ANSYS Fluent Tutorial Guide

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

Release 18.0
January 2017

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001: 2008 companies.

Published in the U.S.A.

Table of Contents

Using This Manual

This preface is divided into the following sections:

1. What's In This Manual

The ANSYS Fluent Tutorial Guide contains a number of tutorials that teach you how to use ANSYS Fluent to solve different types of problems. In each tutorial, features related to problem setup and postprocessing are demonstrated.

The tutorials are written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

and that you are familiar with the ANSYS Fluent navigation pane and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

All of the tutorials include some postprocessing instructions, but Postprocessing (p. 879) is devoted entirely to postprocessing.

2. The Contents of the Fluent Manuals

The manuals listed below form the Fluent product documentation set. They include descriptions of the procedures, commands, and theoretical details needed to use Fluent products. These manuals are all available on the ANSYS Customer Portal (http://support.ansys.com/documentation).

- Fluent Getting Started Guide contains general information about getting started with using Fluent and provides details about starting, running, and exiting the program.
- Fluent Migration Manual contains information about transitioning from the previous release of Fluent, including details about new features, solution changes, and text command list changes.
- Fluent User's Guide contains detailed information about running a simulation using the solution mode of Fluent, including information about the user interface, reading and writing files, defining boundary conditions, setting up physical models, calculating a solution, and analyzing your results.
- Fluent Meshing User's Guide contains detailed information about creating 3D meshes using the meshing mode of Fluent.

- Fluent in Workbench User's Guide contains information about getting started with and using Fluent within the Workbench environment.
- Fluent Theory Guide contains reference information for how the physical models are implemented in Fluent.
- Fluent Customization Manual contains information about writing and using user-defined functions (UDFs).
- Fluent Tutorial Guide contains a number of examples of various flow problems with detailed instructions, commentary, and postprocessing of results.

  Tutorials for release 18.0 are available on the ANSYS Customer Portal. To access tutorials and their input files on the ANSYS Customer Portal, go to http://support.ansys.com/training.
- ANSYS Fluent Meshing Tutorial Guide contains a number of examples of general mesh-generation techniques used in ANSYS Fluent Meshing.

  Tutorials for release 18.0 are available on the ANSYS Customer Portal. To access tutorials and their input files on the ANSYS Customer Portal, go to http://support.ansys.com/training.
- Fluent Text Command List contains a brief description of each of the commands in Fluent's solution mode text interface.
- Fluent Meshing Text Command List contains a brief description of each of the commands in Fluent's meshing mode text interface.
- Fluent Advanced Add-On Modules contains information about the usage of the different advanced Fluent add-on modules, which are applicable for specific modeling needs.
  - Part I: ANSYS Fluent Adjoint Solver contains information about the background and usage of Fluent's Adjoint Solver Module that allows you to obtain detailed sensitivity data for the performance of a fluid system.
  - Part II: ANSYS Fluent Battery Module contains information about the background and usage of Fluent's Battery Module that allows you to analyze the behavior of electric batteries.
  - Part III: ANSYS Fluent Continuous Fiber Module contains information about the background and usage of Fluent's Continuous Fiber Module that allows you to analyze the behavior of fiber flow, fiber properties, and coupling between fibers and the surrounding fluid due to the strong interaction that exists between the fibers and the surrounding gas.
  - Part IV: ANSYS Fluent Fuel Cell Modules contains information about the background and the usage of two separate add-on fuel cell models for Fluent that allow you to model polymer electrolyte membrane fuel cells (PEMFC), solid oxide fuel cells (SOFC), and electrolysis with Fluent.
  - Part V: ANSYS Fluent Magnetohydrodynamics (MHD) Module contains information about the background and usage of Fluent's Magnetohydrodynamics (MHD) Module that allows you to analyze the behavior of electrically conducting fluid flow under the influence of constant (DC) or oscillating (AC) electromagnetic fields.
  - Part VI: ANSYS Fluent Population Balance Module contains information about the background and usage of Fluent's Population Balance Module that allows you to analyze multiphase flows involving size distributions where particle population (as well as momentum, mass, and energy) require a balance equation.

- Fluent as a Server User's Guide contains information about the usage of Fluent as a Server which allows you to connect to a Fluent session and issue commands from a remote client application.
- Running ANSYS Fluent Using a Load Manager contains information about using third-party load managers with ANSYS Fluent.
  - Part I: Running ANSYS Fluent Under LSF contains information about using Fluent with Platform Computing's LSF software, a distributed computing resource management tool.
  - Part II: Running ANSYS Fluent Under PBS Professional contains information about using Fluent with Altair PBS Professional, an open workload management tool for local and distributed environments.
  - Part III: Running ANSYS Fluent Under SGE contains information about using Fluent with Univa Grid Engine (formerly Sun Grid Engine) software, a distributed computing resource management tool.

3. Where to Find the Files Used in the Tutorials

Each of the tutorials uses an existing mesh file. (Tutorials for mesh generation are provided with the mesh generator documentation.) You will find the appropriate mesh file (and any other relevant files used in the tutorial) on the ANSYS Customer Portal. The "Preparation" step of each tutorial will tell you where to find the necessary files. (Note that Tutorial Postprocessing (p. 879) use existing case and data files.)

4. How To Use This Manual

Depending on your familiarity with computational fluid dynamics and the ANSYS Fluent software, you can use this tutorial guide in a variety of ways.

4.1. For the Beginner

If you are a beginning user of ANSYS Fluent you should first read and solve Tutorial 1, in order to familiarize yourself with the interface and with basic setup and solution procedures. You may then want to try a tutorial that demonstrates features that you are going to use in your application. For example, if you are planning to solve a problem using the non-premixed combustion model, you should look at Using the Non-Premixed Combustion Model (p. 583).

You may want to refer to other tutorials for instructions on using specific features, such as custom field functions, mesh scaling, and so on, even if the problem solved in the tutorial is not of particular interest to you. To learn about postprocessing, you can look at Postprocessing (p. 879), which is devoted entirely to postprocessing (although the other tutorials all contain some postprocessing as well).

4.2. For the Experienced User

If you are an experienced ANSYS Fluent user, you can read and/or solve the tutorial(s) that demonstrate features that you are going to use in your application. For example, if you are planning to solve a problem using the non-premixed combustion model, you should look at Using the Non-Premixed Combustion Model (p. 583).

You may want to refer to other tutorials for instructions on using specific features, such as custom field functions, mesh scaling, and so on, even if the problem solved in the tutorial is not of particular interest to you. To learn about postprocessing, you can look at Postprocessing (p. 879), which is devoted entirely to postprocessing (although the other tutorials all contain some postprocessing as well).

5. Typographical Conventions Used In This Manual

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:

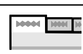
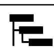
  Iso-Surface dialog box
  `surface/iso-surface` text command

