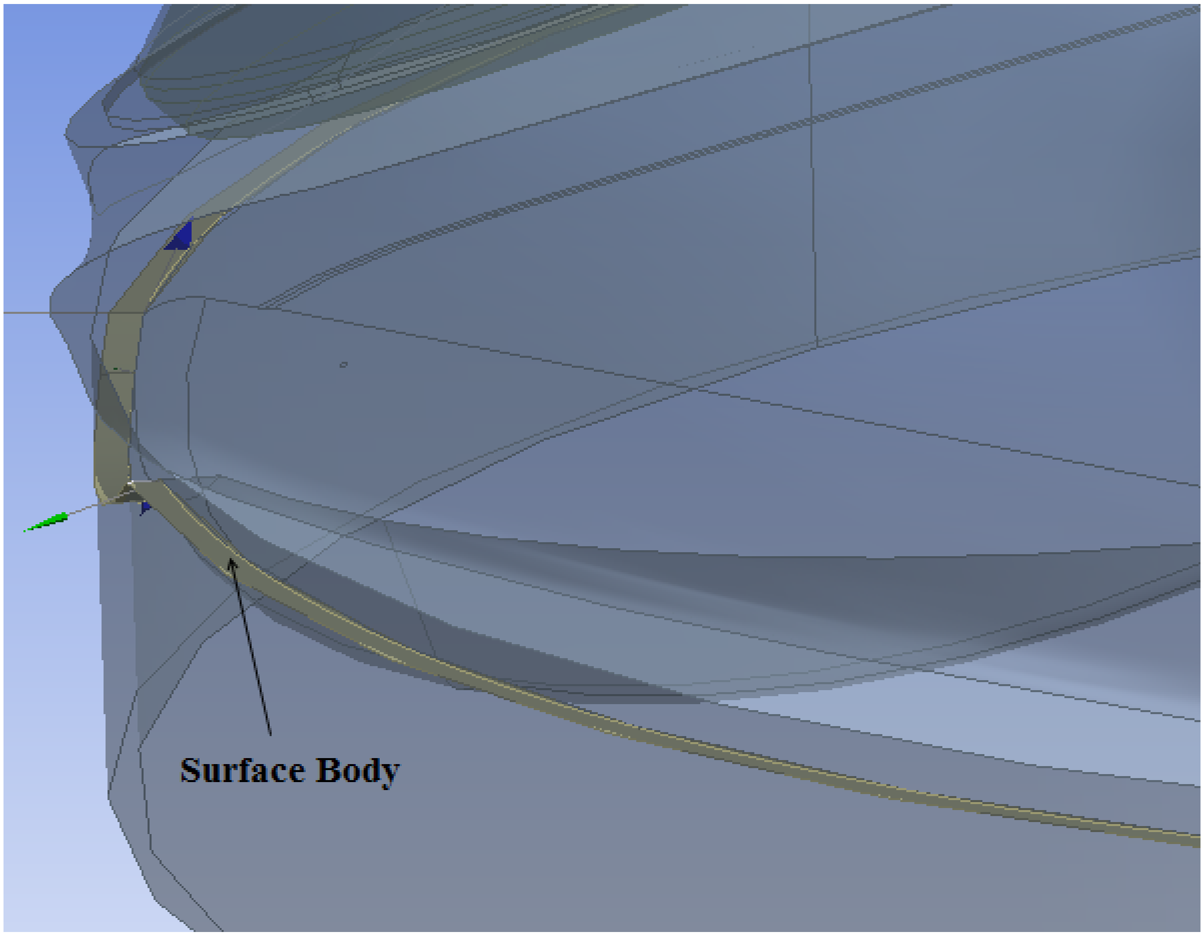
c. For **Operation** select **Add Material** from the drop-down list.

d. Select **Yes** from the **As Thin/Surface?** drop-down list.

e. Enter `0` for **Inward Thickness** and **Outward Thickness**.

f. Click **Generate**.

A **Surface Body** is created under **Parts, Bodies**.



8. Extend the surface inwards and outwards so that the slicing is proper.

> **Note:**
>
> Ensure that the surface extension is adequate. Too much extension will interfere with the piston body. The surface should only separate the crevice from the flow volume and should not enter into the piston part of the flow volume.

Tools → Surface Extension

a. For **Edges** select all the outer edges of the crevice.

b. Retain **Fixed** from the **Extent** drop-down list.

c. Enter an appropriate number for distance.

d. Click **Generate**.

9. Similarly extend the inner edge of the surface.

10. Separate the crevice body.

Create → Body Operation

a. Select **Cut Material** from the **Type** drop-down list.

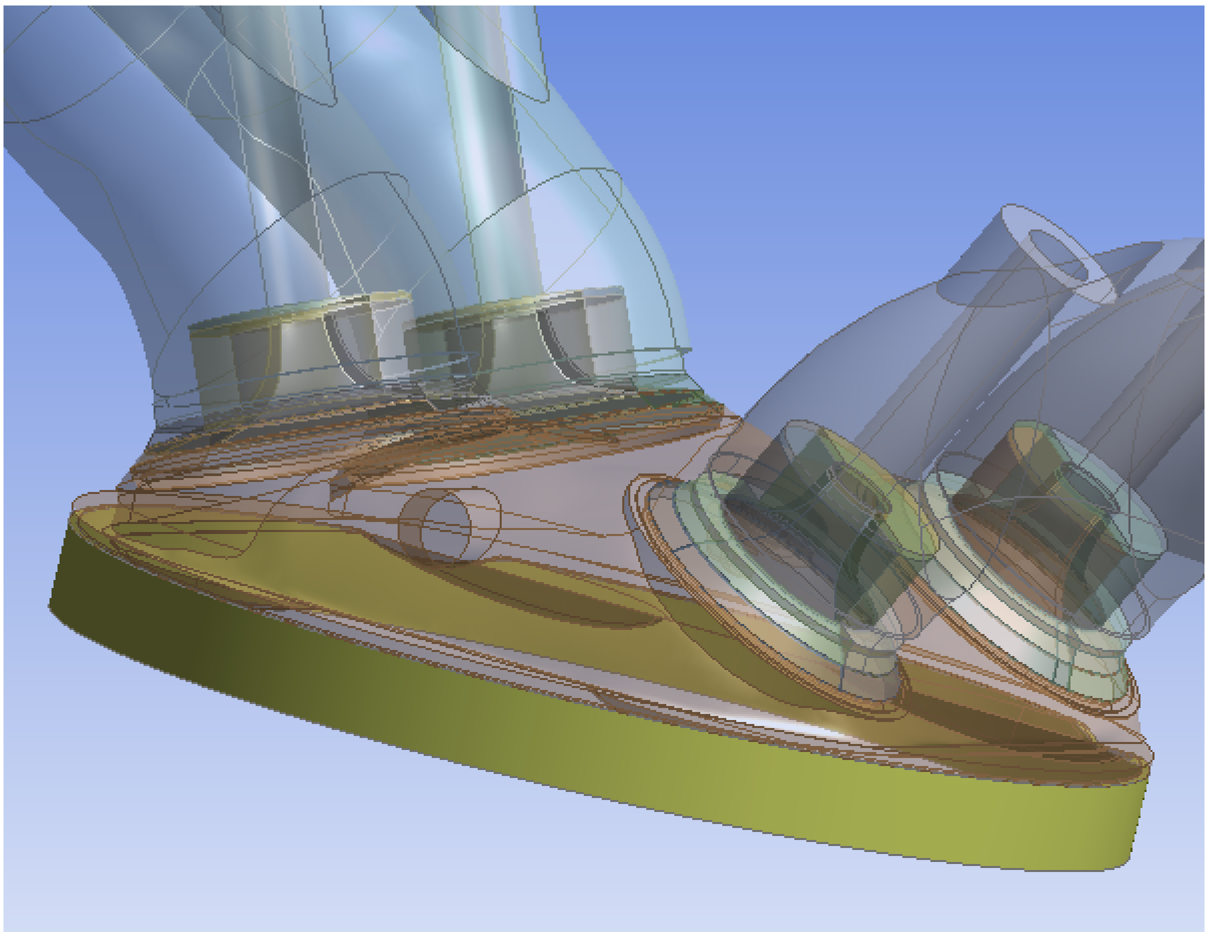
b. Select the surface body created under **Parts, Bodies** for **Body**.

c. Retain **No** for **Preserve Bodies?**.

d. Click **Generate**.

Note:

Before decomposing the geometry delete the 2 line bodies you have created from the tree under **Parts, Bodies**.



16.6. Boundary Conditions, Monitor Settings and Solver Settings

After decomposition, DesignModeler writes a **Default Boundary Conditions and Monitor Settings** file (`icBcSettings.txt`). The solver will read this file to set up the solver settings. This is file is modified by the input parameters defined in the ICE Solver Setup. You can also change the settings by copying the format from this default file and saving your conditions to the **User Boundary Conditions**

and Monitor Settings file (`icUserSettings.txt`). This file will be stored at `~ProjectName_files\dp0\ICE\ICE` . The format is described below.

Note:

The settings in the `icUserSettings.txt` file will override the settings in the default `icBcSettings.txt` file.

Similarly, DesignModeler will also write a solver settings file, `icSolverSettings.txt`. The solver will read this file to set up the simulation located at `~ProjectName_files\dp0\ICE\ICE\`, The solver will read this file to set up the simulation. The format is described below.

16.6.1. Format and Details of an icUserSettings.txt File

The following list shows the format and details of each of the fields.

`solution-type`

Set to `"cold-flow"` or `"port-flow"`, or `"combustion"`depending upon the solution type you have chosen.

`combustion-simulation-type`

Set to `"sector"`, or `"full_engine_ivc_to_evo"`, or `"full_engine_full_cycle"` depending upon the combustion simulation type you have chosen.

`initialize-flow?`

Set to `#t` (true) or `#f` (false) depending upon whether you want to initialize the flow or not.

`report-mesh?`

Set to `#t` (true) or `#f` (false) depending upon whether you want to generate a mesh report after running the calculation.

`report-urf?`

Set to `#t` (true) or `#f` (false) depending upon whether you want the under relaxation factors included in the report or not.

`set-default-models?`

Set to `#t` (true) or `#f` (false) depending upon whether you are using the default models set for this type of solution.

`data-sampling`

Set to `#t` (true) or `#f` (false) depending upon whether sampling of date is enabled during the unsteady calculation.

auto-save-type

Set to "CA" (Crank Angle) or "time" (time steps) depending upon whether you want to save the case and data files at a specific frequency of crank angle or time steps.

auto-save-freq

The number of crank angle or time steps after which the case and data file is saved in Fluent. If it is set to #f, then only the final case and data files are saved.

num-of-CA

Set to the number of crank angles you want the solution to run.

ice-swirl-number

Set to the default swirl number of 1.3 or to the value you chose.

ice-chemkin-file

Set to #f if no file is provided. If file is provided then it shows the path where the file is saved.

ice-them-data-file

Set to #f if no file is provided. If file is provided then it shows the path where the file is saved.

profile-file

Set to #f if no file is provided. If file is provided then it shows the path where the file is saved.

set-default-urf?

Set to#t (true) or #f (false) depending upon whether you are using the default under-relaxation factors or setting your own.

set-bc?

Set to#t (true) or #f (false). When this is #t it will set the boundary conditions specified in the inlet_outlet_bc_list and wall_bc_list.

ice-initialization values

Shows the parameters and their initialization values.

ice-combustion-model-param

Shows the selected combustion model and the mixture material

inlet_outlet_bc_list

Lists the information about the inlets and outlets in the format:

```
(list-no (zone "Zone-Name") (type "Type")(data-profile "no" "profile-
name" "field-name" /value) (temperature-profile "yes/no" value)
(turbulence "Turbulence Specification Method" value1 value2)))
```

Here `list-no` is used to list the serial number of inlets and outlets.

`"Zone-name"` is the name of the zone to which you are applying the boundary conditions.

`"Type"` will be `"pressure-inlet"`, `"pressure-oulet"`, or `"mass-flow-inlet"`.

You can enter `yes` or `no` for the `data-profile` or `temperature-profile`. If you want to provide a profile enter `yes`. The format changes to `(data-profile "yes" "profile-name" "field-name")`. Here `"profile-name"` is the name of the profile in the file provided, and `"field-name"` is the field within the profile that contains the data.

If you specify `no`, then a value should be provided for the parameter.

The `"Turbulence Specification Method"` can be either **Intensity and Viscosity Ratio** (`"int-vis-ratio"`) or **Intensity and Hydraulic Diameter** (`"int-hyd-dia"`).

The `value1` is value of **Turbulent Intensity**. `value2` can be **Turbulent Viscosity Ratio** or **Hydraulic Diameter** depending on the **Specification Method** selected. An example is shown below.

```
inlet_outlet_bc_list (
(1(zone "ice-inlet")(type "pressure-inlet")(data-profile "no" 500)(temperature-profile "no" 400)
(turbulence "int-hyd-dia" 2 5))
(2(zone "ice-outlet")(type "pressure-outlet")(data-profile "no" 0)(temperature-profile "no" 700)
(turbulence "int-hyd-dia" 2 5))
(3(zone "ice-inlet-invalve-1-port")(type "pressure-inlet")(data-profile "yes" "in-mflow-prof")
(temperature-profile "yes" "in-temp-prof" "temp")(turbulence "int-hyd-dia" 2 0.027)) )
```

wall_bc_list

Lists the information about the walls in the format shown:

`((list-no (zone "Zone-Name") (data-profile "no" value)(slipwall "yes")))`

Here `list-no` is used to list the serial number of walls.