- 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,

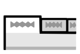
```
solve/initialize/set-fmg-initialization

 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.

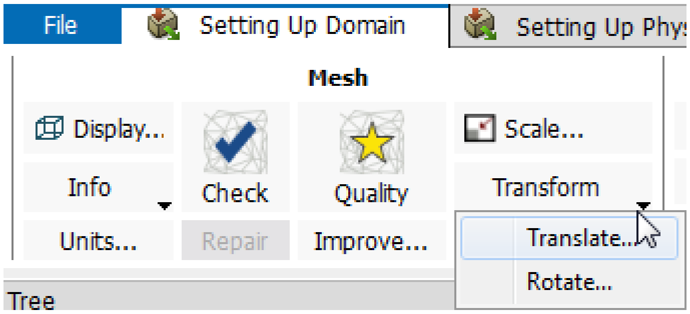
Table 1: Mini Flow Chart Symbol Descriptions

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...**

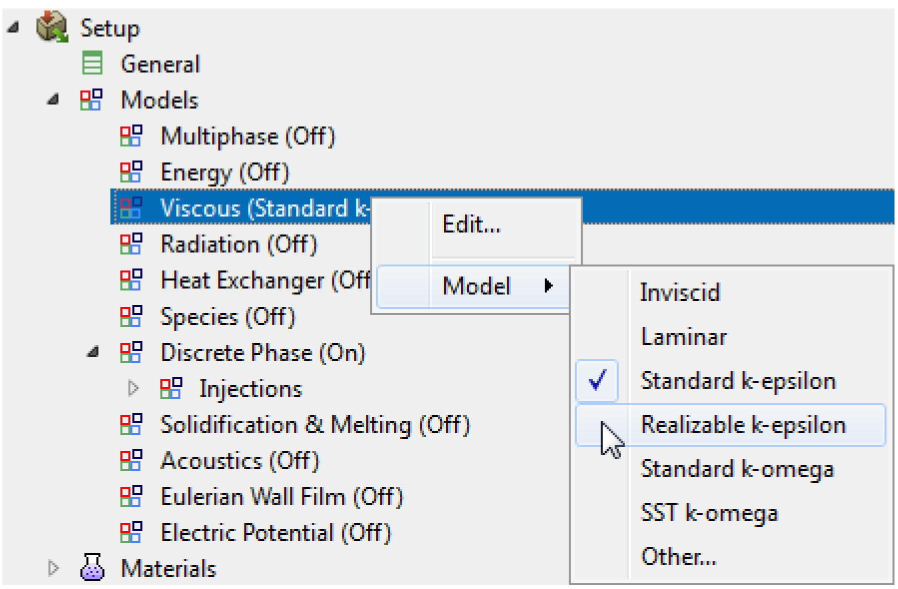
indicates selecting the **Setting Up Domain** ribbon tab, clicking **Transform** (in the **Mesh** group box) and selecting **Translate...**, as indicated in the figure below:



And

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

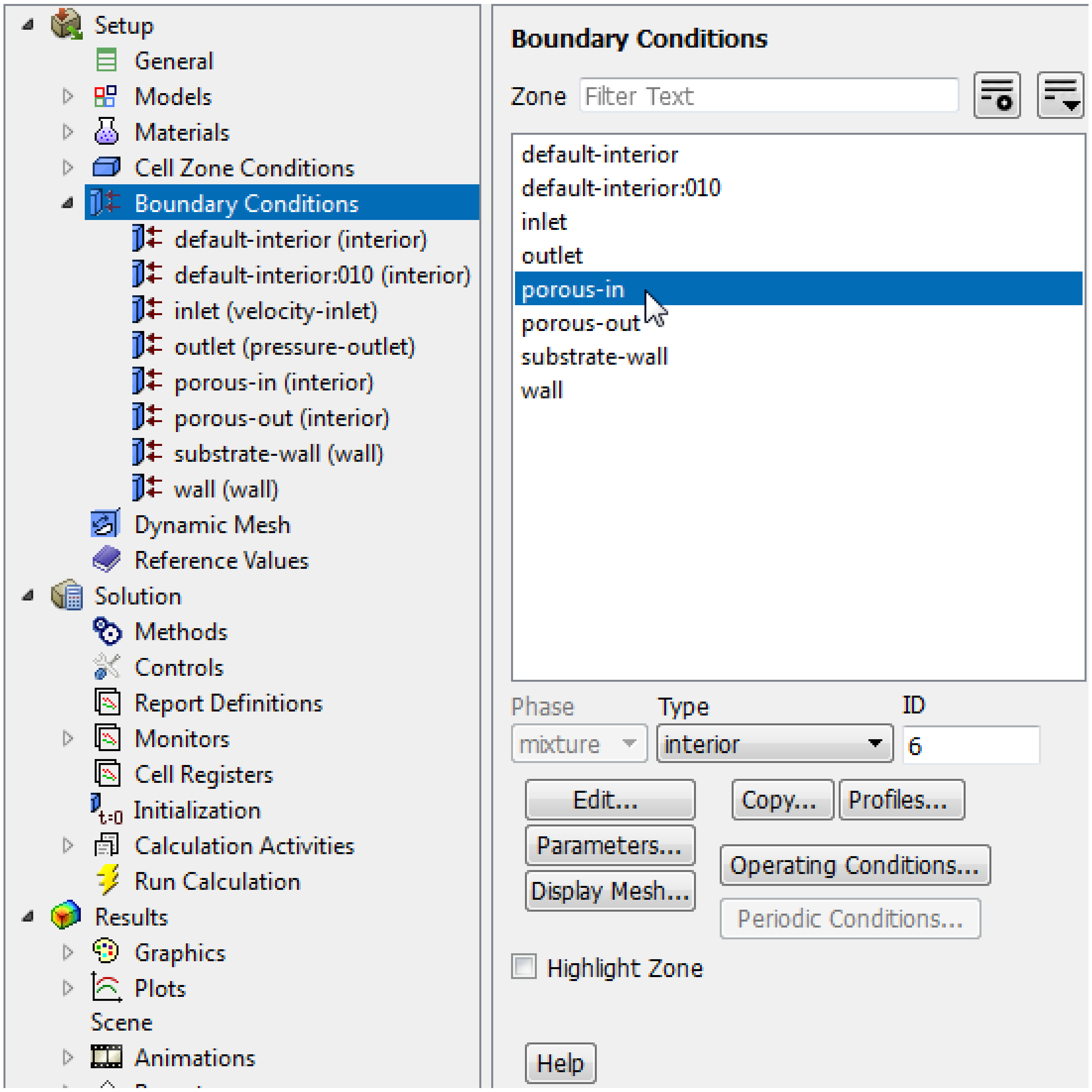
indicates expanding the **Setup** and **Models** branches, right-clicking **Viscous**, and selecting **Realizable k-epsilon** from the **Model** sub-menu, as shown in the following figure:



And

Setup → **Boundary Conditions** → **porous-in**

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).

Chapter 1: Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow

This tutorial is divided into the following sections:

1.1. Introduction

This tutorial illustrates using ANSYS Fluent fluid flow systems in ANSYS Workbench to set up and solve a three-dimensional turbulent fluid-flow and heat-transfer problem in a mixing elbow. It is designed to introduce you to the ANSYS Workbench tool set using a simple geometry. Guided by the steps that follow, you will create the elbow geometry and the corresponding computational mesh using the geometry and meshing tools within ANSYS Workbench. You will use ANSYS Fluent to set up and solve the CFD problem, then visualize the results in both ANSYS Fluent and in the CFD-Post postprocessing tool. Some capabilities of ANSYS Workbench (for example, duplicating fluid flow systems, connecting systems, and comparing multiple data sets) are also examined in this tutorial.

This tutorial demonstrates how to do the following:

- Launch ANSYS Workbench.
- Create a Fluent fluid flow analysis system in ANSYS Workbench.
- Create the elbow geometry using ANSYS DesignModeler.
- Create the computational mesh for the geometry using ANSYS Meshing.
- Set up the CFD simulation in ANSYS Fluent, which includes:
  - Setting material properties and boundary conditions for a turbulent forced-convection problem.
  - Initiating the calculation with residual plotting.
  - Calculating a solution using the pressure-based solver.
  - Examining the flow and temperature fields using ANSYS Fluent and CFD-Post.
- Create a copy of the original Fluent fluid flow analysis system in ANSYS Workbench.
- Change the geometry in ANSYS DesignModeler, using the duplicated system.
- Regenerate the computational mesh.
- Recalculate a solution in ANSYS Fluent.

- Compare the results of the two calculations in CFD-Post.

1.2. Prerequisites

This tutorial assumes that you have little to no experience with ANSYS Workbench, ANSYS DesignModeler, ANSYS Meshing, ANSYS Fluent, or CFD-Post, and so each step will be explicitly described.

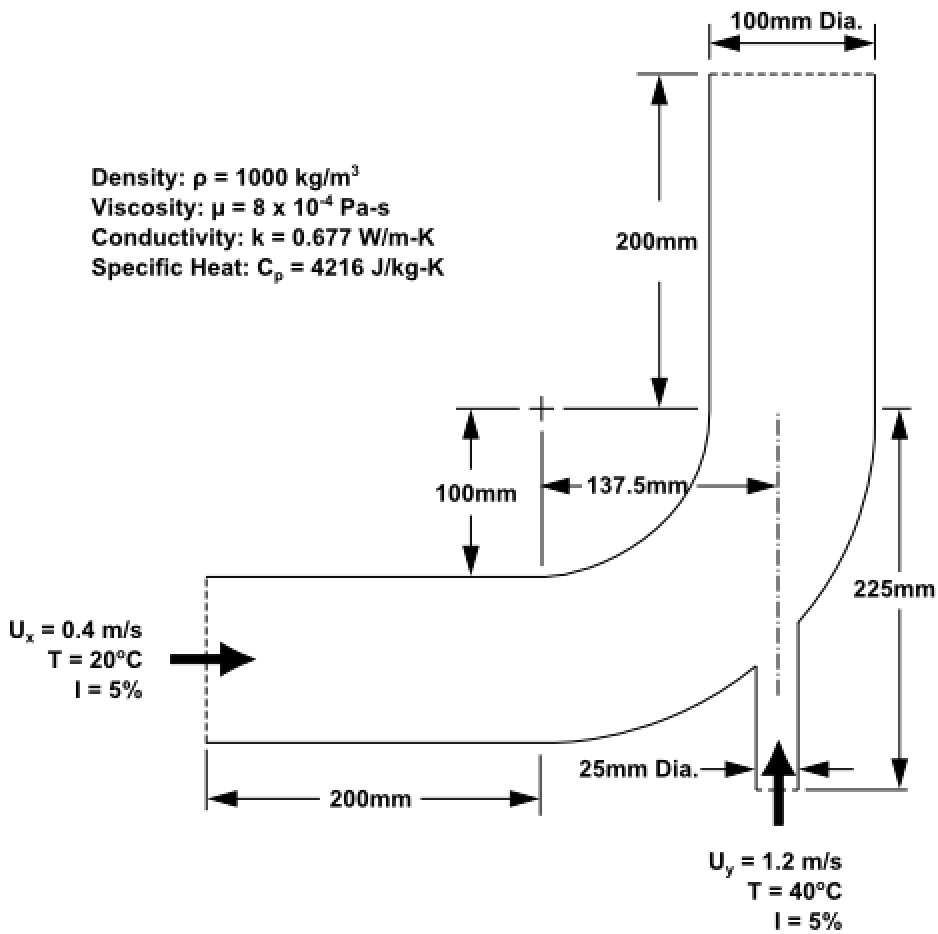
1.3. Problem Description

The problem to be considered is shown schematically in Figure 1.1: Problem Specification (p. 3). A cold fluid at 293.15 K flows into the pipe through a large inlet and mixes with a warmer fluid at 313.15 K that enters through a smaller inlet located at the elbow. The mixing elbow configuration is encountered in piping systems in power plants and process industries. It is often important to predict the flow field and temperature field in the area of the mixing region in order to properly design the junction.

Note

Because the geometry of the mixing elbow is symmetric, only half of the elbow must be modeled.

Figure 1.1: Problem Specification



Note

The functionality to create named selections exists in both ANSYS DesignModeler and ANSYS Meshing. For the purposes of this tutorial, named selections are created in ANSYS Meshing since the meshing application provides more comprehensive and extensive named selection functionality.

1.4. Setup and Solution

To help you quickly identify graphical user interface items at a glance and guide you through the steps of setting up and running your simulation, the ANSYS Fluent Tutorial Guide uses several type styles and mini flow charts. See Typographical Conventions Used In This Manual (p. xvi) for detailed information.

The following sections describe the setup and solution steps for this tutorial:

1.4.1. Preparation

1. Set up a working folder on the computer you will be using.

2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

   Note

   If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

3. Enter the name of this tutorial into the search bar.

4. Narrow the results by using the filter on the left side of the page.

   a. Click **ANSYS Fluent** under **Product**.

   b. Click **18.0** under **Version**.

5. Select this tutorial from the list.

6. Click the **elbow-workbench_R180.zip** link to download the input files.

7. Unzip `elbow-workbench_R180.zip` to your working folder. This file contains a folder, `elbow-workbench`, that holds the following items:

   - two geometry files, `elbow_geometry.agdb` and `elbow_geometry.stp`
   - an ANSYS Workbench project archive, `elbow-workbench.wbpz`

   Tip

   The Workbench project archive contains the project as it will be once you have completed all of the steps of the tutorial and is included for reference. If you want to extract the project archive, start Workbench and select the **File** → **Restore Archive...** menu item. You will be prompted with a dialog box to specify a location in which to extract the project and its supporting files. You may choose any convenient location.

Note

ANSYS Fluent tutorials are prepared using ANSYS Fluent on a Windows system. The screen shots and graphic images in the tutorials may be slightly different than the appearance on your system, depending on the operating system or graphics card.

1.4.2. Creating a Fluent Fluid Flow Analysis System in ANSYS Workbench

In this step, you will start ANSYS Workbench, create a new Fluent fluid flow analysis system, then review the list of files generated by ANSYS Workbench.

1. From the Windows **Start** menu, select **Start > All Programs > ANSYS 18.0** > **Workbench 18.0** to start a new ANSYS Workbench session.

   This displays the ANSYS Workbench application window, which has the **Toolbox** on the left and the **Project Schematic** to its right. Various supported applications are listed in the **Toolbox** and the components of the analysis system will be displayed in the **Project Schematic**.

   Note

   Depending on which other products you have installed, the analysis systems that appear may differ from those in the figures that follow in this tutorial.

   ---

2. Create a new Fluent fluid flow analysis system by double-clicking the **Fluid Flow (Fluent)** option under **Analysis Systems** in the **Toolbox**.

   Tip

   You can also drag-and-drop the analysis system into the **Project Schematic**. A green dotted outline indicating a potential location for the new system initially appears in the **Project Schematic**. When you drag the system to one of the outlines, it turns into a red box to indicate the chosen location of the new system.

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

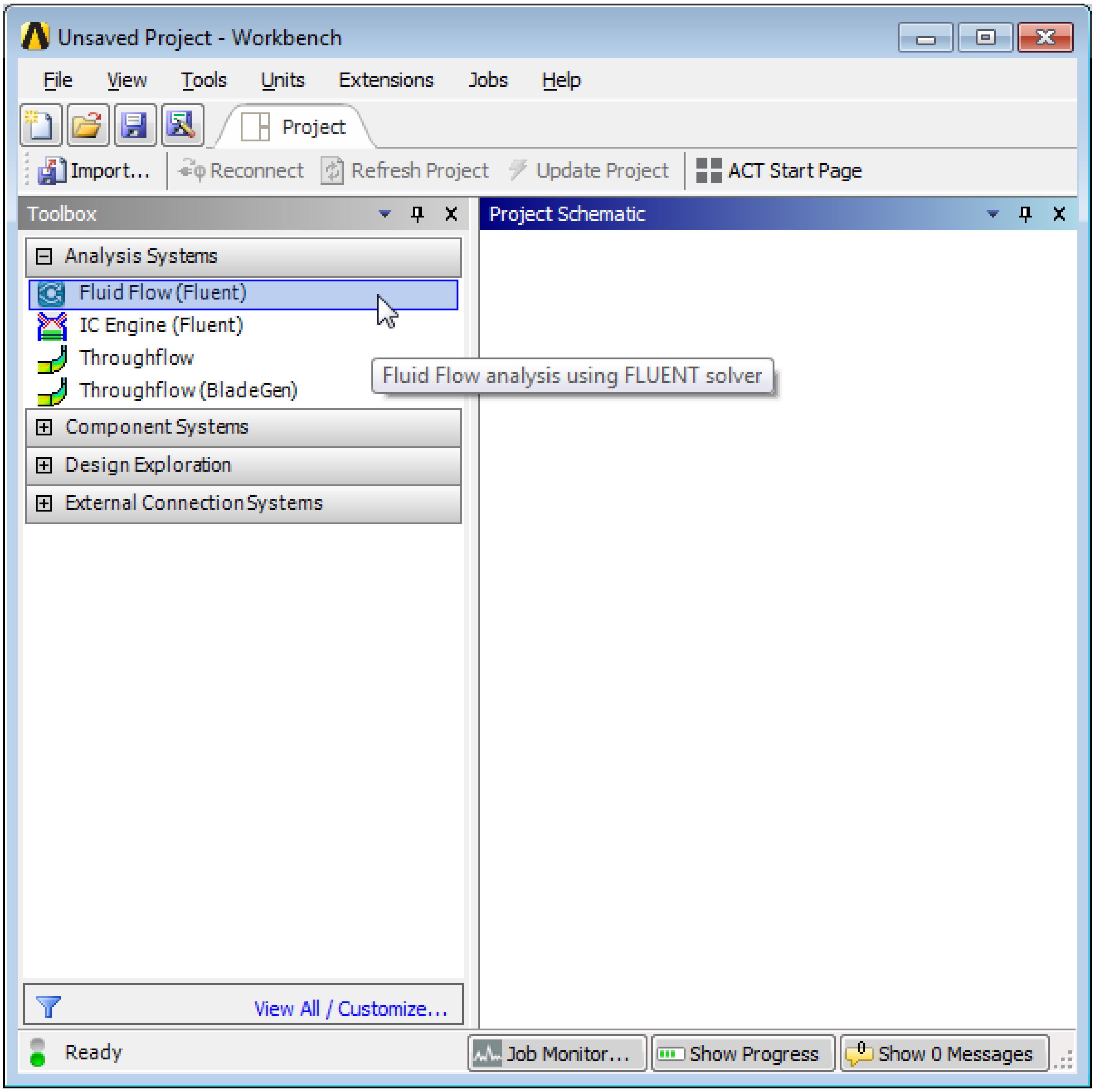
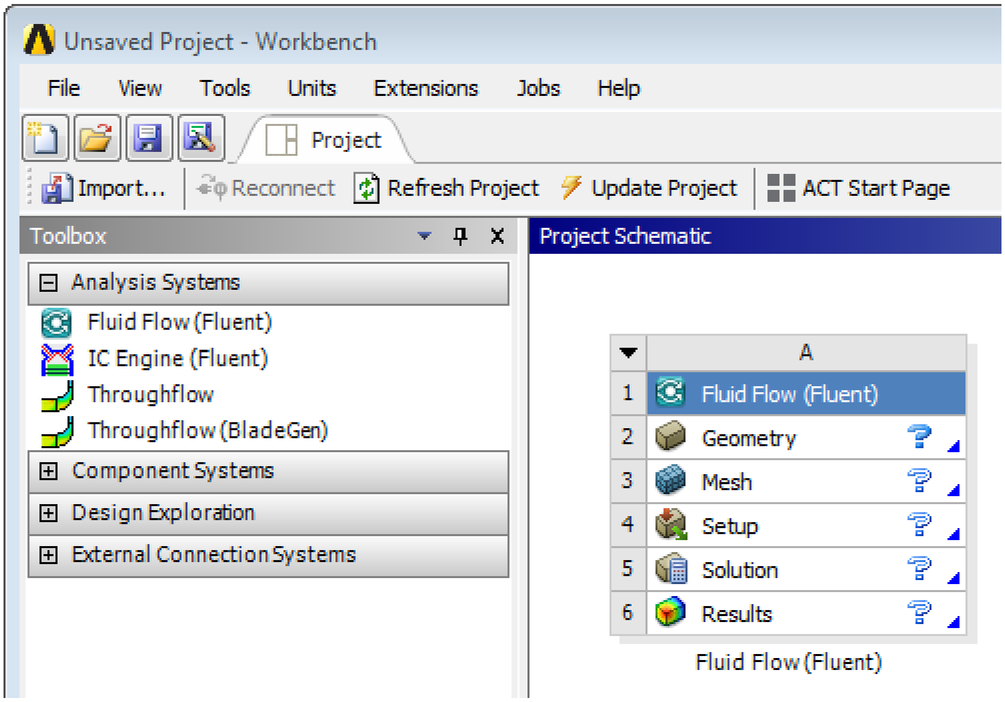


Figure 1.3: ANSYS Workbench with a New Fluent-Based Fluid Flow Analysis System



3. Name the analysis.

   a. Double-click the **Fluid Flow (Fluent)** label underneath the analysis system (if it is not already highlighted).

   b. Enter `elbow` for the name of the analysis system.

4. Save the project.

   a. In ANSYS Workbench, under the **File** menu, select **Save**.

      File → **Save**

      This displays the **Save As** dialog box, where you can browse to your working folder and enter a specific name for the ANSYS Workbench project.

   b. In your working directory, enter `elbow-workbench` as the project **File name** and click the **Save** button to save the project. ANSYS Workbench saves the project with a `.wbpj` extension and also saves supporting files for the project.

      Note that the fluid flow analysis system is composed of various *cells* (**Geometry**, **Mesh**, etc.) that represent the workflow for performing the analysis. ANSYS Workbench is composed of multiple data-integrated and native applications in a single, seamless project flow, where individual cells can obtain data from other cells and provide data to other cells. As a result of this constant flow of data, a cell's state can quickly change. ANSYS Workbench provides a visual indication of a cell's state at any given time via icons on the right side of each cell. Brief descriptions of the various states are provided below:

      - **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 ANSYS 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 ANSYS 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 must generate an ANSYS Fluent mesh file, but the ANSYS 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 that an interactive calculation has been completed successfully. For example, after ANSYS 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 ANSYS Fluent while it is iterating, ANSYS 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.

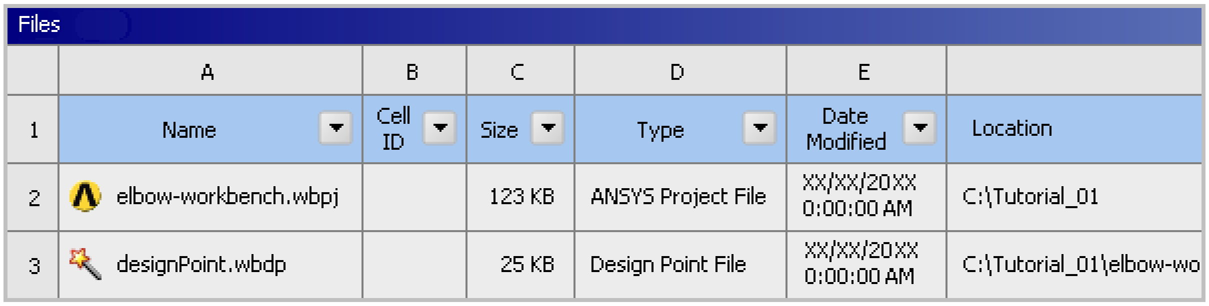
For more information about cell states, see the Workbench User's Guide.

5. View the list of files generated by ANSYS Workbench.

   ANSYS Workbench allows you to easily view the files associated with your project using the **Files** view. To open the **Files** view, select the **Files** option under the **View** menu at the top of the ANSYS Workbench window.

   View → Files

Figure 1.4: ANSYS Workbench Files View for the Project After Adding a Fluent-Based Fluid Flow Analysis System



Files

	A	B	C	D	E	
1	Name	Cell ID	Size	Type	Date Modified	Location
2	elbow-workbench.wbpj		123 KB	ANSYS Project File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01
3	designPoint.wbdp		25 KB	Design Point File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo

In the **Files** view, you will be able to see the name and type of file, the ID of the cell that the file is associated with, the size of the file, the location of the file, and other information. For more information about the **Files** view, see the Workbench User's Guide.

Note

The sizes of the files listed may differ slightly from those portrayed in Figure 1.4: ANSYS Workbench Files View for the Project After Adding a Fluent-Based Fluid Flow Analysis System (p. 9).

From here, you will create the geometry described in Figure 1.1: Problem Specification (p. 3), and later create a mesh and set up a fluid flow analysis for the geometry.

1.4.3. Creating the Geometry in ANSYS DesignModeler

For the geometry of your fluid flow analysis, you can create a geometry in ANSYS DesignModeler, ANSYS SpaceClaim Direct Modeler, or import the appropriate geometry file. In this step, you will create the geometry in ANSYS DesignModeler, then review the list of files generated by ANSYS Workbench.

Important

Note the **Attention Required** icon ( ) within the **Geometry** cell for the system. This indicates that the cell requires data (for example, a geometry). Once the geometry is defined, the state of the cell will change accordingly. Likewise, the state of some of the remaining cells in the system will change.

Note

If you would rather not create the geometry in ANSYS DesignModeler, you can import a pre-existing geometry by right-clicking the **Geometry** cell and selecting the **Import Geometry** option from the context menu. From there, you can browse your file system to locate the `elbow_geometry.agdb` geometry file that is provided for this tutorial. If you do not have access to ANSYS DesignModeler, you can use the `elbow_geometry.stp` file instead.

To learn how to create a mesh from the geometry you imported, go to Meshing the Geometry in the ANSYS Meshing Application (p. 20).

1. Start ANSYS DesignModeler.

In the ANSYS Workbench **Project Schematic**, right-click the **Geometry** cell in the `elbow` fluid flow analysis system to display the context menu, then select **New DesignModeler Geometry...**. This displays the ANSYS DesignModeler application.

2. Set the units in ANSYS DesignModeler.

   When ANSYS DesignModeler first appears, you should select desired system of length units to work from. For the purposes of this tutorial (where you will create the geometry in millimeters and perform the CFD analysis using SI units) set the units to **Millimeter**.

   Units → **Millimeter**

3. Create the geometry.

   The geometry for this tutorial (Figure 1.1: Problem Specification (p. 3)) consists of a large curved pipe accompanied by a smaller side pipe. ANSYS DesignModeler provides various geometry primitives that can be combined to rapidly create geometries such as this one. You will perform the following tasks to create the geometry:

   - Create the bend in the main pipe by defining a segment of a torus.
   - Extrude the faces of the torus segment to form the straight inlet and outlet lengths.
   - Create the side pipe by adding a cylinder primitive.
   - Use the symmetry tool to reduce the model to half of the pipe assembly, thus reducing computational cost.

   a. Create the main pipe:

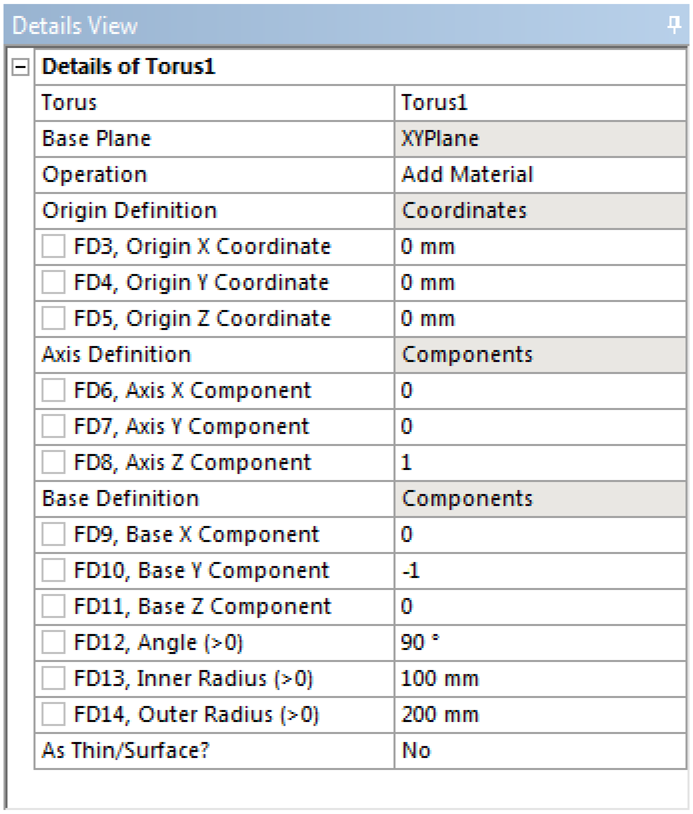
      i. Create a new torus for the pipe bend by choosing the **Create** → **Primitives** → **Torus** menu item from the menubar.

         A preview of the torus geometry will appear in the graphics window. Note that this is a preview and the geometry has not been created yet. First you must specify the parameters of the torus primitive in the next step.

      ii. In the **Details View** for the new torus (**Torus1**), set **Base Y Component** to `-1` by clicking the **1** to the right of **FD10, Base Y Component**, entering `-1`, and pressing **Enter**. This specifies the direction vector from the origin to the center of the circular cross-section at the start of the torus. In the same manner, specify **Angle**; **Inner Radius**; and **Outer Radius** as shown below.

         ### Note

         Enter only the value without the units of `mm`. They will be appended automatically because you specified the units previously.

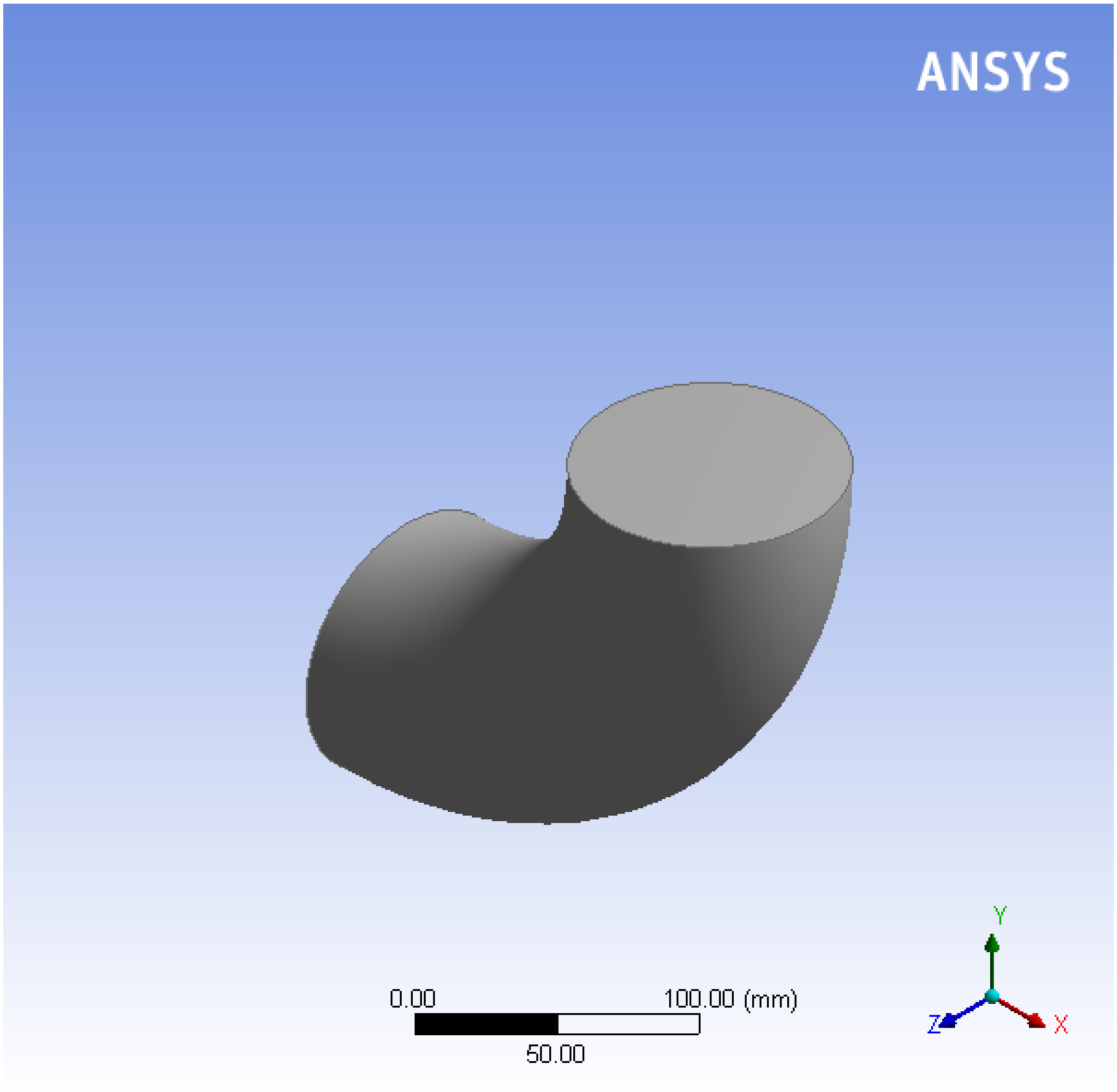


Details View

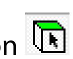
Details of Torus1	
Torus	Torus1
Base Plane	XYPlane
Operation	Add Material
Origin Definition	Coordinates
FD3, Origin X Coordinate	0 mm
FD4, Origin Y Coordinate	0 mm
FD5, Origin Z Coordinate	0 mm
Axis Definition	Components
FD6, Axis X Component	0
FD7, Axis Y Component	0
FD8, Axis Z Component	1
Base Definition	Components
FD9, Base X Component	0
FD10, Base Y Component	-1
FD11, Base Z Component	0
FD12, Angle (>0)	90 °
FD13, Inner Radius (>0)	100 mm
FD14, Outer Radius (>0)	200 mm
As Thin/Surface?	No

iii. To create the torus segment, click the **Generate** button 

that is located in the ANSYS DesignModeler toolbar.



The `Torus1` item appears in the **Tree Outline** view. If you want to delete this item, you can right-click it and select **Delete** from the context menu that opens.

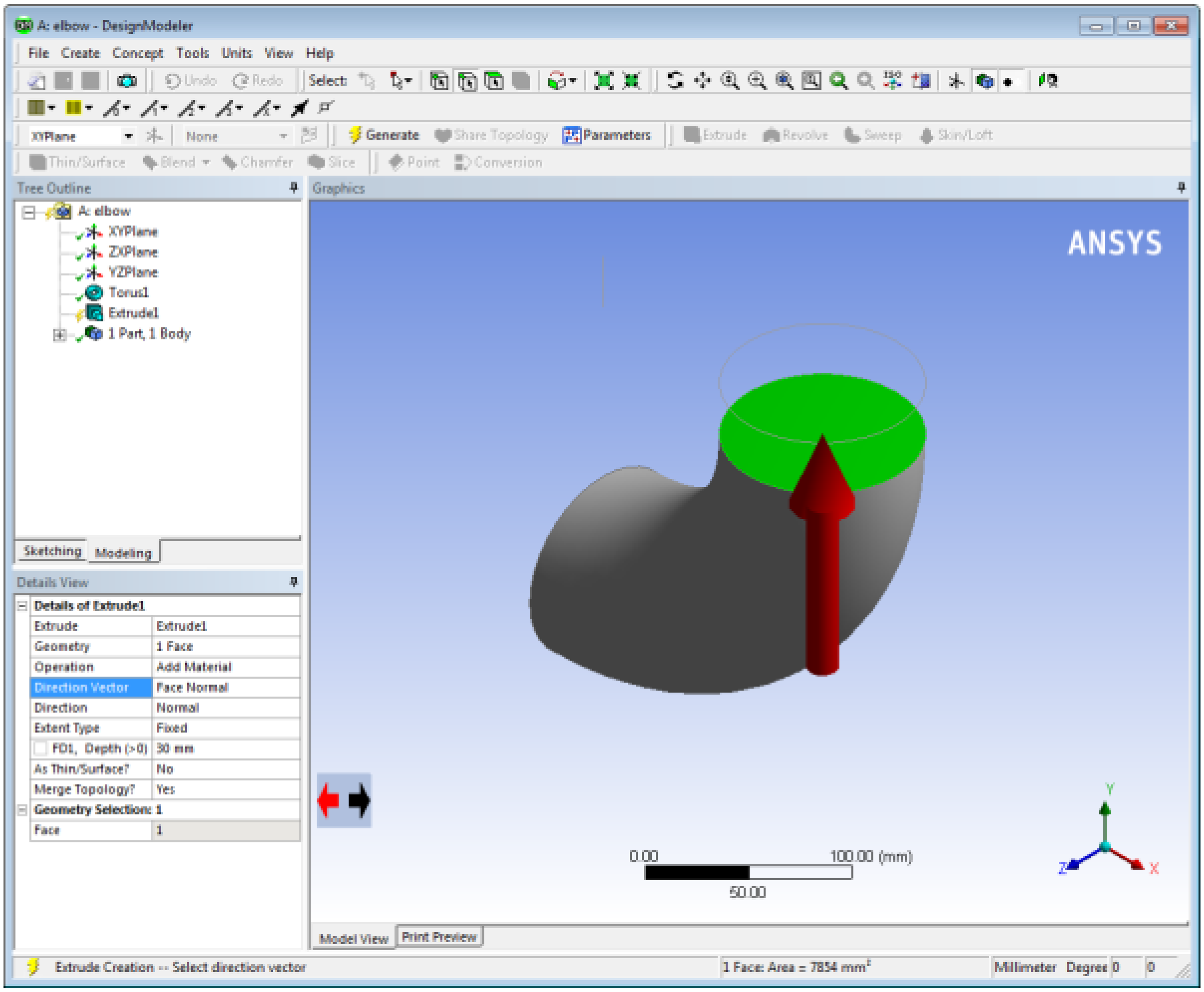
iv. Ensure that the selection filter is set to **Faces**. This is indicated by the **Faces** button  appearing depressed in the toolbar and the appearance of the Face selection cursor,  when you mouse over the geometry.

v. Select the top face (in the positive Y direction) of the elbow and click the **Extrude** button 

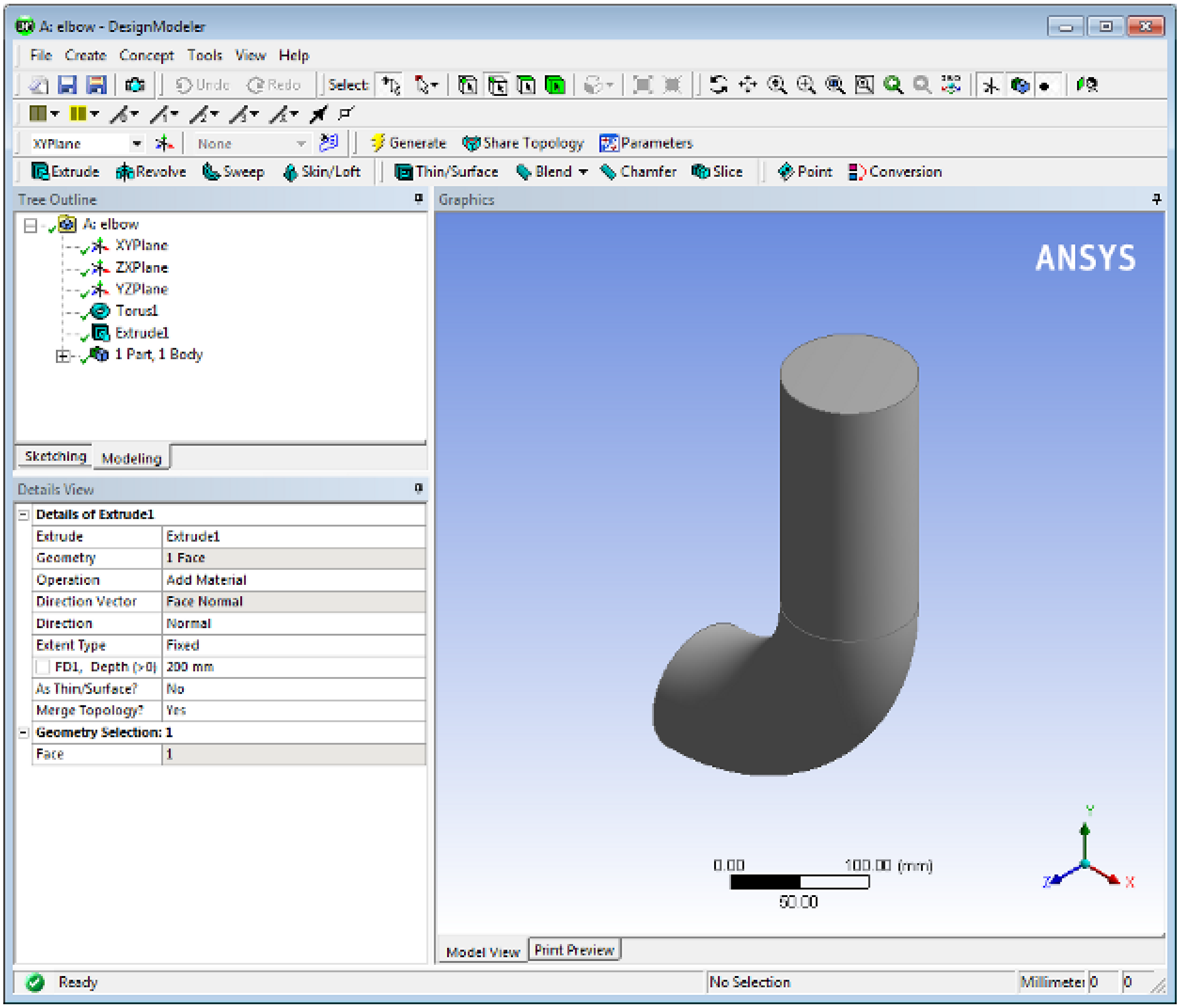
from the **3D Features** toolbar.

vi. In the **Details View** for the new extrusion (**Extrude1**), click **Apply** to the right of **Geometry**. This accepts the face you selected as the base geometry of the extrusion.

vii. Click **None (Normal)** to the right of **Direction Vector**. Again, ensure that the selection filter is set to **Faces**, select the same face on the elbow to specify that the extrusion will be normal to the face and click **Apply**.



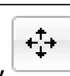
viii. Enter `200` for **FD1, Depth (>0)** and click **Generate**.

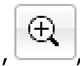


ix. In a similar manner, create an extrusion of the other face of the torus segment to create the 200 mm inlet extension. You will probably find it helpful to rotate the view so that you can easily select the other face of the bend.

You can use the mouse buttons to change your view of the 3D image. The following table describes mouse actions that are available:

Table 1.1: DesignModeler View Manipulation Instructions

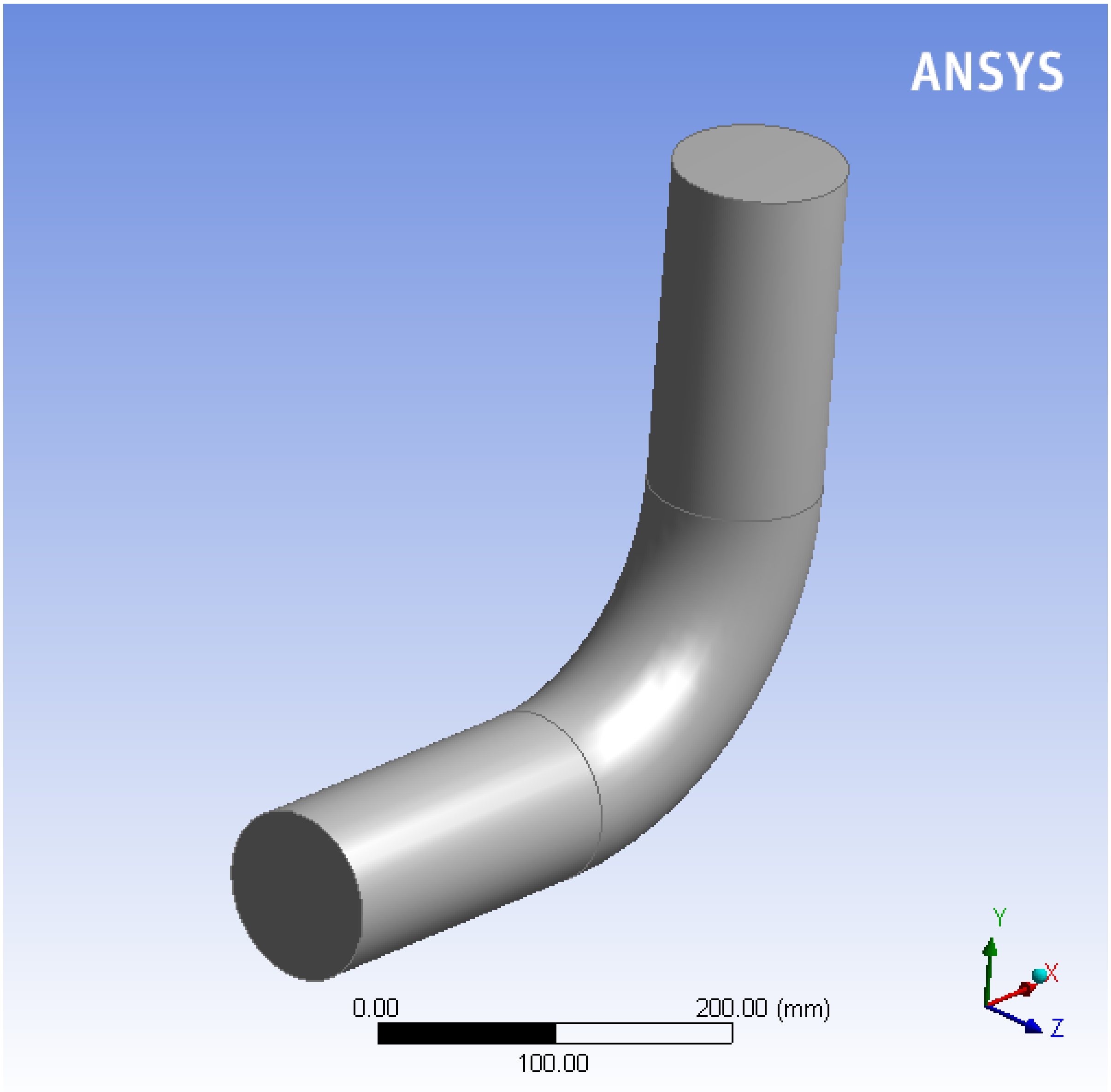
Action	Using Graphics Toolbar Buttons and the Mouse
Rotate view (vertical, horizontal)	After clicking the **Rotate** icon, , press and hold the left mouse button and drag the mouse. Dragging side to side rotates the view about the vertical axis, and dragging up and down rotates the view about the horizontal axis.
Translate or pan view	After clicking the **Pan** icon, , press and hold the left mouse button and drag the object with the mouse until the view is satisfactory.

Action	Using Graphics Toolbar Buttons and the Mouse
Zoom in and out of view	After clicking the **Zoom** icon, , press and hold the left mouse button and drag the mouse up and down to zoom in and out of the view.
Box zoom	After clicking the **Box Zoom** icon, , press and hold the left mouse button and drag the mouse diagonally across the screen. This action will cause a rectangle to appear in the display. When you release the mouse button, a new view will be displayed that consists entirely of the contents of the rectangle.

Clicking the **Zoom to Fit** icon, , will cause the object to fit exactly and be centered in the window.

After entering the extrusion parameters and clicking **Generate**, the geometry should appear as in Figure 1.5: Elbow Main Pipe Geometry (p. 16).

Figure 1.5: Elbow Main Pipe Geometry



b. Next you will use a cylinder primitive to create the side pipe.

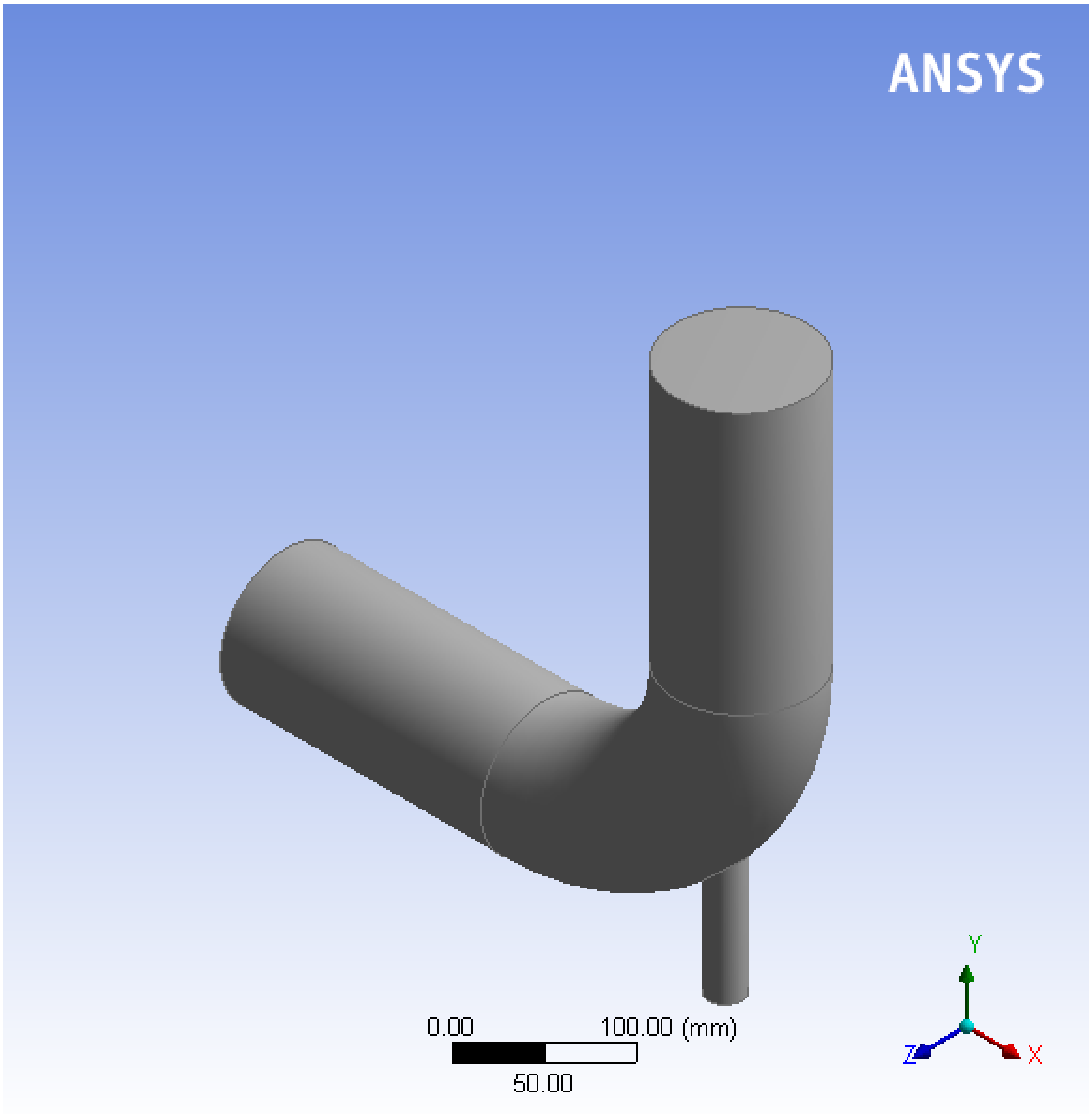
   i. Choose **Create** → **Primitives** → **Cylinder** from the menubar.

   ii. In the **Details View**, set the parameters for the cylinder as follows and click **Generate**:

Setting	Value
BasePlane	`XYPlane`
FD3, Origin X Coordinate	`137.5`
FD4, Origin Y Coordinate	`-225`
FD5, Origin Z Coordinate	`0`
FD6, Axis X Component	`0`
FD7, Axis Y Component	`125`

Setting	Value
FD8, Axis Z Component	`0`
FD10, Radius (>0)	`12.5`

The Origin Coordinates determine the starting point for the cylinder and the Axis Components determine the length and orientation of the cylinder body.



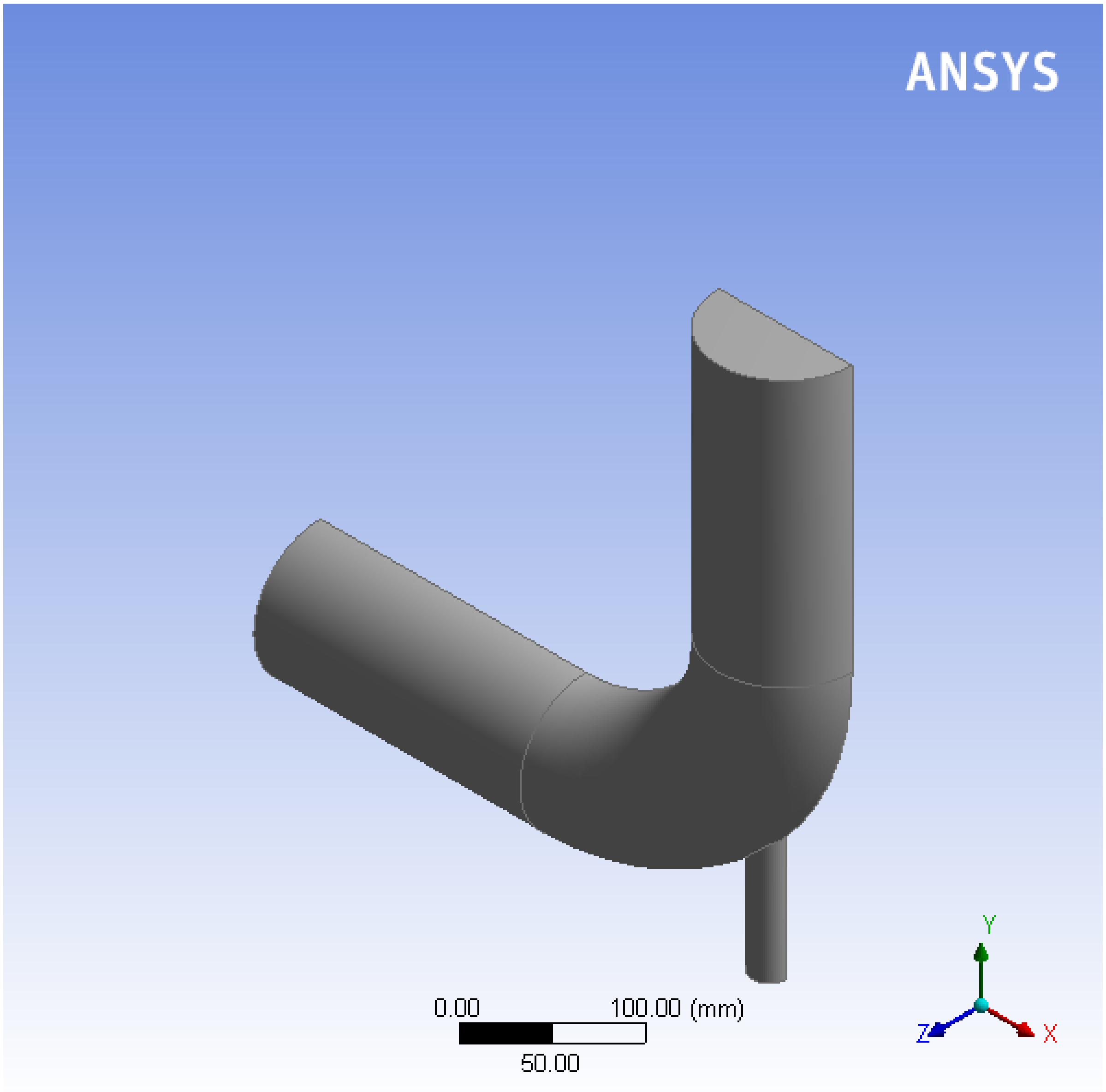
c. The final step in creating the geometry is to split the body on its symmetry plane which will halve the computational domain.

   i. Choose **Tools** → **Symmetry** from the menu bar.

   ii. Select the **XYPlane** in the **Tree Outline**.

   iii. Click **Apply** next to **Symmetry Plane 1** in the **Details** view.

iv. Click **Generate**.



The new surface created with this operation will be assigned a symmetry boundary condition in Fluent so that the model will accurately reflect the physics of the complete elbow geometry even though only half of it is meshed.

d. Specify the geometry as a fluid body.

i. In the **Tree Outline**, open the **1 Part, 1 Body** branch and select **Solid**.

ii. In the **Details View** of the body, change the name of the **Body** from **Solid** to **Fluid**.

iii. In the **Fluid/Solid** section, select **Fluid** from the drop-down list.

Details View

Details of Body	
Body	Fluid
Volume	2.5159e+006 mm³
Surface Area	1.7636e+005 mm²
Faces	8
Edges	18
Vertices	12
Fluid/Solid	Fluid
Shared Topology Method	Automatic
Geometry Type	DesignModeler

iv. Click **Generate**.

Tip

In addition to the primitives you used in this tutorial, ANSYS DesignModeler offers a full suite of 2D sketching and 3D solid modeling tools for creating arbitrary geometry. Refer to the DesignModeler User's Guide for more information.

4. Close ANSYS DesignModeler by selecting **File** → **Close DesignModeler** or by clicking the 'X' icon in the upper right-hand corner. ANSYS Workbench automatically saves the geometry and updates the **Project Schematic** accordingly. The question mark in the **Geometry** cell is replaced by a check mark, indicating that there is a geometry now associated with the fluid flow analysis system.

5. View the list of files generated by ANSYS Workbench by selecting **View** → **Files**.

Figure 1.6: ANSYS Workbench Files View for the Project After Creating the Geometry

Files

	A	B	C	D	E	
1	Name	Cell ID	Size	Type	Date Modified	Location
2	elbow-workbench.wbpj		124 KB	ANSYS Project File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01
3	FFF.agdb	A2	2 MB	Geometry File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
4	designPoint.wbdp		26 KB	Design Point File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo

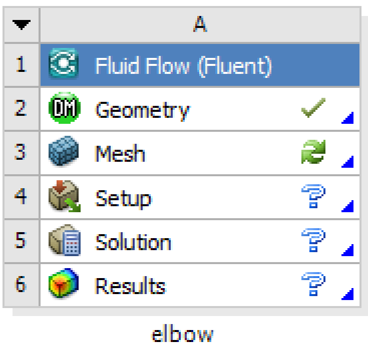
Note the addition of the geometry file (`FFF.agdb`, where FFF indicates a Fluent-based fluid flow system) to the list of files. If you had imported the geometry file provided for this tutorial rather than creating the geometry yourself, the `elbow_geometry.agdb` (or the `elbow_geometry.stp`) file would be listed instead.

1.4.4. Meshing the Geometry in the ANSYS Meshing Application

Now that you have created the mixing elbow geometry, you must generate a computational mesh throughout the flow volume. For this section of the tutorial, you will use the ANSYS Meshing application to create a mesh for your CFD analysis, then review the list of files generated by ANSYS Workbench.

Important

Note the **Refresh Required** icon ( ) within the **Mesh** cell for the system. This indicates that the state of the cell requires a refresh and that upstream data has changed since the last refresh or update (such as an update to the geometry). Once the mesh is defined, the state of the **Mesh** cell will change accordingly, as will the state of the next cell in the system, in this case the **Setup** cell.

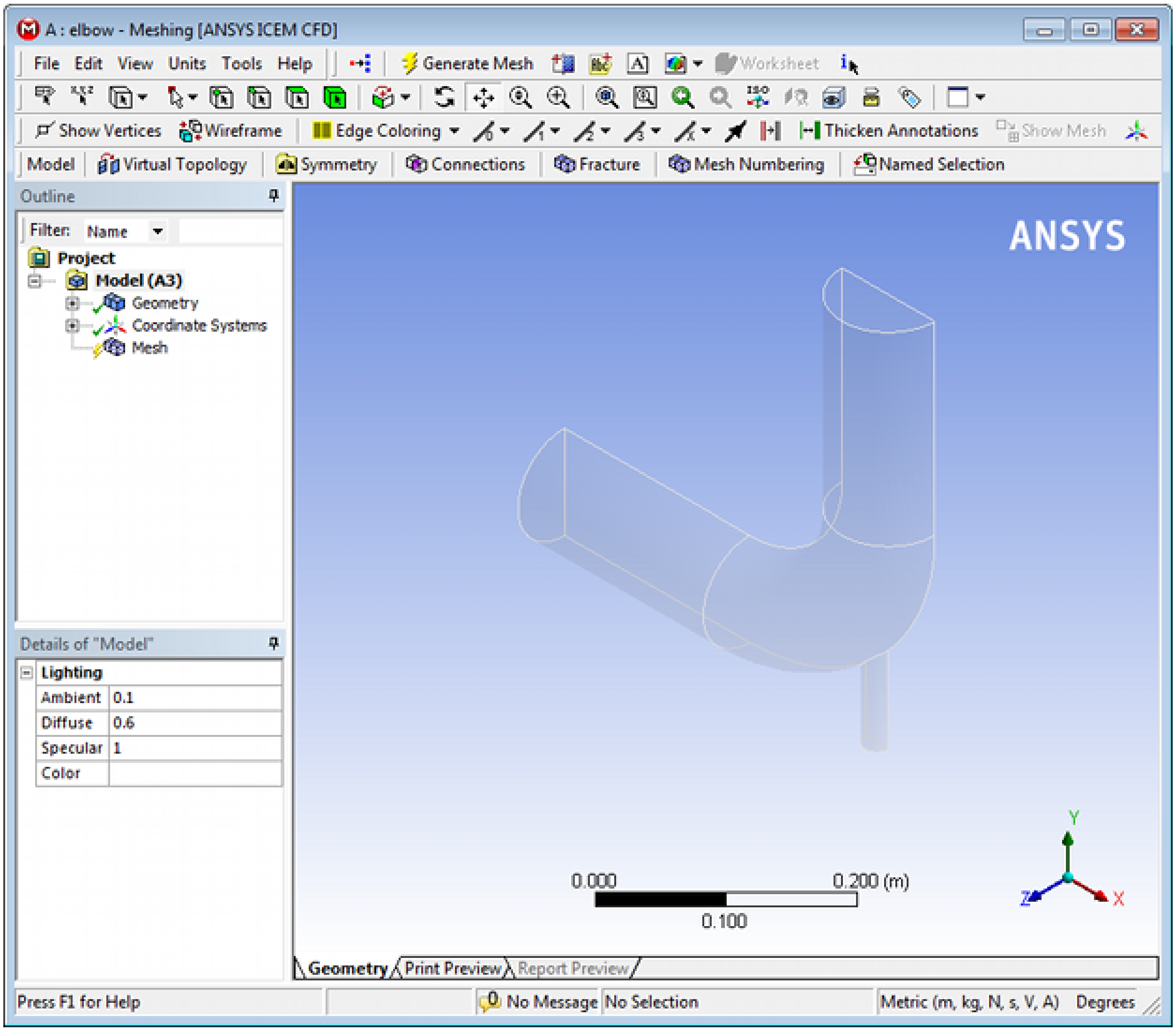


---

1. Open the ANSYS Meshing application.

   In the ANSYS Workbench **Project Schematic**, double-click the **Mesh** cell in the `elbow` fluid flow analysis system (cell A3). This displays the ANSYS Meshing application with the elbow geometry already loaded. You can also right-click the **Mesh** cell to display the context menu where you can select the **Edit...** option.

Figure 1.7: The ANSYS Meshing Application with the Elbow Geometry Loaded



2. Create named selections for the geometry boundaries.

   In order to simplify your work later on in ANSYS Fluent, you should label each boundary in the geometry by creating named selections for the pipe inlets, the outlet, and the symmetry surface (the outer wall boundaries are automatically detected by ANSYS Fluent).

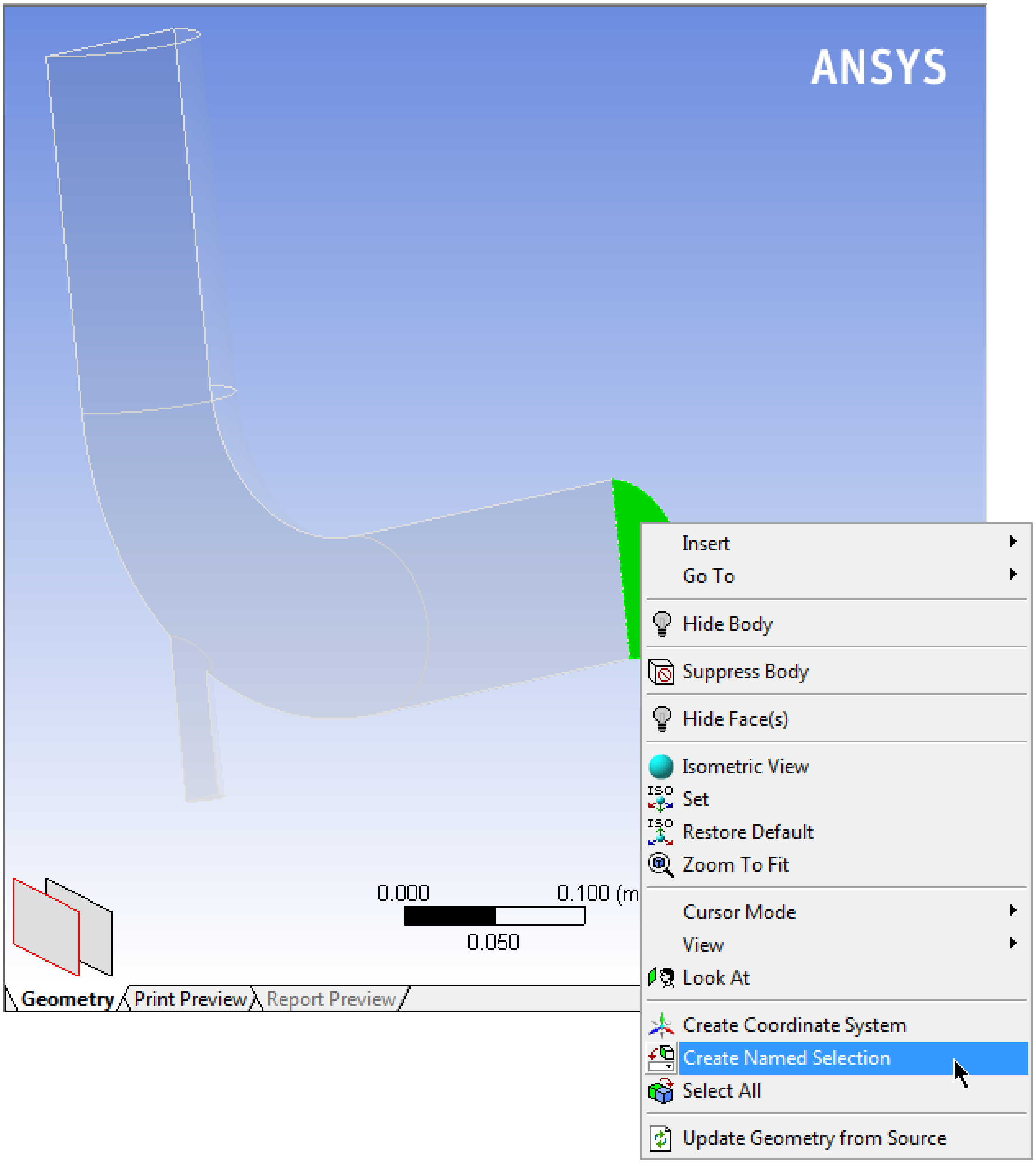
   a. Select the large inlet in the geometry that is displayed in the ANSYS Meshing application.

   ### Tip

   - Use the Graphics Toolbar buttons and the mouse to manipulate the image until you can easily see the pipe openings and surfaces.
   - To select the inlet, the *Single select* () mode must be active.

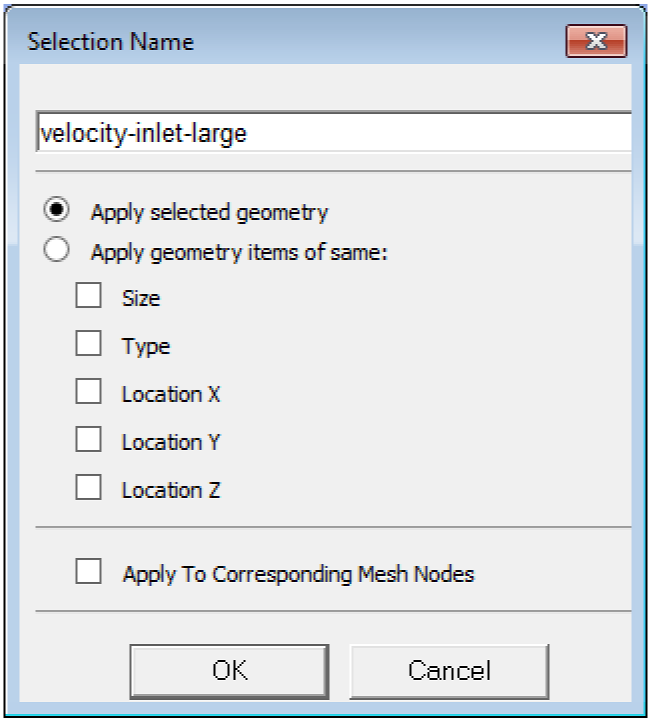
   b. Right-click and select the **Create Named Selection** option.

Figure 1.8: Selecting a Face to Name



This displays the **Selection Name** dialog box.

Figure 1.9: Applying a Name to a Selected Face



c. In the **Selection Name** dialog box, enter `velocity-inlet-large` for the name and click **OK**.

d. Perform the same operations for:

- The small inlet (`velocity-inlet-small`)
- The large outlet (`pressure-outlet`)
- The symmetry plane (`symmetry`).

The named selections that you have created appear under the **Named Selections** item in the **Outline** view.

Important

It is important to note that by using the strings “velocity inlet” and “pressure outlet” in the named selections (with or without hyphens or underscore characters), ANSYS Fluent automatically detects and assigns the corresponding boundary types accordingly.

3. Create a named selection for the fluid body.

a. Change the selection filter to **Body** in the **Graphics Toolbar** ().

b. Click the elbow in the graphics display to select it.

c. Right-click, select the **Create Named Selection** option and name the body `Fluid`.

By creating a named selection called `Fluid` for the fluid body you will ensure that ANSYS Fluent automatically detects that the volume is a fluid zone and treats it accordingly.

4. Set basic meshing parameters for the ANSYS Meshing application.

   For this analysis, you will adjust several meshing parameters to obtain a finer mesh.

   a. In the **Outline** view, select **Mesh** under **Project/Model** to display the **Details of "Mesh"** view below the **Outline** view.

      Important

      Note that because the ANSYS Meshing application automatically detects that you are going to perform a CFD fluid flow analysis using ANSYS Fluent, the **Physics Preference** is already set to **CFD** and the **Solver Preference** is already set to **Fluent**.

      ---

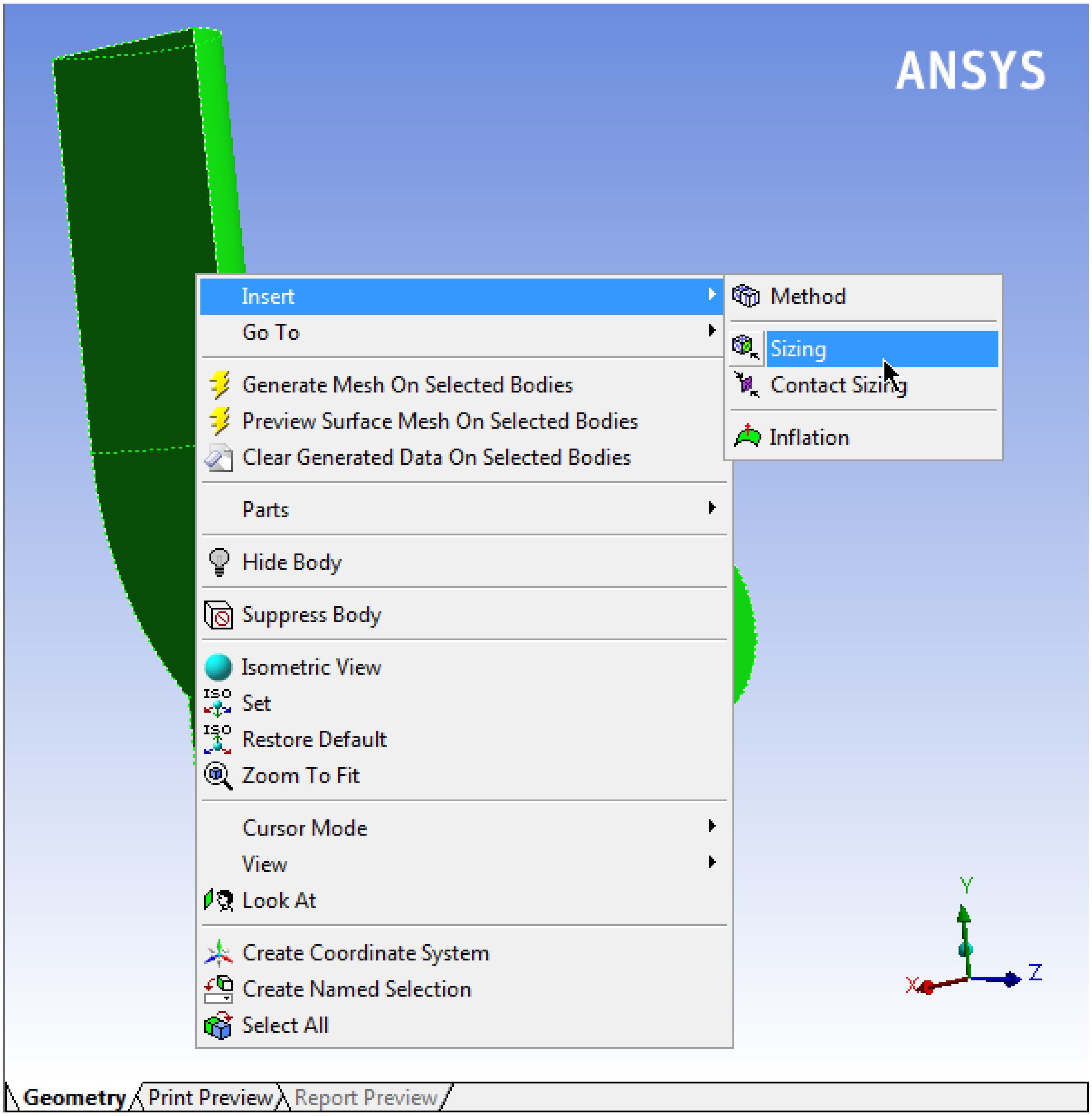
   b. Expand the **Sizing** node by clicking the "+" sign to the left of the word **Sizing** to reveal additional sizing parameters. Change **Relevance Center** to **Fine** by clicking the default value, **Coarse**, and selecting **Fine** from the drop-down list.

   c. Expand the **Quality** node to reveal additional quality parameters. Change **Smoothing** to **High**.

   d. Add a **Body Sizing** control.

      i. With **Mesh** still selected in the **Outline** tree, click the elbow in the graphics display to select it.

      ii. Right click in the graphics area and select **Insert** → **Sizing** from the context menu.



*A new **Body Sizing** entry appears under **Mesh** in the project **Outline** tree*

iii. In the **Outline** tree, click the new **Body Sizing** control.

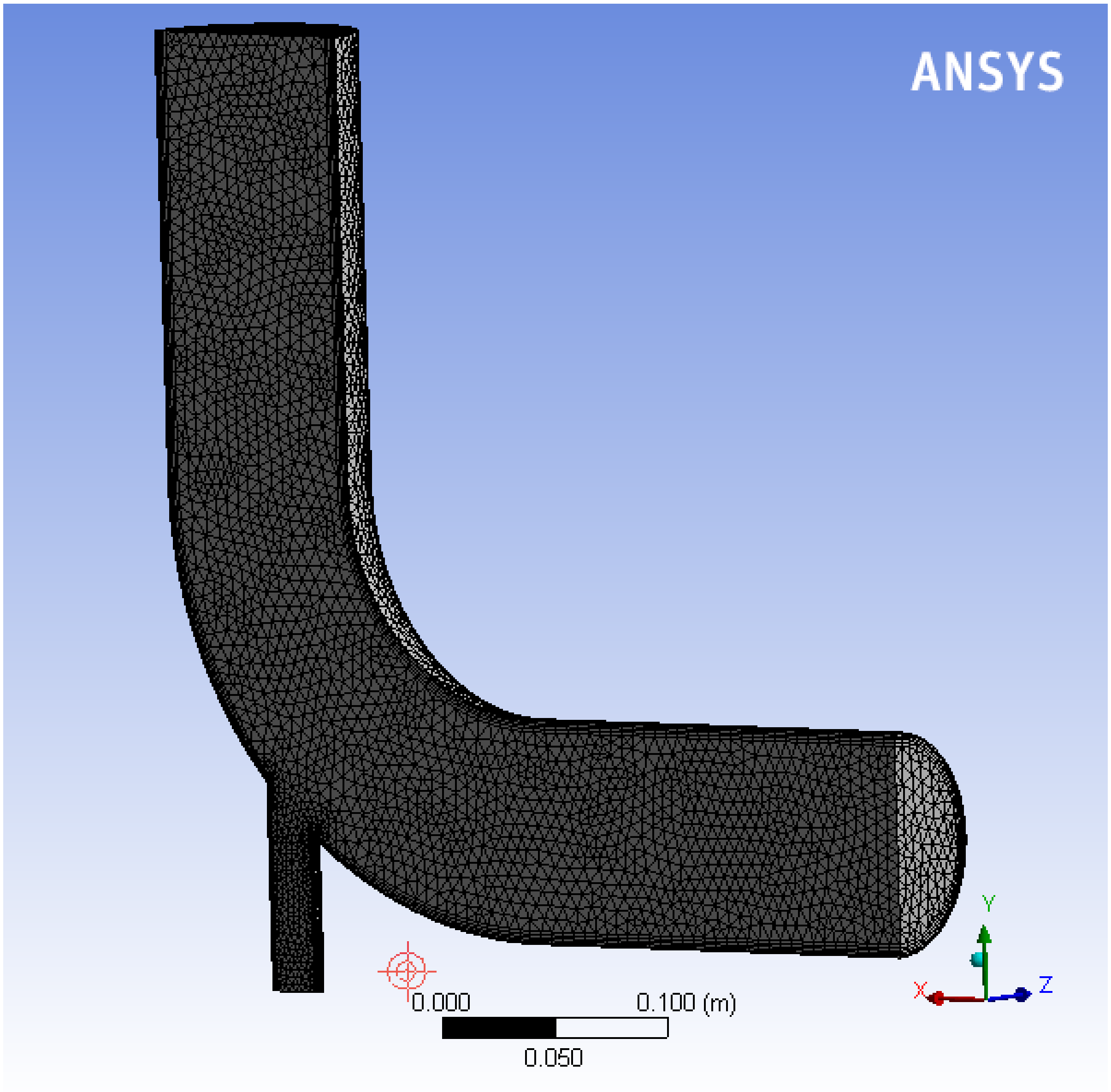
iv. Enter 6e-3 for **Element Size** and press **Enter**.

e. Click again **Mesh** in the **Outline** view and in the **Details of "Mesh"** view, expand the **Inflation** node to reveal additional inflation parameters. Change **Use Automatic Inflation** to **Program Controlled**.

5. Generate the mesh.

Right-click **Mesh** in the project **Outline** tree, and select **Update** in the context menu.

Figure 1.10: The Computational Mesh for the Elbow Geometry in the ANSYS Meshing Application



Important

Using the **Generate Mesh** option creates the mesh, but does not actually create the relevant mesh files for the project and is optional if you already know that the mesh is acceptable. Using the **Update** option automatically generates the mesh, creates the relevant mesh files for your project, and updates the ANSYS Workbench cell that references this mesh.

Note

Once the mesh is generated, you can view the mesh statistics by opening the **Statistics** node in the **Details of "Mesh"** view. This will display information such as the number of nodes and the number of elements.

---

6. Close the ANSYS Meshing application.

   You can close the ANSYS Meshing application without saving it because ANSYS Workbench automatically saves the mesh and updates the **Project Schematic** accordingly. The **Refresh Required** icon in the **Mesh** cell has been replaced by a check mark, indicating that there is a mesh now associated with the fluid flow analysis system.

7. View the list of files generated by ANSYS Workbench.

   View → **Files**

   Figure 1.11: ANSYS Workbench Files View for the Project After Mesh Creation

   Files

	A	B	C	D	E	
1	Name	Cell ID	Size	Type	Date Modified	Location
2	elbow-workbench.wbpj		122 KB	ANSYS Project File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01
3	FFF.agdb	A2	1 MB	Geometry File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
4	FFF.msh	A3	8 MB	Fluent Mesh File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
5	FFF.mshdb	A3	2 MB	Mesh Database Files	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
6	designPoint.wbdp		25 KB	Design Point File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo

   Note the addition of the mesh files (`FFF.msh` and `FFF.mshdb`) to the list of files. The `FFF.msh` file is created when you update the mesh, and the `FFF.mshdb` file is generated when you close the ANSYS Meshing application.

1.4.5. Setting Up the CFD Simulation in ANSYS Fluent

Now that you have created a computational mesh for the elbow geometry, you will set up a CFD analysis using ANSYS Fluent, then review the list of files generated by ANSYS Workbench.

1.4.5.1. Starting ANSYS Fluent

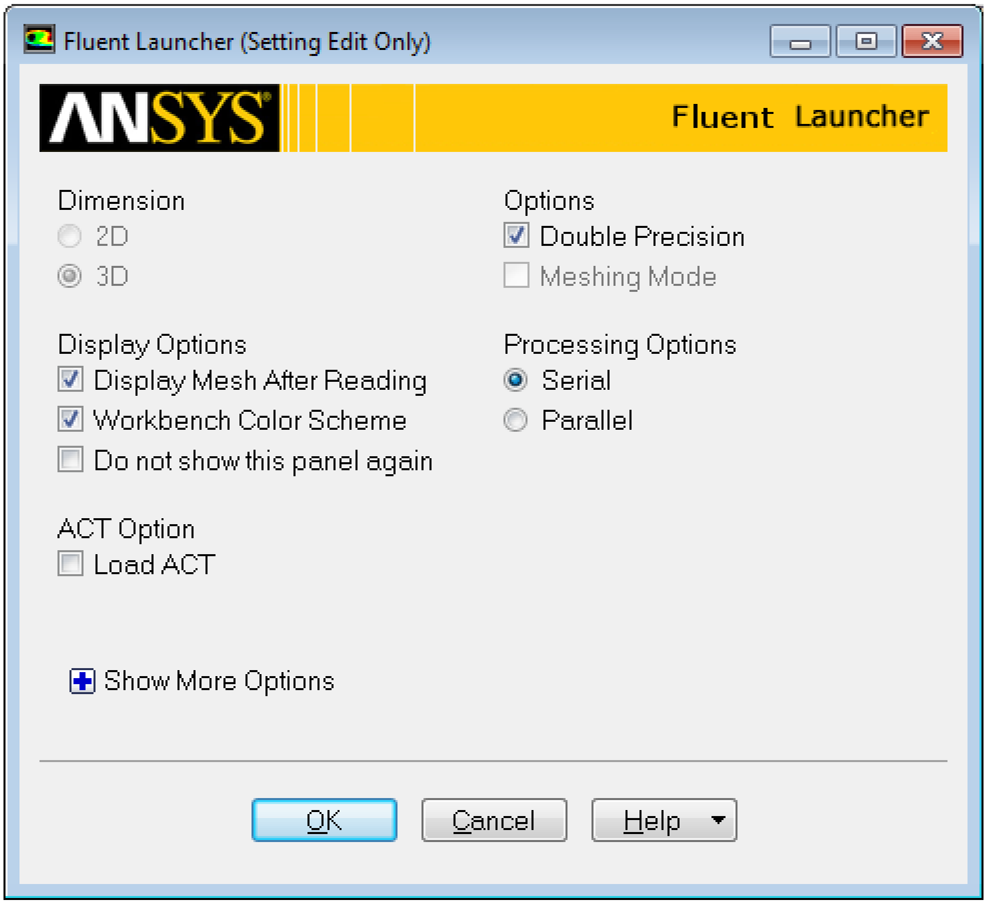
1. In the ANSYS Workbench **Project Schematic**, double-click the **Setup** cell in the `elbow` fluid flow analysis system. You can also right-click the **Setup** cell to display the context menu where you can select the **Edit...** option.

   When ANSYS Fluent is first started, the Fluent Launcher is displayed, enabling you to view and/or set certain ANSYS Fluent start-up options.

   Note

   The Fluent Launcher allows you to decide which version of ANSYS Fluent you will use, based on your geometry and on your processing capabilities.

Figure 1.12: Fluent Launcher



2. Ensure that the proper options are enabled.

Important

Note that the **Dimension** setting is already filled in and cannot be changed, since ANSYS Fluent automatically sets it based on the mesh or geometry for the current system.

---

a. Ensure that **Serial** from the **Processing Options** list is enabled.

b. Select **Double Precision** under **Options**.

c. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.

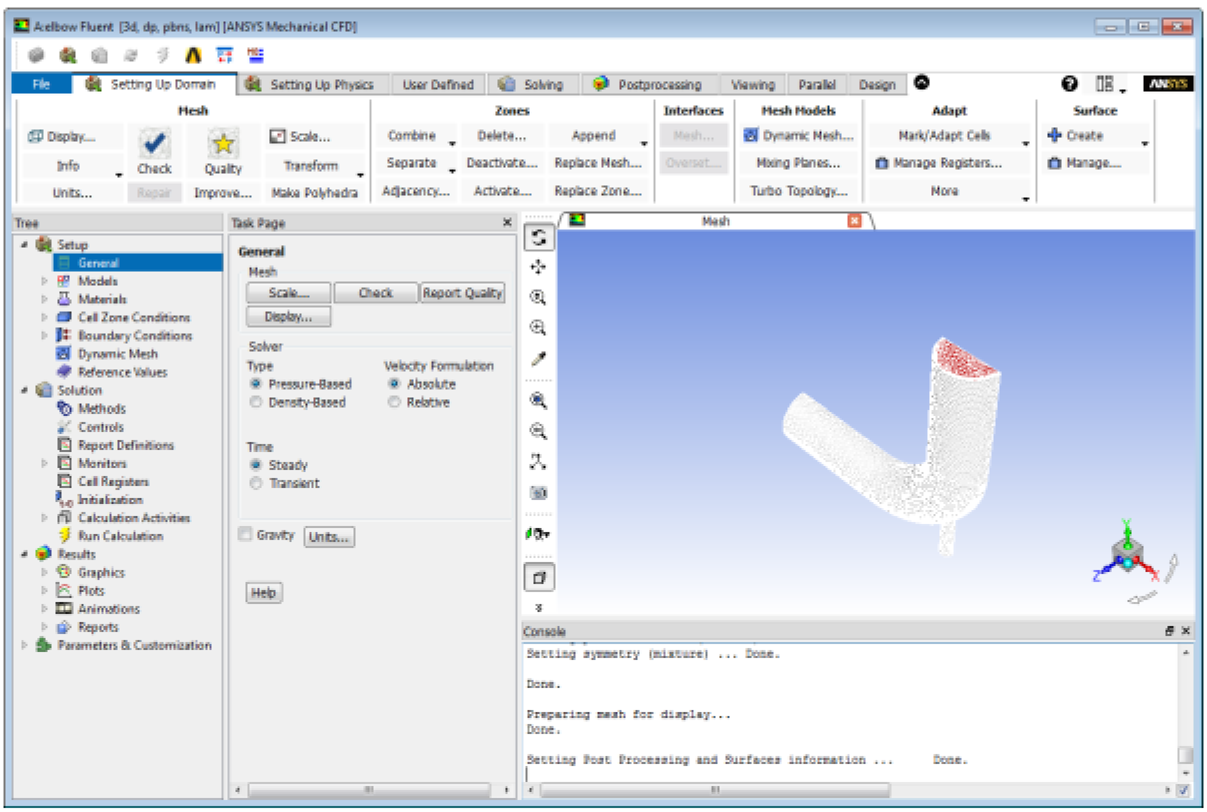
> **Note**
>
> An option is enabled when there is a check mark in the check box, and disabled when the check box is empty. To change an option from disabled to enabled (or vice versa), click the check box or the text.

> **Note**
>
> Fluent will retain your preferences for future sessions.

3. Click **OK** to launch ANSYS Fluent.

> **Note**
>
> The ANSYS Fluent settings file (`FFF.set`) is written as soon as ANSYS Fluent opens.

Figure 1.13: The ANSYS Fluent Application



1.4.5.2. Setting Up Domain

In this step, you will perform the mesh-related activities using the **Setting Up Domain** ribbon tab (**Mesh** group).

Note

These controls are also available in the **General** task page that can be accessed by clicking the **Setup/General** tree item.

1. Change the units for length.

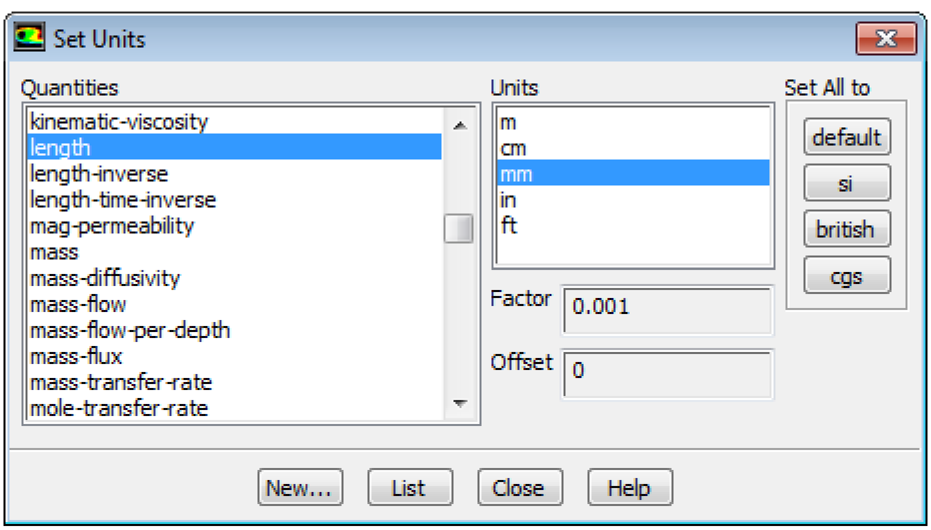
   Because you want to specify and view values based on a unit of length in millimeters from within ANSYS Fluent, change the units of length within ANSYS Fluent from meters (the default) to millimeters.

   ### Important

   Note that the ANSYS Meshing application automatically converts and exports meshes for ANSYS Fluent using meters (m) as the unit of length regardless of what units were used to create them. This is so you do not have to scale the mesh in ANSYS Fluent under ANSYS Workbench.

   Setting Up Domain → **Mesh** → **Units...**

   This displays the **Set Units** dialog box.



   a. Select **length** in the **Quantities** list.

   b. Select **mm** in the **Units** list.

c. Close the dialog box.

Note

Now, all subsequent inputs that require a value based on a unit of length can be specified in millimeters rather than meters.

2. Check the mesh.

Setting Up Domain → Mesh → Check

Note

ANSYS Fluent will report the results of the mesh check in the console.