`"Zone-name"` is the name of the zone to which you are applying the boundary conditions. If you want to set one value on all the zones, set `"Zone-name"` to `"all"`. You can add individual zones thereafter that will overwrite the conditions set for `"all"` zones.

You can enter `yes` or `no` for the `data-profile`. If you want to provide a profile enter `yes`. The format changes to `(data-profile "yes" "profile-name" "field-name")`. Here `"profile-name"` is the name of the profile in the file provided, and `"field-name"` is the field within the profile that contains the data.

If you specify `no`, then a value should be provided for the parameter.

You can enter `yes` or `no` for the `slipwall`. If you choose `"yes"` then **Specified Shear** is selected from **Shear Condition** group box and you can provide the **Shear Stress** in the wall boundary condition dialog box in Fluent. For `"no"`, **No Slip** is selected from **Shear Condition** group box. `slipwall` term is present only for **Port Flow Simulation**.

patching_values

Temperature and pressure of multiple zones can be patched to specific values using the following string. This term is present only for **Cold Flow Simulation**.

`(("pressure" value (zone1 zone2...)))(("temperature" value (zone1 zone2...)))`

post_planes

Lists the location and the name of the plane created for postprocessing the results, default planes created and any planes you added after decomposition. You can also add your planes here in the file using the format given. Ensure that the plane name starts with `"ice_user_plane_"`.

This takes input in the form of `(("plane" (x y z) (x y z) (x y z)))`

You can add a plane using **IC_setting_manager**. After decomposition this feature is created in the **Tree Outline**. Right-click **IC_setting_manager** and select **Insert** and then **New Plane** from the context menu. Ensure that the plane name starts with `"ice_user_plane_"`. Click **Generate** to complete the procedure.

If you would like to add a swirl plane for port flow simulation, follow the procedure of creating a plane using the **IC_setting_manager** feature. Ensure that the name starts with `"ice_swirl_plane_"`.

Insert → **New Plane**

After you add the new planes using **IC_setting_manager** they are automatically added to the `icBcSettings.txt` file under the tag of `post_planes`.

post_plane_images

This term has different formats depending on how you want the images captured.

```
post_plane_images (
```

- When you want only animation from the images and do not require a table, use the following format:

```
( (name "imagename1") (surfaces ("plane1" "inlet1" "outlet1")) (view "ice-plane1-view1")
(quantities ("velocity-magnitude" "pressure")) (interval-tag ((start-CA 15) (end-CA 45) (frequency 2)) )
(anim-properties (animation? #t)) (table-properties (table? #f) (columns #f) (rows #f) (tag "")) )
```

- When you require an animation and also a table for some of the images, use the following format:

```
( (name "imagename2") (surfaces ("plane1" "inlet1" "outlet1")) (view "ice-plane1-view1")
(quantities ("velocity-magnitude" "pressure")) (interval-tag ((start-CA 15) (end-CA 45) (frequency 2)) )
(anim-properties (animation? #t)) (table-properties (table? #t)  (columns #3) (rows #5) (tag "")) )
```

- When you do not require an animation but a table for images taken in intervals so that all images appear in the table, use the following format:

```
( (name "imagename3") (surfaces ("plane1" "inlet1" "outlet1")) (view "ice-plane1-view1")
(quantities ("velocity-magnitude" "pressure")) (interval-tag ((start-CA 15) (end-CA 45) (frequency 2)) )
(anim-properties (animation? #f)) (table-properties (table? #t) (columns #3) (rows #5) (tag "my-table-new"
```

- When the frequency is `0`, only the last image of the simulation is captured. To arrange the multiple images in a table use the following format:

```
( (name "imagename4") (surfaces ("plane1")) (view "ice-plane1-view1") (quantities ("velocity-magnitude"))
(interval-tag ((start-CA #f) (end-CA #f) (frequency #f)) ) (anim-prperties (animation? #f) )
(table-properties (table? #t) (columns #3) (rows #5) (tag "my-table")) )

( (name "imagename5") (surfaces ("plane2")) (view "ice-plane1-view1") (quantities ("velocity-magnitude"))
(interval-tag ((start-CA #f) (end-CA #f) (frequency #f)) ) (anim-prperties (animation? #f) )
(table-properties (table? #t) (columns #3) (rows #5) (tag "my-table")) )
```

- When you need just the images without any animation or the table, use the following format:

```
( (name "imagename6") (surfaces ("plane2")) (view "ice-plane1-view1")
(quantities ("velocity-magnitude" )) (interval-tag ((start-CA 0) (end-CA 360) (frequency 20)) )
(anim-prperties (animation? #f) ) (table-properties (table? #f) (columns #3) (rows #5) (tag "")) )
```

Note:

In this case the images are stored in the **Report** folder. The images will not appear in the **Report.html**.

In the above formats there are different tags, like `name`, `surfaces`, `view`, `interval-tag`, and so on. Each tag is explained in detail below.

`name`

The name given to each sublist of the `post_plane_image` list. Each list should be named differently.

`surfaces`

These are the surfaces on which the images are displayed. You can choose multiple valid surfaces as `("plane1" "inlet1" "outlet1")` or any plane you have created. Every image will contain all the surfaces you have entered. If you require a different image for another surface then you should create a different sublist with a different name.

`view`

View is required to capture an image. So you have to specify at which view you want the image captured. Some views are created by default, based on the number of `post_planes` in the format of `"plane_name-view"`. Apart from these views one `ice_iso_view` is also created.

quantities

You can specify any parameter for the contour you require, for example, velocity, temperature, pressure, and so on. For every quantity one image will be captured on the number of surfaces you have provided.

interval-tag

You can use this feature when you require images in between some crank angles. It gives you the flexibility to specify the intervals and the frequency at which you require the images.

The tags `start-CA` and `end-CA` are for transient solutions where crank angle is taken as input. So in between the specified `start-CA` and `end-CA` images will be captured at the specified `frequency` of crank angle. For example, if `frequency` is set to 2 then images are captured at every 2 crank angles, from `start-CA` to `end-CA`.

For steady cases the tags are changed to `start-iter` and `end-iter`, where iteration is taken as input. For a steady case you can give input as frequency only, and the rest two tags can be set to `#f`. In this case the images will be captured at that frequency of iterations, starting from the beginning of the solution. For example, if `frequency` is set to 2 then images are captured at every 2 iterations.

anim-properties

If you require animation of some images you must provide the `animation?` tag as `#t`. Then the animation of the images will be created in the report.

table-properties

Creates a table of the images captured. The table is created only when `table?` tag is set to `#t`. You then have to mention the table size by providing values for `columns` and `rows` tag.

`tag` is useful if you want only the final images of different surfaces in a single table. In this case you have to provide the same `tag` to each sublist. The row and column numbers of the first sublist of the same `tag` will be considered.

swirl_def

This term helps in monitoring swirl ratio on the swirl planes. This term is present only for **Port Flow Simulation**. The swirl definition is as follows:

$\overrightarrow{am} = \vec{r} \times \vec{v}$

SN = [$\overrightarrow{am} \cdot \overrightarrow{sa}$] * mass flow rate

where

$\overrightarrow{am}$ = angular momentum per unit mass

SN = swirl number

r = radial coordinate, (that is, facet center-swirl center)

$\vec{v}$ = velocity

$\vec{sa}$ = swirl axis

The `swirl_def` term is defined in the following way in the `icBcSettings.txt` file.

```
(
((name "swirl1")
(swirl-center ( X Y Z ))
(swirl-axis ( X1 Y1 Z1 ))
(function "(X1*am_x_swirl1) + (Y1*am_y_swirl1) + (Z1*am_z_swirl1)") )
)
```

Here a custom field function is created in Fluent with the name given for the `name` tag. `swirl-center` signifies the location of swirl center, and `swirl-axis` signifies the direction of swirl axis. `function` tag is used to create the custom field function using the corresponding formula.

Custom field functions with the names `am_x_name`, `am_y_name`, and `am_z_name` are created automatically based on `swirl-center`. These terms are the components of the vector `am`. You can use these terms and create any formula using the `function` tag.

If you would like to add a swirl plane for port flow simulation, use the feature **IC_setting_manager**. After decomposition this feature is created in the **Tree Outline**. Right-click **IC_setting_manager** and select **Insert** and then **New Plane** from the context menu. Ensure that the plane name starts with `"ice_swirl_plane_"`. Click **Generate** to complete the procedure.

IC_setting_manager → **Insert** → **New Plane**

After you add the new planes using **IC_setting_manager** they are automatically added to the `icBcSettings.txt` file under the tag of `post_planes` and the custom field function is also automatically added under the `swirl_def` tag.

`monitor_def`

Defines the default monitors.

`("Name of the monitor" "Monitor Type" "Report Type" "Field Variable" (Cell Zones))`

The `Monitor Type` can be `volume` for **Volume Monitor** or `surface` for **Surface Monitor**.

For `volume` monitor the `Report Type` can be `Volume-Average`, `Volume`, `Sum`, `Max`, `Min`, `Volume-Integral`, `Mass Integral`, or `Mass Average`.

For `surface` monitor the `Report Type` can be `Integral`, `Standard Deviation`, `Flow Rate`, `Mass Flow Rate`, `Volume Flow Rate`, `Area-Weighted Average`, `Mass-Weighted Average`, `Sum`, `Uniformity Index - Mass Weighted`, `Uniformity Index - Area Weighted`, `Facet Average`, `Facet Minimum`, `Facet Maximum`, `Vertex Average`, `Vertex Minimum`, or `Vertex Maximum`.

You can choose the field you want to monitor by entering it for `Field Variable`.

You can also choose the zone you want to monitor by entering it for `(Cell Zones)`.

In the example here, the first monitor name is `vol-avg-pressure-mon`. The `Monitor Type` is `volume` (**Volume Monitor**). The `Report Type` chosen is `Volume-Average` and the `Field Variable` is `pressure`. The `Cell Zones` on which the variable is monitored are `fluid-ch fluid-piston-layer`.

`swirl_data`

term helps in plotting swirl ratio as a function of CA. This term is present only for **Cold Flow Simulation**.

```
(( 0 0 0 ) zone-name1 zone-name2)
```

Here `( 0 0 0 )` is the location of swirl axis. `zone-name1 zone-name2` are the zones from dynamic mesh zones where data is calculated.

16.6.2. Format and Details of the Solver Settings File

icSolverSettings.txt - Notepad

File Edit Format View Help

```
solution-type                   "cold-flow"
rpm                             2000
crank_angle_step_size           0.250000
crank_radius                    0.045000
piston_pin_offset               0.000000
piston_stroke_cutoff            0.013300
connecting_rod_length           0.144300
min_valve_lift                  0.000500
auto_activate_deactivate        #t
symmetry_engine                 #t
ice_mesh_topology               "yes"
symm_normal                      ( -0.000000    -1.000000       -
0.000000 )
point_on_symmetry                ( -0.001316    0.000000        0.000020
)
hybrid/layering                 "h"
conf/nonconf                     "n"
withbowl                         #f
piston_type                      1
no_of_valves                     2
cylinder_axis                    ( 1.000000     -0.000000       -
0.000000 )
cylinder_axis_origin             ( -0.012281    0.000000        0.000001
)
cylinder_radius                    0.041999
decomposition_type                       1
start_crank_angle               329.600000
insert_angle                    46.750000
delete_angle                    312.875000
reference_mesh_size             0.000930
setup_valve_list                (
(                               1       "invalve1"      "invalve1"
329.600000      552.200000      0.015602        ( 0.970296      -
0.000000        -0.241922 )     ( 0.008894      -0.021600       -
0.015445 )       (smoothing #t)          (end-valve-open-smoothing
348.4)  (begin-valve-close-smoothing 528.7))
(                               2       "exvalve1"      "exvalve1"
166.400000      390.400000      0.014000        ( 0.944949      -
0.000000        0.327218 )      ( 0.003423      -0.021000       0.017835
)        (smoothing #t)          (end-valve-open-smoothing 185.3)
(begin-valve-close-smoothing 365.4))
                                )
profile-file    ("ICE/ICE/ice_master_profile.prof")
```

Format of Solver Settings File

The following table shows the format and details of each of the fields.

Keys	Values
solution-type	`"combustion"`
combustion_simulation_type	`"full_eng_full_cycle"`
rpm	`1800`
crank_angle_step_size	`0.25`
crank_radius	`0.045`
piston_pin_offset	`0`
piston_stroke_cutoff	`0.013`
connecting_rod_length	`0.144`
min_valve_lift	`0.0002`
auto_activate_deactivate	`#t`
symmetry_engine	`#t`
ice_mesh_topology **ice_mesh_topology**	`#t`
symm_normal	`(0 1 0)`
point_on_symmetry	`(0 0.0002)`
hybrid/layering	`"h"`
conf/nonconf	`n`
withbowl	`#f`
piston_type	`3`
no_of_valves	`2`
cylinder_axis	`(0 0 1)`
cylinder_axis_origin	`(0 -0 0.011)`
cylinder_radius	`0.042`
start_crank_angle	`0`
insert_angle	`47`
delete_angle	`313`
reference_mesh_size	`0.000930`

setup_valve_list	`((1 "invalve1" "invalve1" 323 563 0.015 (0 -0.24 0.97) (-0.02 -0.15 0.009) (smoothing #t) (end-valve-open-smoothing 384.4 ) (be- gin-valve-close-smoothing 528.7))`
	
	`(2 "exvalve1" "exvalve1" 160 402 0.014 (-0 0.327 0.94) (-0.02 0.0197 0.0088) (smoothing #t) (end-valve-open-smoothing`

	`185.3) (begin-valve-close-smoothing 365.4)) )`

Note:

- The order of the keys is not important in the file.
- All the units are in SI.