```
 Domain Extents:
   x-coordinate: min (m) = -2.000000e-01, max (m) = 2.000000e-01
   y-coordinate: min (m) = -2.250000e-01, max (m) = 2.000000e-01
   z-coordinate: min (m) = 0.000000e+00, max (m) = 5.000000e-02
 Volume statistics:
   minimum volume (m3): 1.144763e-10
   maximum volume (m3): 5.871098e-08
     total volume (m3): 2.511309e-03
 Face area statistics:
   minimum face area (m2): 2.051494e-07
   maximum face area (m2): 3.429518e-05
 Checking mesh..........................
Done.
```

Note

The minimum and maximum values may vary slightly when running on different platforms. The mesh check will list the minimum and maximum x and y values from the mesh in the default SI unit of meters. It will also report a number of other mesh features that are checked. Any errors in the mesh will be reported at this time. Ensure that the minimum volume is not negative as ANSYS Fluent cannot begin a calculation when this is the case.

3. Review the mesh quality.

Setting Up Domain → Mesh → Quality

Note

ANSYS Fluent will report the results of the mesh quality below the results of the mesh check in the console.

```
Mesh Quality:
Minimum Orthogonal Quality = 1.98483e-01
(Orthogonal Quality ranges from 0 to 1, where values close to 0 correspond to low quality.)
Maximum Ortho Skew = 7.19694e-01
```