Details of Fields

solution-type

is the type of simulation you have chosen. Here it is set to `"combustion"`.

combustion_simulation_type

is the sub type of combustion simulation you have chosen. Here it is set to `"full_engine_full_cycle"`.

rpm

is the engine speed, which will be used along with crank angle step size to calculate the time step size.

crank_angle_step_size

will be used to calculate the time step. `0.25` is a good default value.

$\Delta t=\theta/6N(RPM)$

crank_radius

is the crank radius in meter when the piston is at the Bottom Dead Center (BDC) position.

piston_pin_offset

is piston offset value in meter.

piston_stroke_cutoff

is the distance till which the layering part of the mesh will move as a rigid body. After this it will stop moving, but the piston head will continue to move until it reaches BDC. Thus the layering process will begin from the cutoff distance. This is required to create a large enough space in the upper part of the cylinder, that is meshed with tetrahedra. It ensures that the valves can fully move in that zone and the smoothing and remeshing process can happen. This value along with the value of **min_valve_lift** are used to control the actual values of the valve lift and piston stroke, such that:

$\nu_{lift} = \max (v^{c}{}_{lift}, v^{min}{}_{lift})$

$p_s = \min(p^c{}_s, p^{min}{}_s)$

connecting_rod_length

is the length of the connecting rod in meters.

min_valve_lift

takes the floating numerical value of the minimum valve lift in meters. Fluent assumes that once you have set up the mesh topology, the mesh topology is unchanged throughout the entire simulation. Therefore, Fluent does not allow you to completely close the valves such that the cells between the valve and the valve seat degenerate (become flat cells) when these surfaces come in contact. (Removing these flat cells would require the creation of new boundary face zones). To prevent this collapse, you need to define a minimum valve lift and Fluent will automatically stop the motion of the valve when the valve lift is smaller than the minimum valve lift value.

auto_activate_deactivate

is set to Boolean `#t` or `#f`. This will be used to deactivate some of the fluid zones which are not part of the simulation.

symmetry_engine

will take the value `#t` or `#f`, depending on whether the engine has symmetry defined or not.

ice_mesh_topology

will take the value `#t` if the mesh topology is the default ICE topology.

symm_normal

takes the format of `(x y z)`. If symmetry is defined, you have to provide the normal at the given point.

point_on_symmetry

takes the format of `(x y z)`. If symmetry is defined, you have to provide a point on the symmetry.

hybrid/layering

takes the input `"h"` or `"l"` with quotes. If you are opting for a hybrid mesh i.e. layering in the valve region and combination of re-meshing and layering in the cylinder and dome region, you need to set it as `"h"`. If you are opting for a layered mesh in the valve, dome, and the cylinder region, then you need to set it as `"l"`.

conf/nonconf

takes the input `"n"` or `“c”` (with quotes) depending on whether the mesh has non conformal interfaces (`"n"`) or conformal (`"c"`) interfaces.

withbowl

takes the Boolean value `#t` or `#f`, and will be used to define proper interfaces.

piston_type

will be used in the case of a hybrid approach. It takes input `1` or `3`. `1` is for straight valve, or canted valve engine with chamber decomposition. `3` is used for straight or canted valve engines, without chamber decomposition. In the case of layered approach this value will not be used.

no_of_valves

is the number of valves in the setup valve list.

cylinder_axis

takes the input in the form of `(x y z)`. It is the axis of the cylinder.

cylinder_axis_origin

takes the input in the form of `(x y z)`. It signifies the point of origin of the cylinder axis.

cylinder_radius

is the radius of the cylinder in meters.

Values of cylinder axis, origin, and radius are used to define the rigid body motion for the piston and also for remeshing of the dynamic cylinder zone.

start_crank_angle

is the angle at which decomposition takes place. It is set to `0` by default.

insert_angle

is used to insert an interior zone in case of canted or straight valve engines without chamber decomposition. This is in the case where there is no initial layering zone.

delete_angle

is used to delete the interior zone inserted by the `insert_angle`.

reference_mesh_size

is a reference value. Some global and local mesh setting values are dependent on this term. This term is used to set layering height for **vlayer**, **ch-lower**, and **ch-valve** bodies.

- For **vlayer** layer height is equal to (0.4 X **reference_mesh_size**).
- For **ch-lower** and **ch-valve** layer height is equal to (0.8 X **reference_mesh_size**).

setup_valve_list

contains all the information related to valve in the format shown.

```
((Valve-Index “Valve-Name” “Valve-Profile-Name” Valve-Open-Angle
Valve-Close-Angle (valve Axis vector) (Valve origin coordinates)
Variable-urf? Variable-timestep? (smoothing #t) (end-valve-open-
smoothing 384.4) (begin-valve-close-smoothing 528.7))(Valve-Index
……..))
```

If the field `Variable-urf?` is `#t` Fluent automatically increases and decreases the URF (under relaxation factors) values of the flow.

Similarly if `Variable-timestep` is `#t` Fluent will modify the time step during valve openings.

If `smoothing` is set to `#t`, solver will only do smoothing while valve is opening, till reaching the crank angle `end-valve-open-smoothing`. (Here for the **invalve** you have set it to `384.4`). Also solver will stop layering and start smoothing while closing the valve at `begin-valve-close-smoothing`. (In this example it for the **invalve** it is set at `528.7` degrees).

16.7. Calculating Compression Ratio

The following procedure shows how the compression ratio for any engine is calculated internally in this system.

Compression Ratio = Volume at BDC / Volume at TDC

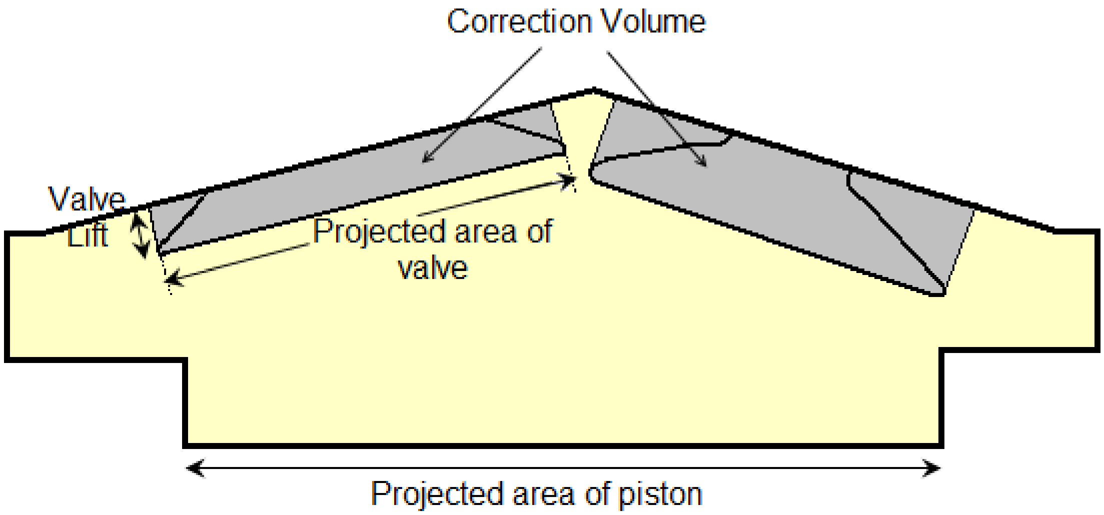
Volume at BDC = Volume at TDC + Stroke Volume