```
(Ortho Skew ranges from 0 to 1, where values close to 1 correspond to low quality.)
Maximum Aspect Ratio = 2.05643e+01
```

Note

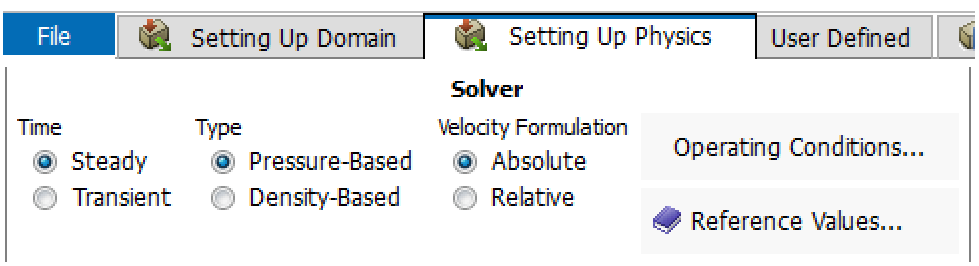
The quality of the mesh plays a significant role in the accuracy and stability of the numerical computation. Checking the quality of your mesh is, therefore, an important step in performing a robust simulation. Minimum cell orthogonality is an important indicator of mesh quality. Values for orthogonality can vary between 0 and 1 with lower values indicating poorer quality cells. In general, the minimum orthogonality should not be below 0.01 with the average value significantly larger. The high aspect ratio cells in this mesh are near the walls and are a result of the boundary layer inflation applied in the meshing step. For more information about the importance of mesh quality refer to the Fluent User's Guide.

1.4.5.3. Setting Up Physics

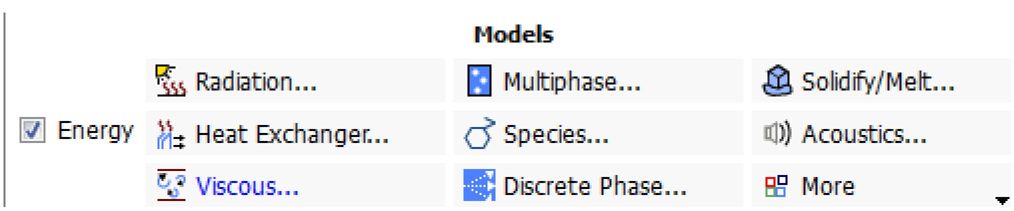
In the steps that follow, you will select a solver and specify physical models, material properties, and zone conditions for your simulation using the **Setting Up Physics** ribbon tab.

1. In the **Solver** group of the **Setting Up Physics** ribbon tab, retain the default selection of the steady pressure-based solver.

   Setting Up Physics → Solver



2. Set up your models for the CFD simulation using the **Models** group of the **Setting Up Physics** ribbon tab.



Note

You can also use the **Models** task page that can be accessed by double-clicking the **Setup/Models** tree item.

---

   a. Enable heat transfer by activating the energy equation.

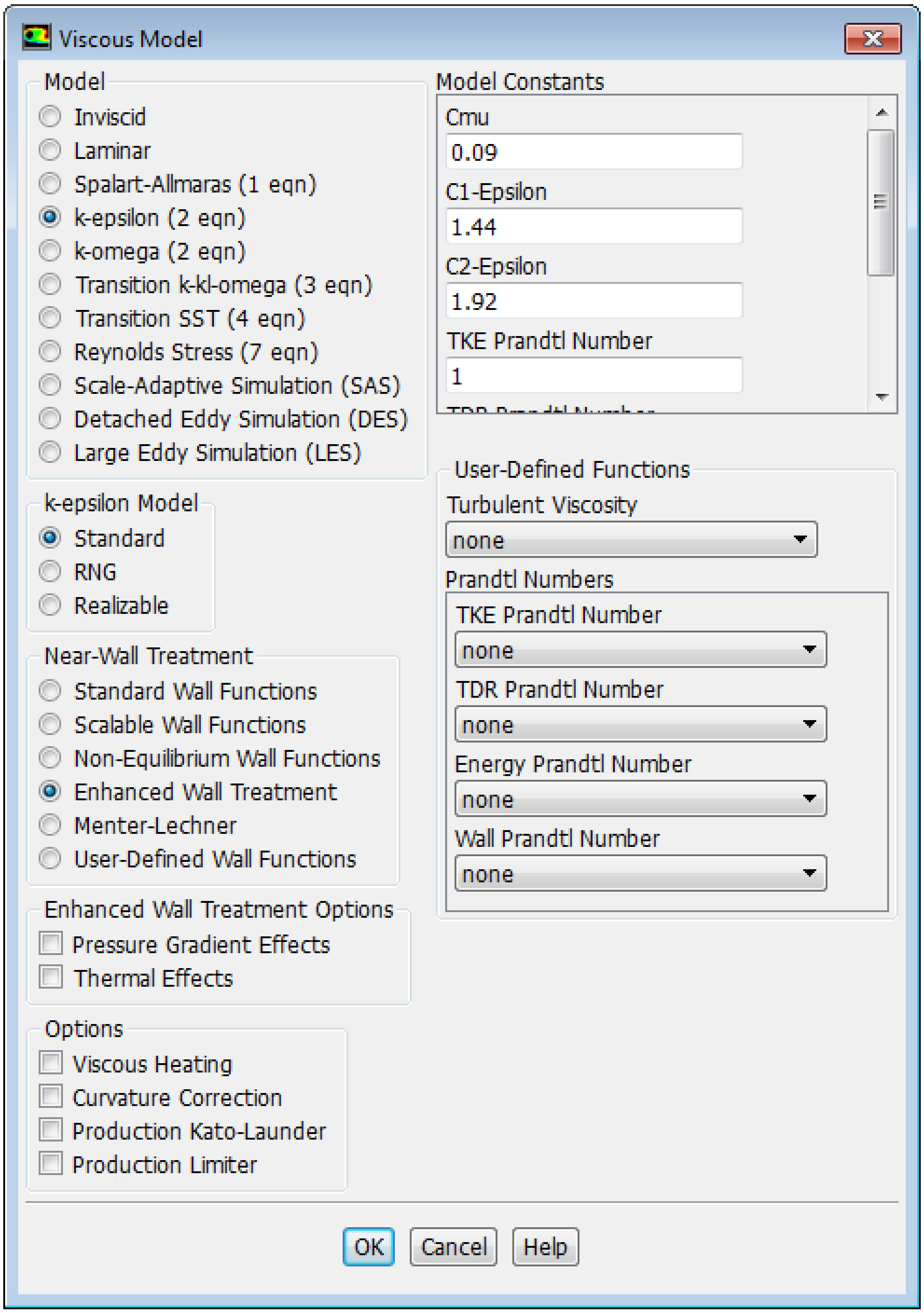
      In the **Setting Up Physics** ribbon tab, select **Energy** (**Models** group).

Setting Up Physics → Models → Energy

b. Enable the k-ε turbulence model.

   i. In the **Setting Up Physics** ribbon tab, click **Viscous...** (**Models** group).

   Setting Up Physics → Models → Viscous...



ii. In the **Viscous Model** dialog box, select **k-epsilon** from the **Model** list.

> **Note**
>
> The **Viscous Model** dialog box will expand.

iii. Use the default **Standard** from the **k-epsilon Model** group.

iv. Select **Enhanced Wall Treatment** for the **Near-Wall Treatment**.

> **Note**
>
> The default Standard Wall Functions are generally applicable if the first cell center adjacent to the wall has a y+ larger than 30. In contrast, the Enhanced Wall Treatment option provides consistent solutions for all y+ values. Enhanced Wall Treatment is recommended when using the k-epsilon model for general single-phase fluid flow problems. For more information about Near Wall Treatments in the k-epsilon model refer to the Fluent User's Guide.

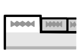
v. Click **OK** to accept the model and close the **Viscous Model** dialog box.

Note that the **Viscous...** label in the ribbon is displayed in blue to indicate that the Viscous model is enabled.

3. Set up the materials for your CFD simulation using the **Materials** group of the **Setting Up Physics** ribbon tab.

Create a new material called **water** using the **Create/Edit Materials** dialog box (Figure 1.14: The Create/Edit Materials Dialog Box (p. 35)).

a. In the **Setting Up Physics** ribbon tab, click **Create/Edit...** (**Materials** group).

 **Setting Up Physics → Materials → Create/Edit...**

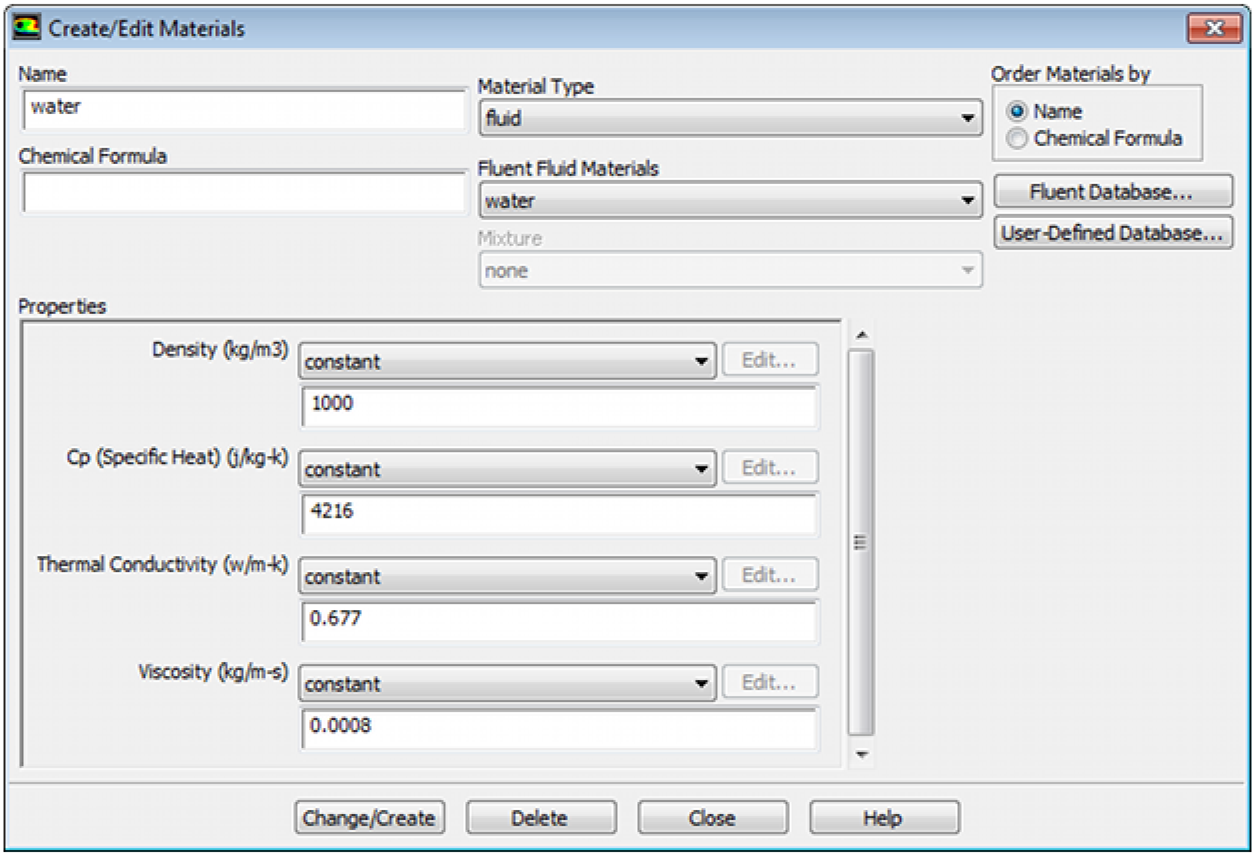
b. In the **Create/Edit Materials** dialog box, type `water` for **Name**.

c. In the **Properties** group, enter the following values:

Property	Value
Density	`1000` [kg/m^3]
c_p (Specific Heat)	`4216` [J/kg-K]
Thermal Conductivity	`0.677` [W/m-K]

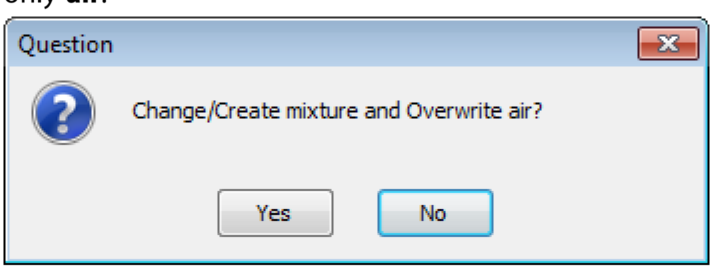
Property	Value
Viscosity	`8e-04` [kg/m-s]

Figure 1.14: The Create/Edit Materials Dialog Box



d. Click **Change/Create**.

A **Question** dialog box will open, asking if you want to overwrite air. Click **No** so that the new material **water** is added to the **Fluent Fluid Materials** list of materials that originally contained only **air**.



Extra

You could have copied the material **water-liquid (h2o < l >)** from the materials database (accessed by clicking the **ANSYS Fluent Database...** button). If the properties in the database are different from those you want to use, you can edit the values in the **Properties** group in the **Create/Edit Materials** dialog box and click **Change/Create** to update your local copy. The original copy will not be affected.

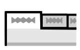
e. Ensure that there are now two materials (water and air) defined locally by examining the **Fluent Fluid Materials** drop-down list in the **Create/Edit Materials** dialog box .

> **Note**
>
> Both the materials will also be listed under **Fluid** in the **Materials** task page and under the **Materials** tree branch.

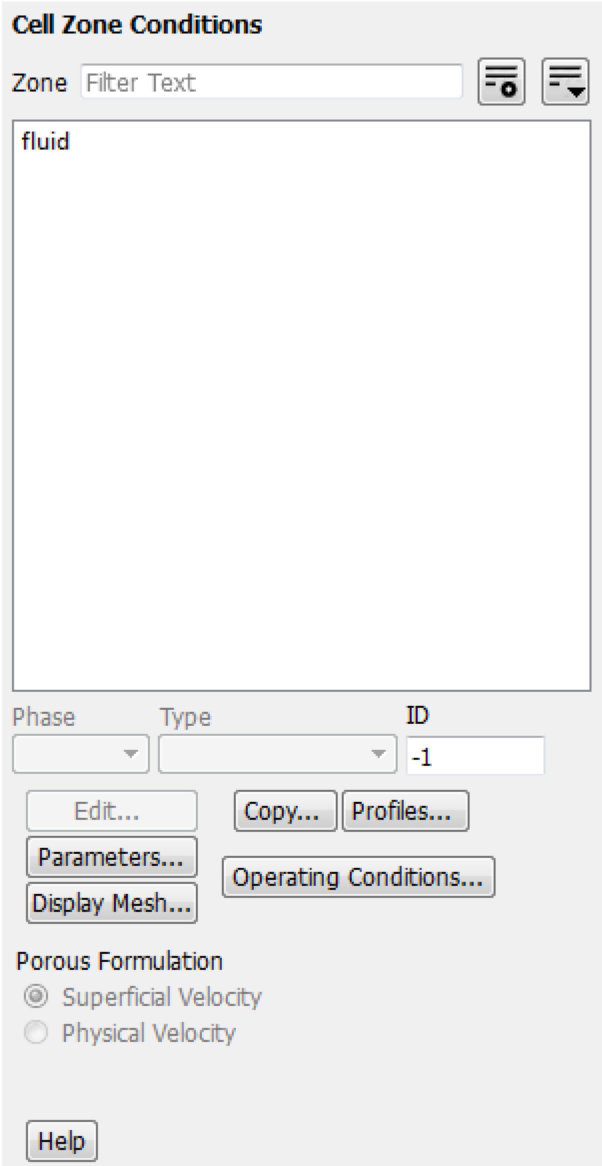
f. Close the **Create/Edit Materials** dialog box.

4. Set up the cell zone conditions for the CFD simulation using the **Zones** group of the **Setting Up Physics** ribbon tab.

a. In the **Setting Up Physics** tab, click **Cell Zones**.

 **Setting Up Physics → Zones → Cell Zones**

This opens the **Cell Zone Conditions** task page.

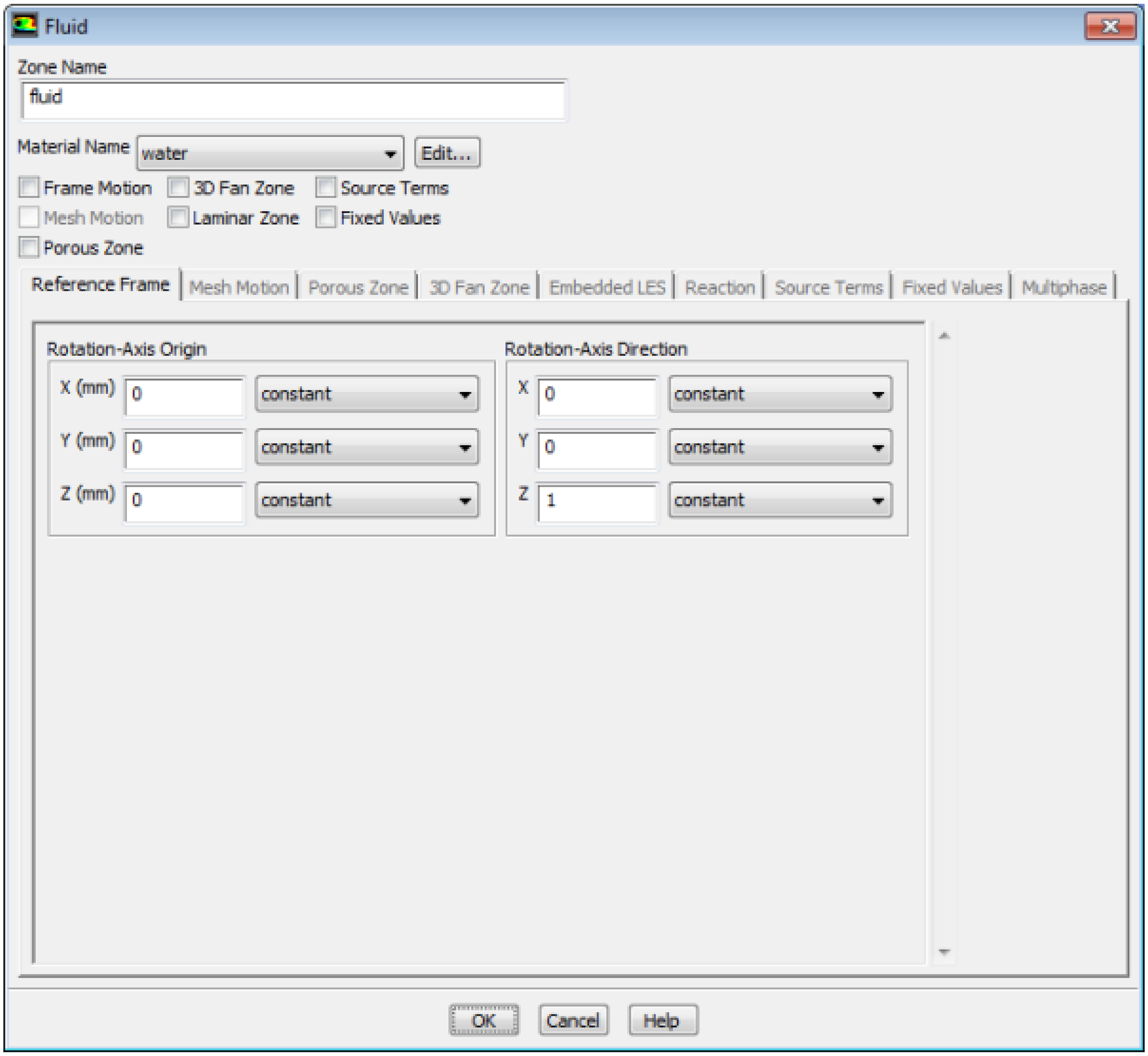


b. Set the cell zone conditions for the fluid zone.

   i. In the **Cell Zone Conditions** task page, in the **Zone** list, select **fluid** and click **Edit...** to open the **Fluid** dialog box.

Note

You can also double-click **fluid** in the **Cell Zones Conditions** task page or under the **Setup/Cell Zone Conditions** tree branch in order to open the corresponding dialog box.

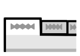


ii. In the **Fluid** dialog box, select **water** from the **Material Name** drop-down list.

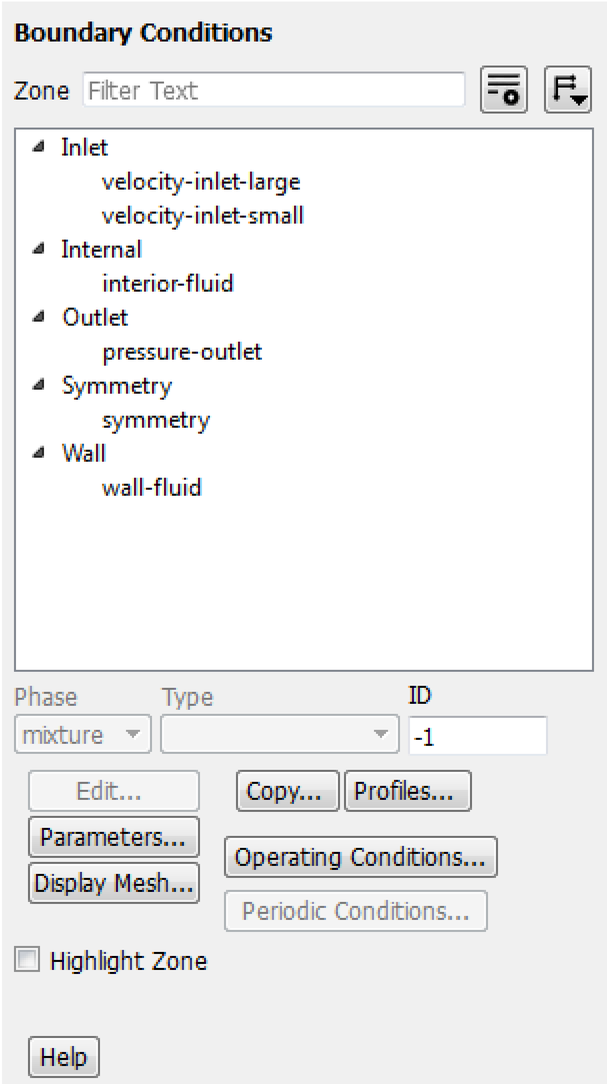
iii. Click **OK** to close the **Fluid** dialog box.

5. Set up the boundary conditions for the CFD analysis using the **Zones** group of the **Setting Up Physics** ribbon tab.

a. In the **Setting Up Physics** tab, click **Boundaries** (**Zones** group).

 **Setting Up Physics** → **Zones** → **Boundaries**

This opens the **Boundary Conditions** task page.



Note

To display boundary zones grouped by zone type (as shown above), click the **Toggle Tree View** button in the upper right corner of the **Boundary Conditions** task page and under the **Group By** category select **Zone Type** from the drop-down list.

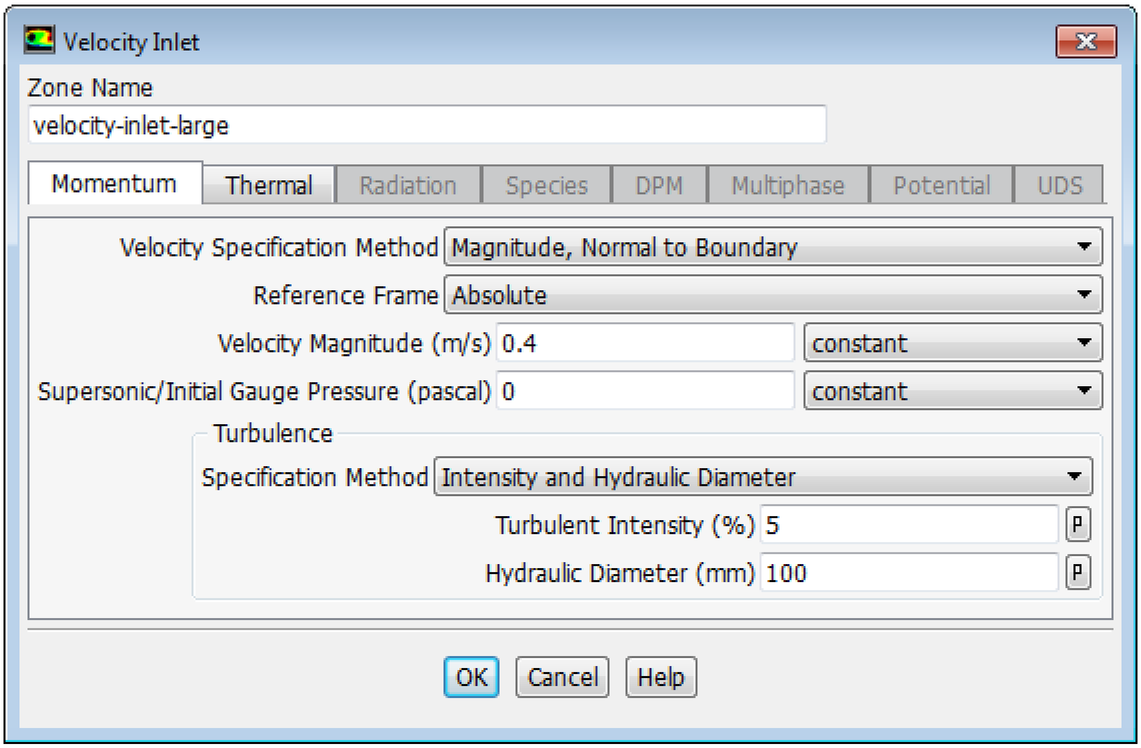
b. Set the boundary conditions at the cold inlet (**velocity-inlet-large**).

Tip

If you are unsure of which inlet zone corresponds to the cold inlet, you can use the mouse to probe for mesh information in the graphics window. If you click the right mouse button with the pointer on any node in the mesh, information about the associated zone will be displayed in the ANSYS Fluent console, including the name of the zone. The zone you probed will be automatically selected from the **Zone** selection list in the **Boundary Conditions** task page.

Alternatively, you can click the probe button () in the graphics toolbar and click the left mouse button on any node. This feature is especially useful when you have several zones of the same type and you want to distinguish between them quickly. The information will be displayed in the console.

i. From the **Zone** selection list, select **velocity-inlet-large** and click **Edit...**.



ii. In the **Velocity Inlet** dialog box, ensure that **Magnitude, Normal to Boundary** is selected for **Velocity Specification Method**.

iii. Enter `0.4` m/s for **Velocity Magnitude**.

iv. In the **Turbulence** group, from the **Specification Method** drop-down list, select **Intensity and Hydraulic Diameter**.

v. Retain the default of 5 $\%$ for **Turbulent Intensity**.

vi. Enter `100` mm for **Hydraulic Diameter**.

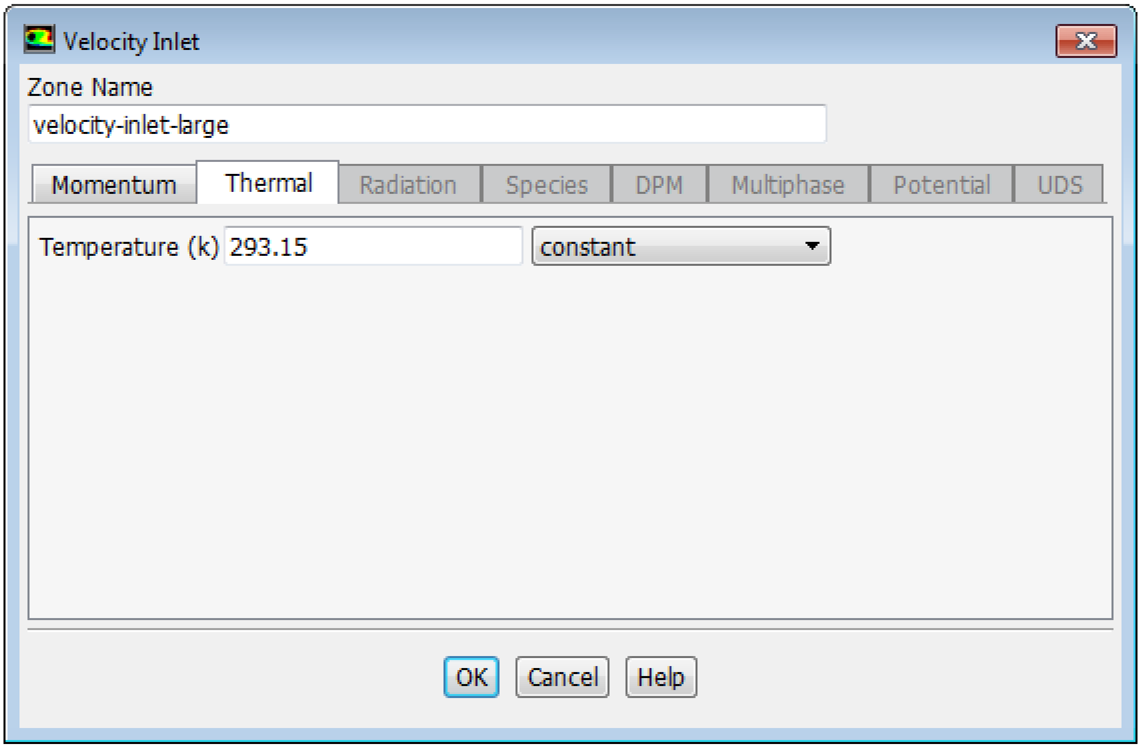
Note

The hydraulic diameter D_h is defined as:

$$D_h=\frac{4A}{P_w}$$

where A is the cross-sectional area and P_w is the wetted perimeter.

vii. Click the **Thermal** tab.



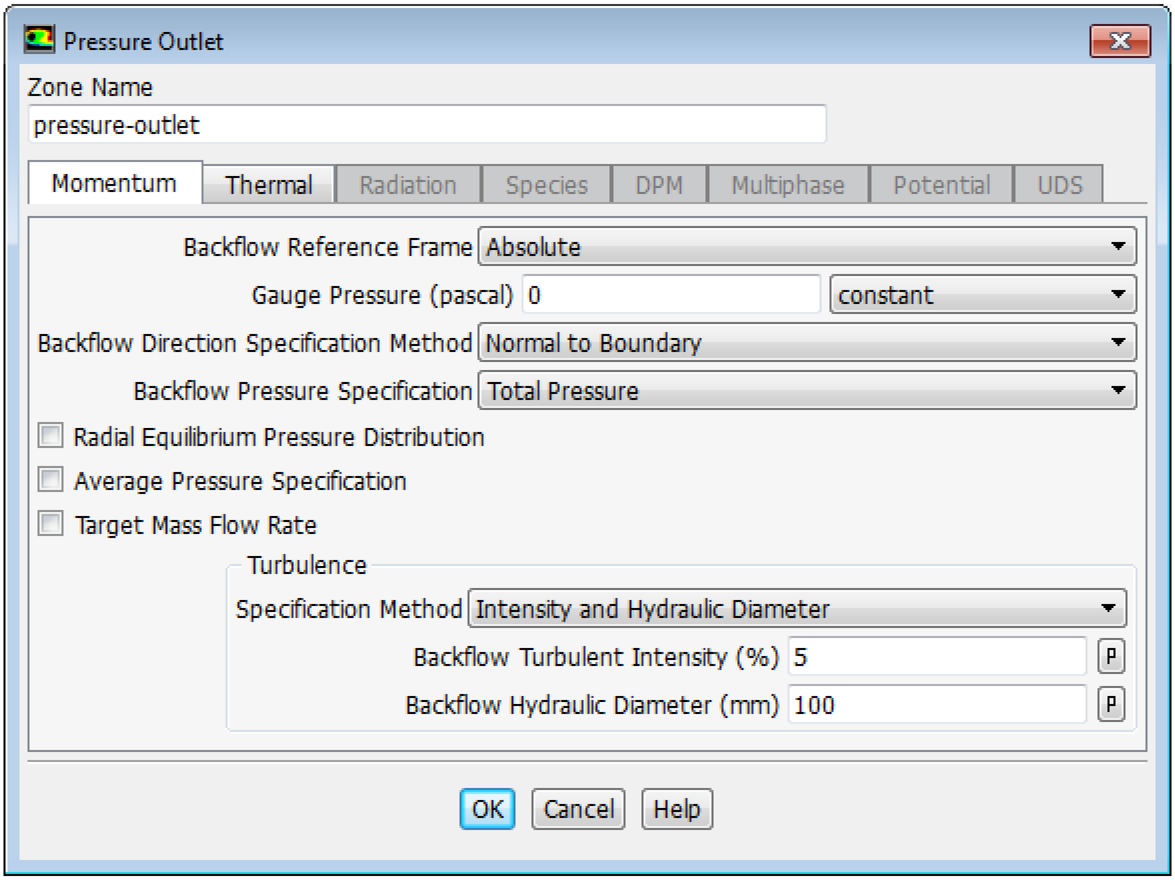
viii. Enter `293.15` K for **Temperature**.

ix. Click **OK** to close the **Velocity Inlet** dialog box.

c. In a similar manner, set the boundary conditions at the hot inlet (**velocity-inlet-small**), using the values in the following table:

Setting	Value
Velocity Specification Method	Magnitude, Normal to Boundary
Velocity Magnitude	**1.2** [m/s]
Specification Method	Intensity & Hydraulic Diameter
Turbulent Intensity	**5**%
Hydraulic Diameter	**25** [mm]
Temperature	**313.15**[K]

d. Double-click **pressure-outlet** and set the boundary conditions at the outlet, as shown in the **Pressure Outlet** dialog box.



Note

ANSYS Fluent will use the backflow conditions only if the fluid is flowing into the computational domain through the outlet. Since backflow might occur at some point during the solution procedure, you should set reasonable backflow conditions to prevent convergence from being adversely affected.

1.4.6. Solving

In the steps that follow, you will set up and run the calculation using the **Solving** ribbon tab.

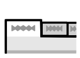
Note

You can also use the task pages listed under the **Solution** branch in the tree to perform solution-related activities.

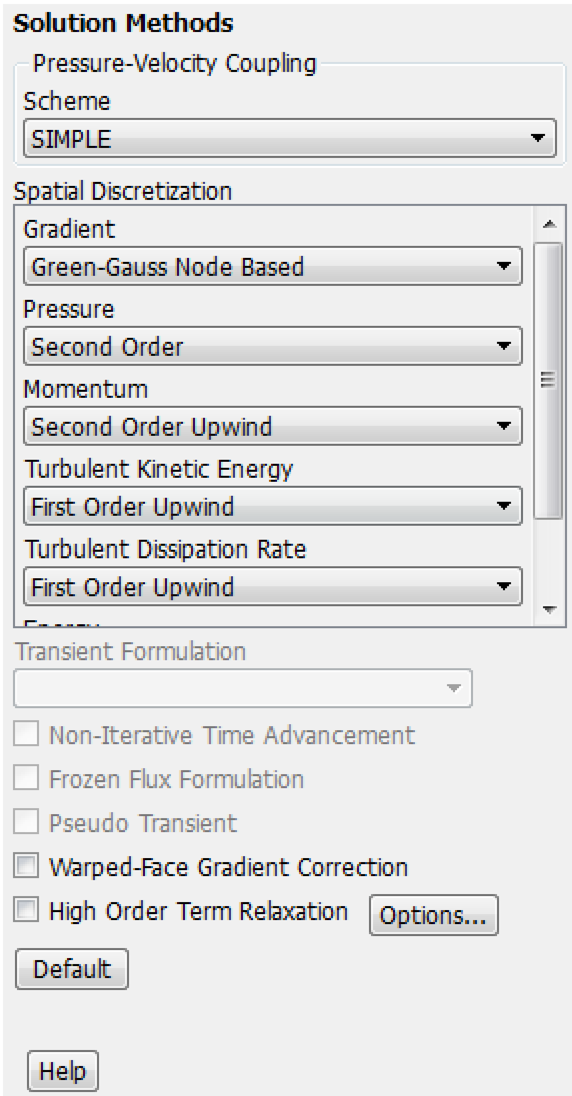
1. Set up solution parameters for the CFD simulation.

   a. Change the Gradient method.

      i. In the **Solving** ribbon tab, click **Methods...** (**Solution** group).

       **Solving** → **Solution** → **Methods...**

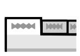
This will open the **Solution Methods** task page.

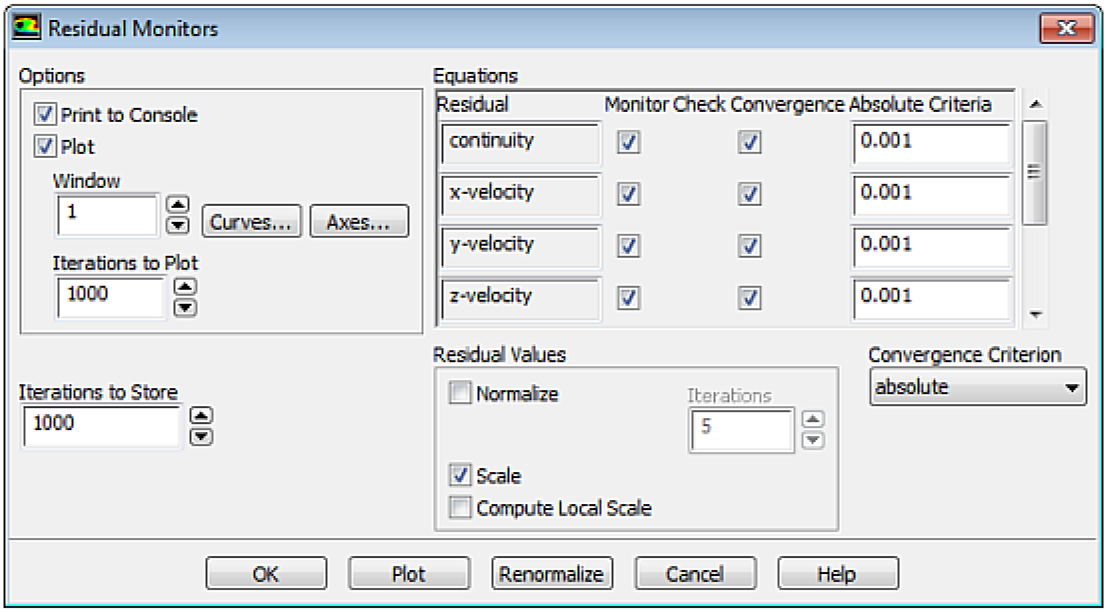


ii. In the **Spatial Discretization** group of the **Solution Methods** task page, change the **Gradient** to **Green-Gauss Node Based**. This gradient method is suggested for tetrahedral meshes.

b. Enable the plotting of residuals during the calculation.

i. In the **Solving** ribbon tab, click **Residuals...** (**Reports** group).

 **Solving** → **Reports** → **Residuals...**



In the **Residual Monitors** dialog box, ensure that **Plot** is enabled in the **Options** group.

ii. Keep the default values for the **Absolute Criteria** of the **Residuals**, as shown in the **Residual Monitors** dialog box.

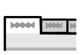
iii. Click **OK** to close the **Residual Monitors** dialog box.

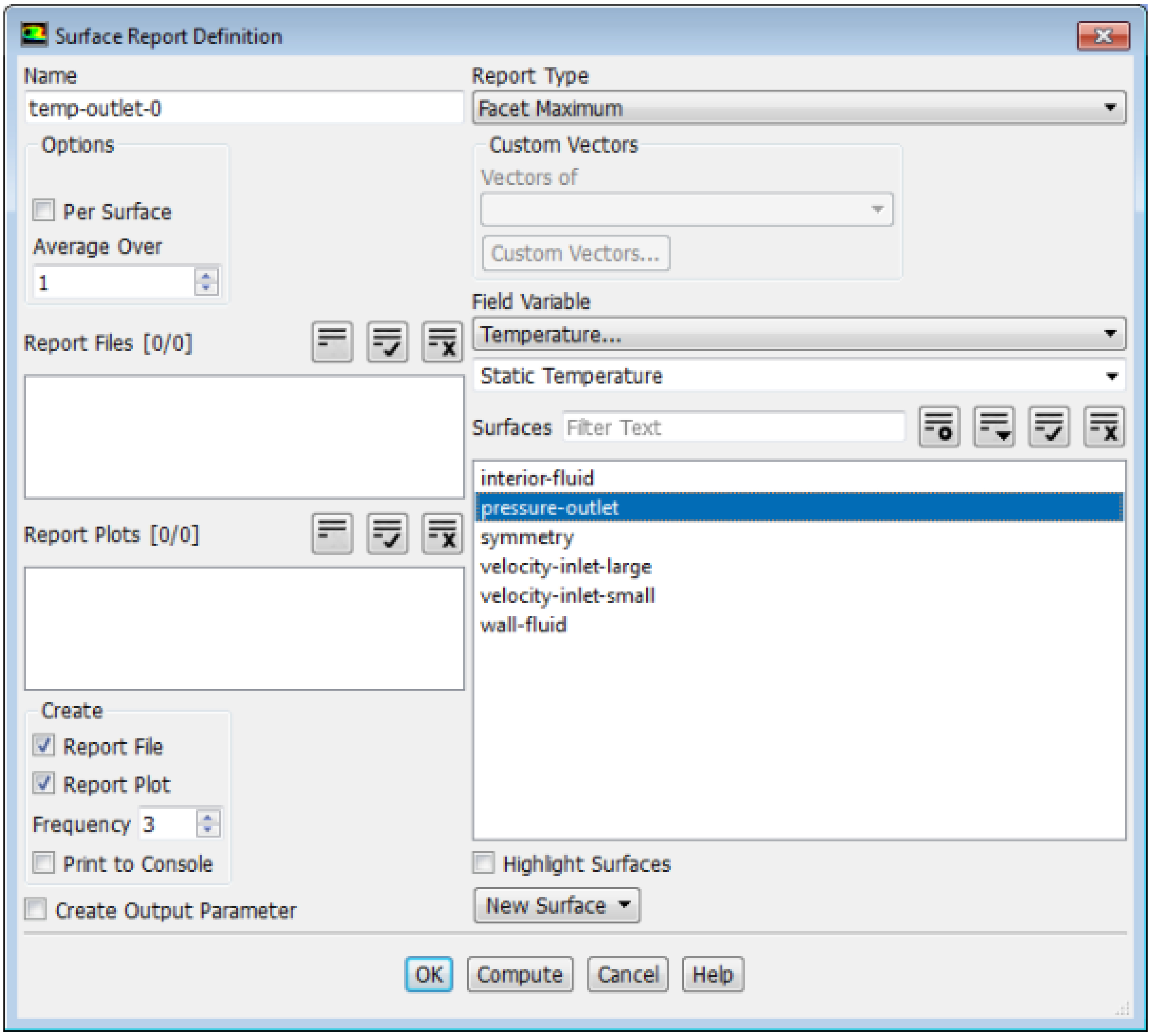
Note

By default, all variables will be monitored and checked by ANSYS Fluent as a means to determine the convergence of the solution.

---

c. Create a surface report definition at the outlet (**pressure-outlet**).

 **Solving** → **Reports** → **Definitions** → **New** → **Surface Report** → **Facet Maximum**



i. In the **Surface Report Definition** dialog box, enter `temp-outlet-0` for the **Name**.

ii. Under the **Create** group, enable **Report File** and **Report Plot**.

During a solution run, ANSYS Fluent will write solution convergence data in a report file and plot the solution convergence history in a graphics window.

It is good practice to monitor physical solution quantities in addition to equation residuals when assessing convergence.

iii. Set **Frequency** to 3 by clicking the up-arrow button.

This setting instructs ANSYS Fluent to update the plot of the surface report and write data to a file after every 3 iterations during the solution.

iv. Select **Temperature...** and **Static Temperature** from the **Field Variable** drop-down lists.

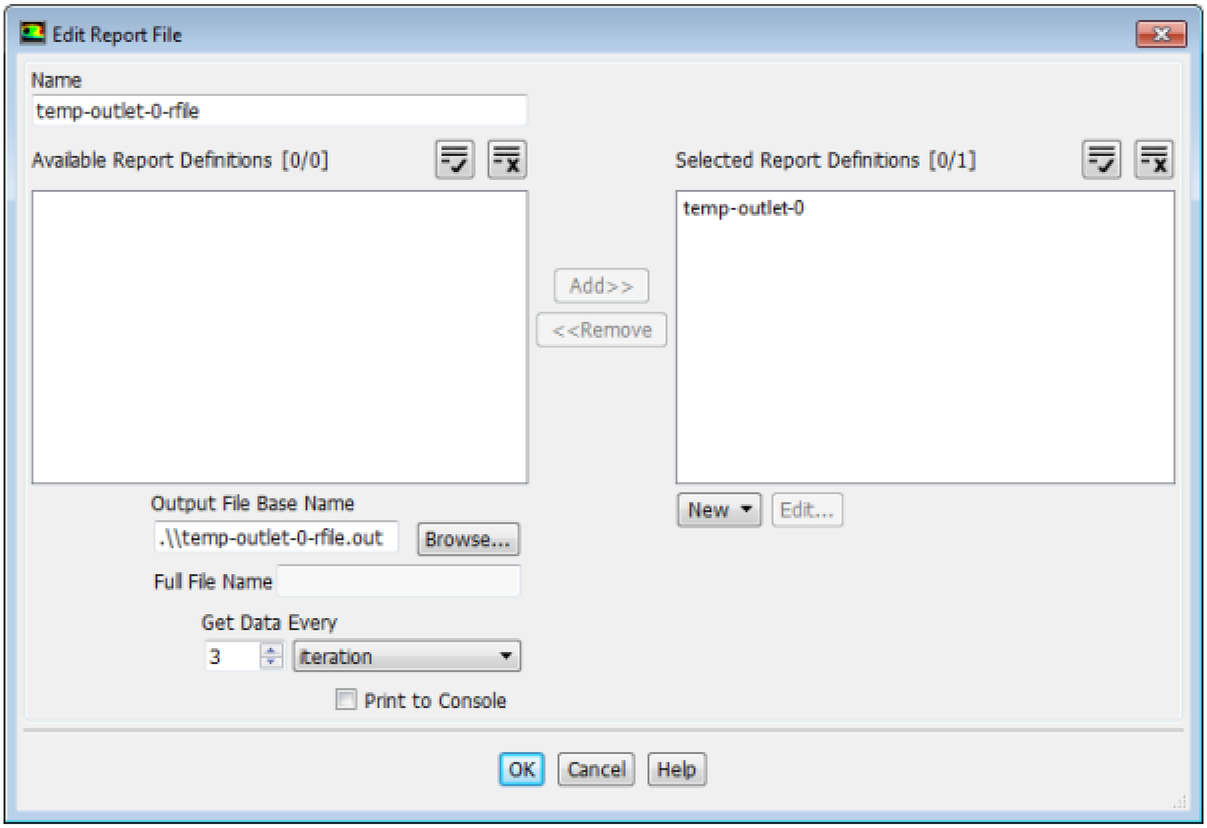
v. Select **pressure-outlet** from the **Surfaces** selection list.

vi. Click **OK** to save the surface report definition and close the **Surface Report Definition** dialog box.

The new surface report definition `temp-outlet-0` appears under the **Solution/Report Definitions** tree item. ANSYS Fluent also automatically creates the following items:

- `temp-outlet-0-rfile` (under the **Solution/Monitors/Report Files** tree branch)
- `temp-outlet-0-rplot` (under the **Solution/Monitors/Report Plots** tree branch)

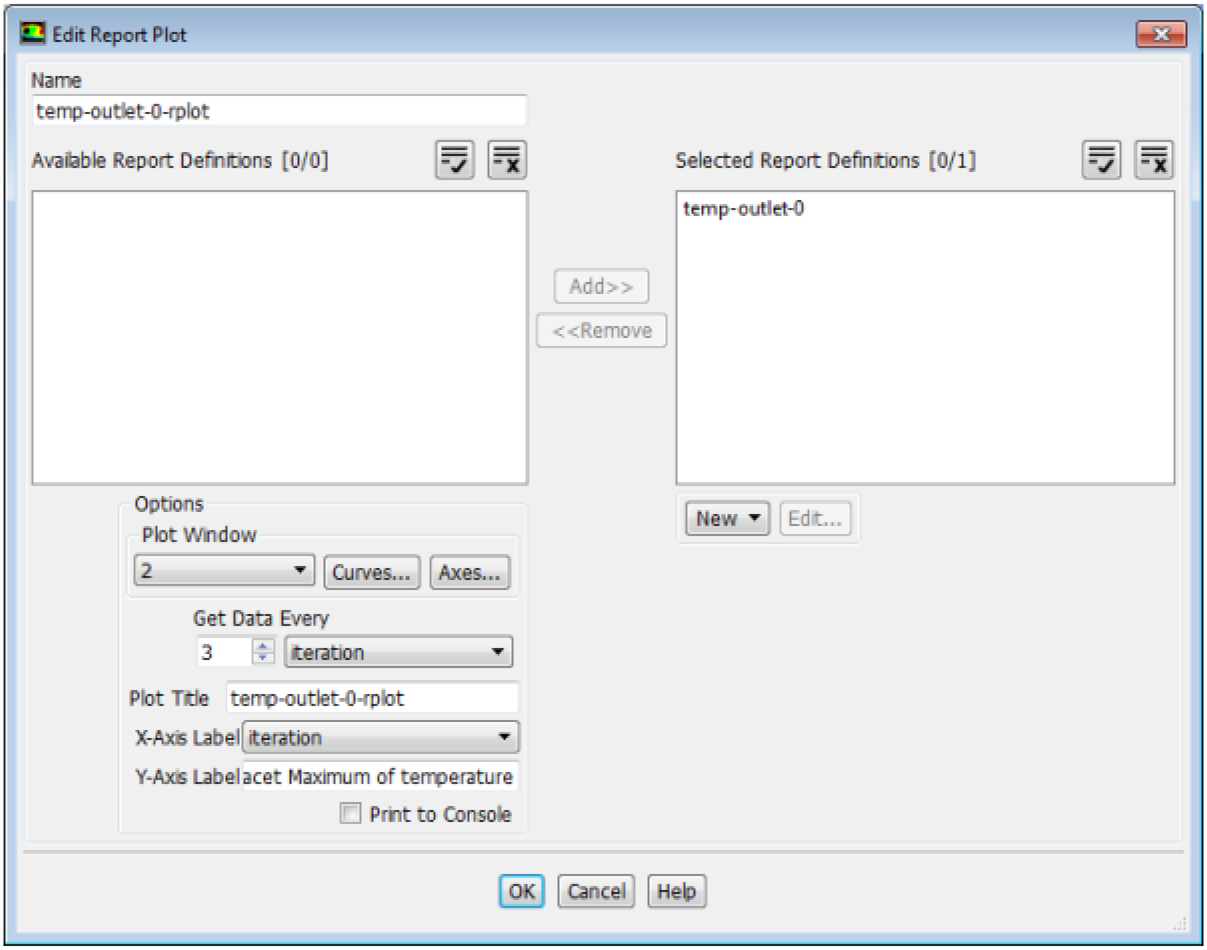
d. In the tree, double-click the **temp-outlet-0-rfile** (under the **Solution/Monitors/Report Files**) and examine the report file settings in the **Edit Report File** dialog box.



The dialog box is automatically populated with data from the `temp-outlet-0` report definition. The report that will be written in a report file during a solution is listed under **Selected Report Definitions**.

Retain the default output file name and click **OK**.

e. In the tree, double-click **temp-outlet-0-rplot** (under **Solution/Monitors/Report Plots**) and examine the report plot settings in the **Edit Report Plot** dialog box.



The dialog box is automatically populated with data from the `temp-outlet-0` report definition. As the solution progresses, the report that is listed under **Selected Report Definitions** will be plotted in a graphics tab window with the title specified in **Plot Title**.

Retain the default names for **Plot Title** and **Y-Axis Label** and click **OK**.

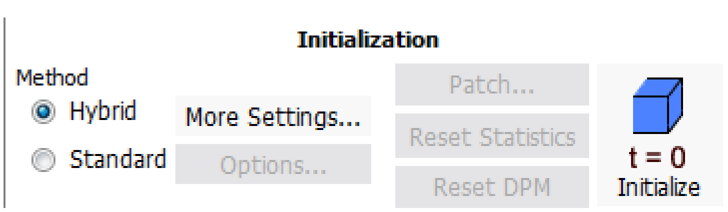
Note

You can create report definitions for different boundaries and plot them in the same graphics window. However, report definitions in the same **Report Plot** must have the same units.

---

f. Initialize the flow field using the **Initialization** group of the **Solving** ribbon tab.

Solving → Initialization

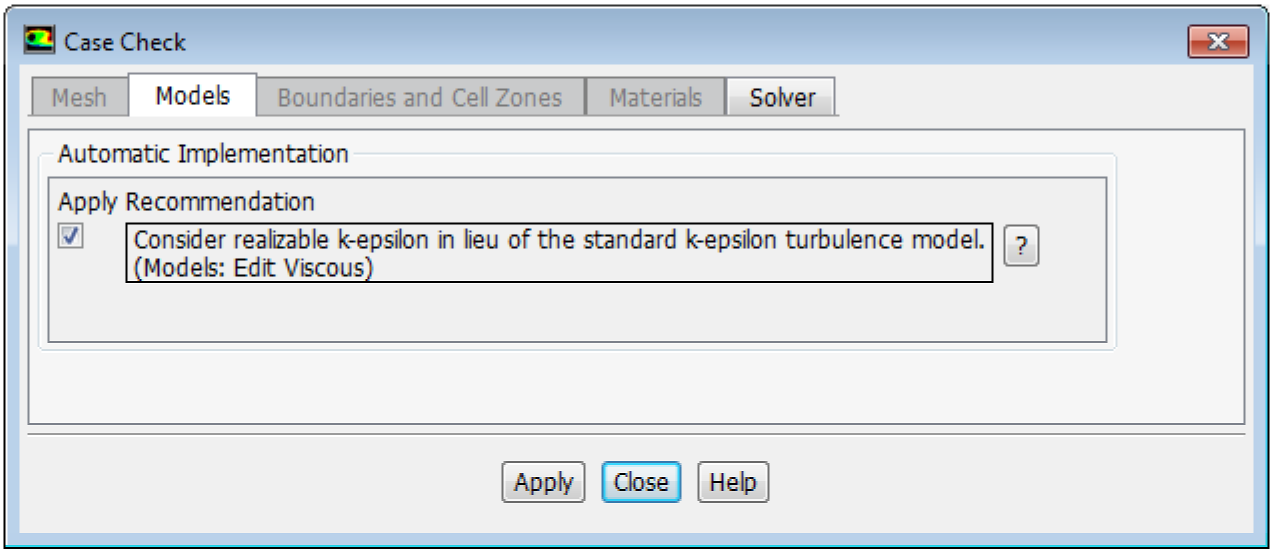


i. Keep the **Method** at the default of **Hybrid**.

ii. Click **Initialize**.

g. Check to see if the case conforms to best practices.

Solving → Run Calculation → Check Case

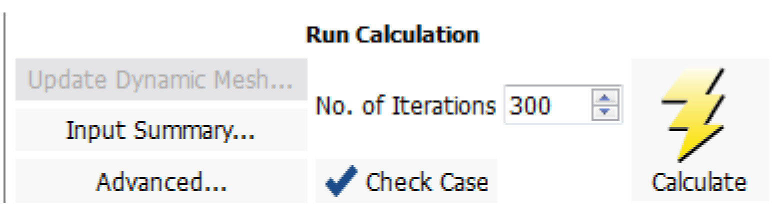


i. Click the **Models** and **Solver** tabs and examine the **Recommendation** in each. These recommendations can be ignored for this tutorial. The issues they raise will be addressed in later tutorials.

ii. Close the **Case Check** dialog box.

2. Calculate a solution using the **Run Calculation** group of the **Solving** tab.

Solving → Run Calculation



a. Start the calculation by requesting 300 iterations.

i. Enter `300` for **No. of Iterations**.

ii. Click **Calculate**.

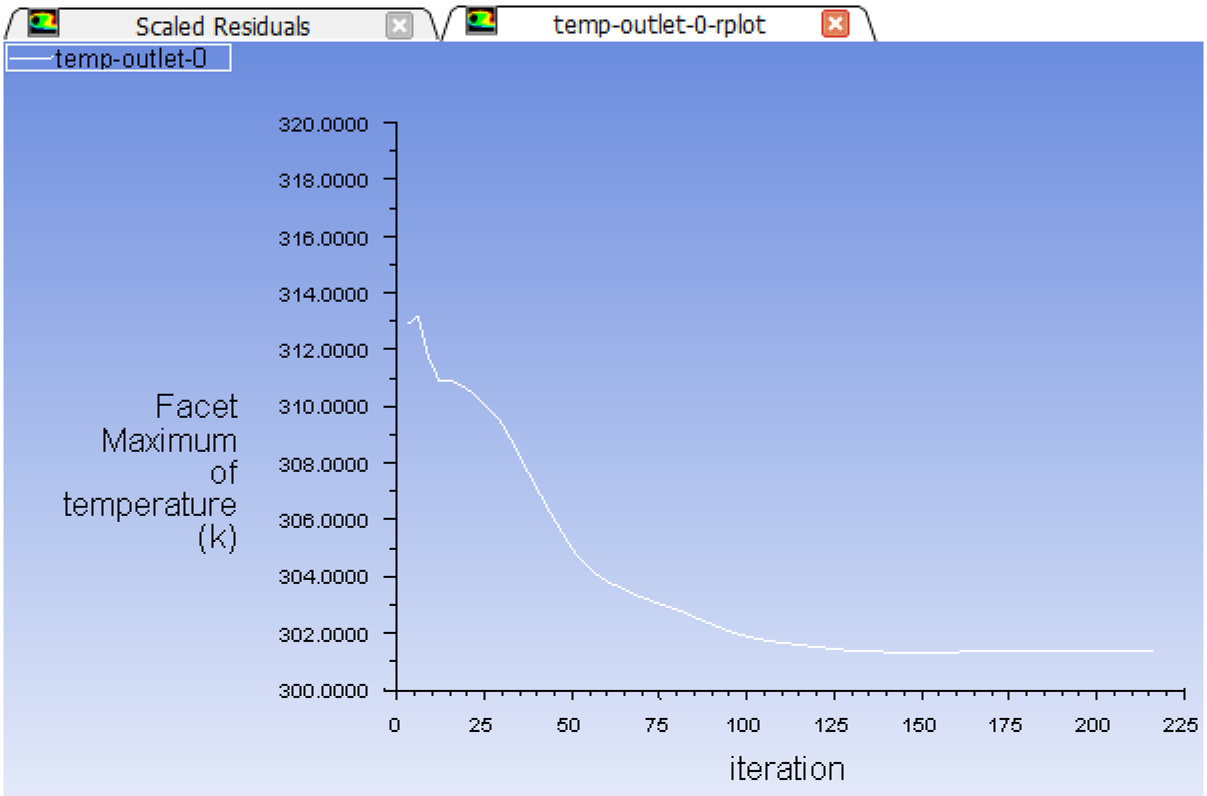
Note that while the program is calculating the solution, the states of the **Setup** and **Solution** cells in the fluid flow ANSYS Fluent analysis system in ANSYS Workbench are changing. For example:

- The state of the **Setup** cell becomes **Up-to-Date** and the state of the **Solution** cell becomes **Refresh Required** after the **Run Calculation** task page is visited and the number of iterations is specified.

- The state of the **Solution** cell is **Update Required** while iterations are taking place.
- The state of the **Solution** cell is **Up-to-Date** when the specified number of iterations are complete (or if convergence is reached).

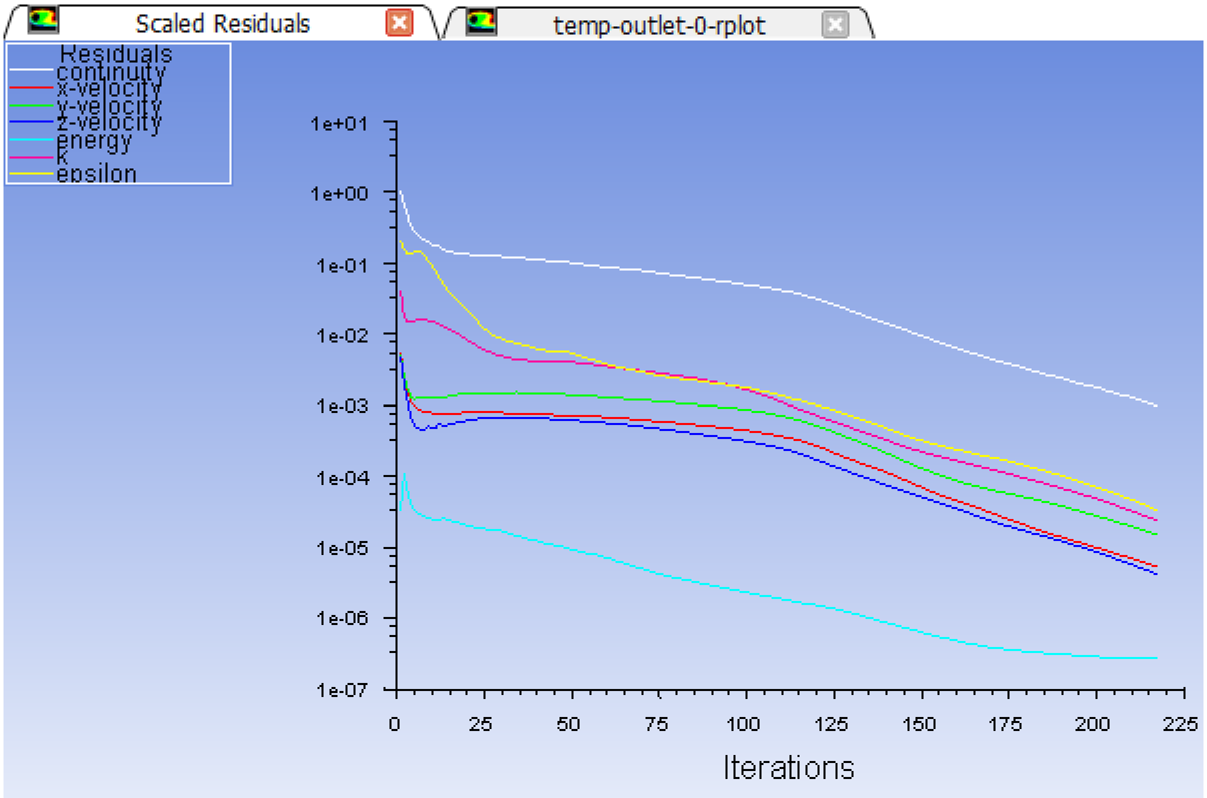
As the calculation progresses, the surface report plot will be plotted in the graphics window (**temp-outlet-0–rplot** tab window) (Figure 1.15: Convergence History of the Maximum Temperature at Pressure Outlet (p. 49)).

Figure 1.15: Convergence History of the Maximum Temperature at Pressure Outlet



The residuals history will be plotted in the **Scaled Residuals** tab window (Figure 1.16: Residuals for the Converged Solution (p. 50)).

Figure 1.16: Residuals for the Converged Solution



Note

The solution will be stopped by ANSYS Fluent when the residuals reach their specified values or after 300 iterations. The exact number of iterations will vary depending on the platform being used. An **Information** dialog box will open to alert you that the calculation is complete. Click **OK** in the **Information** dialog box to proceed.

Because the residual values vary slightly by platform, the plot that appears on your screen may not be exactly the same as the one shown here.

Note that the ANSYS Fluent settings file (`FFF.set`) is updated in the **Files** view of the ANSYS Workbench before the calculation begins.

b. Examine the plots for convergence (Figure 1.16: Residuals for the Converged Solution (p. 50) and Figure 1.15: Convergence History of the Maximum Temperature at Pressure Outlet (p. 49)).

Note

There are no universal metrics for judging convergence. Residual definitions that are useful for one class of problem are sometimes misleading for other classes of problems. Therefore it is a good idea to judge convergence not only by examining residual levels, but also by monitoring relevant integrated quantities and checking for mass and energy balances.

There are three indicators that convergence has been reached:

- The residuals have decreased to a sufficient degree.

  The solution has converged when the **Convergence Criterion** for each variable has been reached. The default criterion is that each residual will be reduced to a value of less than 10^{-3}, except the **energy** residual, for which the default criterion is 10^{-6}.

- The solution no longer changes with more iterations.

  Sometimes the residuals may not fall below the convergence criterion set in the case setup. However, monitoring the representative flow variables through iterations may show that the residuals have stagnated and do not change with further iterations. This could also be considered as convergence.

- The overall mass, momentum, energy, and scalar balances are obtained.

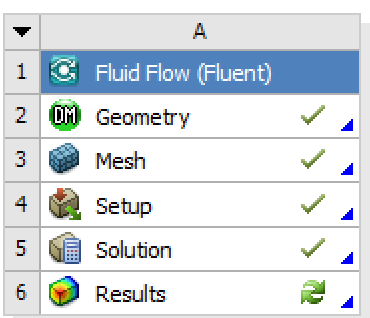
  You can examine the overall mass, momentum, energy and scalar balances in the **Flux Reports** dialog box. The net imbalance should be less than 0.2 % of the net flux through the domain when the solution has converged. In the next step you will check to see if the mass balance indicates convergence.

3. With ANSYS Fluent still running, go back to ANSYS Workbench and view the list of generated files.

   View → **Files**

   Note the addition of the surface report definition file `temp-outlet-0.out` to the list of files.

   The status of the **Solution** cell is now up-to-date.



elbow

1.4.7. Displaying Results in ANSYS Fluent and CFD-Post

In this step, you will display the results of the simulation in ANSYS Fluent, display the results in CFD-Post, and then review the list of files generated by ANSYS Workbench.

1. Display results in ANSYS Fluent using the **Postprocessing** ribbon tab.

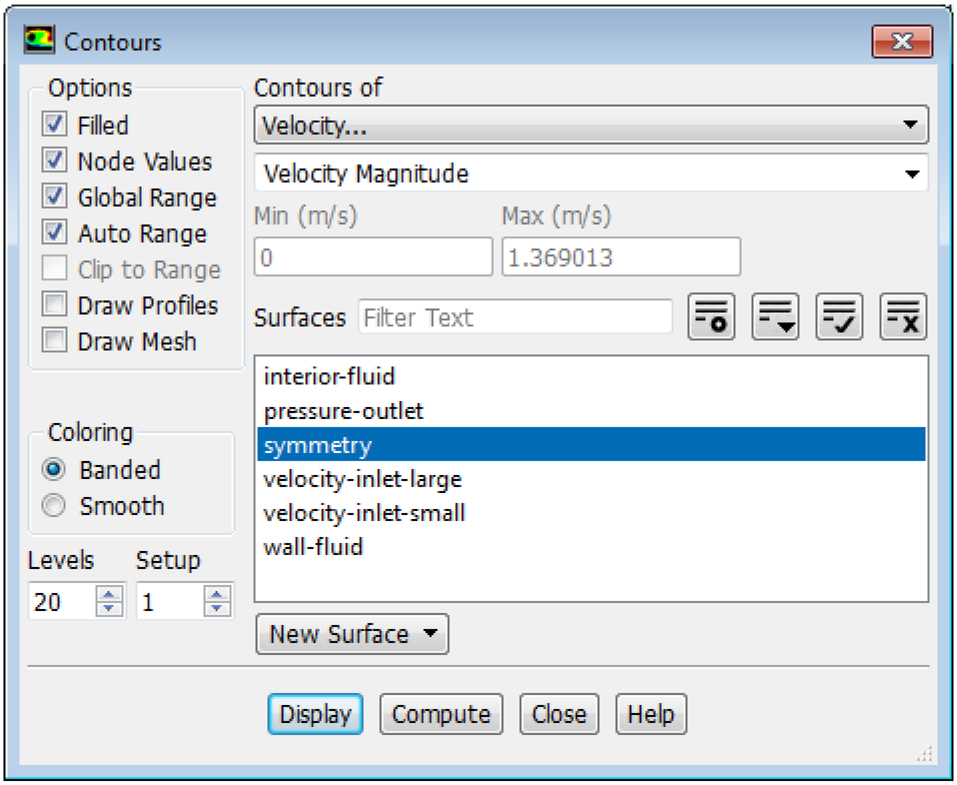
   In ANSYS Fluent, you can perform a simple evaluation of the velocity and temperature contours on the symmetry plane. Later, you will use CFD-Post (from within ANSYS Workbench) to perform the same evaluation.

a. Display filled contours of velocity magnitude on the symmetry plane (Figure 1.17: Velocity Distribution Along Symmetry Plane (p. 53)).

Postprocessing → **Graphics** → **Contours** → **Edit...**

Note

You can also double-click the **Results/Graphics/Contours** tree item.



i. In the **Contours** dialog box, in the **Options** group, enable **Filled**.

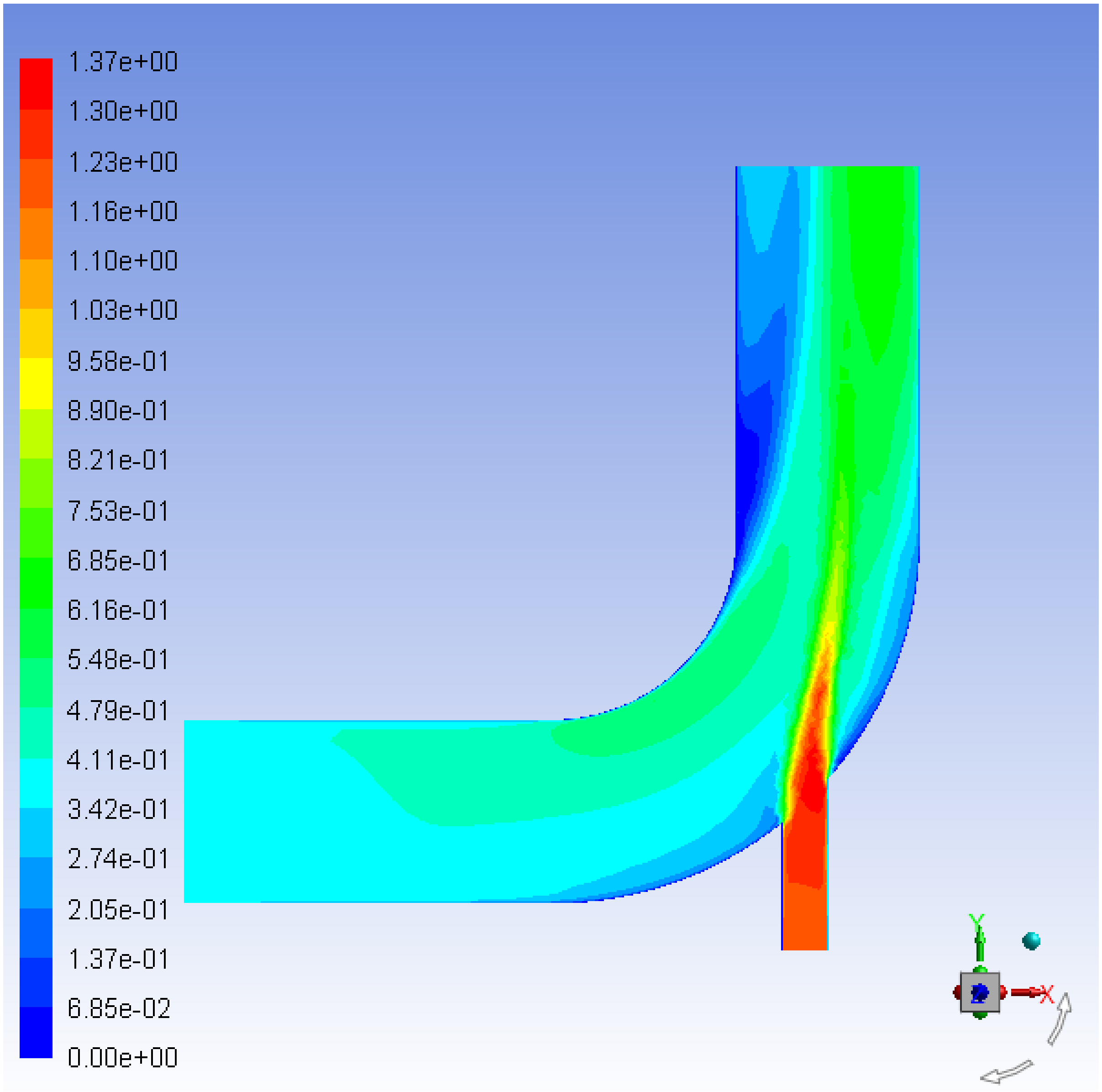
ii. Ensure that **Node Values** is enabled in the **Options** group.

iii. From the **Contours of** drop-down lists, select **Velocity...** and **Velocity Magnitude**.

iv. From the **Surfaces** selection list, deselect all items by clicking and then select **symmetry**.

v. Click **Display** to display the contours in the active graphics window.

Figure 1.17: Velocity Distribution Along Symmetry Plane

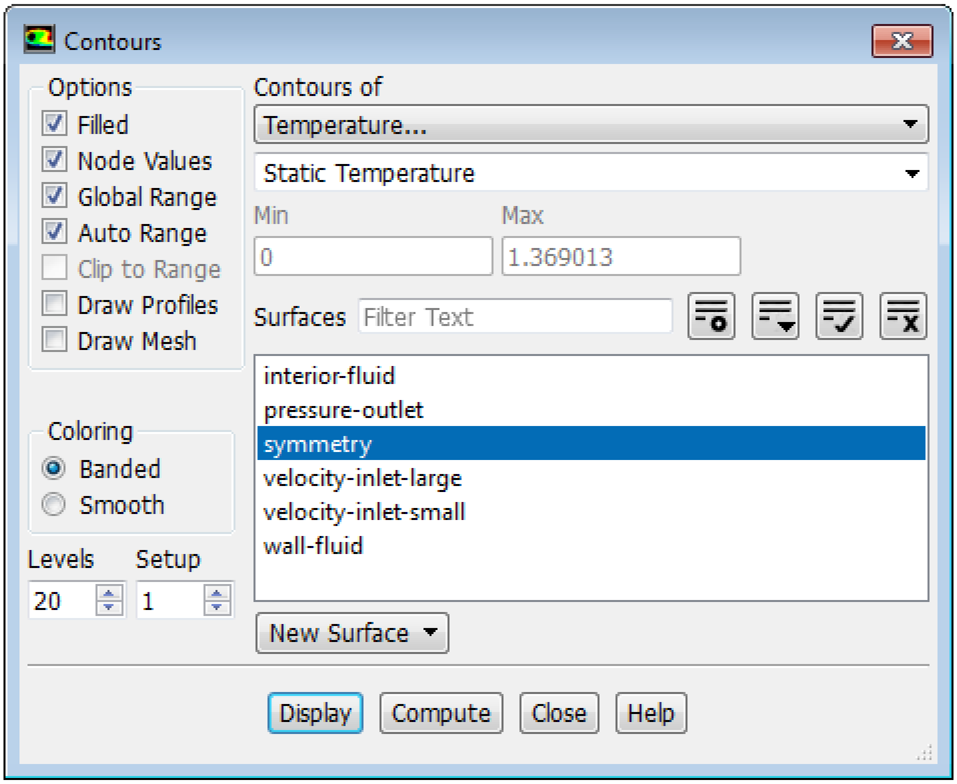


Note

You may want to clear **Lighting** in the **Viewing** ribbon tab (**Display** group).

b. Display filled contours of temperature on the symmetry plane (Figure 1.18: Temperature Distribution Along Symmetry Plane (p. 55)).

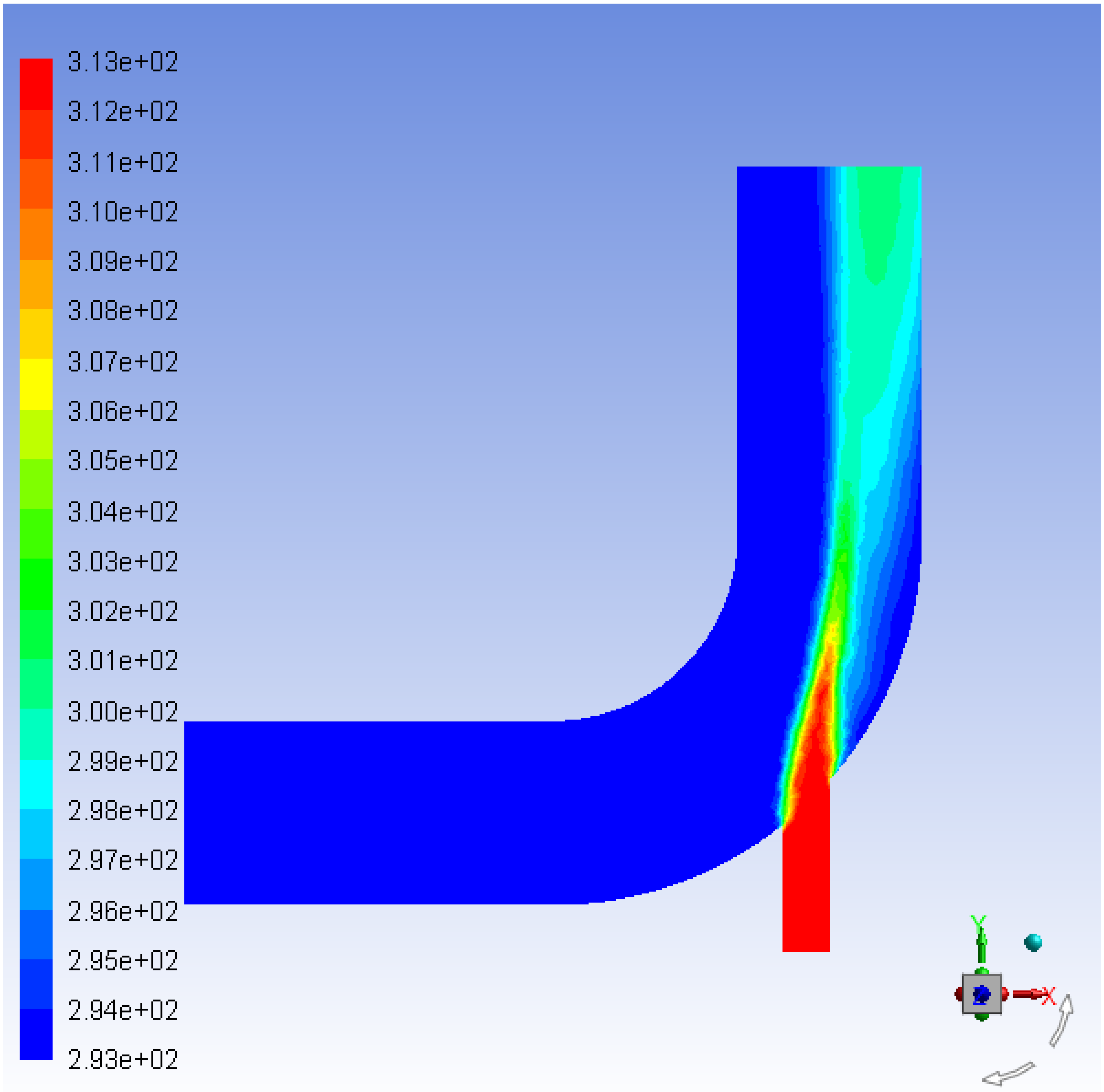
Postprocessing → **Graphics** → **Contours** → **Edit...**



i. Select **Temperature...** and **Static Temperature** from the **Contours of** drop-down lists.

ii. Click **Display** and close the **Contours** dialog box.

Figure 1.18: Temperature Distribution Along Symmetry Plane



c. Close the ANSYS Fluent application.

File → Close Fluent

Important

Note that the ANSYS Fluent case and data files are automatically saved when you exit ANSYS Fluent and return to ANSYS Workbench.

---

d. View the list of files generated by ANSYS Workbench.

View → Files

Note the addition of the compressed ANSYS Fluent case and data files to the list of files. These will have names like `FFF-1.cas.gz` and `FFF-1-00222.dat.gz`. Note that the digit(s) following FFF may be different if you have had to restart the meshing or calculation steps for any reason and that the name of the data file is based on the number of iterations. Thus your file names may be slightly different than those shown here.

2. Display results in CFD-Post.

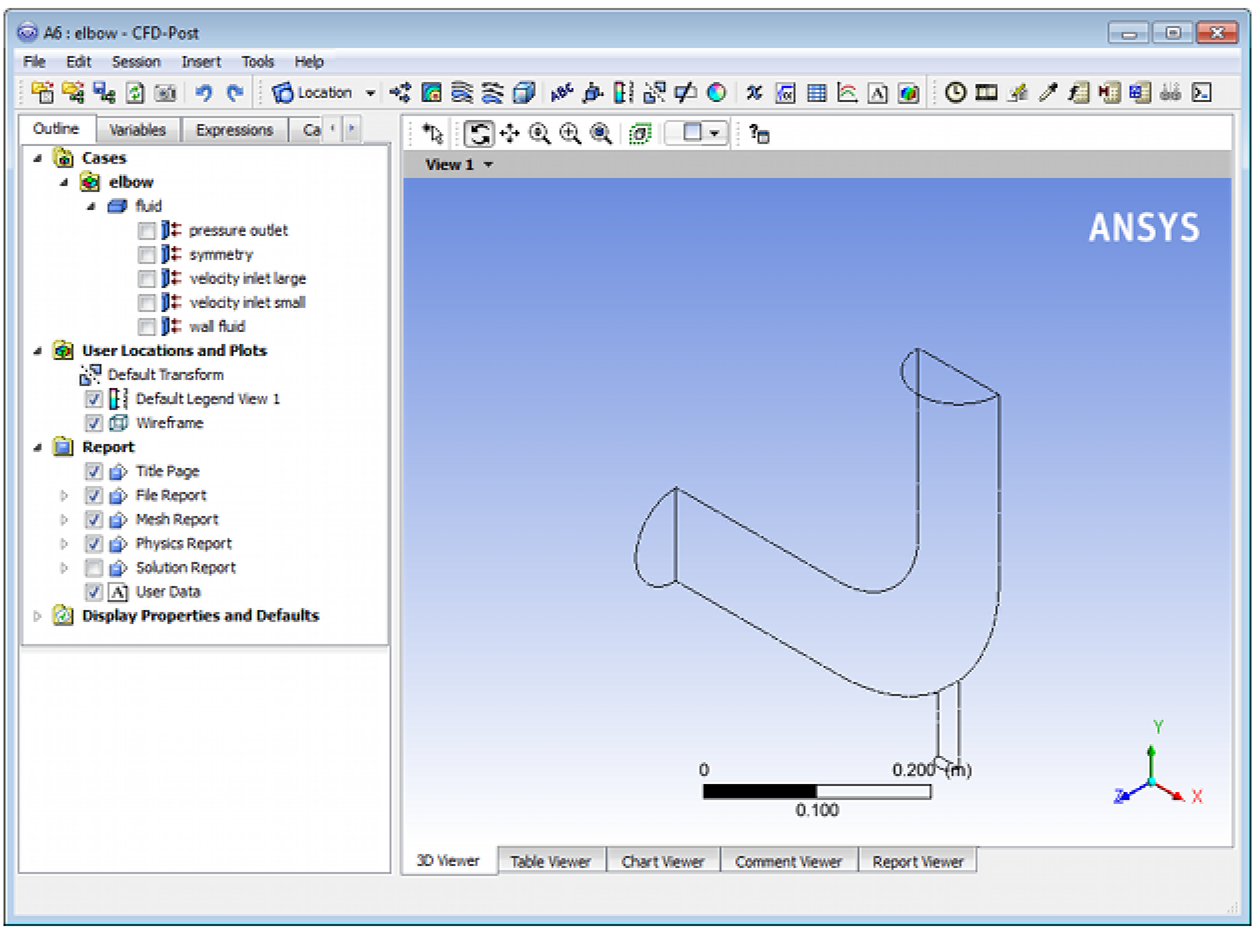
   a. Start CFD-Post.

      In the ANSYS Workbench **Project Schematic**, double-click the **Results** cell in the `elbow` fluid flow analysis system (cell A6). This displays the CFD-Post application. You can also right-click the **Results** cell to display the context menu where you can select the **Edit...** option.

      Note

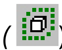
      The elbow geometry is already loaded and is displayed in outline mode. ANSYS Fluent case and data files are also automatically loaded into CFD-Post.

      Figure 1.19: The Elbow Geometry Loaded into CFD-Post



   b. Reorient the display.

      Click the blue Z axis on the axis triad in the bottom right hand corner of the graphics display to orient the display so that the view is of the front of the elbow geometry.

c. Ensure that *Highlighting* () is disabled.

d. Display contours of velocity magnitude on the symmetry plane (Figure 1.20: Velocity Distribution Along Symmetry Plane (p. 58)).

   i. Insert a contour object using the **Insert** menu item at the top of the CFD-Post window.

      Insert → **Contour**

      This displays the **Insert Contour** dialog box.

   ii. Keep the default name of the contour (`Contour 1`) and click **OK** to close the dialog box. This displays the **Details of Contour 1** view below the **Outline** view in CFD-Post. This view contains all of the settings for a contour object.

   iii. In the **Geometry** tab, from the **Domains** drop-down list, select **fluid**.

   iv. In the **Locations** list, select **symmetry**.

   v. In the **Variable** list, select **Velocity**.

   vi. Click **Apply**.

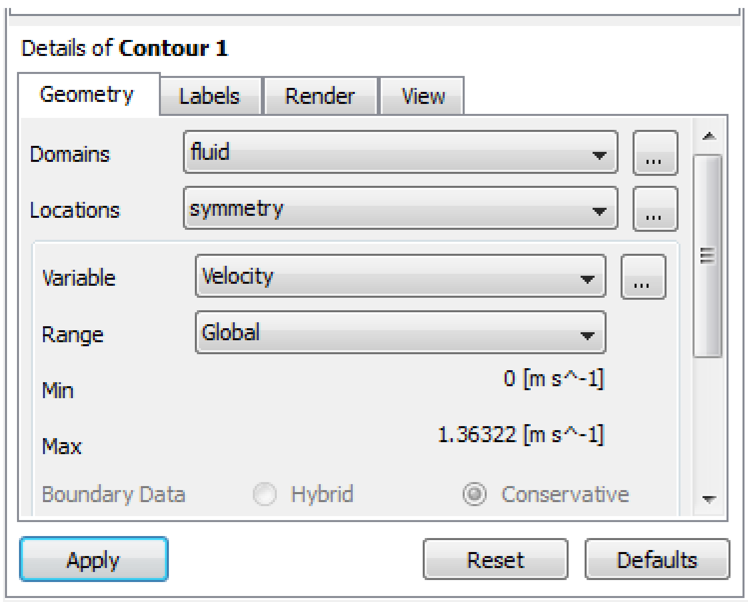
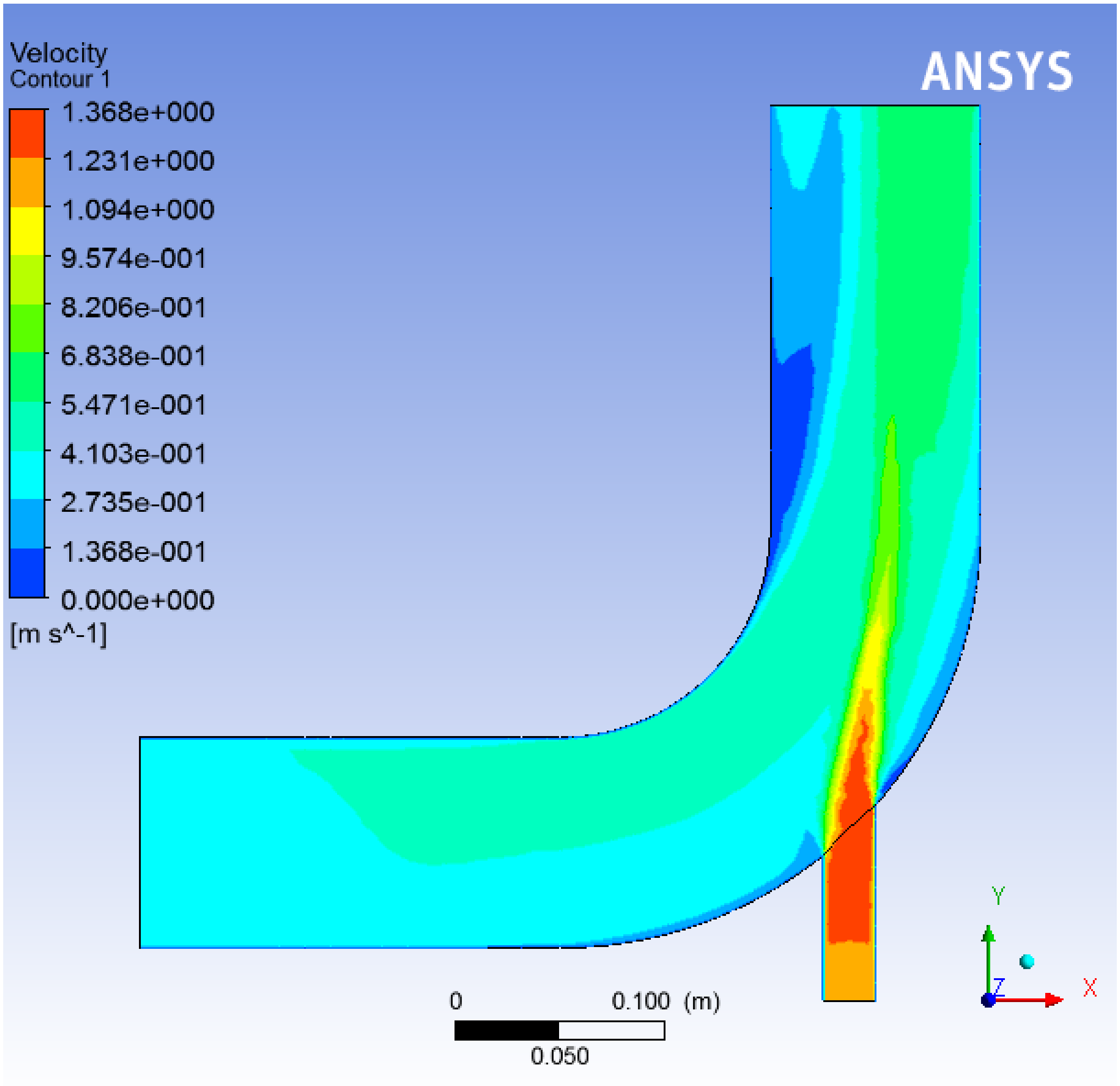


Figure 1.20: Velocity Distribution Along Symmetry Plane



e. Display contours of temperature on the symmetry plane (Figure 1.21: Temperature Distribution Along Symmetry Plane (p. 59)).

   i. In the **Outline** tree view, under **User Locations and Plots**, clear the check box beside the **Contour 1** object to disable the **Contour 1** object and hide the first contour display.

   ii. Insert a contour object.

      Insert → **Contour**

      This displays the **Insert Contour** dialog box.

   iii. Keep the default name of the contour (`Contour 2`) and click **OK** to close the dialog box. This displays the **Details of Contour 2** view below the **Outline** view.

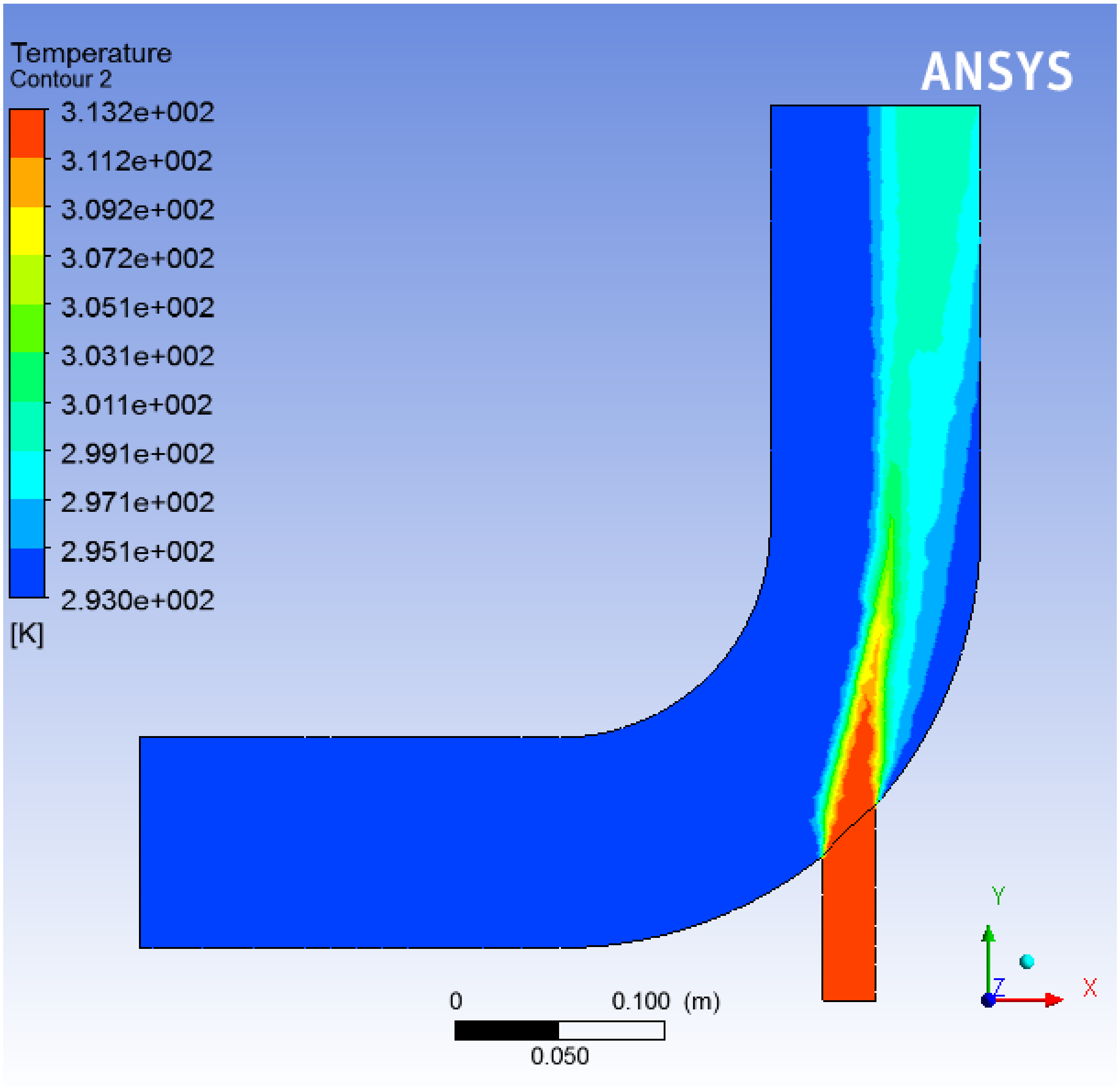
   iv. In the **Geometry** tab, from the **Domains** drop-down list, select **fluid**.

v. In the **Locations** list, select **symmetry**.

vi. In the **Variable** list, select **Temperature**.

vii. Click **Apply**.

Figure 1.21: Temperature Distribution Along Symmetry Plane



3. Close the CFD-Post application by selecting **File** → **Close CFD-Post** or by clicking the 'X' in the top right corner of the window.

Important

Note that the CFD-Post state files are automatically saved when you exit CFD-Post and return to ANSYS Workbench.

4. Save the `elbow-workbench` project in ANSYS Workbench.