Stroke Volume = (The projected area of the piston) X (The stroke length)

Volume at TDC = Volume calculated by DesignModeler at TDC + Correction Volume

Due to the minimum valve lift the valves are positioned inside the chamber. While calculating the volume at TDC, the volume occupied by the valves is not included in the volume computation by DesignModeler due to valve margin slice. Therefore correction is required for the calculation of the TDC volume.

Figure 16.19: Volume at TDC



Correction volume = (The projected area of the valve) X (The valve lift)

This is how the compression ratio is calculated.

Bibliography

[1] Lumley, John L. *Engines: An Introduction*. Cambridge University Press. 1999.

[2] Stone, Richard. *Introduction to Internal Combustion Engines*. 3rd Edition, SAE International. 1999.

[3] Heywood, John B. *Engines: An Introduction*. McGraw-Hill, Inc., NY. 1988.

[4] Bedford Frederick C., Ph.D. *Modeling production and abatement of nitrogen oxide pollutants in direct injection diesel engines*. University of Wisconsin-Madison. 2001.

Appendix A. The Fluent Ribbon Tab Under Workbench

When Fluent is running within Workbench, the Fluent **File** ribbon tab is slightly different. The differences are described in the sections that follow:

The common functionality for stand-alone Fluent is documented in the separate Fluent User's Guide.

A.1. File/Refresh Input Data

The **File/Refresh Input Data** ribbon tab item allows you to refresh your Fluent input data from within Fluent. This option is only enabled if new input data exists, or if a parameter value has changed. For more information, see Refreshing Fluent Input Data (p. 59).

A.2. File/Recorded Mesh Operations...

The **File/Recorded Mesh Operations** ribbon tab item allows you to view common mesh manipulation operations that you perform within Fluent while working in Workbench. For more information, see Recording Mesh Manipulation Operations and Resolving Mesh Incompatibility in Fluent (p. 76).

A.3. File/Save Project

The **File/Save Project** ribbon tab item allows you to save your current Workbench project within Fluent, along with your current Fluent case and/or data files. For more information, see Saving Your Work in Fluent with Workbench (p. 41).

A.4. File/Reload

The **File/Reload** ribbon tab item allows you to reload the Workbench project's cell information into Fluent. Available if the case or mesh information is present. For more information, see Reloading Data and Synchronizing Fluent with Workbench (p. 60).

A.5. File/Sync Workbench

The **File/Sync Workbench** ribbon tab item allows you to synchronize recent changes in Fluent to the corresponding Workbench project. Not available if the case or mesh information is not present. For more information, see Reloading Data and Synchronizing Fluent with Workbench (p. 60).

A.6. File/Import/Mesh...

The **File/Import/Mesh...** ribbon tab item opens the **Select File** dialog box which allows you to select the appropriate mesh file to be read. For more information see Importing Fluent files in Workbench (p. 49).

A.7. File/Import/Case...

The **File/Import/Case...** ribbon tab item is used to read in a Fluent case file (extension `.cas.h5`), or a mesh file (extension `.msh.h5`, `.msh`, `.grd`, `.MSH.h5`, `.MSH`, or `.GRD`) that has been saved in the native format for Fluent. See the User's Guide for details.

The **File/Import/Case...** ribbon tab item opens the **Select File** dialog box which allows you to select the appropriate file to be read. For more information see Importing Fluent files in Workbench (p. 49).

A.8. File/Import/Data...

The **File/Import/Data...** ribbon tab item is used to read in a Fluent data file (which has a `.dat.h5` extension). This ribbon tab item will not be available until you read in a case or mesh file. See the separate Fluent User's Guide for details.

The **File/Import/Data...** ribbon tab item opens the **Select File** dialog box which allows you to select the appropriate file to be read. For more information see Importing Fluent files in Workbench (p. 49).

A.9. File/Import/Case and Data...

The **File/Import/Case & Data...** ribbon tab item is used to read in a Fluent case file and the corresponding data file (for example, `myfile.cas.h5` and `myfile.dat.h5`) together. See the separate Fluent User's Guide for details.

The **File/Import/Case & Data...** ribbon tab item opens the **Select File** dialog box which allows you to select the appropriate files to be read. Select the appropriate case file, and the corresponding data file (that is, the file having the same name with a `.dat.h5` extension) will also be read in. For more information see Importing Fluent files in Workbench (p. 49).

A.10. File/Export/...

When running under Workbench, several commands located under the **Write** option under the **File** ribbon tab have been moved to the **Export** option under the **File** ribbon tab. The new commands are:

File → Export → Case

File → Export → Data

File → Export → Case & Data

File → Export → PDF

File → Export → ISAT Table

File → Export → Flamelet

File → Export → Surface Clusters

File → Export → Profile

File → Export → Boundary Mesh

These items are used when you want to manually export a file independent of the project. Files exported in this way are not used by the project unless you later import them into a new system. When you use an **Export** command, you can export the files to the location of your choice with a name of your choice.

There is no need to export files since Workbench always saves the files it needs automatically. These export commands are provided for your convenience when you want to save a specific file for later use.

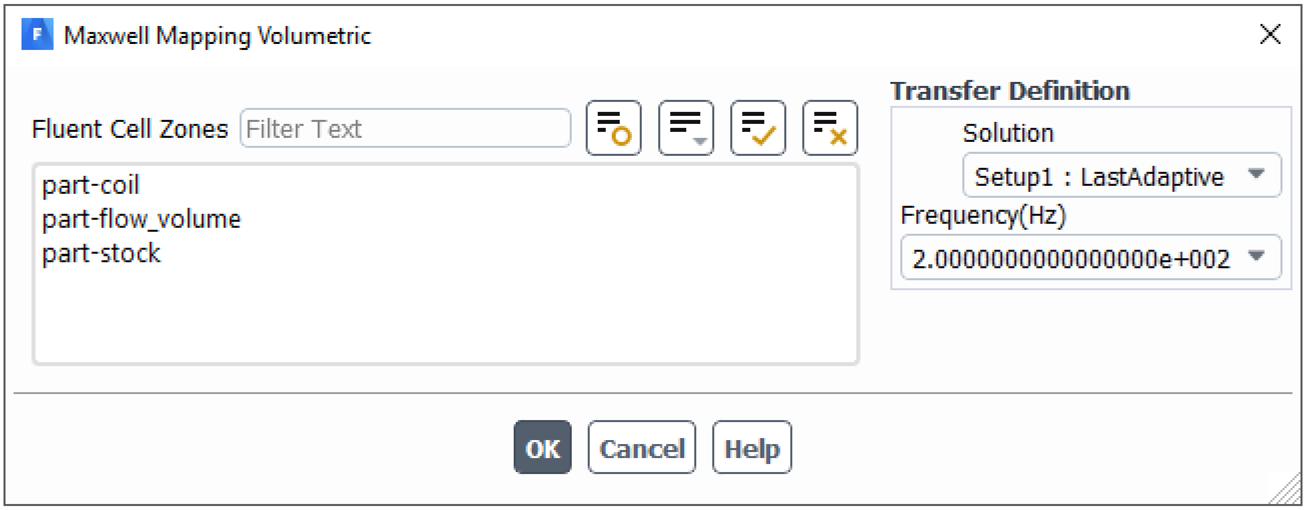
A.11. File/EM Mapping/Volumetric Energy Source...

The **File/EM Mapping/Volumetric Energy Source...** ribbon tab item opens the Maxwell Mapping Volumetric Dialog Box (p. 597).

A.11.1. Maxwell Mapping Volumetric Dialog Box

The **Maxwell Mapping Volumetric** dialog box allows you to map the volumetric loss results from an Ansys Maxwell electromagnetic simulation onto the centroids of cell zones in the Fluent mesh as a heat source.

See Performing Fluent and Maxwell Coupling in Workbench (p. 97) for details.



Controls

Fluent Cell Zones

contains a list of cell zones from the Fluent mesh, onto which the loss information can be mapped. For these zones, Fluent requests the heat source (loss) terms from Maxwell.

Transfer Definition

contains elements related to the transfer of data, including:

Solution

contains available solution sets. Since Maxwell may have multiple solutions, Fluent will request the generated heat source data for the selected solution.

Frequency

(only available for steady simulations) contains available frequencies. Fluent will request that Maxwell provide the heat source data for the selected frequency.

Start time

(only available for transient simulations) contains the simulation start time. Maxwell will request to consider the selected time as the start time.

End time

(only available for transient simulations) contains the simulation end time. Maxwell will request to consider the selected time as the end time.

A.12. File/EM Mapping/Surface Energy Source...

The **File/EM Mapping/Surface Energy Source...** ribbon tab item opens the Maxwell Mapping Surface Dialog Box (p. 599).

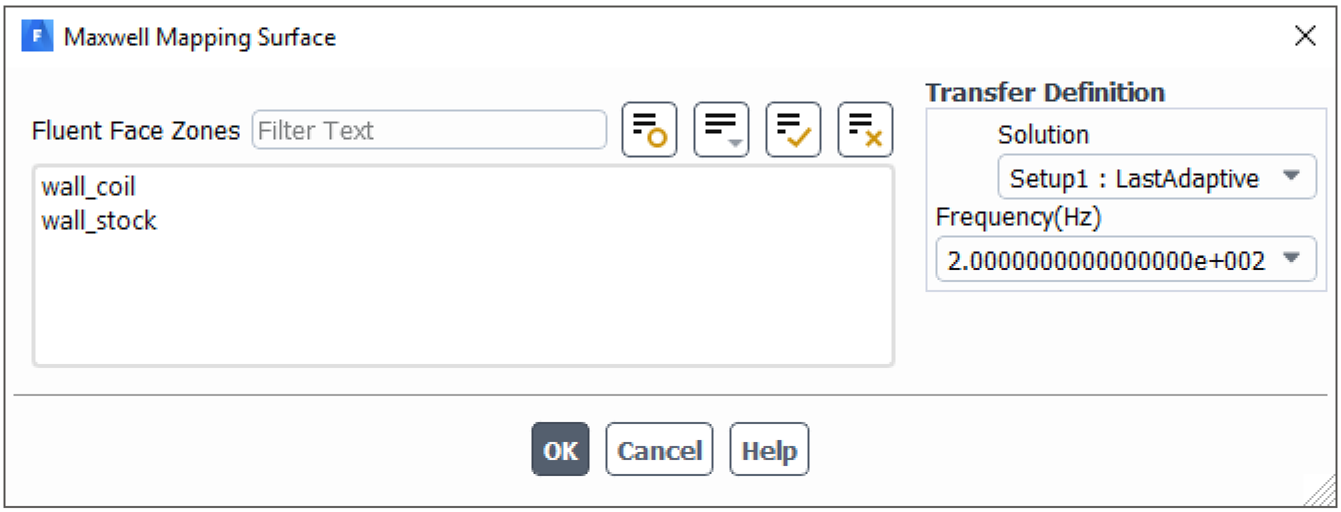
A.12.1. Maxwell Mapping Surface Dialog Box

The **Maxwell Mapping Surface** dialog box allows you to map the surface loss results from an Ansys Maxwell electromagnetic simulation onto the faces of solid zones in the Fluent mesh as a heat source.

Note:

To import heat loss (source terms) for an interior zone, split the zone into wall and wall-shadow. These face zones will then be available for Surface EM Mapping.

See Performing Fluent and Maxwell Coupling in Workbench (p. 97) for details.



Controls

Fluent Face Zones

contains a list of face zones (adjacent to the solid zones) from the Fluent mesh, onto which the loss information can be mapped. For these zones, Fluent requests the heat source (loss) terms from Maxwell.

Transfer Definition

contains elements related to the transfer of data, including:

Solution

contains available solution sets. Since Maxwell may have multiple solutions, Fluent will request the generated heat source data for the selected solution.

Frequency

(only available for steady simulations) contains available frequencies. Fluent will request that Maxwell provide the heat source data for the selected frequency.

Start time

(only available for transient simulations) contains the simulation start time. Maxwell will request to consider the selected time as the start time.

End time

(only available for transient simulations) contains the simulation end time. Maxwell will request to consider the selected time as the end time.

A.13. File/Close Without Save

The **File/Close Without Save** ribbon tab item allows you to close Fluent without saving the most recent changes. Not available if the case or mesh information is not present.

Appendix B. The Fluent Meshing Menu under Workbench

When Fluent Meshing is running within Workbench, the Fluent Meshing **File** menu is slightly different from stand-alone Fluent Meshing. The differences are described below:

The common functionality for stand-alone Fluent Meshing is documented in the separate Fluent User's Guide.

Note:

If you read a case file into Fluent Meshing and click **Switch to Solution**, only the mesh will be transferred to the Fluent solver (no case settings).