5. View the list of files generated by ANSYS Workbench.

   View → Files

   Note the addition of the CFD-Post state file (`elbow.cst`) to the list of files. For more information about CFD-Post (and the files associated with it), see the CFD-Post documentation.

1.4.8. Duplicating the Fluent-Based Fluid Flow Analysis System

At this point, you have a completely defined fluid flow system that has a geometry, a computational mesh, a CFD setup and solution, and corresponding results. In order to study the effects upon the flow field that may occur if you were to alter the geometry, another fluid flow analysis is required. One approach would be to use the current system and change the geometry, however you would overwrite the data from your previous simulation. A more suitable and effective approach would be to create a copy, or duplicate, of the current system, and then make the appropriate changes to the duplicate system.

In this step, you will create a duplicate of the original Fluent-based fluid flow system, then review the list of files generated by ANSYS Workbench.

1. In the **Project Schematic**, right-click the title cell of the **Fluid Flow (Fluent)** system and select **Duplicate** from the context menu.

   Figure 1.22: Duplicating the Fluid Flow System

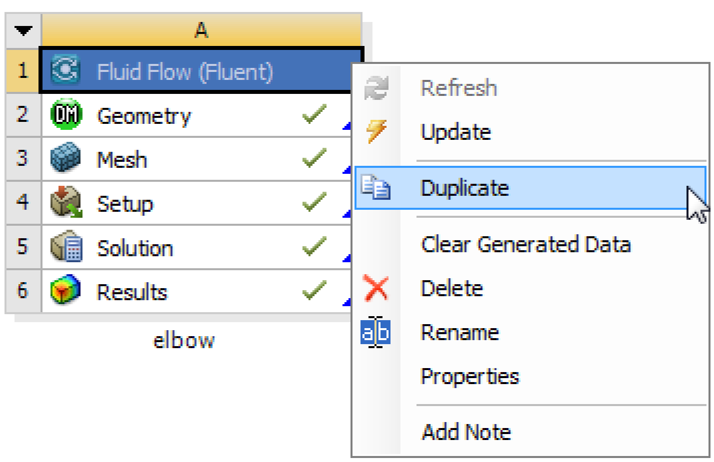
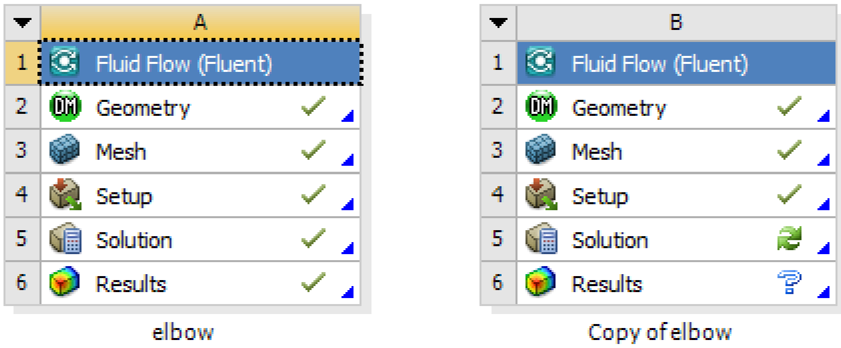


Figure 1.23: The Original Fluid Flow System and Its Duplicate



Note

Notice that in the duplicated system, the state of the **Solution** cell indicates that the cell requires a refresh while the state of the **Results** cell indicates that the cell requires attention. This is because when a system is duplicated, the case and data files are not copied to the new system, therefore, the new system does not yet have solution data associated with it.

---

2. Rename the duplicated system to `new-elbow`.

3. Save the `elbow-workbench` project in ANSYS Workbench.

1.4.9. Changing the Geometry in ANSYS DesignModeler

Now that you have two separate, but equivalent, Fluent-based fluid flow systems to work from, you can make changes to the second system without impacting the original system. In this step, you will make a slight alteration to the elbow geometry in ANSYS DesignModeler by changing the diameter of the smaller inlet, then review the list of files generated by ANSYS Workbench.

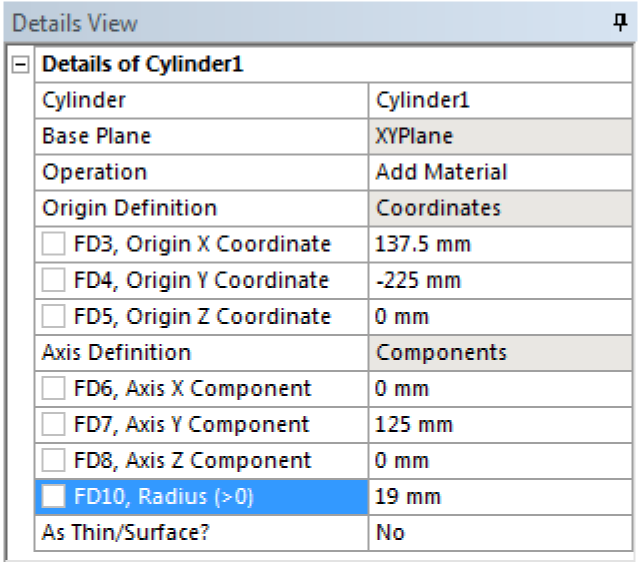
1. Open ANSYS DesignModeler.

   Right-click the **Geometry** cell of the `new-elbow` system (cell B2) and select **Edit Geometry in DesignModeler...** from the context menu to display the geometry in ANSYS DesignModeler.

2. Change the diameter of the small inlet (`velocity-inlet-small`).

   a. In the **Tree Outline**, select **Cylinder1** to open the **Details View** of the small inlet pipe.

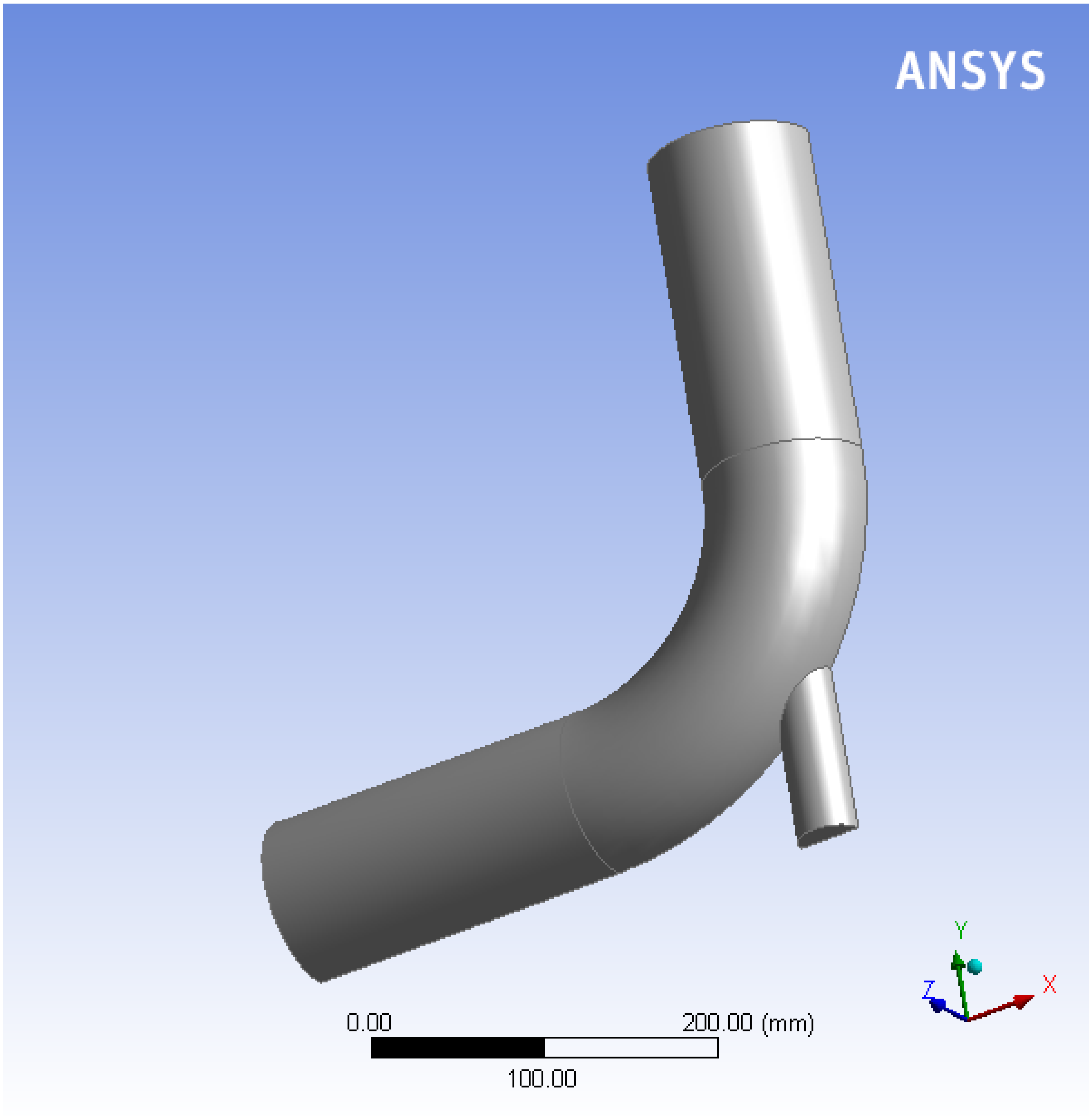
   b. In the **Details View**, change the **FD10, Radius (>0)** value from `12.5` millimeters to `19` millimeters.


Details View

Details of Cylinder1	
Cylinder	Cylinder1
Base Plane	XYPlane
Operation	Add Material
Origin Definition	Coordinates
FD3, Origin X Coordinate	137.5 mm
FD4, Origin Y Coordinate	-225 mm
FD5, Origin Z Coordinate	0 mm
Axis Definition	Components
FD6, Axis X Component	0 mm
FD7, Axis Y Component	125 mm
FD8, Axis Z Component	0 mm
FD10, Radius (>0)	19 mm
As Thin/Surface?	No

c. Click the **Generate** button to generate the geometry with your new values.

Figure 1.24: Changing the Diameter of the Small Inlet in ANSYS DesignModeler



3. Close ANSYS DesignModeler.

4. View the list of files generated by ANSYS Workbench.

   View → **Files**

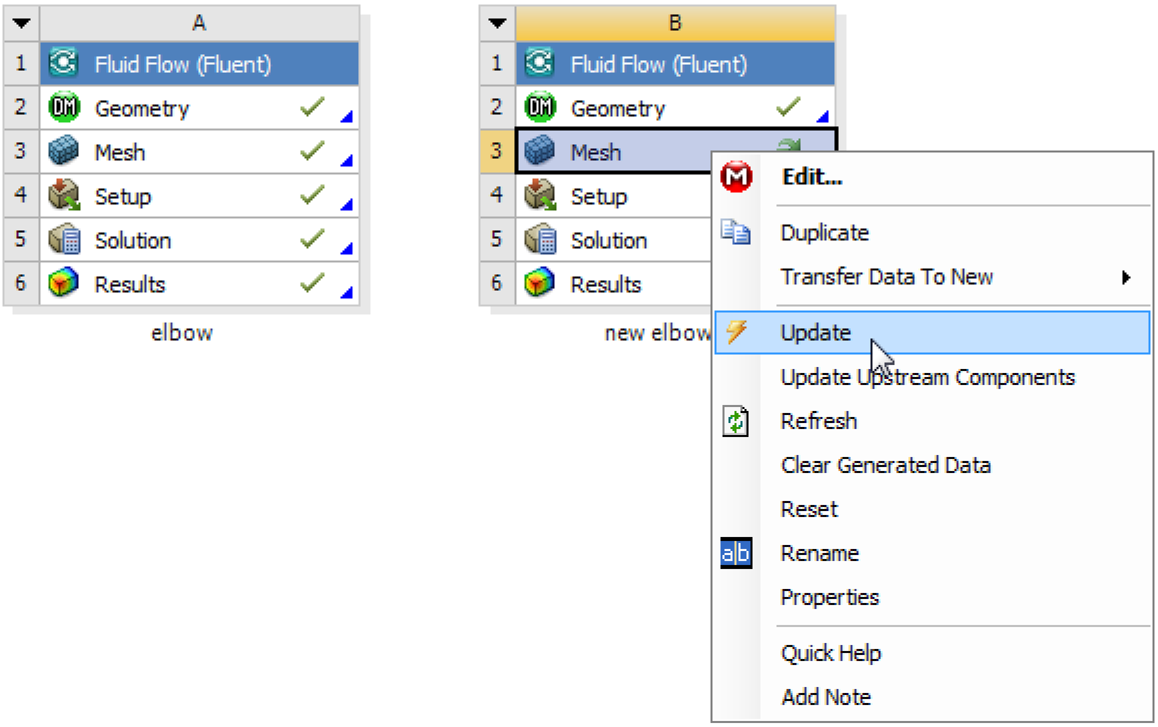
   Note the addition of the geometry, mesh, and ANSYS Fluent settings files now associated with the new, duplicated system.

1.4.10. Updating the Mesh in the ANSYS Meshing Application

The modified geometry now requires a new computational mesh. The mesh settings for the duplicated system are retained in the duplicated system. In this step, you will update the mesh based on the mesh settings from the original system, then review the list of files generated by ANSYS Workbench.

In the **Project Schematic**, right-click the **Mesh** cell of the `new-elbow` system (cell B3) and select **Update** from the context menu. This will update the mesh for the new geometry based on the mesh settings you specified earlier in the ANSYS Meshing application without having to open the ANSYS Meshing editor to regenerate the mesh.

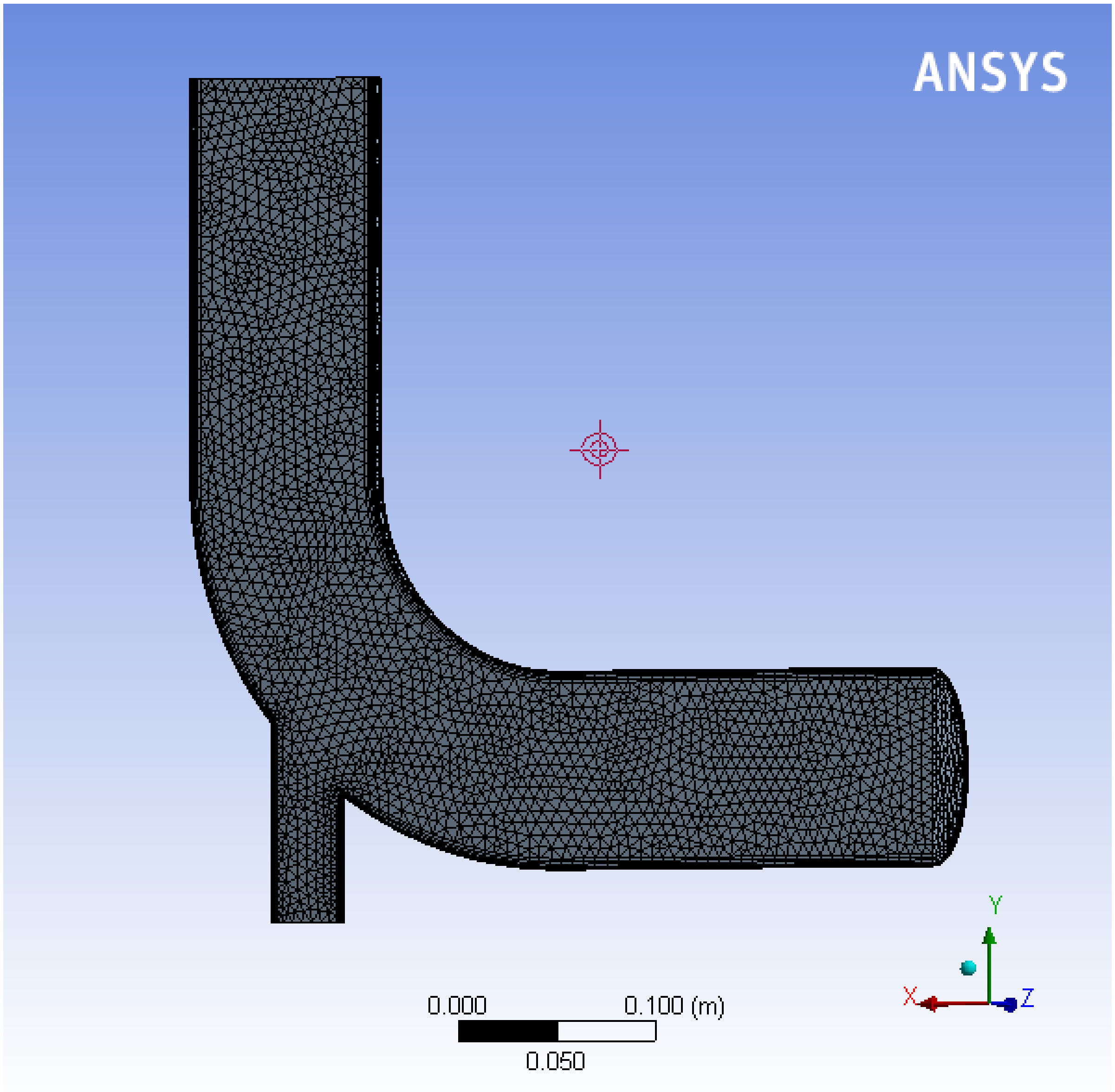
Figure 1.25: Updating the Mesh for the Changed Geometry



It will take a few moments to update the mesh. Once the update is complete, the state of the **Mesh** cell is changed to up-to-date, symbolized by a green check mark.

For illustrative purposes of the tutorial, the new geometry and the new mesh are displayed below.

Figure 1.26: The Updated Geometry and Mesh in the ANSYS Meshing Application



Inspecting the files generated by ANSYS Workbench reveals the updated mesh file for the duplicated system.

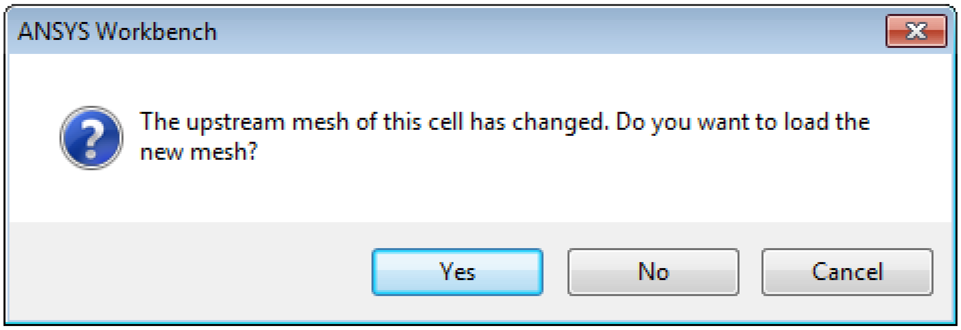
View → **Files**

1.4.11. Calculating a New Solution in ANSYS Fluent

Now that there is an updated computational mesh for the modified geometry in the duplicated system, a new solution must be generated using ANSYS Fluent. In this step, you will revisit the settings within ANSYS Fluent, calculate another solution, view the new results, then review the list of files generated by ANSYS Workbench.

1. Open ANSYS Fluent.

In the **Project Schematic**, right-click the **Setup** cell of the `new-elbow` system (cell B4) and select **Edit...** from the context menu. Since the mesh has been changed, you are prompted as to whether you want to load the new mesh into ANSYS Fluent or not.



Click **Yes** to continue, and click **OK** when Fluent Launcher is displayed in order to open ANSYS Fluent.

2. Ensure that the unit of length is set to millimeters.

   Setting Up Domain → **Mesh** → **Units...**

3. Check the mesh (optional).

   Setting Up Domain → **Mesh** → **Check**

4. Revisit the boundary conditions for the small inlet.

   Setup → **Boundary Conditions** → **velocity-inlet-small** **Edit...**

   In the **Velocity Inlet** dialog box for **velocity-inlet-small**, you must set the hydraulic diameter to 38 mm based on the new dimensions of the small inlet.

5. Re-initialize the solution.

   Solving → **Initialization**

   Keep the **Method** at the default of **Hybrid** and click **Initialize**.

6. Recalculate the solution.

   Solving → **Run Calculation** → **Calculate**

   Keep the **No. of Iterations** set to 300 and click **Calculate**.

7. Close ANSYS Fluent.

8. Revisit the results of the calculations in CFD-Post.

   Double-click the **Results** cell of the `new-elbow` fluid flow system to re-open CFD-Post where you can review the results of the new solution.

9. Close CFD-Post.

10. Save the `elbow-workbench` project in ANSYS Workbench.

11. View the list of files generated by ANSYS Workbench.

    View → Files

    Note the addition of the solution and state files now associated with new duplicated system.

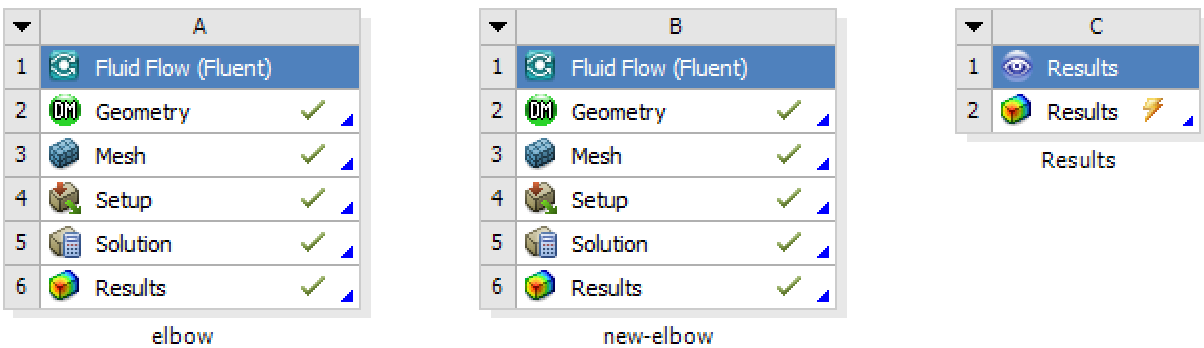
1.4.12. Comparing the Results of Both Systems in CFD-Post

In this step, you will create a new **Results** system in ANSYS Workbench, use that system to compare the solutions from each of the two Fluent-based fluid flow analysis systems in CFD-Post at the same time, then review the list of files generated by ANSYS Workbench.

1. Create a **Results** system.

   In ANSYS Workbench, drag a **Results** system from the **Component Systems** section of the **Toolbox** and drop it into the **Project Schematic**, next to the fluid flow systems.

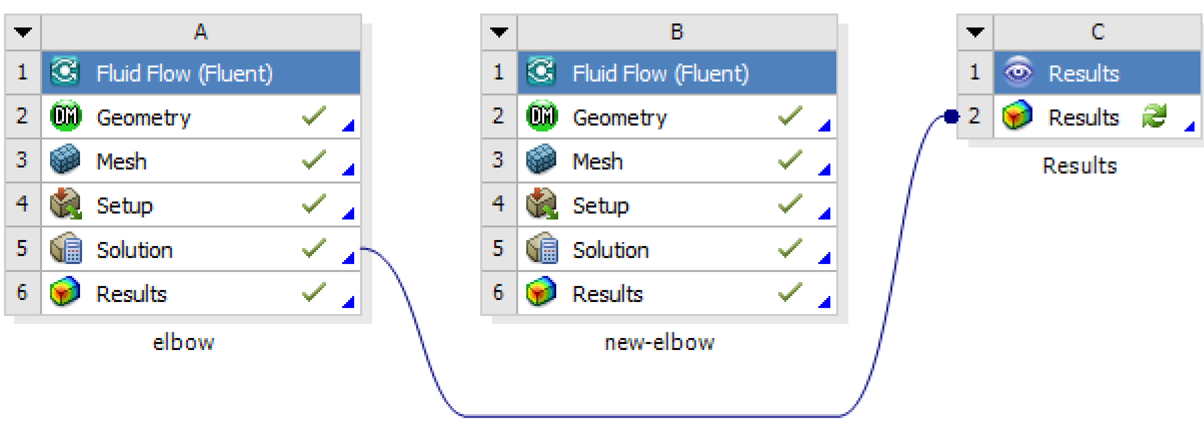
   Figure 1.27: The NewResults System in the Project Schematic



2. Add the solutions of each of the systems to the **Results** system.

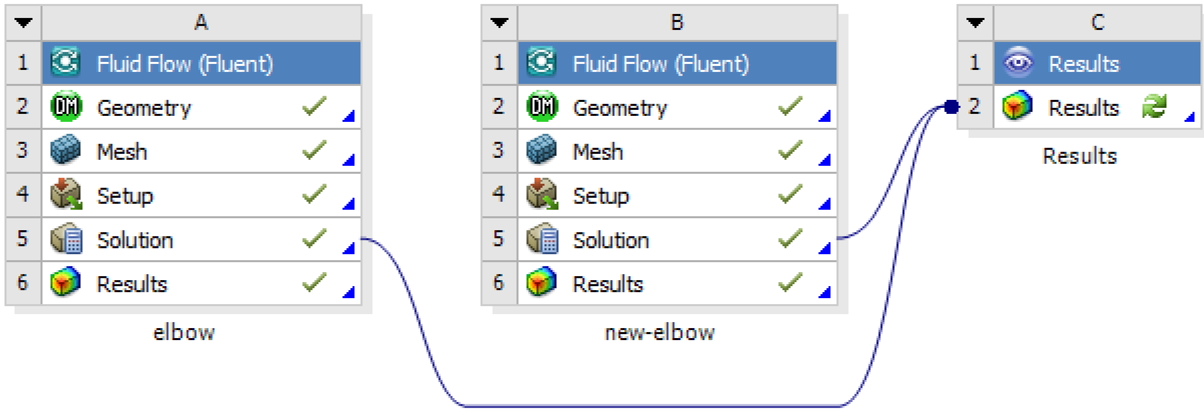
   a. Select the **Solution** cell in the first Fluid Flow analysis system (cell A5) and drag it over the **Results** cell in the **Results** system (cell C2). This creates a transfer data connection between the two systems.

   Figure 1.28: Connecting the First Fluid Flow System to the New Results System



b. Select the **Solution** cell in the second Fluid Flow analysis system (cell B5) and drag it over the **Results** cell in the **Results** system (cell C2). This creates a transfer data connection between the two systems.

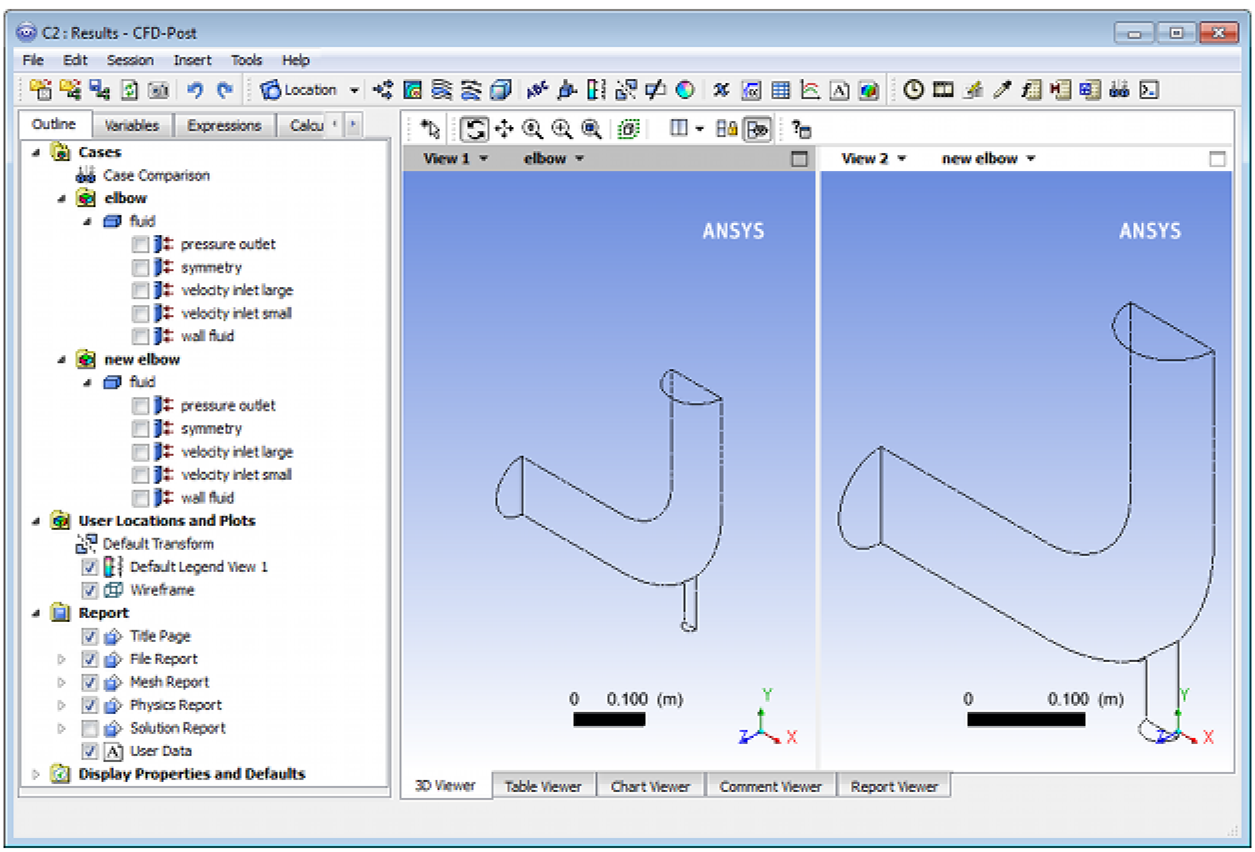
Figure 1.29: Connecting the Second Fluid Flow System to the New Results System



3. Open CFD-Post to compare the results of the two fluid flow systems.

   Now that the two fluid flow systems are connected to the **Results** system, double-click the **Results** cell in the **Results** system (cell C2) to open CFD-Post. Within CFD-Post, both geometries are displayed side by side.

Figure 1.30: CFD-Post with Both Fluid Flow Systems Displayed



a. Re-orient the display.

In each view, click the blue Z axis on the axis triad in the bottom right hand corner of the graphics display to orient the display so that the view is of the front of the elbow geometry.

Important

Alternatively, you can select the synchronization tool () in the **3D Viewer Toolbar** to synchronize the views, so that when you re-orient one view, the other view is automatically updated.

---

b. Display contours of velocity magnitude on the symmetry plane.

  i. Insert a contour object.

  Insert → **Contour**

  This displays the **Insert Contour** dialog box.

  ii. Keep the default name of the contour (`Contour 1`) and click **OK** to close the dialog box. This displays the **Details of Contour 1** view below the **Outline** view in CFD-Post. This view contains all of the settings for a contour object.

  iii. In the **Geometry** tab, from the **Domains** list, select **fluid**.

  iv. In the **Locations** list, select **symmetry**.

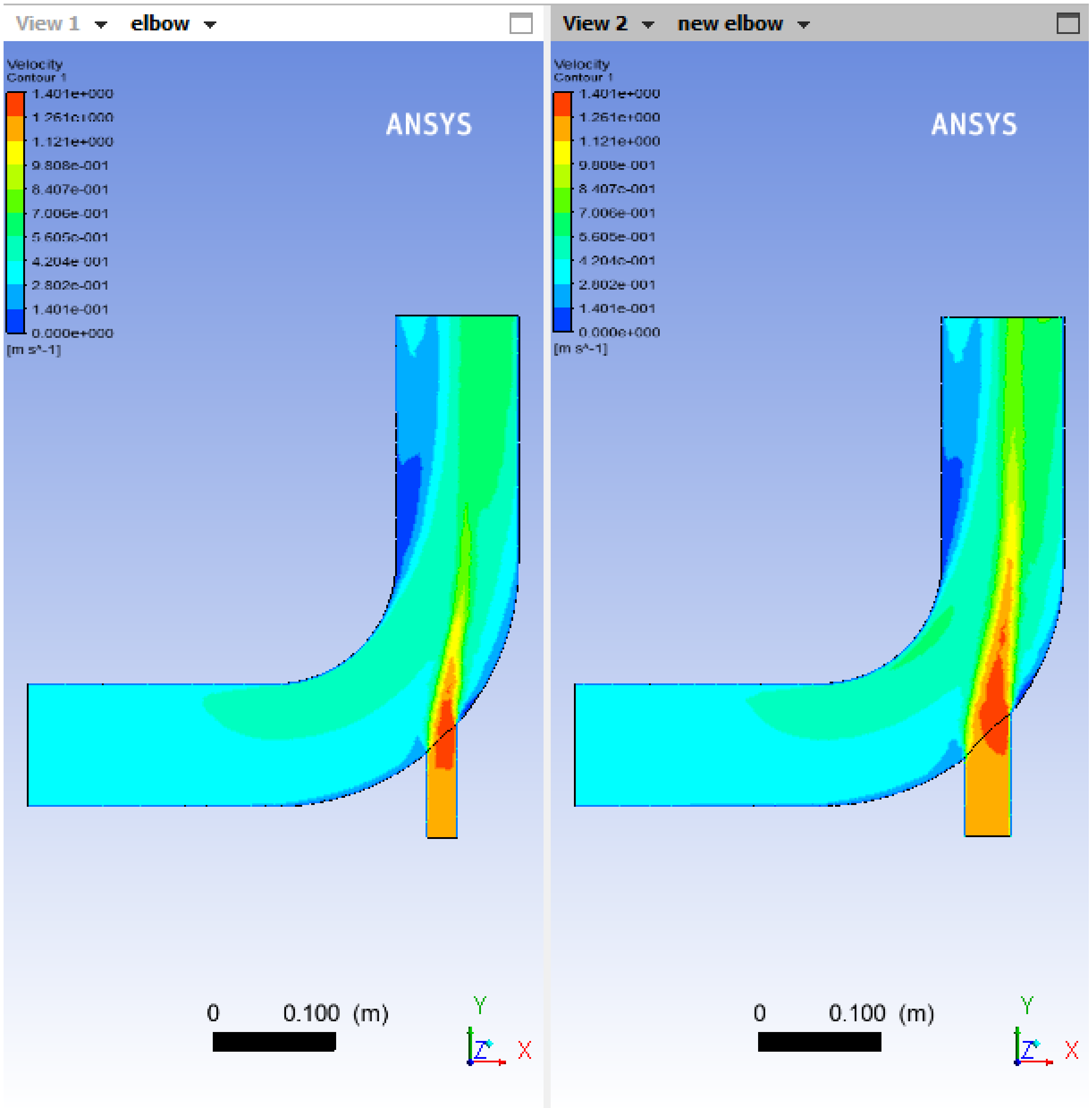
  v. In the **Variable** list, select **Velocity**.

  vi. Click **Apply**. The velocity contours are displayed in each view.

Note

To better visualize the velocity display, you can clear the **Wireframe** view option under **User Locations and Plots** in the **Outline** tree view.

Figure 1.31: CFD-Post Displaying Velocity Contours for Both Geometries



c. Display contours of temperature on the symmetry plane.

   i. In the **Outline** tree view, under **User Locations and Plots**, deselect the **Contour 1** object to hide the first contour display in CFD-Post.

   ii. Insert another contour object.

   Insert → **Contour**

   This displays the **Insert Contour** dialog box.

   iii. Keep the default name of the contour (`Contour 2`) and click **OK** to close the dialog box. This displays the **Details of Contour 2** view below the **Outline** view in CFD-Post.

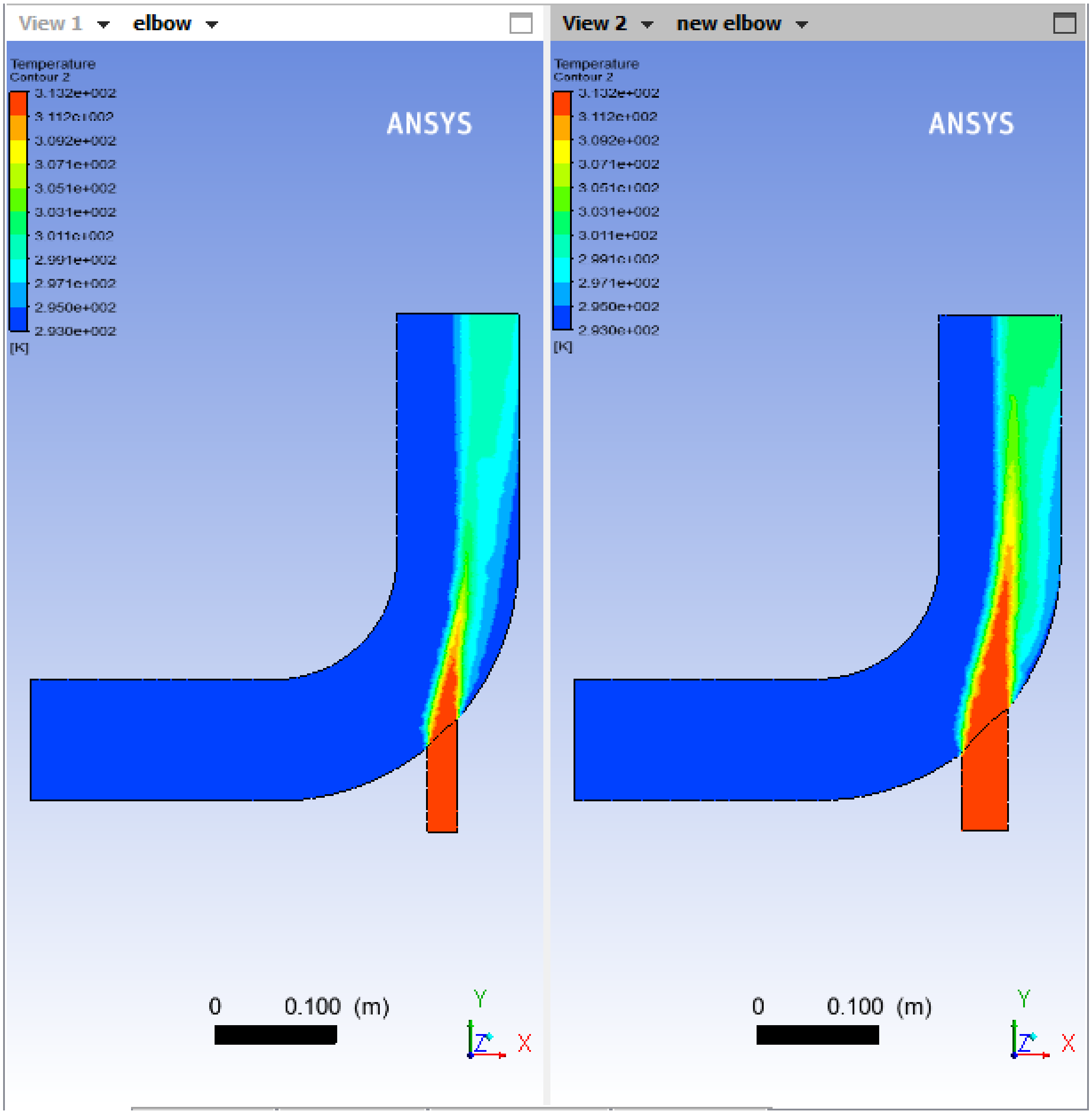
iv. In the **Geometry** tab, select **fluid** in the **Domains** list.

v. Select **symmetry** in the **Locations** list.

vi. Select **Temperature** in the **Variable** list.

vii. Click **Apply**. The temperature contours are displayed in each view.

Figure 1.32: CFD-Post Displaying Temperature Contours for Both Geometries



4. Close the CFD-Post application.

5. Save the `elbow-workbench` project in ANSYS Workbench.

6. View the list of files associated with your project using the **Files** view.

   View → **Files**

Note the addition of the **Results** system and its corresponding files.

1.5. Summary

In this tutorial, portions of ANSYS Workbench were used to compare the fluid flow through two slightly different geometries. ANSYS DesignModeler was used to create a mixing elbow geometry, ANSYS Meshing was used to create a computational mesh, ANSYS Fluent was used to calculate the fluid flow throughout the geometry using the computational mesh, and CFD-Post was used to analyze the results. In addition, the geometry was altered, a new mesh was generated, and a new solution was calculated. Finally, ANSYS Workbench was set up so that CFD-Post could directly compare the results of both calculations at the same time.

Chapter 2: Parametric Analysis in ANSYS Workbench Using ANSYS Fluent

This tutorial is divided into the following sections:

2.1. Introduction

This tutorial illustrates using an ANSYS Fluent fluid flow system in ANSYS Workbench to set up and solve a three-dimensional turbulent fluid flow and heat transfer problem in an automotive heating, ventilation, and air conditioning (HVAC) duct system. ANSYS 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 ANSYS Fluent that can be used in your ANSYS Workbench project. You can also define parameters in other applications including 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. This tutorial is designed to introduce you to the parametric analysis utility available in ANSYS Workbench.

The tutorial starts with a Fluid Flow (Fluent) analysis system with pre-defined geometry and mesh components. Within this tutorial, you will redefine the geometry parameters created in ANSYS DesignModeler by adding constraints to the input parameters. You will use ANSYS Fluent to set up and solve the CFD problem. While defining the problem set-up, you will also learn to define input parameters in ANSYS Fluent. The tutorial will also provide information on how to create output parameters in ANSYS CFD-Post.

This tutorial demonstrates how to do the following:

- Add constraints to the ANSYS DesignModeler input parameters.
- Create an ANSYS Fluent fluid flow analysis system in ANSYS Workbench.
- Set up the CFD simulation in ANSYS Fluent, which includes:
  - Setting material properties and boundary conditions for a turbulent forced convection problem.
  - Defining input parameters in Fluent
- Define output parameters in CFD-Post
- Create additional design points in ANSYS Workbench.
- Run multiple CFD simulations by updating the design points.

- Analyze the results of each design point project in ANSYS CFD-Post and ANSYS Workbench.

Important

The mesh and solution settings for this tutorial are designed to demonstrate a basic parameterization simulation within a reasonable solution time-frame. Ordinarily, you would use additional mesh and solution settings to obtain a more accurate solution.

2.2. Prerequisites

This tutorial assumes that you are already familiar with the ANSYS Workbench interface and its project workflow (for example, ANSYS DesignModeler, ANSYS Meshing, ANSYS Fluent, and ANSYS CFD-Post). This tutorial also assumes that you have completed Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1), and that you are familiar with the ANSYS Fluent graphical user interface. Some steps in the setup and solution procedure will not be shown explicitly.

2.3. Problem Description

In the past, evaluation of vehicle air conditioning systems was performed using prototypes and testing their performance in test labs. However, the design process of modern vehicle air conditioning (AC) systems improved with the introduction of Computer Aided Design (CAD), Computer Aided Engineering (CAE) and Computer Aided Manufacturing (CAM). The AC system specification will include minimum performance requirements, temperatures, control zones, flow rates, and so on. Performance testing using CFD may include fluid velocity (air flow), pressure values, and temperature distribution. Using CFD enables the analysis of fluid through very complex geometry and boundary conditions.

As part of the analysis, a designer can change the geometry of the system or the boundary conditions such as the inlet velocity, flow rate, and so on, and view the effect on fluid flow patterns. This tutorial illustrates the AC design process on a representative automotive HVAC system consisting of both an evaporator for cooling and a heat exchanger for heating requirements. This HVAC system is symmetric, so the geometry has been simplified using a plane of symmetry to reduce computation time.

Figure 2.1: Automotive HVAC System

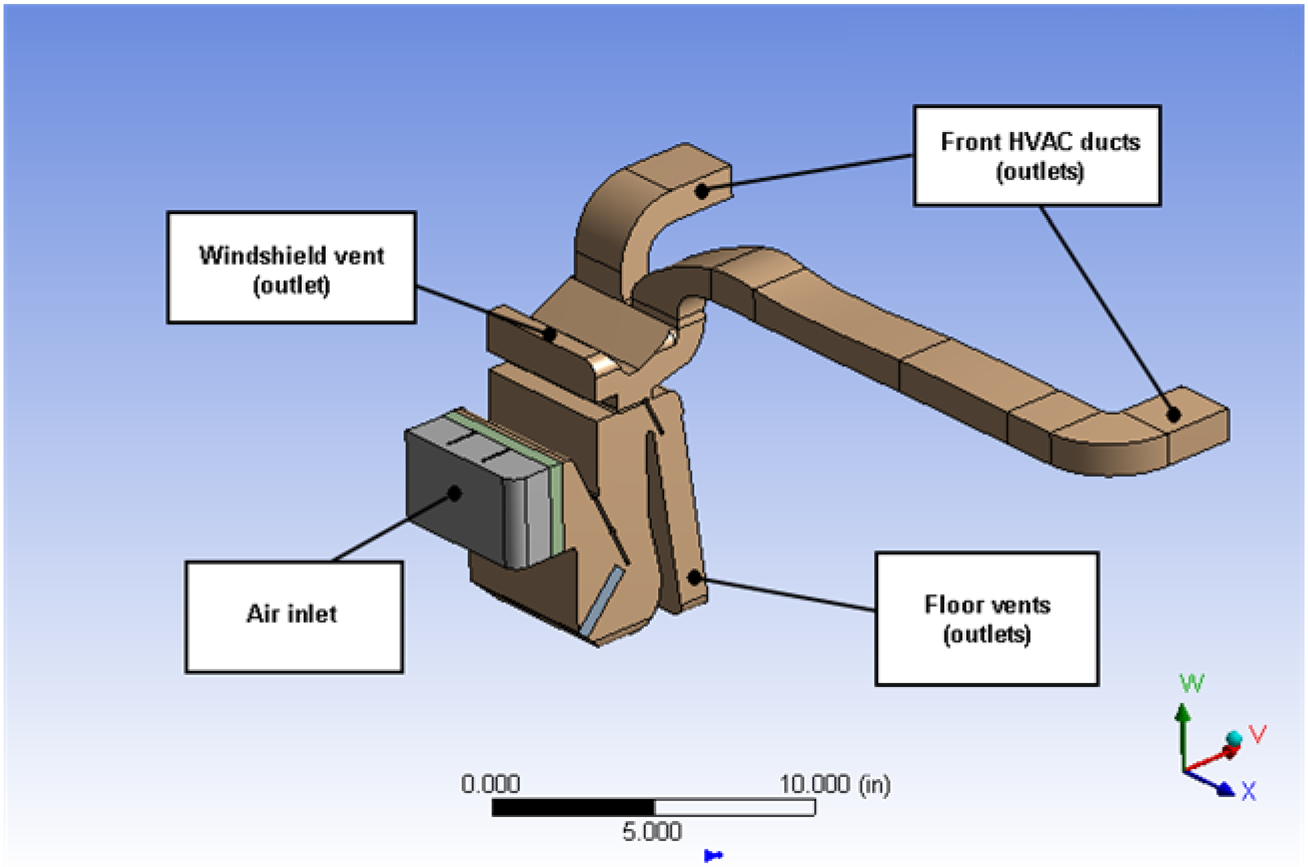


Figure 2.2: HVAC System Valve Location Details

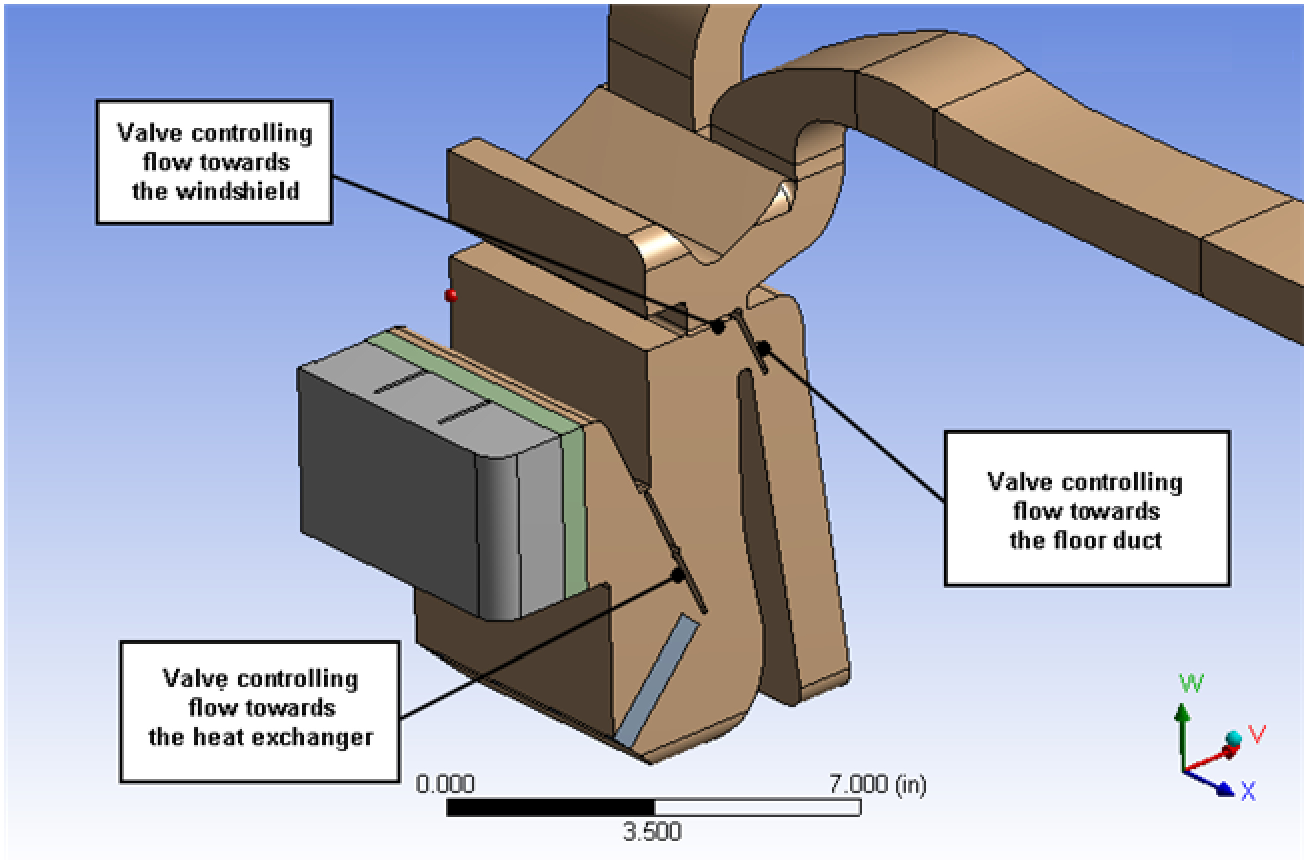


Figure 2.1: Automotive HVAC System (p. 75) shows a representative automotive HVAC system. The system has three valves (as shown in Figure 2.2: HVAC System Valve Location Details (p. 76)), which control the flow in the HVAC system. The three valves control:

- Flow over the heat exchanger coils
- Flow towards the duct controlling the flow through the floor vents
- Flow towards the front vents or towards the windshield

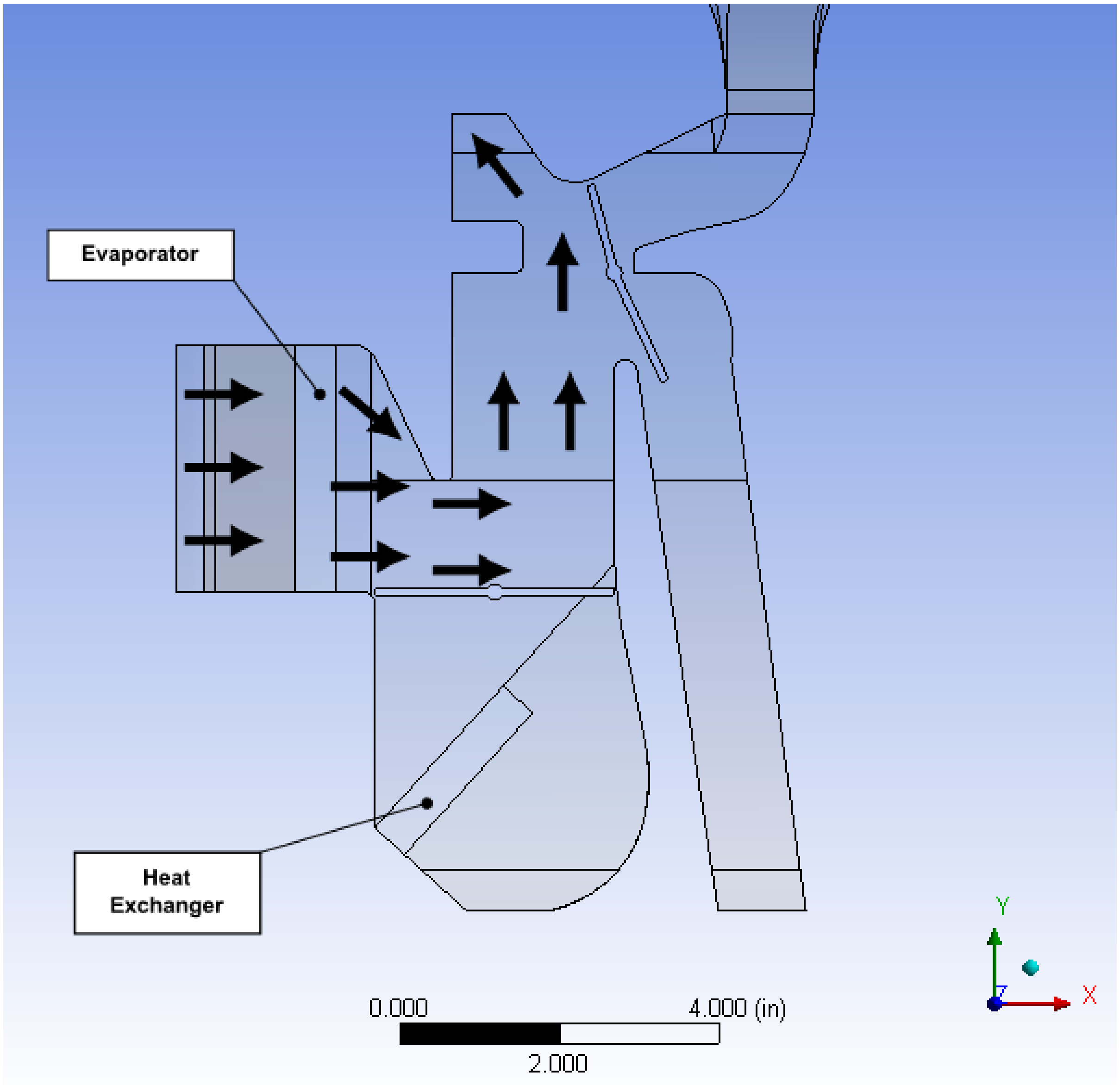
Air enters the HVAC system at 310 K with a velocity of 0.5 m/sec through the air inlet and passes to the evaporator and then, depending on the position of the valve controlling flow to the heat exchanger, flows over or bypasses the heat exchanger. Depending on the cooling and heating requirements, either the evaporator or the heat exchanger would be operational, but not both at the same time. The position of the other two valves controls the flow towards the front panel, the windshield, or towards the floor ducts.

The motion of the valves is constrained. The valve controlling flow over the heat exchanger varies between 25° and 90°. The valve controlling the floor flow varies between 20° and 60°. The valve controlling flow towards front panel or windshield varies between 15° and 175°.

The evaporator load is about 200 W in the cooling cycle. The heat exchanger load is about 150 W.

This tutorial illustrates the easiest way to analyze the effects of the above parameters on the flow pattern/distribution and the outlet temperature of air (entering the passenger cabin). Using the parametric analysis capability in ANSYS Workbench, a designer can check the performance of the system at various design points.

Figure 2.3: Flow Pattern for the Cooling Cycle



2.4. Setup and Solution

To help you quickly identify graphical user interface items at a glance and guide you through the steps of setting up and running your simulation, the ANSYS Fluent Tutorial Guide uses several type styles and mini flow charts. See Typographical Conventions Used In This Manual (p. xvi) for detailed information.

The following sections describe the setup and solution steps for this tutorial:

2.4.1. Preparation

To prepare for running this tutorial:

1. Set up a working folder on the computer you will be using.

2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

   Note

   If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

3. Enter the name of this tutorial into the search bar.

4. Narrow the results by using the filter on the left side of the page.

   a. Click **ANSYS Fluent** under **Product**.

   b. Click **18.0** under **Version**.

5. Select this tutorial from the list.

6. Click the **workbench-parameter-tutorial_R180.zip** link to download the input files.

7. Unzip the `workbench-parameter-tutorial_R180.zip` file to your working folder.

   The extracted `workbench-parameter-tutorial` folder contains a single archive file `fluent-workbench-param.wbpz` that includes all supporting input files of the starting ANSYS Workbench project and a folder called `final_project_files` that includes the archived final version of the project. The final result files incorporate ANSYS Fluent and ANSYS CFD-Post settings and all already defined design points (all that is required is to update the design points in the project to generate corresponding solutions).

Note

ANSYS Fluent tutorials are prepared using ANSYS Fluent on a Windows system. The screen shots and graphic images in the tutorials may be slightly different than the appearance on your system, depending on the operating system or graphics card.

2.4.2. Adding Constraints to ANSYS DesignModeler Parameters in ANSYS Workbench

In this step, you will start ANSYS Workbench, open the project file, review existing parameters, create new parameters, and add constraints to existing ANSYS DesignModeler parameters.

1. From the Windows **Start** menu, select **Start > All Programs > ANSYS 18.0** > **Workbench 18.0** to start ANSYS Workbench.

   This displays the ANSYS Workbench application window, which has the **Toolbox** on the left and the **Project Schematic** to its right. Various supported applications are listed in the **Toolbox**, and the components of the analysis system are displayed in the **Project Schematic**.

   Note

   When you first start ANSYS Workbench, the **Getting Started** message window is displayed, offering assistance through the online help for using the application. You can keep the window open, or close it by clicking **OK**. If you need to access the online help at any time, use the **Help** menu, or press the **F1** key.

   ---

2. Restore the archive of the starting ANSYS Workbench project to your working directory.

   File → Restore Archive...

   The **Select Archive to Restore** dialog box appears.

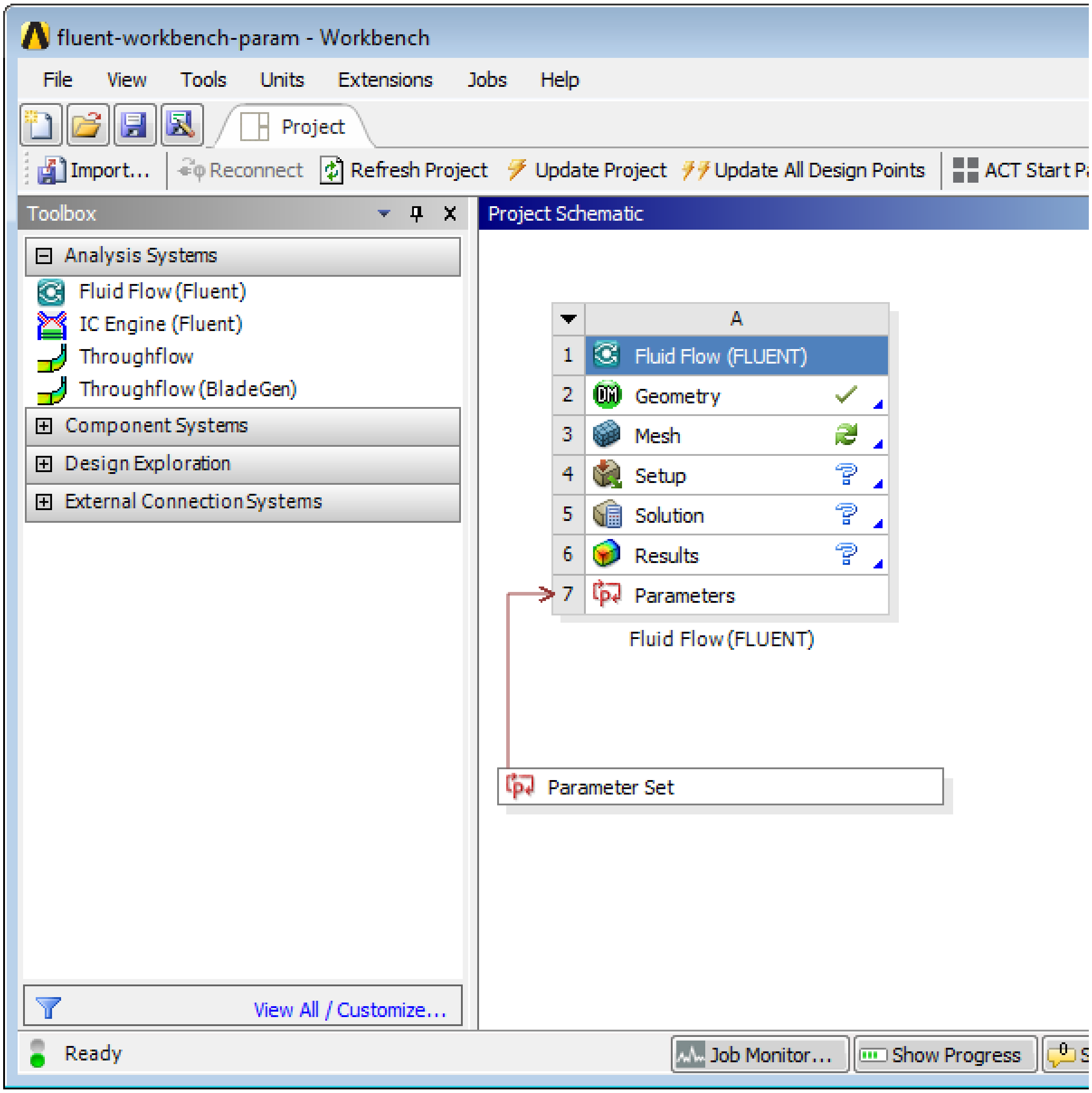
   a. Browse to your working directory, select the project archive file `fluent-workbench-param.wbpz`, and click **Open**.

      The **Save As** dialog box appears.

   b. Browse, if necessary, to your working folder and click **Save** to restore the project file, `fluent-workbench-param.wbpj`, and a corresponding project folder, `fluent-workbench-param_files`, for this tutorial.

   Now that the project archive has been restored, the project will automatically open in ANSYS Workbench.

Figure 2.4: The Project Loaded into ANSYS Workbench

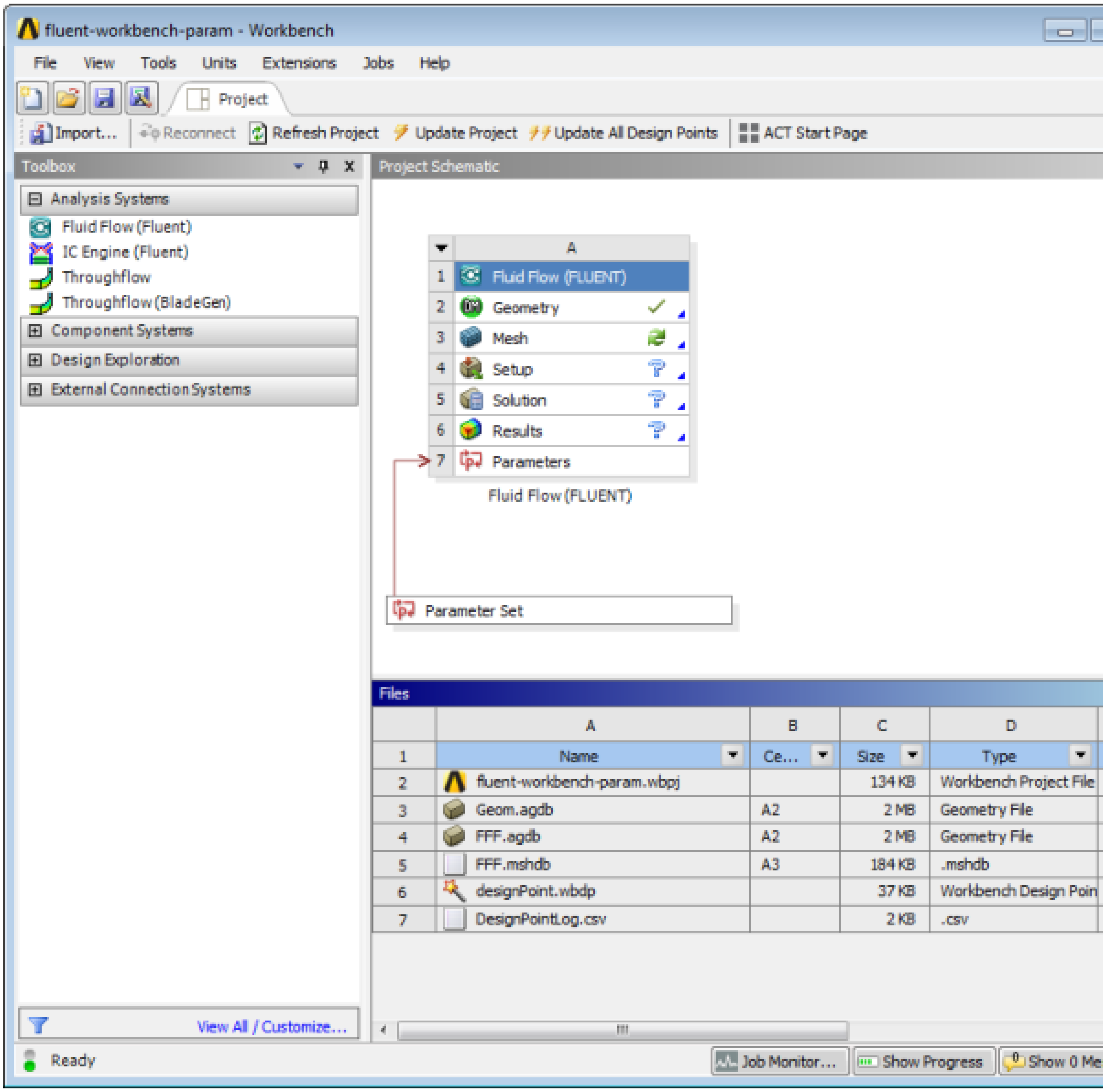


The project (`fluent-workbench-param.wbpj`) already has a Fluent-based fluid flow analysis system that includes the geometry and mesh, as well as some predefined parameters. You will first examine and edit parameters within Workbench, then later proceed to define the fluid flow model in ANSYS Fluent.

3. Open the **Files** view in ANSYS Workbench so you can view the files associated with the current project and are written during the session.

   View → **Files**

Figure 2.5: The Project Loaded into ANSYS Workbench Displaying Properties and Files View



Note the types of files that have been created for this project. Also note the states of the cells for the Fluid Flow (Fluent) analysis system. Since the geometry has already been defined, the status of the ***Geometry*** *cell is Up-to-Date ( ). Since the mesh is not complete, the* ***Mesh*** *cell's state is Refresh Required ( ), and since the ANSYS Fluent setup is incomplete and the simulation has yet to be performed, with no corresponding results, the state for the* ***Setup***, ***Solution***, *and* ***Results*** *cells is Unfulfilled ( ). For more information about cell states, see the Workbench User's Guide.*

4. Review the input parameters that have already been defined in ANSYS DesignModeler.

   a. Double-click the **Parameter Set** bus bar in the ANSYS Workbench **Project Schematic** to open the **Parameters Set** tab.

Note

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

b. In the **Outline of All Parameters** view (Figure 2.6: Parameters Defined in ANSYS DesignModeler (p. 82)), review the following existing parameters:

- The parameter `hcpos` represents the valve position that controls the flow over the heat exchanger. When the valve is at an angle of 25°, it allows the flow to pass over the heat exchanger. When the angle is 90°, it completely blocks the flow towards the heat exchanger. Any value in between allows some flow to pass over the heat exchanger giving a mixed flow condition.

- The parameter `ftpos` represents the valve position that controls flow towards the floor duct. When the valve is at an angle of 20°, it blocks the flow towards the floor duct and when the valve angle is 60°, it unblocks the flow.

- The parameter `wsfpos` represents the valve position that controls flow towards the windshield and the front panel. When the valve is at an angle of 15°, it allows the entire flow to go towards the windshield. When the angle is 90°, it completely blocks the flow towards windshield as well as the front panel. When the angle is 175°, it allows the flow to go towards the windshield and the front panel.