B.1. File/Refresh Input Data

The **File/Refresh Input Data** menu command is similar to the **File/Refresh Input Data** ribbon tab command in Fluent. For details, see **File/Refresh Input Data** (p. 595).

B.2. File/Save Project

The **File/Save Project** menu command is similar to the **File/Save Project** ribbon tab command in Fluent. For details, see **File/Save Project** (p. 595).

B.3. File/Import

The following commands found under the **File/Read** main menu item in stand-alone Fluent Meshing have been moved to under the **File/Import** main menu item:

- **File/Import/Mesh...**
- **File/Import/Case...**
- **File/Import/Boundary Mesh...**
- **File/Import/Size Fields...**

The **File/Import** commands enable you to import a previously generated mesh, case, boundary mesh, or size fields file directly into Fluent Meshing. The action of these commands is identical to those in stand-alone Fluent Meshing.

For more information, see Reading and Writing Files in the *Fluent User's Guide*.

B.4. File/Export

The following commands found under the **File/Write** main menu item in a stand-alone Fluent Meshing have been moved to under the **File/Export** main menu item:

- **File/Export/Mesh...**
- **File/Export/Case...**
- **File/Export/Boundaries…**
- **File/Export/Domains...**
- **File/Export/Size Fields...**

You can use these commands when you want to manually export mesh, case, boundaries, domains or size fields data into a file independent of the project. Files exported in this way are not used by the project. Files created by the export commands could be imported into a new system. When you use an **Export** command, you can specify the file name and file location.

There is no need to export files since Workbench always saves the files it needs automatically. These export commands are provided for your convenience when you want to save a specific file for later use.

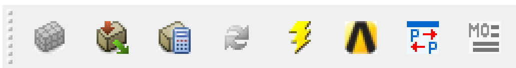
For more information, see Reading and Writing Files in the *Fluent User's Guide*.

B.5. File/Close Without Save

The **File/Close Without Save** ribbon tab item allows you to close Fluent Meshing without saving the most recent changes. Not available if the case or mesh information is not present.

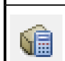
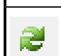
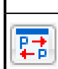
Appendix C. The Workbench Tools Toolbar Commands

When Fluent is running within Workbench, the **Workbench Tools** toolbar is displayed in the editor window.



The **Workbench Tools** toolbar provides direct access to general Workbench and Fluent functions from within Fluent and allows you to quickly setup, update, and reset your case, define output parameters, exchange data between Fluent and Workbench, and so on. You can hover over the toolbar icons to display the function of the tools, as a tooltip.

Table 1: Workbench Tools Toolbar Commands

Icon	Command	Description
	**Mesh Cell Commands** (Fluent Meshing mode)	Contains an expandable list of the **Mesh** cell commands. See Table 2: Mesh Cell Commands (p. 604) for details.
	**Setup Cell Commands**	Contains an expandable list of the **Setup** cell commands. See Table 3: Setup Cell Commands (p. 604) for details.
	**Solution Cell Commands**	Contains an expandable list of the **Solution** cell commands. See Table 4: Solution Cell Commands (p. 604) for details.
	**Refresh input data of Mesh/Setup/Solution**	Refreshes cell input data from upstream cells, and refreshes properties and input parameters of the modified cell (**Mesh**, **Setup** and **Solution**). This command is only enabled if new input data exists, or if a parameter value has changed. Note: If the input data for a **Mesh** cell of a **Fluent (with Fluent Meshing)** system has changed when the editor is in the solution mode, then on refresh, the editor will switch to the meshing mode and reread the mesh data from the **Mesh** cell.
	**Update Mesh/Setup/Solution**	Contains an expandable list of commands for controlling the solution process. See Table 5: Update Mesh/Setup/Solution (p. 605) for details.
	**Synchronize WB cell status**	Synchronizes recent changes in Fluent to the corresponding Workbench project.
	**Parameter System**	Opens the **Parameter** dialog box.

Icon	Command	Description
	**Recorded Mesh Operations**	Opens the **Recorded Mesh Operations and Incoming Zones** dialog box.

Table 2: Mesh Cell Commands

Command	Description
Reload	Reloads the mesh file or, if the mesh file has not yet been generated, re-imports available CAD files. Any unsaved changes you may have previously made will be discarded.
Clear Generated Data	Discards all unsaved changes and the current mesh in the editor (Fluent Meshing) session, deletes the generated mesh file associated with the **Mesh** cell, and re-imports available CAD files.
Reset	Performs the same actions as the **Clear Generated Data** command, and in addition, sets the **Mesh** cell property values to their defaults.

Table 3: Setup Cell Commands

Command	Description
Reload	Reads available input files (case, mesh, settings, and so on) into the **Setup** cell and executes the recorded mesh operations (if available). Any unsaved changes you may have previously made will be discarded.
Clear Generated Data	Discards all unsaved changes, deletes the *name*`-Setup-Output.cas.h5` file and generated mesh file associated with the **Mesh** cell, reads all available input files (case, mesh, settings, and so on), and executes the available recorded mesh operations.
Reset	Discards any unsaved changes and recorded mesh operations, deletes *name*`-Setup-Output.cas.h5` file, sets the **Setup** cell property values to their defaults, and reads the input files from the upstream cells (if available).
Import Settings	Allows you to import previously saved settings files.
Export Settings	Allows you to export a settings file (`.set`). Note: The exported settings file can become incompatible with your case if the mesh has been modified either upstream or in the Fluent editor.

Table 4: Solution Cell Commands

Command	Description
Reload	Reloads the latest available solution (case/data) files from the project directory. If a data file is not available, the case is initialized using the specified method. Any unsaved changes you may have previously made will be discarded.

Command	Description
Clear Generated Data	Deletes all generated files and loads the input data (reads the mesh/case files and initializes the case using the specified method).
Reset	Performs the same actions as the **Clear Generated Data** command, and in addition, sets the **Solution** cell property values to their defaults.

Table 5: Update Mesh/Setup/Solution

Command	Description
Update	Performs the calculation as specified in the case settings. Once the calculation is complete, the state of the **Solution** cell becomes **Up-To-Date**. This command is available only if no calculation has been performed yet.
Restart	Reinitializes the case using the selected initialization method and then performs the calculation. This command allows you to re-run the case after you have performed some iterations and changed the settings for you case. This command is available only after some calculations have been performed.
Continue	Continues the calculation from the current solution state until the specified number of iterations or time steps is reached, or the solution meets the convergence criteria. This command allows you to continue your calculation after altering the case settings. The changed settings will be saved for future calculations. This command is available only after some calculations have been performed. Note: • Any case modifications that you make after starting the calculation will not be saved or recorded for future calculations. • For parametric studies, you can enable **Automatically Initialize and Modify Case** in the **Calculation Activities** task page in Fluent in order to automatically apply the case modifications for each design point calculation.