Figure 2.6: Parameters Defined in ANSYS DesignModeler

	A	B	C	D
1	ID	Parameter Name	Value	Unit
2	Input Parameters			
3	Fluid Flow (FLUENT) (A1)			
4	P1	hcpos	90	
5	P2	ftpos	25	
6	P3	wsfpos	175	
*	New input parameter	New name	New expression	
8	Output Parameters			
*	New output parameter		New expression	
10	Charts			

5. In the **Outline of All Parameters** view, create three new named input parameters.

   a. In the row that contains **New input parameter**, click the parameter table cell with **New name** (under the **Parameter Name** column) and enter **`input_hcpos`**. Note the ID of the parameter that appears in column **A** of the table. For the new input parameter, the parameter **ID** is **P4**. In the **Value** column, enter **`15`**.

   b. In a similar manner, create two more parameters named **`input_ftpos`** and **`input_wsfpos`**. In the **Value** column, enter **`25`**, and **`90`** for each new parameter (**P5** and **P6**), respectively.

Figure 2.7: New Parameters Defined in ANSYS Workbench

	A	B	C	D
1	ID	Parameter Name	Value	Unit
2	Input Parameters			
3	Fluid Flow (FLUENT) (A1)			
4	P1	hcpos	90	
5	P2	ftpos	25	
6	P3	wsfpos	175	
7	P4	input_hcpos	15	
8	P5	input_ftpos	25	
9	P6	input_wsfpos	90	
*	New input parameter	New name	New expression	
11	Output Parameters			
*	New output parameter		New expression	
13	Charts			

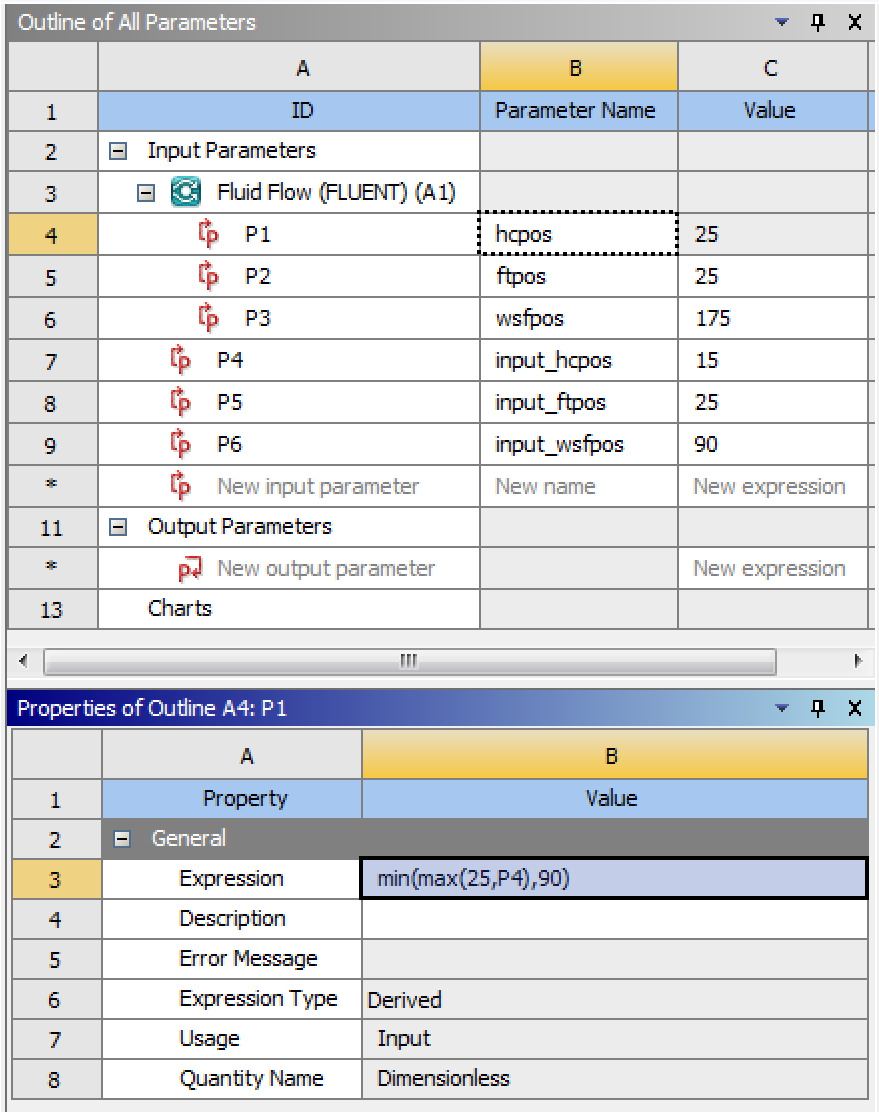
6. Select the row (or any cell in the row) that corresponds to the `hcpos` parameter. In the **Properties of Outline** view, change the value of the `hcpos` parameter in the **Expression** field from 90 to the expression `min(max(25,P4),90)`. This puts a constraint on the value of `hcpos`, so that the value always remains between 25° and 90°. The redefined parameter `hcpos` is automatically passed to ANSYS DesignModeler. Alternatively the same constraint can also be set using the expression `max(25, min(P4,90))`.

   After defining this expression, the parameter becomes a derived parameter that is dependent on the value of the parameter `input_hcpos` with ID **P4**. The derived parameters are unavailable for editing in the **Outline of All Parameters** view and could be redefined only in the **Properties of Outline** view.

Important

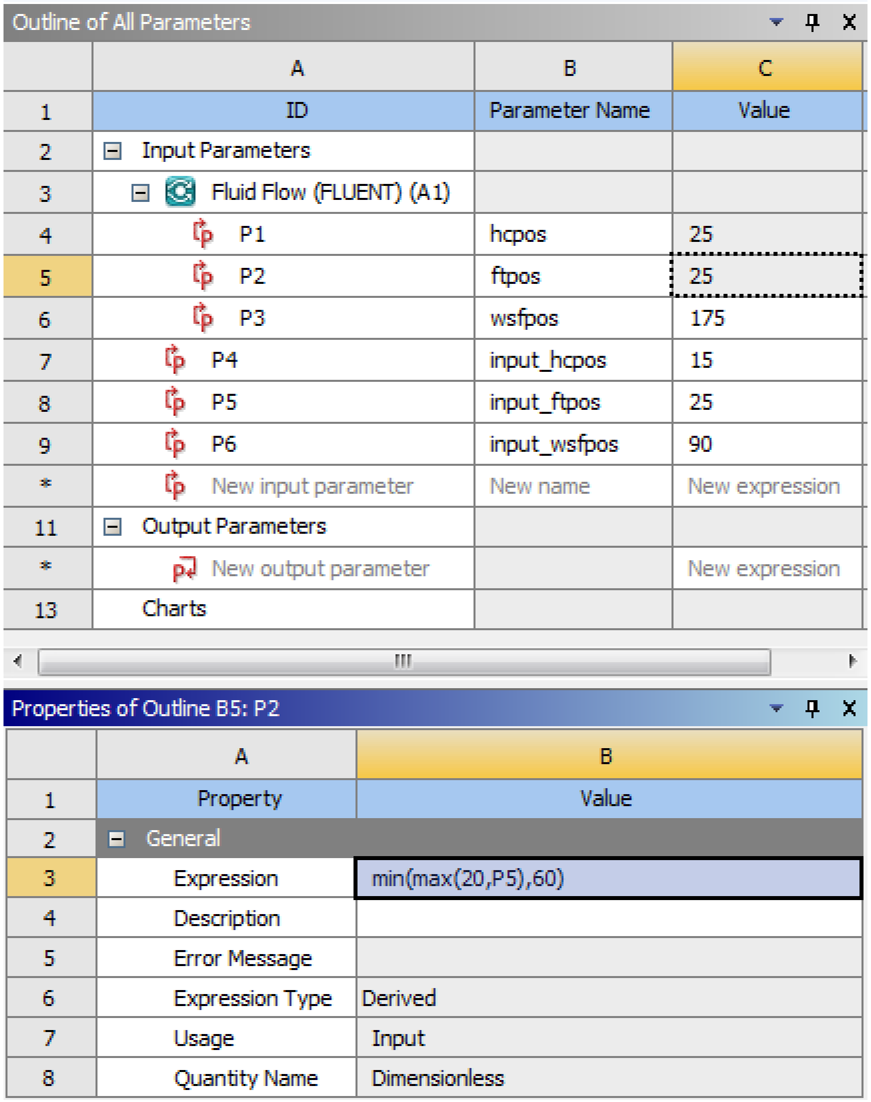
When entering expressions, you must use the list and decimal delimiters associated with your selected language in the Workbench regional and language settings, which correspond to the regional settings on your machine. The instructions in this tutorial assume that your systems uses "." as a decimal separator and "," as a list separator.

Figure 2.8: Constrained Parameterhcpos



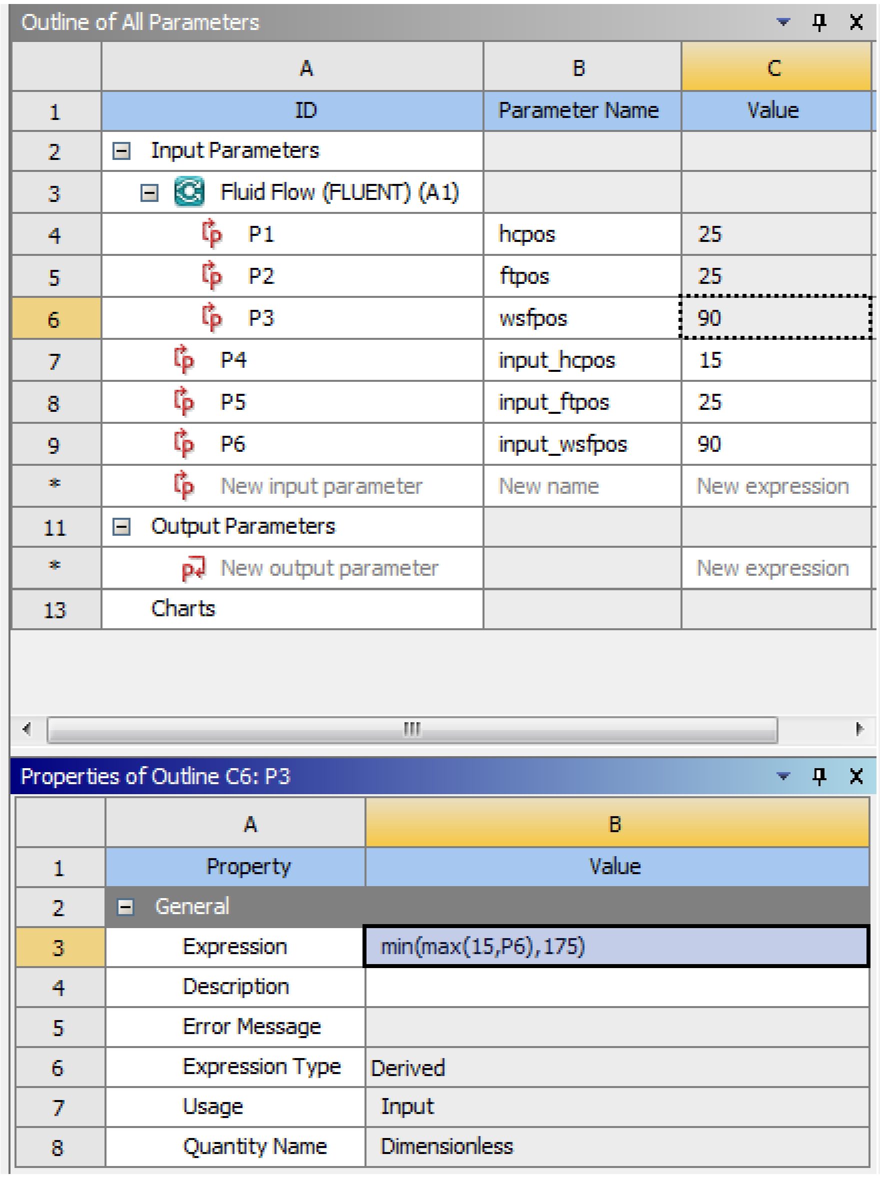
7. Select the row or any cell in the row that corresponds to the `ftpos` parameter and create a similar expression for `ftpos`: `min(max(20,P5),60)`.

Figure 2.9: Constrained Parameter`ftpos`



8. Create a similar expression for `wsfpos`:`min(max(15,P6),175)`.

Figure 2.10: Constrained Parameter`wsfpos`



9. Click the **X** on the right side of the **Parameters Set** tab to close it and return to the **Project Schematic**.

   Note the new status of the cells in the Fluid Flow (Fluent) analysis system. Since we have changed the values of `hcpos`, `ftpos`, and `wsfpos` to their new expressions, the **Geometry** and **Mesh** cells now indicates Refresh Required ( ).

10. Update the **Geometry** and **Mesh** cells.

    a. Right-click the **Geometry** cell and select the **Update** option from the context menu.

    b. Likewise, right-click the **Mesh** cell and select the **Refresh** option from the context menu. Once the cell is refreshed, then right-click the **Mesh** cell again and select the **Update** option from the context menu.

11. Save the project in ANSYS Workbench.

    In the main menu, select **File** → **Save**

2.4.3. Setting Up the CFD Simulation in ANSYS Fluent

Now that you have edited the parameters for the project, you will set up a CFD analysis using ANSYS Fluent. In this step, you will start ANSYS Fluent, and begin setting up the CFD simulation.

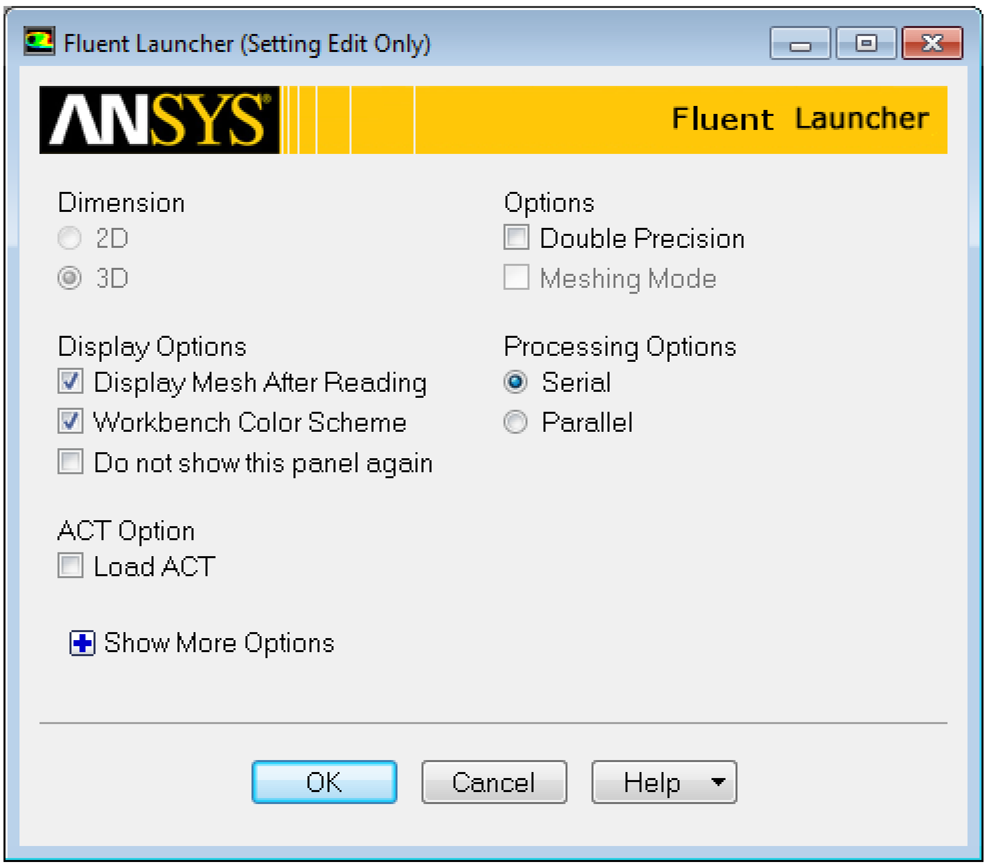
2.4.3.1. Starting ANSYS Fluent

In the ANSYS Workbench **Project Schematic**, double-click the **Setup** cell in the ANSYS Fluent fluid flow analysis system. You can also right-click the **Setup** cell to display the context menu where you can select the **Edit...** option.

When ANSYS Fluent is first started, Fluent Launcher is displayed, allowing you to view and/or set certain ANSYS Fluent start-up options.

Fluent Launcher allows you to decide which version of ANSYS Fluent you will use, based on your geometry and on your processing capabilities.

Figure 2.11: ANSYS Fluent Launcher



1. Ensure that the proper options are enabled.

Important

Note that the **Dimension** setting is already filled in and cannot be changed, since ANSYS Fluent automatically sets it based on the mesh or geometry for the current system.

a. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.

Note

An option is enabled when there is a check mark in the check box, and disabled when the check box is empty. To change an option from disabled to enabled (or vice versa), click the check box or the text.

b. Ensure that **Serial** is selected from the **Processing Options** list.

Note

Parallel processing offers a substantial reduction in computational time. Refer to Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121) in this manual and the Fluent User's Guide for further information about using the parallel processing capabilities of ANSYS Fluent.

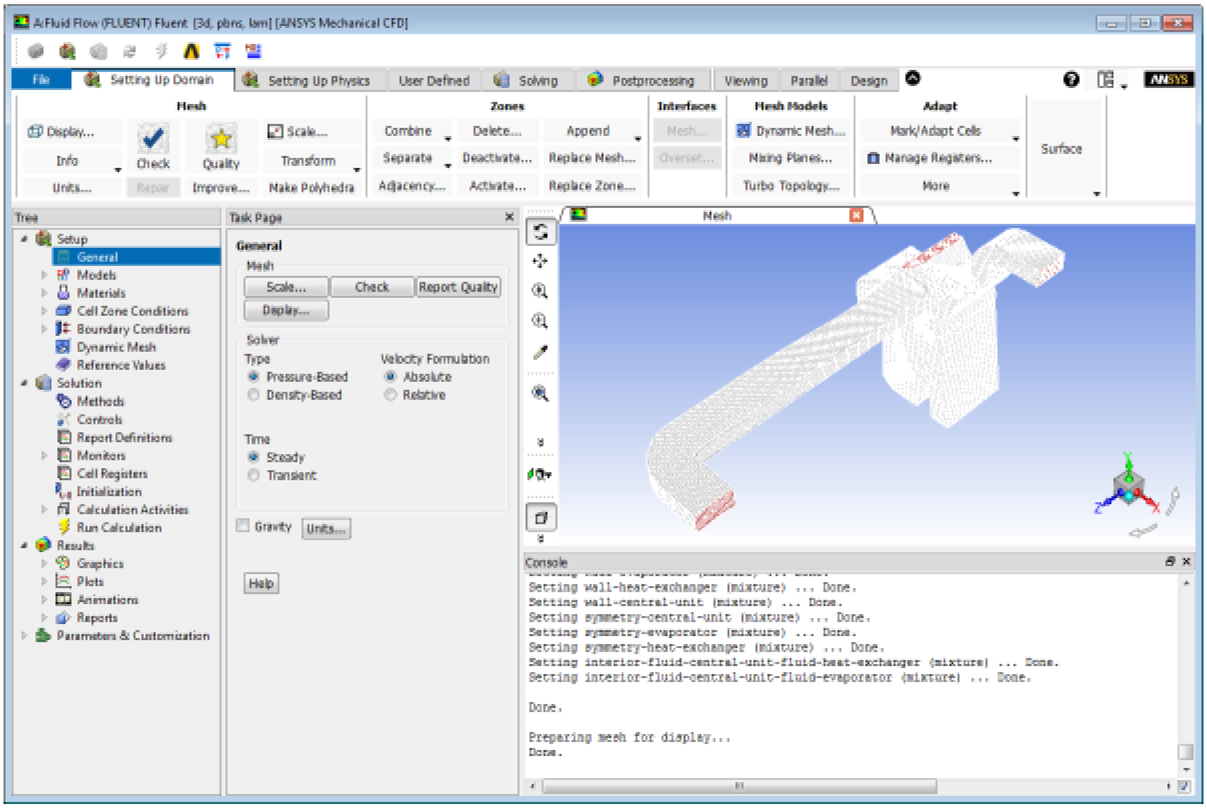
c. Ensure that the **Double Precision** option is disabled.

Note

Fluent will retain your preferences for future sessions.

2. Click **OK** to launch ANSYS Fluent.

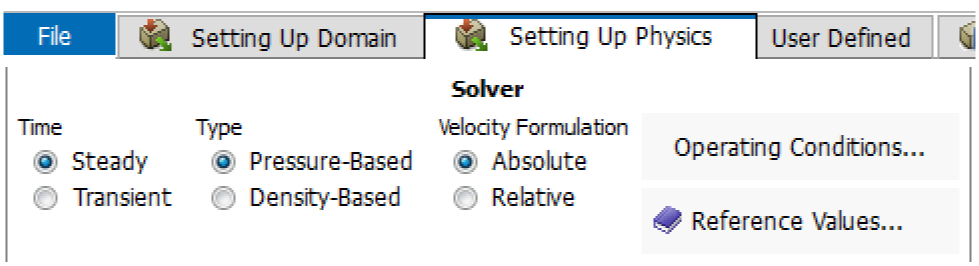
Figure 2.12: The ANSYS Fluent Application



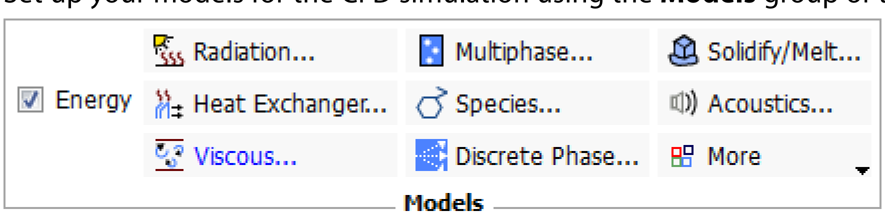
2.4.3.2. Setting Up Physics

1. In the **Solver** group of the **Setting Up Physics** ribbon tab, retain the default selection of the steady pressure-based solver.

   Setting Up Physics → Solver



2. Set up your models for the CFD simulation using the **Models** group of the **Setting Up Physics** ribbon tab.



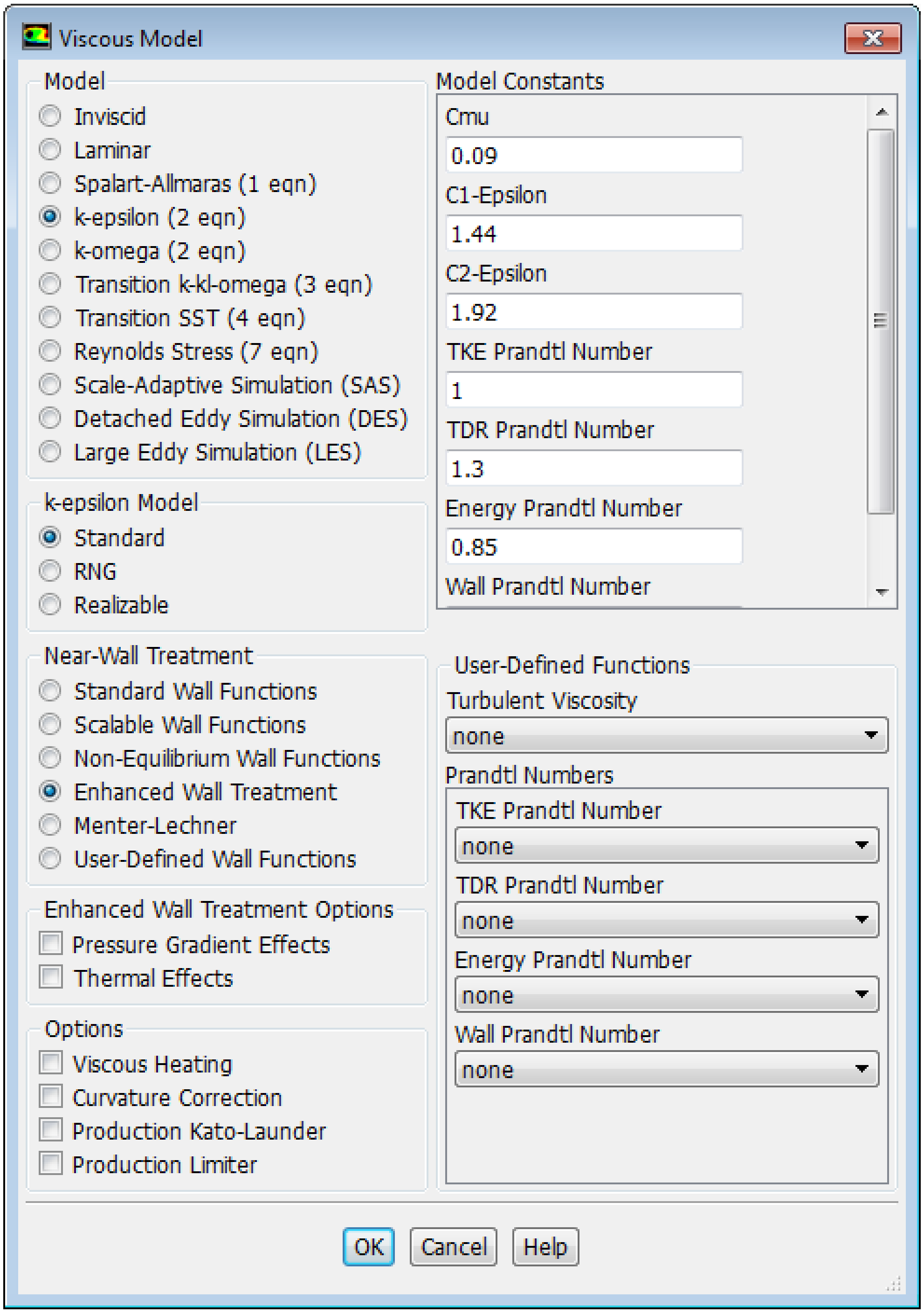
   a. Enable heat transfer by activating the energy equation.

In the **Setting Up Physics** ribbon tab, select **Energy** (**Models** group).

Setting Up Physics → Models → Energy

b. Enable the k- ε turbulence model.

Setting Up Physics → Models → Viscous...



i. Select **k-epsilon (2 eqn)** from the **Model** group box.

ii. Select **Enhanced Wall Treatment** from the **Near-Wall Treatment** group box.

The default Standard Wall Functions are generally applicable when the cell layer adjacent to the wall has a y+ larger than 30. In contrast, the Enhanced Wall Treatment option provides consistent solutions for all y+ values. Enhanced Wall Treatment is recommended when using the k-epsilon model for general single-phase fluid flow problems. For more information about Near Wall Treatments in the k-epsilon model refer to the Fluent User's Guide.

iii. Click **OK** to retain the other default settings, enable the model, and close the **Viscous Model** dialog box.

Note that the **Viscous...** label in the ribbon is displayed in blue to indicate that the Viscous model is enabled.

3. Define a heat source cell zone condition for the evaporator volume.

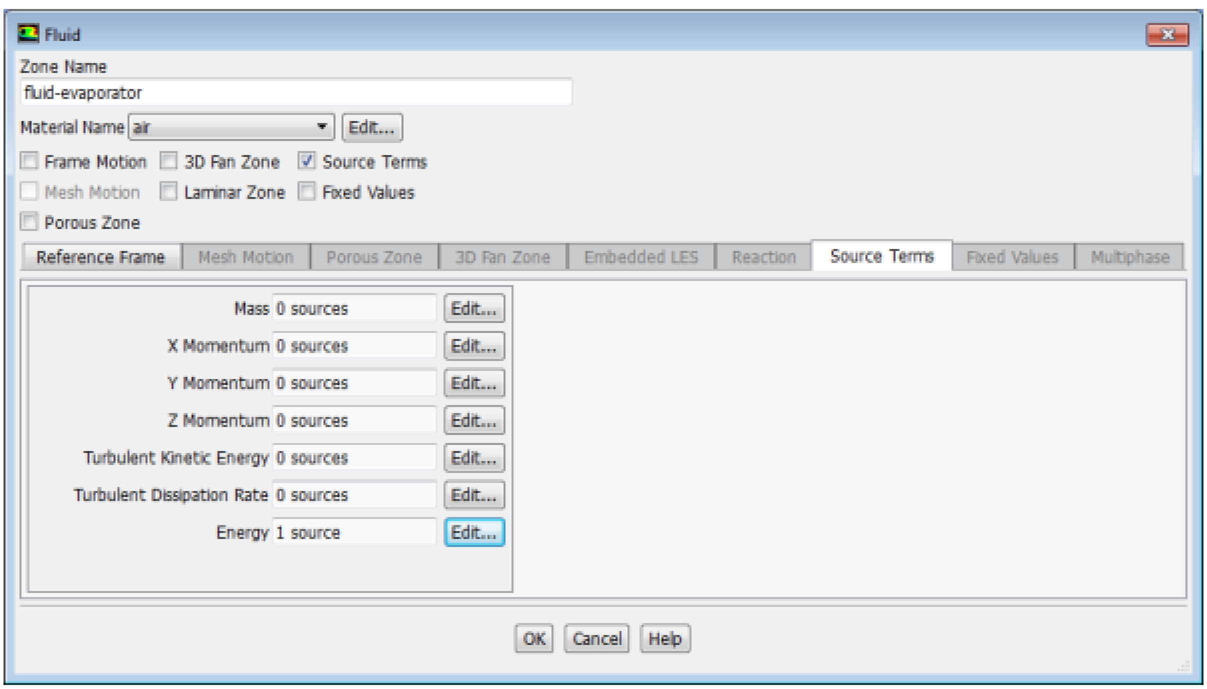
Setting Up Physics → Zones → Cell Zones

Note

All cell zones defined in your simulation are listed in the **Cell Zone Conditions** task page and under the **Setup/Cell Zone Conditions** tree branch.

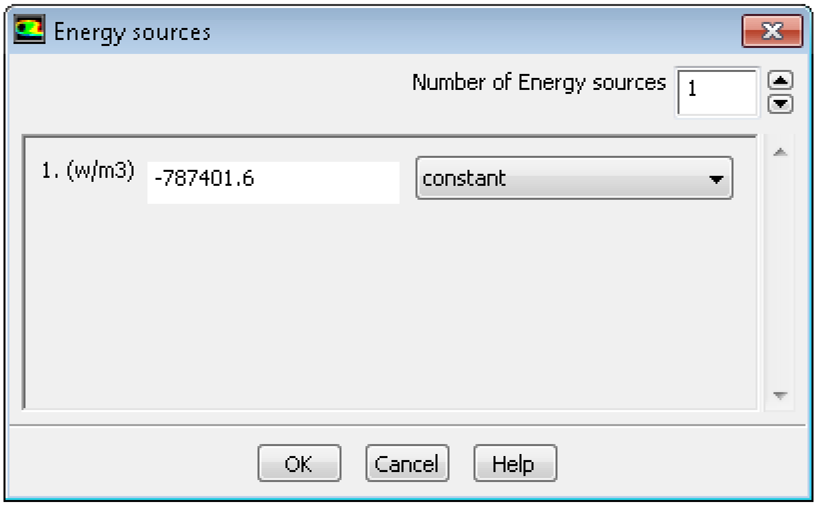
---

a. In the **Cell Zone Conditions** task page, under the **Zone** list, select **fluid-evaporator** and click **Edit...** to open the **Fluid** dialog box.



b. In the **Fluid** dialog box, enable **Source Terms**.

c. In the **Source Terms** tab, click the **Edit...** button next to **Energy**.



d. In the **Energy sources** dialog box, change the **Number of Energy sources** to 1.

e. For the new energy source, select **constant** from the drop-down list, and enter `-787401.6` W/m^3 — based on the evaporator load (200 W) divided by the evaporator volume (0.000254 m^3) that was computed earlier.

f. Click **OK** to close the **Energy Source** dialog box.

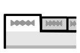
g. Click **OK** to close the **Fluid** dialog box.

2.4.4. Defining Input Parameters in ANSYS Fluent

You have now started setting up the CFD analysis using ANSYS Fluent. In this step, you will define boundary conditions and input parameters for the velocity inlet.

1. Define an input parameter called `in_velocity` for the velocity at the inlet boundary.

   a. In the **Setting Up Physics** tab, click **Boundaries** (**Zones** group).

    **Setting Up Physics → Zones → Boundaries**

   This opens the **Boundary Conditions** task page.

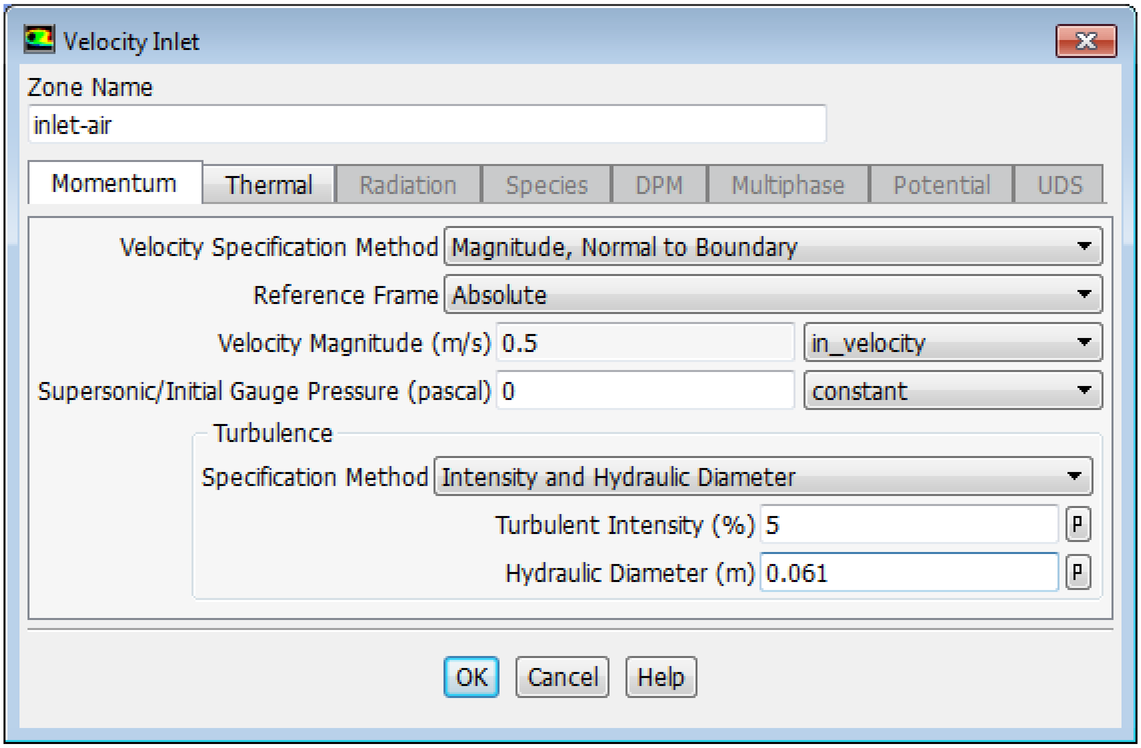
   Note

   All boundaries defined in the case are also displayed under the **Setup/Boundary Conditions** tree branch.

   ---

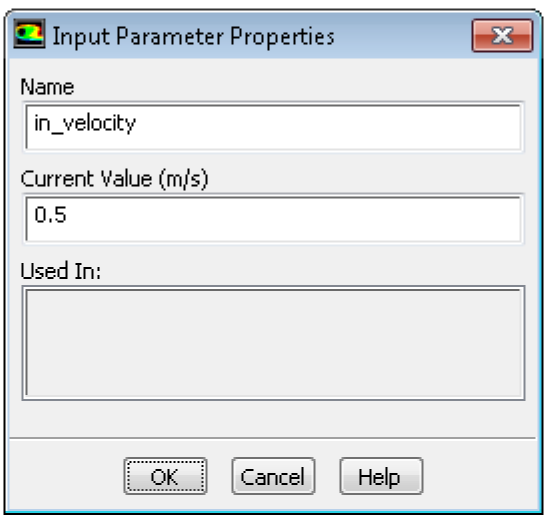
   b. In the **Boundary Conditions** task page, click the **Toggle Tree View** button (in the upper right corner), and under the **Group By** category, select **Zone Type**. This displays boundary zones grouped by zone type.

   c. Under the **Inlet** zone type, double-click **inlet-air**.



d. In the **Velocity Inlet** dialog box, from the **Velocity Magnitude** drop-down list, select **New Input Parameter...**.

*This displays the **Input Parameter Properties** dialog box.*



e. Enter `in_velocity` for the **Name**, and enter `0.5` m/s for the **Current Value**.

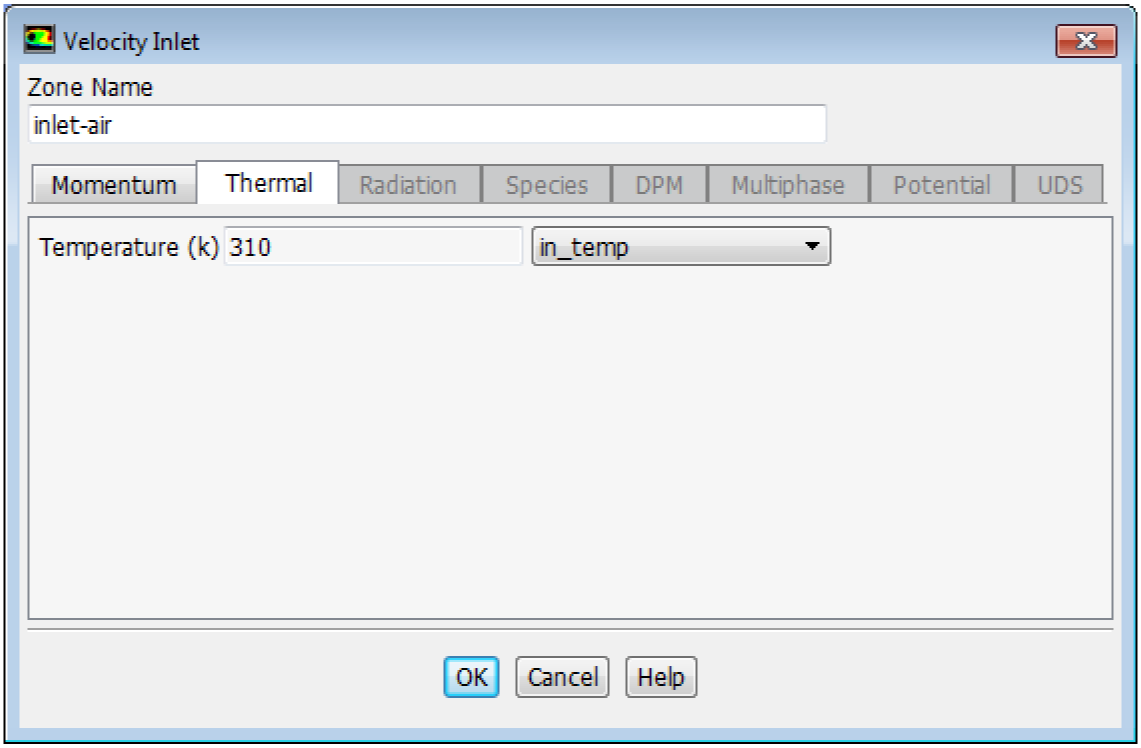
f. Click **OK** to close the **Input Parameter Properties** dialog box.

g. Under the **Turbulence** group box, from the **Specification Method** drop-down list, select **Intensity and Hydraulic Diameter**.

h. Retain the value of 5 % for **Turbulent Intensity**.

i. Enter `0.061` for **Hydraulic Diameter (m)**.

2. Define an input parameter called `in_temp` for the temperature at the inlet boundary.



a. In the **Thermal** tab of the **Velocity Inlet** dialog box, select **New Input Parameter...** from the **Temperature** drop-down list.

b. Enter `in_temp` for the **Name** and enter `310` K for the **Current Value** in the **Input Parameter Properties** dialog box.

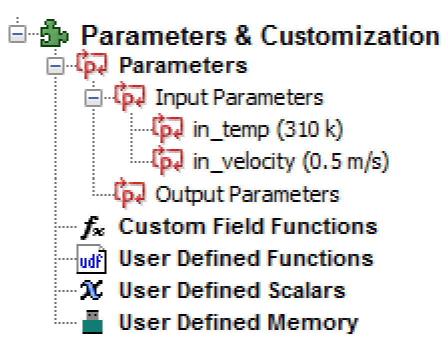
c. Click **OK** to close the **Input Parameter Properties** dialog box.

d. Click **OK** to close the **Velocity Inlet** dialog box.

3. Review all of the input parameters that you have defined in ANSYS Fluent under the **Parameters & Customization/Parameters** tree branch.

Parameters & Customization → **Parameters** → **Input Parameters**

Figure 2.13: The Input Parameters Sub-Branch in ANSYS Fluent



These parameters are passed to ANSYS Fluent component system in ANSYS Workbench and are available for editing in ANSYS Workbench (see Figure 2.14: The Parameters View in ANSYS Workbench (p. 95)).

Figure 2.14: The Parameters View in ANSYS Workbench

Outline of All Parameters

	A	B	C	D
1	ID	Parameter Name	Value	Unit
2	Input Parameters			
3	Fluid Flow (FLUENT) (A1)			
4	P1	hcpos	25	
5	P2	ftpos	25	
6	P3	wsfpos	90	
7	P7	in_velocity	0.5	m s^-1
8	P9	in_temp	310	K
9	P4	input_hcpos	15	
10	P5	input_ftpos	25	
11	P6	input_wsfpos	90	
*	New input parameter	New name	New expression	
13	Output Parameters			
*	New output parameter		New expression	
15	Charts			

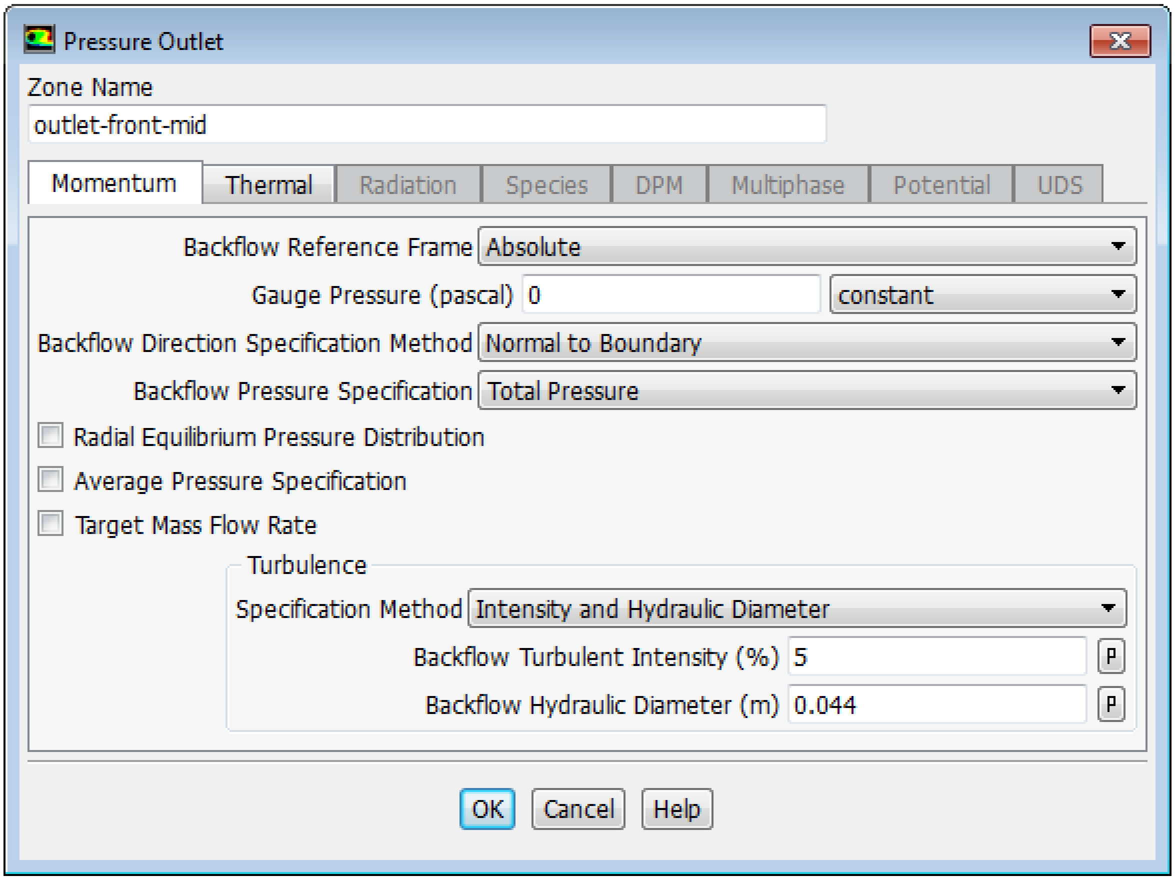
4. Set the turbulence parameters for backflow at the front outlets and foot outlets.

   a. In the **Boundary Conditions** task page, type **`outlet`** in the **Zone** filter text entry field. Note that as you type, the names of the boundary zones beginning with the characters you entered appear in the boundary condition zone list.

   Note

   - The search string can include wildcards. For example, entering **`*let*`** will display all zone names containing `let`, such as `inlet` and `outlet`.
   - To display all zones again, click the red **X** icon in the **Zone** filter.

   b. Double-click **outlet-front-mid**.



c. In the **Pressure Outlet** dialog box, under the **Turbulence** group box, select **Intensity and Hydraulic Diameter** from the **Specification Method** drop-down list.

d. Retain the value of 5 for **Backflow Turbulent Intensity (%)**.

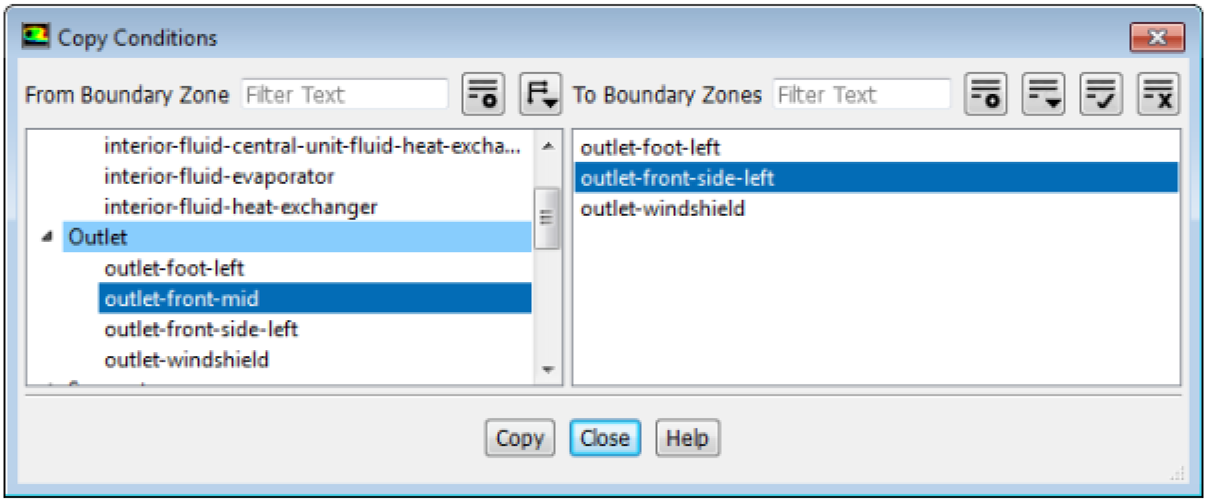
e. Enter `0.044` for **Backflow Hydraulic Diameter (m)**.

   These values will only be used if reversed flow occurs at the outlets. It is a good idea to set reasonable values to prevent adverse convergence behavior if backflow occurs during the calculation.

f. Click **OK** to close the **Pressure Outlet** dialog box.

g. Copy the boundary conditions from **outlet-front-mid** to the other front outlet.

   Setup → **Boundary Conditions** → **outlet-front-mid** **Copy...**



i. Confirm that **outlet-front-mid** is selected in the **From Boundary Zone** selection list.

ii. Select **outlet-front-side-left** in the **To Boundary Zones** selection list.

iii. Click **Copy** to copy the boundary conditions.

Fluent will display a dialog box asking you to confirm that you want to copy the boundary conditions.

iv. Click **OK** to confirm.

v. Close the **Copy Conditions** dialog box.

h. In a similar manner, set the backflow turbulence conditions for **outlet-foot-left** using the values in the following table:

Parameter	Value
Specification Method	**Intensity and Hydraulic Diameter**
Backflow Turbulent Intensity (%)	5
Backflow Hydraulic Diameter (m)	`0.052`

2.4.5. Solving

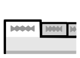
In the steps that follow, you will set up and run the calculation using the **Solving** ribbon tab.

Note

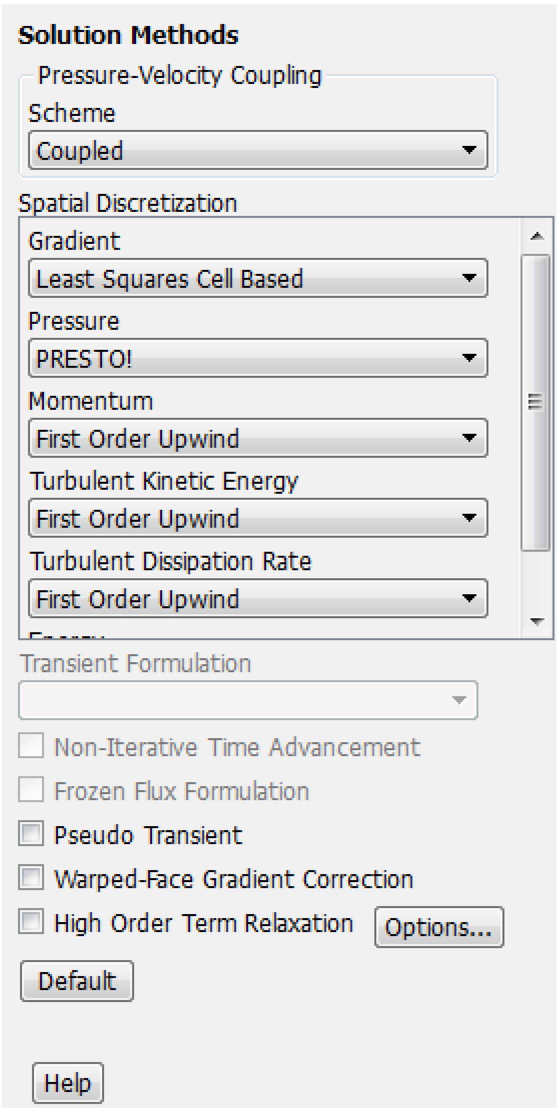
You can also use the task pages listed under the **Solution** branch in the tree to perform solution-related activities.

---

1. Set the Solution Methods.

 **Solving** → **Solution** → **Methods...**

This will open the **Solution Methods** task page.



a. From the **Scheme** drop-down list, select **Coupled**.

   The pressure-based coupled solver is the recommended choice for general fluid flow simulations.

b. In the **Spatial Discretization** group box, configure the following settings:

Setting	Value
Pressure	PRESTO!
Momentum	First Order Upwind
Energy	First Order Upwind

This tutorial is primarily intended to demonstrate the use of parameterization and design points when running Fluent from Workbench. Therefore, you will run a simplified analysis using first order discretization, which will yield faster convergence. These settings were chosen to speed up solution time for this tutorial. Usually, for cases like this, we would recommend higher order discretization settings to be set for all flow equations to ensure improved results accuracy.

2. Initialize the flow field using the **Initialization** group of the **Solving** ribbon tab.

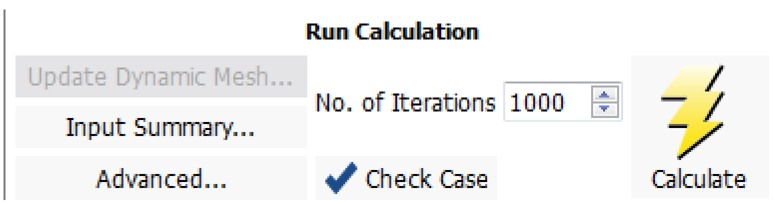
   Solving → Initialization

   a. Retain the default selection of **Hybrid Initialization**.

   b. Click the **Initialize** button.

3. Run the simulation in ANSYS Fluent from the **Run Calculation** group of the **Solving** tab.

   Solving → Run Calculation



   a. For **Number of Iterations**, enter `1000`.

   b. Click the **Calculate** button.

   The solution converges within approximately 55 iterations.

   Throughout the calculation, Fluent displays a warning in the console regarding reversed flow at the outlets. This behavior is expected in this case since air is redirected to the outlets, creating small regions of recirculation.

   Note

   The warning message can be switched off by setting the `solve/set/flow-warnings` text user interface (TUI) command to `no` in the console.

   ---

4. Close Fluent.

   File → Close Fluent

5. Save the project in ANSYS Workbench.

File → Save

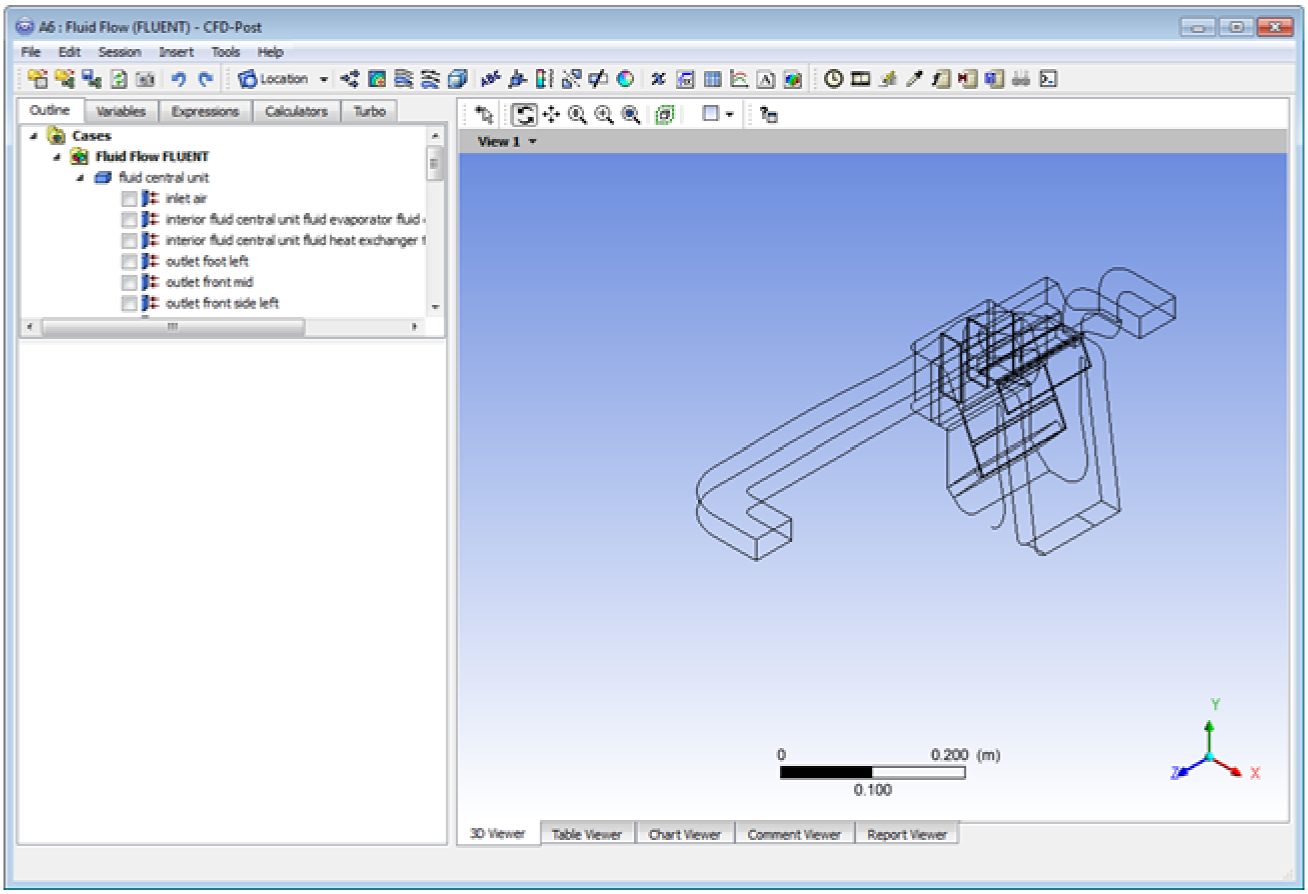
2.4.6. Postprocessing and Setting the Output Parameters in ANSYS CFD-Post

In this step, you will visualize the results of your CFD simulation using ANSYS CFD-Post. You will plot vectors that are colored by pressure, velocity, and temperature, on a plane within the geometry. In addition, you will create output parameters within ANSYS CFD-Post for later use in ANSYS Workbench.

In the ANSYS Workbench **Project Schematic**, double-click the **Results** cell in the ANSYS Fluent fluid flow analysis system to start CFD-Post. You can also right-click the **Results** cell to display the context menu where you can select the **Edit...** option.

The CFD-Post application appears with the automotive HVAC geometry already loaded and displayed in outline mode. Note that ANSYS Fluent results (that is, the case and data files) are also automatically loaded into CFD-Post.

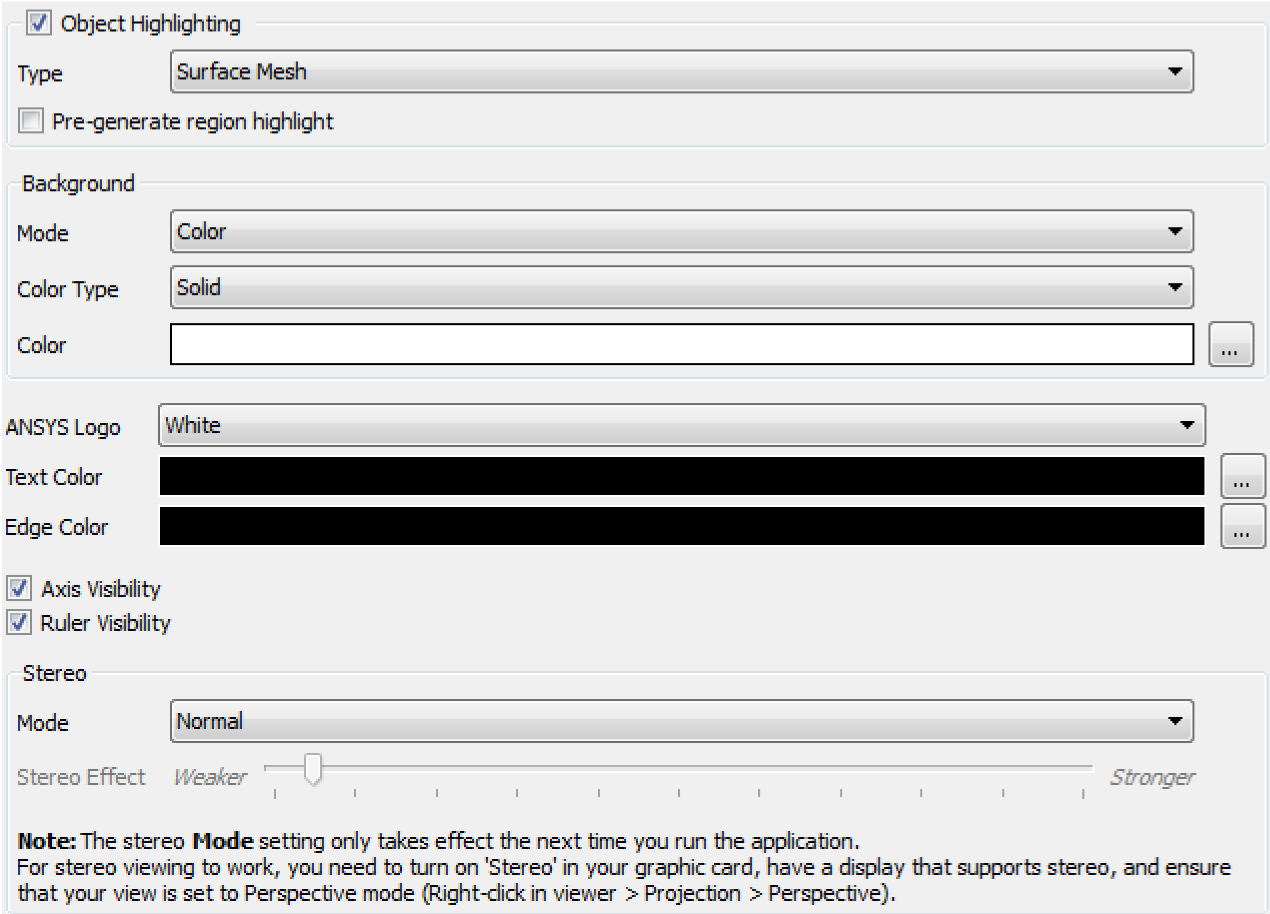
Figure 2.15: The Automotive HVAC Geometry Loaded into CFD-Post



1. Edit some basic settings in CFD-Post (for example, changing the background color to white).

   Edit → Options...

   a. In the **Options** dialog box, select **Viewer** under **CFD-Post** in the tree view.



b. Under the Background group, from the **Color Type** drop-down list, select **Solid**.

c. Click the **Color** sample bar to cycle through common color swatches until it displays white.

Tip

You can also click the ellipsis icon  to bring up a color selector dialog box from which you can choose an arbitrary color.

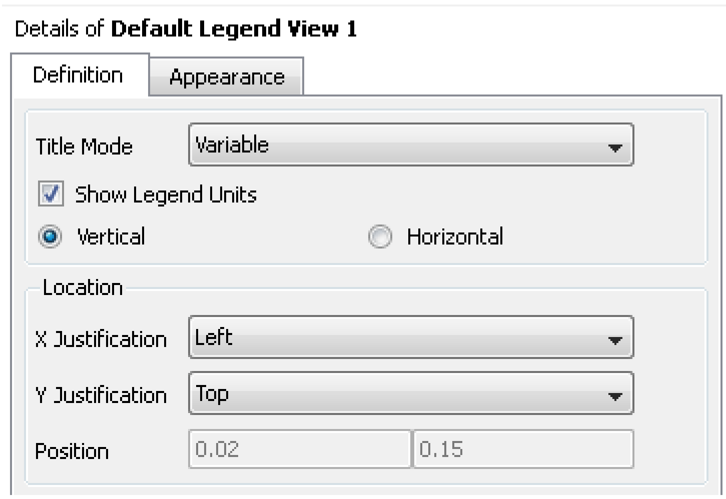
---

d. Click **OK** to set the white background color for the display and close the **Options** dialog box.

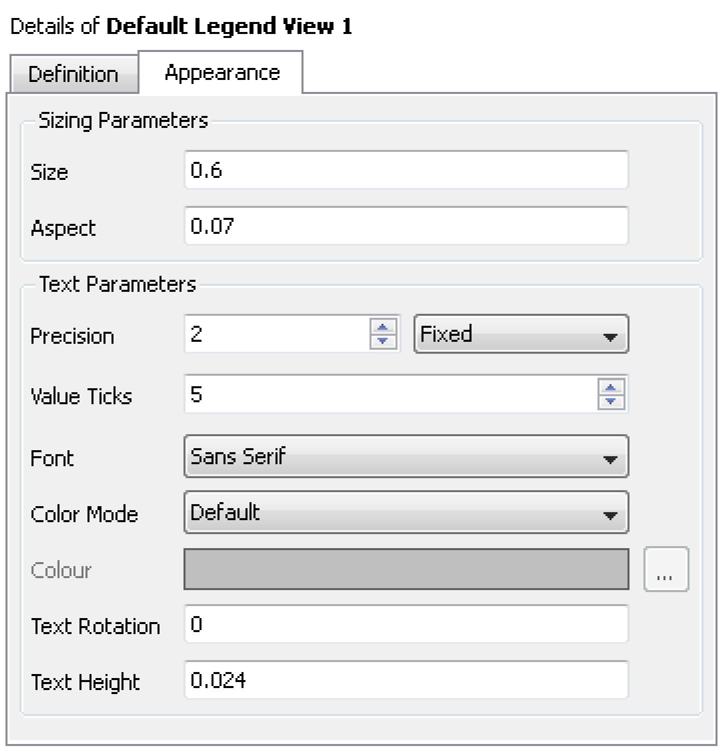
2. Adjust the color-map legend to show the numbers in floating format.

   a. In the **Outline** tree view, double-click **Default Legend View 1** to display the **Details** view for the default legend to be used for your plots.

   b. In the **Definition** tab of the **Details** view, from the **Title Mode** drop-down list, select **Variable**.



c. In the **Appearance** tab, set the **Precision** to **2** and **Fixed**.



d. Click **Apply** to set the display.

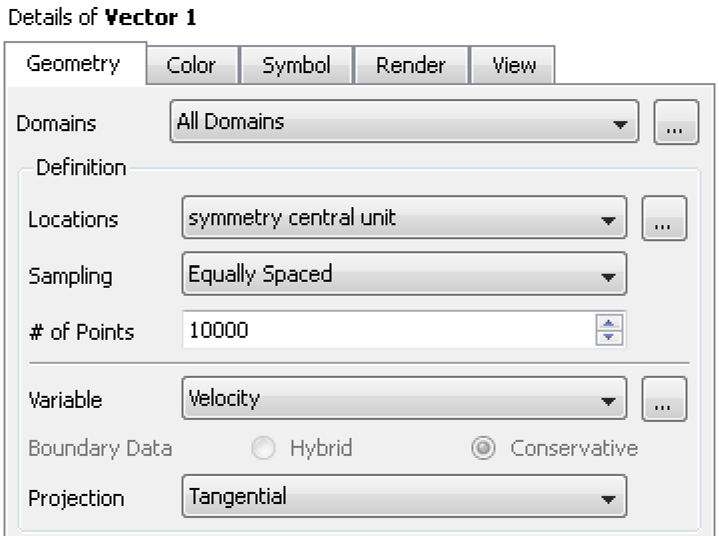
3. Plot vectors colored by pressure.

   a. From the main menu, select **Insert** → **Vector** or click  in the ANSYS CFD-Post toolbar.

   *This displays the **Insert Vector** dialog box.*

   b. Keep the default name of **Vector 1** by clicking **OK**.

   c. In the **Details** view for **Vector 1**, under the **Geometry** tab, configure the following settings.



i. Ensure **All Domains** is selected from the **Domains** drop-down list.

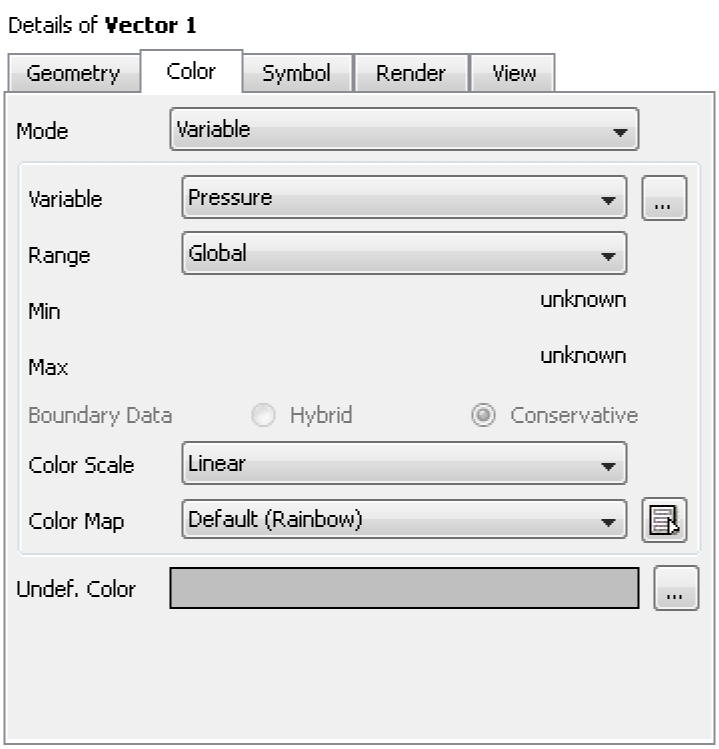
ii. From the **Locations** drop-down list, select **symmetry central unit**.

iii. From the **Sampling** drop-down list, select **Equally Spaced**.

iv. Set the **# of Points** to `10000`.

v. From the **Projection** drop-down list, select **Tangential**.

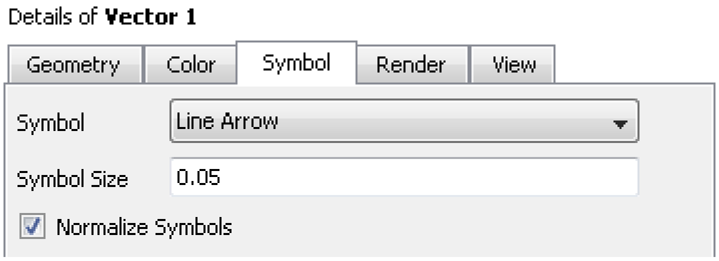
d. In the **Color** tab, configure the following settings.



i. From the **Mode** drop-down list, select **Variable**.

ii. From the **Variable** drop-down list, select **Pressure**.

e. In the **Symbol** tab, configure the following settings.



i. Set the **Symbol Size** to `0.05`.

ii. Enable **Normalize Symbols**.

f. Click **Apply**.

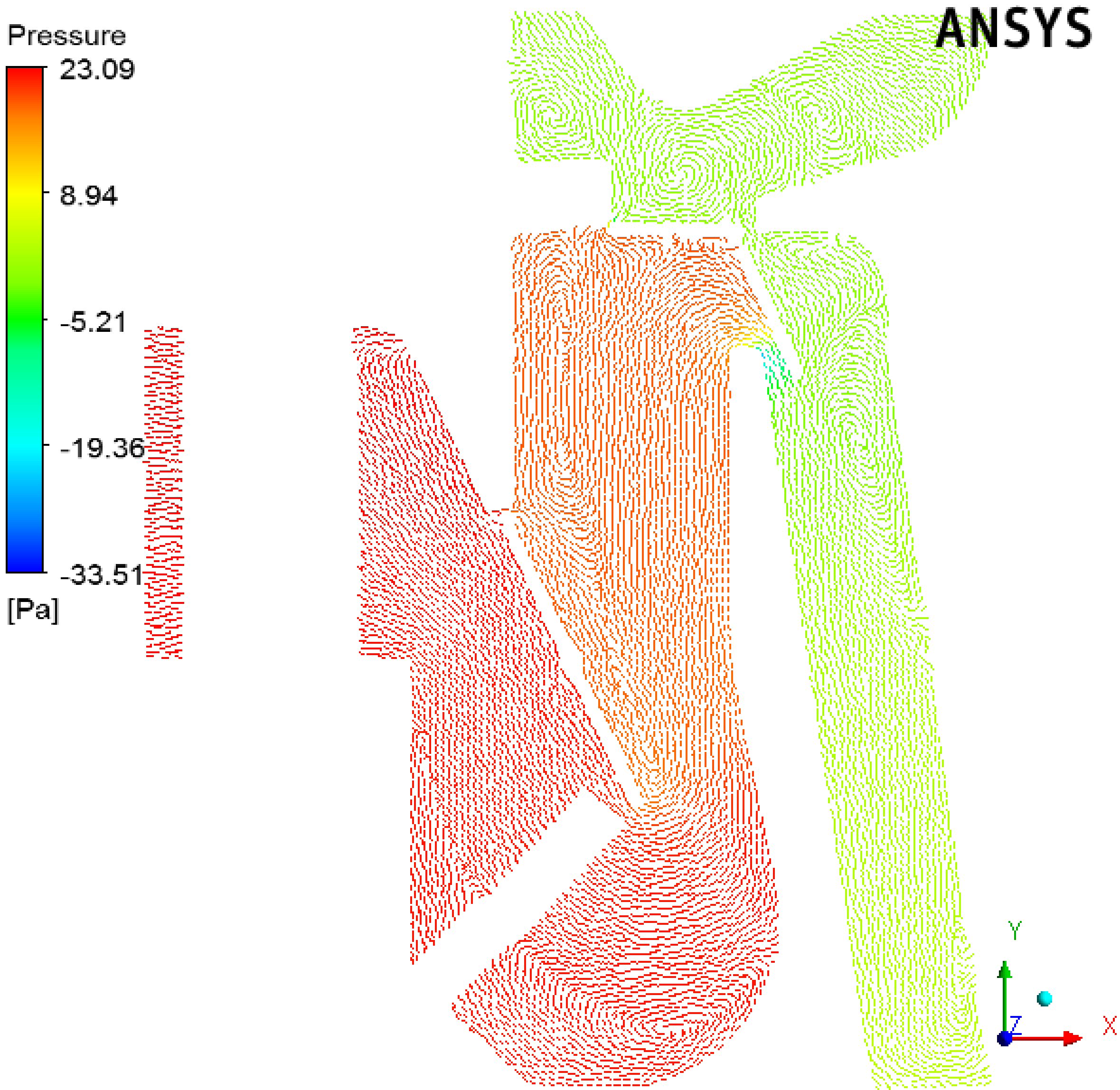
Vector 1 appears under **User Locations and Plots** in the **Outline** tree view.

*In the graphics display window, note that **symmetry-central-unit** shows the vectors colored by pressure. Use the controls in CFD-Post to rotate the geometry (for example, clicking the dark blue axis in the axis triad of the graphics window). Zoom into the view as shown in* *Figure 2.16: Vectors Colored by Pressure (p. 105)**.*

Note

To better visualize the vector display, you can deselect the **Wireframe** view option under **User Locations and Plots** in the **Outline** tree view.

Figure 2.16: Vectors Colored by Pressure



4. Plot vectors colored by velocity.

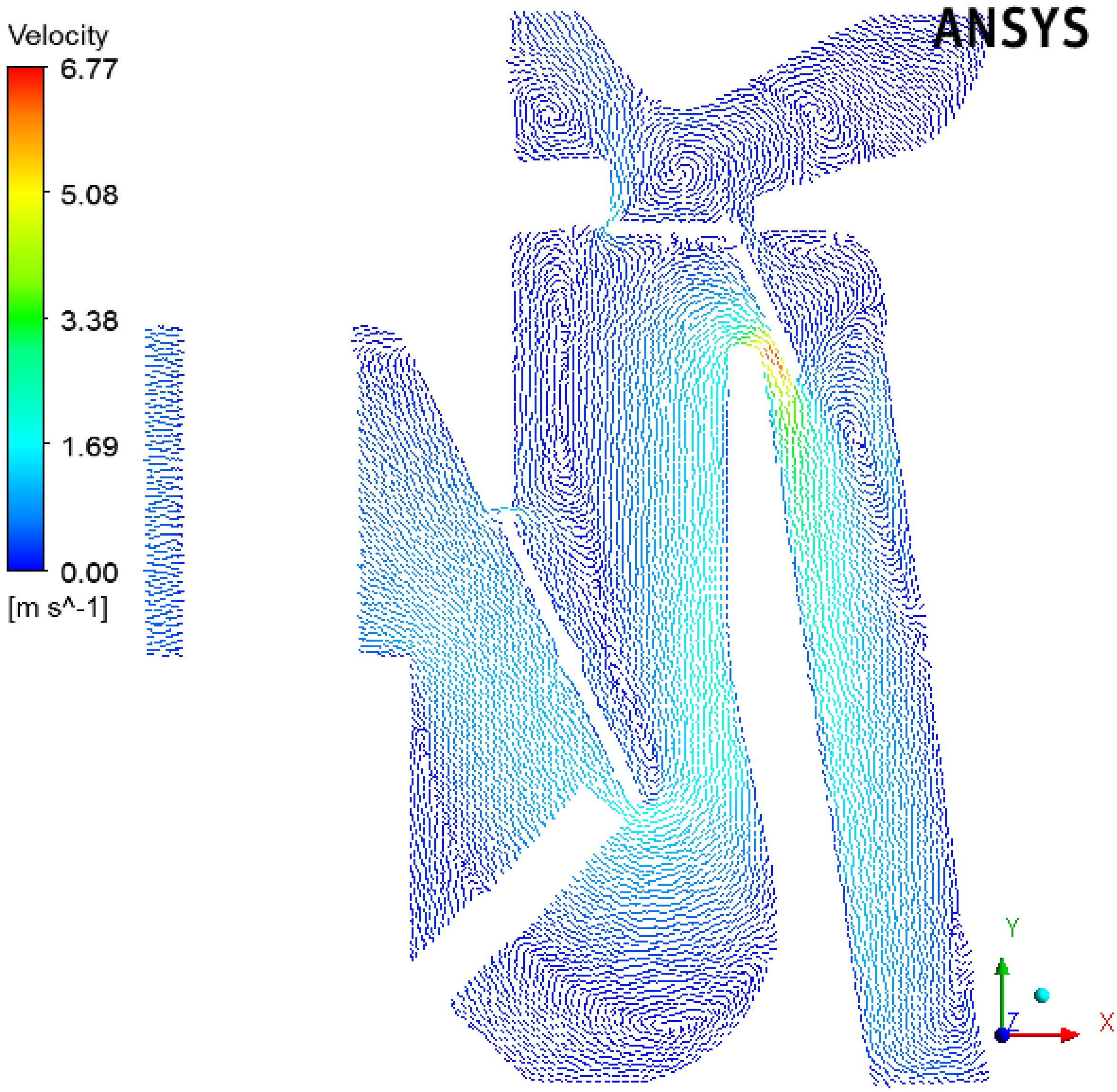
   a. In the **Details** view for **Vector 1**, under the **Color** tab, configure the following settings.

      i. Select **Velocity** from the **Variable** drop-down list.

      ii. Click **Apply**.

      The velocity vector plot appears on the **symmetry-central-unit** symmetry plane.

Figure 2.17: Vectors Colored by Velocity



5. Plot vectors colored by temperature.

    a. In the **Details** view for **Vector 1**, under the **Color** tab, configure the following settings.

        i. Select **Temperature** from the **Variable** drop-down list.

        ii. Select **User Specified** from the **Range** drop-down list.

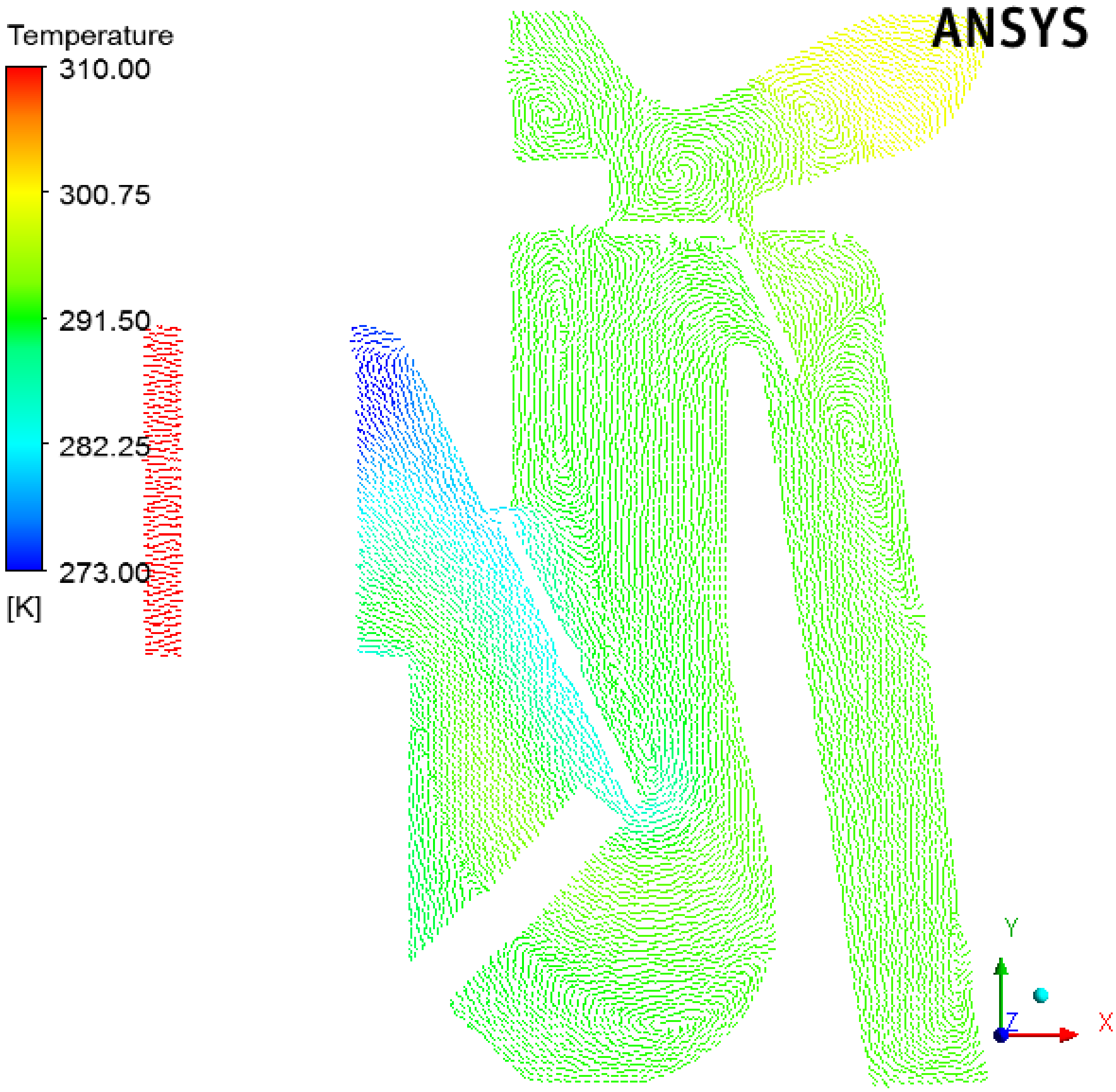
        iii. Enter `273` K for the **Min** temperature value.

        iv. Enter `310` K for the **Max** temperature value.

        v. Click **Apply**.

        The user-specified range is selected much narrower than the Global and Local ranges in order to better show the variation.

Figure 2.18: Vectors Colored by Temperature



Note the orientation of the various valves and how they impact the flow field. Later in this tutorial, you will change these valve angles to see how the flow field changes.

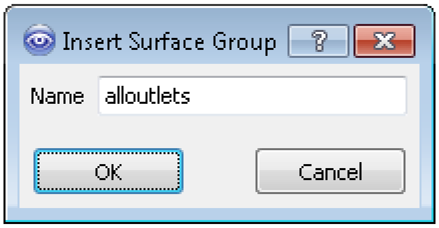
6. Create two surface groups.

   Surface groups are collections of surface locations in CFD-Post. In this tutorial, two surface groups are created in CFD-Post that will represent all of the outlets and all of the front outlets. Once created, specific commands (or expressions) will be applied to these groups in order to calculate a particular numerical value at that surface.

   a. Create a surface group consisting of all outlets.

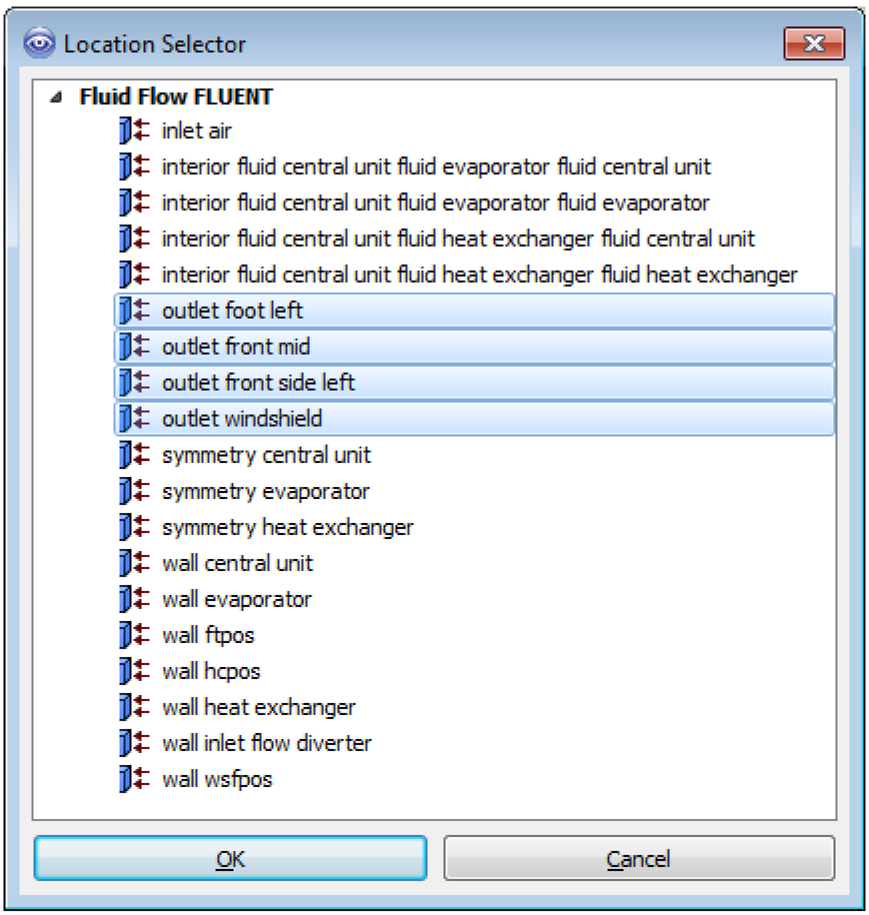
      i. With the **Outline** tree view open in the CFD-Post tree view, open the **Insert Surface Group** dialog box.

         Insert → **Location** → **Surface Group**



ii. Enter `alloutlets` for the **Name** of the surface group, and click **OK** to close the **Insert Surface Group** dialog box.

iii. In the **Details** view for the `alloutlets` surface group, in the **Geometry** tab, click the ellipsis icon  next to **Locations** to display the **Location Selector** dialog box.



iv. Select all of the outlet surfaces (`outlet foot left`, `outlet front mid`, `outlet front side left`, and `outlet windshield`) in the **Location Selector** dialog box (hold **Ctrl** for multiple selection) and click **OK**.

v. Click **Apply** in the **Details** view for the new surface group.

`alloutlets` appears under **User Locations and Plots** in the **Outline** tree view.

b. Create a surface group for the front outlets.

Perform the same steps as described above to create a surface group called `frontoutlets` with locations for the front outlets (`outlet front mid` and `outlet front side left`).

7. Create expressions in CFD-Post and mark them as ANSYS Workbench output parameters.

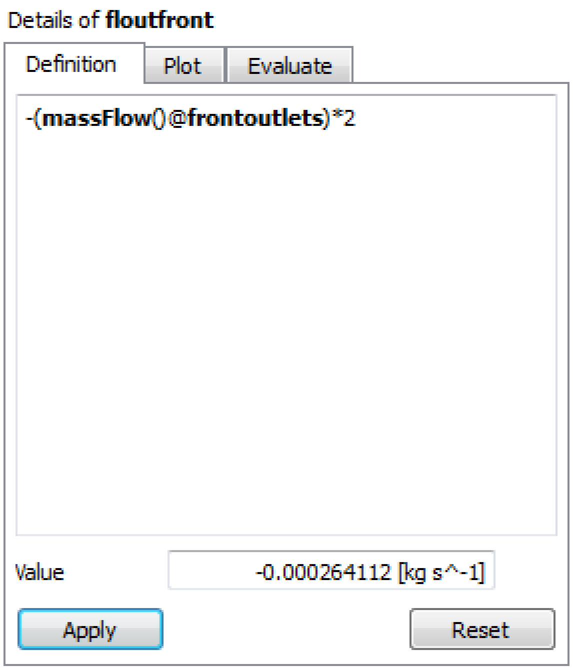
   In this tutorial, programmatic commands or expressions are written to obtain numerical values for the mass flow rate from all outlets, as well as at the front outlets, windshield, and foot outlets. The surface groups you defined earlier are used to write the expressions.

   a. Create an expression for the mass flow from all outlets.

      i. With the **Expressions** tab open in the CFD-Post tree view, open the **Insert Expression** dialog box.

         Insert → **Expression**

      ii. Enter `floutfront` for the **Name** of the expression and click **OK** to close the **Insert Expression** dialog box.



      iii. In the **Details** view for the new expression, enter the following in the **Definition** tab.

```
-(massFlow()@frontoutlets)*2
```

         *The sign convention for **massFlow()** is such that a positive value represents flow into the domain and a negative value represents flow out of the domain. Since you are defining an expression for outflow from the ducts, you use the negative of the **massFlow()** result in the definition of the expression.*

      iv. Click **Apply** to obtain a **Value** for the expression.

         *Note the new addition in the list of expressions in the **Expressions** tab in CFD-Post.*

         In this case, there is a small net backflow into the front ducts.

v. Right-click the new expression and select **Use as Workbench Output Parameter** from the context menu. A small "P" with a right-pointing arrow appears on the expression's icon.

b. Create an expression for the mass flow from the wind shield.
   i. Perform the same steps as described above to create an expression called `floutwindshield` with the following definition:

```
-(massFlow()@outlet windshield)*2
```

   ii. Right-click the new expression and select **Use as Workbench Output Parameter** from the context menu.

c. Create an expression for the mass flow from the foot outlets.
   i. Perform the same steps as described above to create an expression called `floutfoot` with the following definition:

```
-(massFlow()@outlet foot left)*2
```

   ii. Right-click the new expression and select **Use as Workbench Output Parameter** from the context menu.

d. Create an expression for the mass weighted average outlet temperature.
   i. Perform the same steps as described above to create an expression called `outlettemp` with the following definition:

```
massFlowAveAbs(Temperature)@alloutlets
```

   ii. Right-click the new expression and select **Use as Workbench Output Parameter** from the context menu.

8. Close ANSYS CFD-Post.

   In the main menu, select **File** → **Close CFD-POST** to return to ANSYS Workbench.

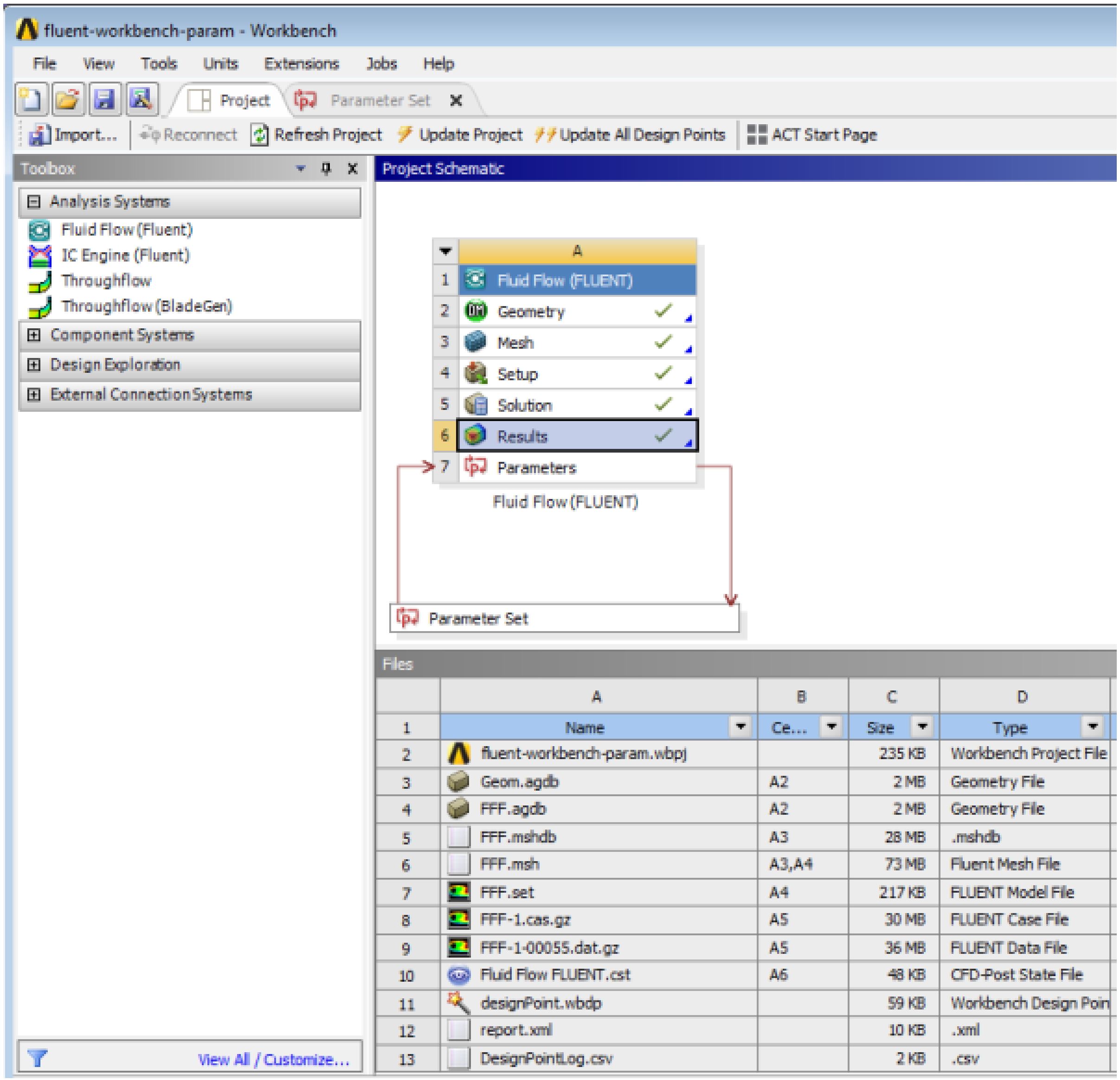
9. In the **Outline of All Parameters** view of the **Parameter Set** tab (double-click **Parameter Set**), review the newly-added output parameters that you specified in ANSYS CFD-Post and when finished, click the **Project** tab to return to the **Project Schematic**.

10. If any of the cells in the analysis system require attention, update the project by clicking the **Update Project** button in the ANSYS Workbench toolbar.

11. Optionally, review the list of files generated by ANSYS Workbench. If the **Files** view is not open, select **View** → **Files** from the main menu.

    You will notice additional files associated with the latest solution as well as those generated by CFD-Post.

Figure 2.19: The Updated Project Loaded into ANSYS Workbench Displaying the Files View



12. Save the project in ANSYS Workbench.

    In the main menu, select **File** → **Save**

Note

You can also select the **Save Project** option from the CFD-Post **File** ribbon tab.

2.4.7. Creating Additional Design Points in ANSYS Workbench

Parameters and design points are tools that allow you to analyze and explore a project by giving you the ability to run optimization and what-if scenarios. Design points are based on sets of parameter values. When you define input and output parameters in your ANSYS Workbench project, you are essentially working with a design point. To perform optimization and what-if scenarios, you create multiple design points based on your original project. In this step, you will create additional design points for your project where you will be able to perform a comparison of your results by manipulating input parameters (such as the angles of the various valves within the automotive HVAC geometry). ANSYS Workbench provides a Table of Design Points to make creating and manipulating design points more convenient.

1. Open the Table of Design Points.

   a. In the Project Schematic, double-click the **Parameter Set** bus bar to open the Table of Design Points view. If the table is not visible, select **Table** from the **View** menu in ANSYS Workbench.

      View → Table

      The table of design points initially contains the current project as a design point (DP0), along with its corresponding input and output parameter values.

      Figure 2.20: Table of Design Points (with DP0)

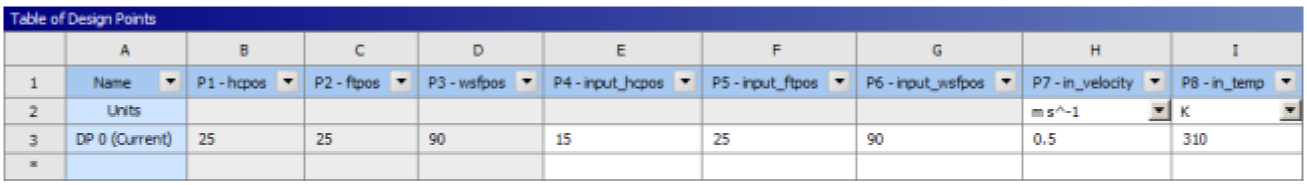


Table of Design Points

	A	B	C	D	E	F	G	H	I
1	Name	P1 - hcpos	P2 - ftpos	P3 - wsfpos	P4 - input_hcpos	P5 - input_ftpos	P6 - input_wsfpos	P7 - in_velocity	P8 - in_temp
2	Units							m s^-1	K
3	DP 0 (Current)	25	25	90	15	25	90	0.5	310
*									

      From this table, you can create new design points (or duplicate existing design points) and edit them (by varying one or more input parameters) to create separate analyses for future comparison of data.

2. Create a design point (DP1) by duplicating the current design point (DP0).

   a. Right-click the **Current** design point and select **Duplicate Design Point** from the context menu.

      The cells autofill with the values from the **Current** row.

   b. Scroll over to the far right to expose the **Retain** column in the table of design points, and ensure the check box in the row for the duplicated design point **DP 1** (cell N4) is selected.

      This allows the data from this new design point to be saved before it is exported for future analysis.

3. Create another design point (DP2) by duplicating the DP1 design point.

   a. Right-click the **DP1** design point and select **Duplicate Design Point** from the context menu.

      Since this is a duplicate of DP1, this design point will also have its data retained.

4. Edit values for the input parameters for DP1 and DP2.

   For DP1 and DP2, edit the values for your input parameters within the Table of Design Points as follows:

	`input_hcpos`	`input_ftpos`	`input_wsfpos`	`in_velocity`	`in_temp`
DP1	45	45	45	0.6	300
DP2	90	60	15	0.7	290

Figure 2.21: Table of Design Points (with DP0, DP1, and DP2 Defined)

Table of Design Points

	A	B	C	D	E	F	G	H	I
1	Name	P1 - hcpos	P2 - ftpos	P3 - wsfpos	P4 - input_hcpos	P5 - input_ftpos	P6 - input_wsfpos	P7 - in_velocity	P8 - in_temp
2	Units							m s^-1	K
3	DP 0 (Current)	25	25	90	15	25	90	0.5	310
4	DP 1	45	45	45	45	45	45	0.6	300
5	DP 2	90	60	15	90	60	15	0.7	290
*									

For demonstration purposes of this tutorial, in each design point, you are slightly changing the angles of each of the valves, and increasing the inlet velocity and the inlet temperature. Later, you will see how the results in each case vary.

5. Update all of your design points.

   Click the **Update all Design Points** button in the ANSYS Workbench toolbar. Alternatively, you can also select one or more design points, right-click, and select **Update Selected Design Points** from the context menu. Click **OK** to acknowledge the information message notifying you that some open editors may close during the update process. By updating the design points, ANSYS Workbench takes the new values of the input parameters for each design point and updates the components of the associated system (for example, the geometry, mesh, settings, solution, and results), as well as any output parameters that have been defined.

   Note

   It may take significant time and/or computing resources to re-run the simulations for each design point.

   ---

6. Export the design points to separate projects.

   This will allow you to work with calculated data for each design point.

   a. Select the three design points, **DP0**, **DP1**, and **DP2** (hold **Shift** for multiple selection).

   b. Right-click the selected design points and select **Export Selected Design Points**.

      Note the addition of three more ANSYS Workbench project files (and their corresponding folders) in your current working directory (`fluent-workbench-param_dp0.wbpj`, `fluent-workbench-param_dp1.wbpj` and `fluent-workbench-param_dp2.wbpj`). You can open each of these projects up separately and examine the results of each parameterized simulation.

   Tip

   You can easily access files in your project directory directly from the **Files** view by right-clicking any cell in the corresponding row and selecting **Open Containing Folder** from the menu that opens.

   ---

7. Inspect the output parameter values in ANSYS Workbench.

Once all design points have been updated, you can use the table of design points to inspect the values of the output parameters you created in CFD-Post (for example, the mass flow parameters at the various outlets: `floutfront`, `floutfoot`, `floutwindshield`, and `outlettemp`). These, and the rest of the output parameters are listed to the far right in the table of design points.

Figure 2.22: Table of Design Points (Showing Output Parameters for DP0, DP1, and DP2)

Table of Design Points

	A	E	F	G	H	I	J	K	L	M
1	Name	P4 - input_hcpos	P5 - input_ftpos	P6 - input_wsfpos	P7 - in_velocity	P8 - in_temp	P9 - floutfront	P10 - floutwindshield	P11 - floutfoot	P12 - outlettemp
2	Units				m s^-1	K	kg s^-1	kg s^-1	kg s^-1	K
3	DP 0 (Current)	15	25	90	0.5	310	-0.00026376	0.0011285	0.01016	292.34
4	DP 1	45	45	45	0.6	300	9.5785E-05	0.0071258	0.0060084	284.98
5	DP 2	90	60	15	0.7	290	0.0016708	0.0081473	0.0056169	277.13
*										

8. Click the **Project** tab, just above the ANSYS Workbench toolbar to return to the **Project Schematic**.

9. View the list of files generated by ANSYS Workbench (optional).

   View → **Files**

   The additional files for the new design points are stored with their respective project files since you exported them.

10. Save the project in the current state in ANSYS Workbench.

    In the main menu, select **File** → **Save**.

11. Quit ANSYS Workbench.

    In the main menu, select **File** → **Exit**.

2.4.8. Postprocessing the New Design Points in CFD-Post

In this section, you will open the ANSYS Workbench project for each of the design points and inspect the vector plots based on the new results of the simulations.

1. Study the results of the first design point (DP1).

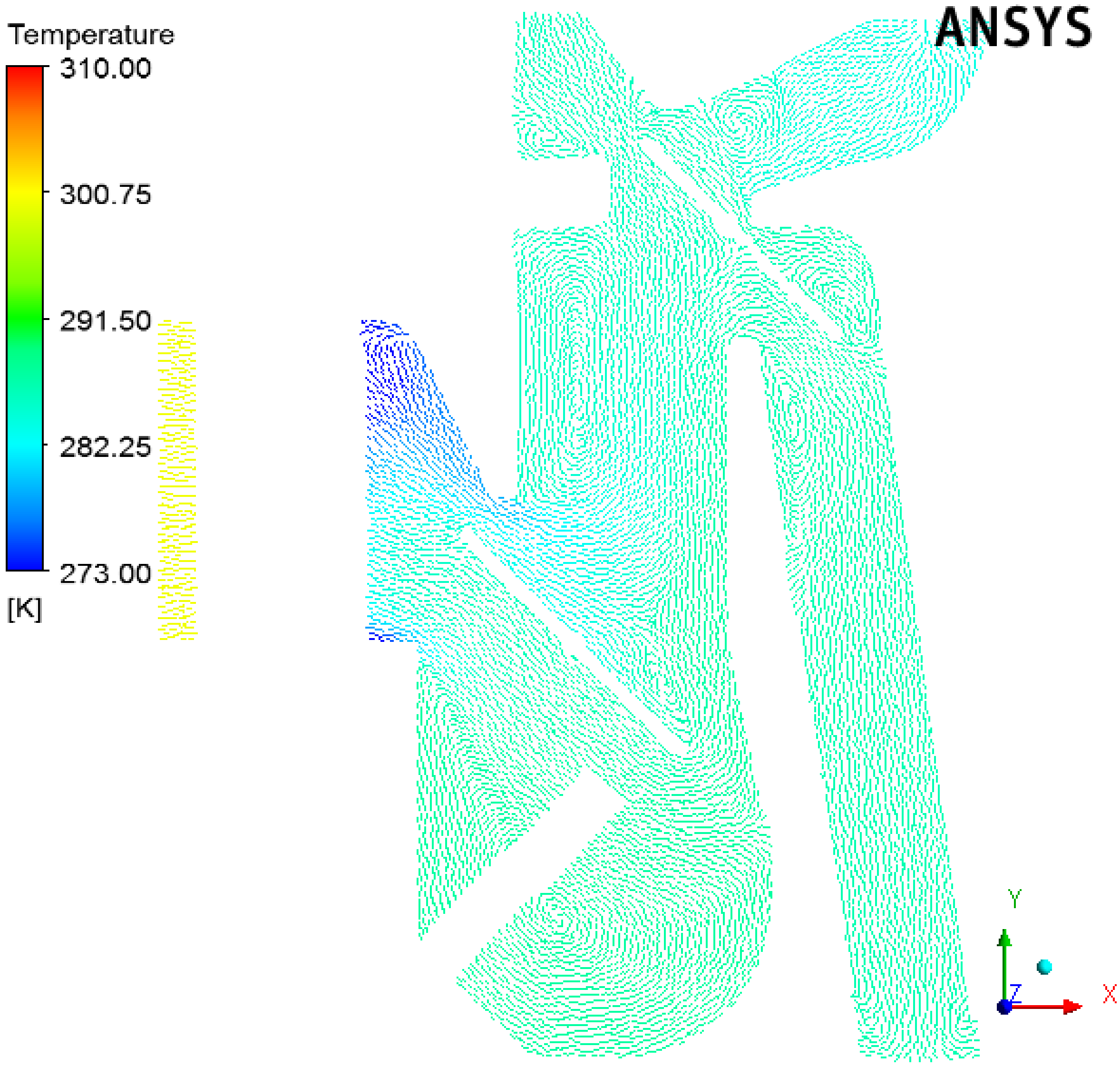
   a. Open the ANSYS Workbench project for the first design point (DP1).

      In your current working folder, double-click the `fluent-workbench-param_dp1.wbpj` file to open ANSYS Workbench.

   b. Open CFD-Post by double-clicking the **Results** cell in the Project Schematic for the Fluid Flow (Fluent) analysis system.

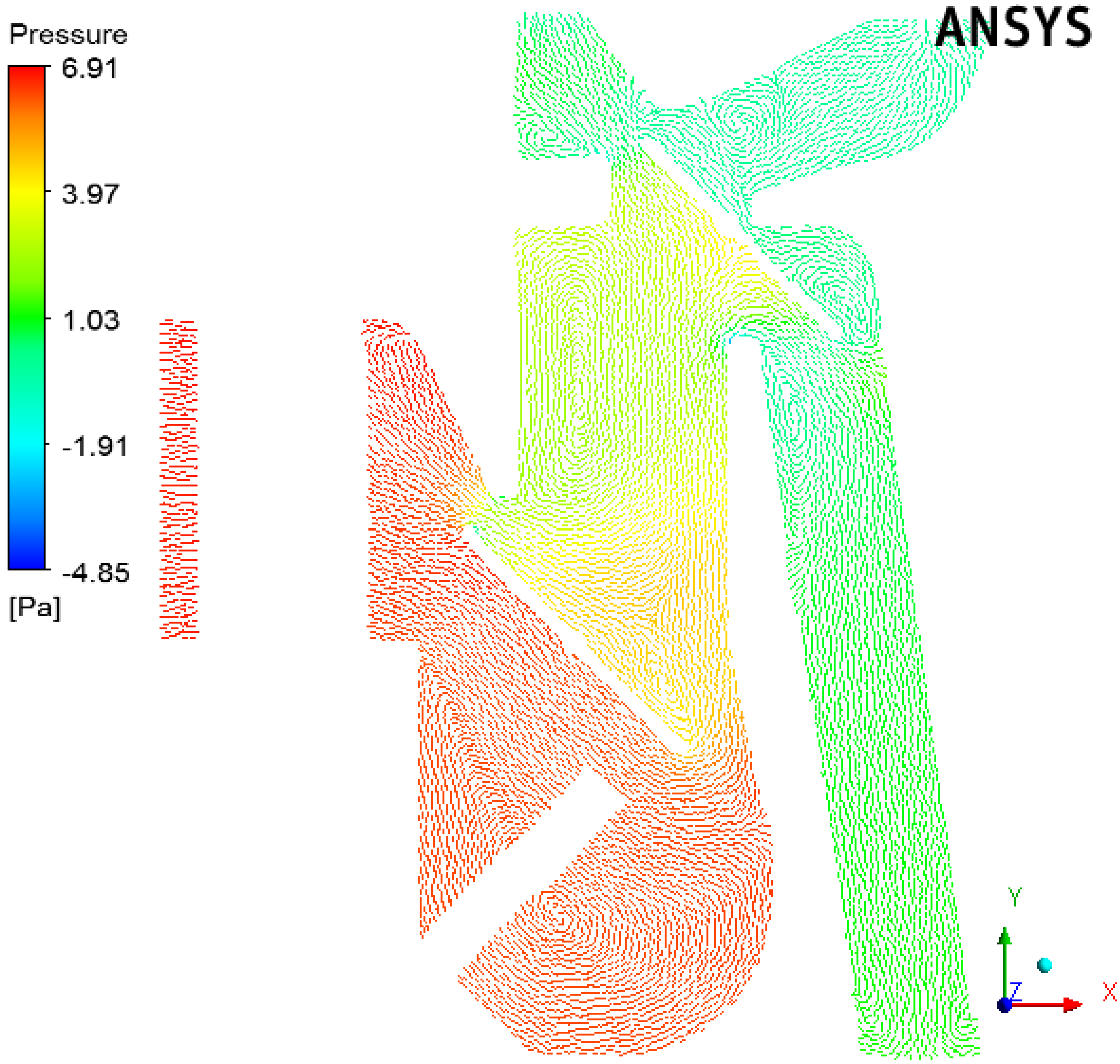
   c. View the vector plot colored by temperature. Ensure that **Range** in the **Color** tab is set to **User Specified** and the **Min** and **Max** temperature values are set to `273 K` and `310 K`, respectively.

Figure 2.23: Vectors Colored by Temperature (DP1)



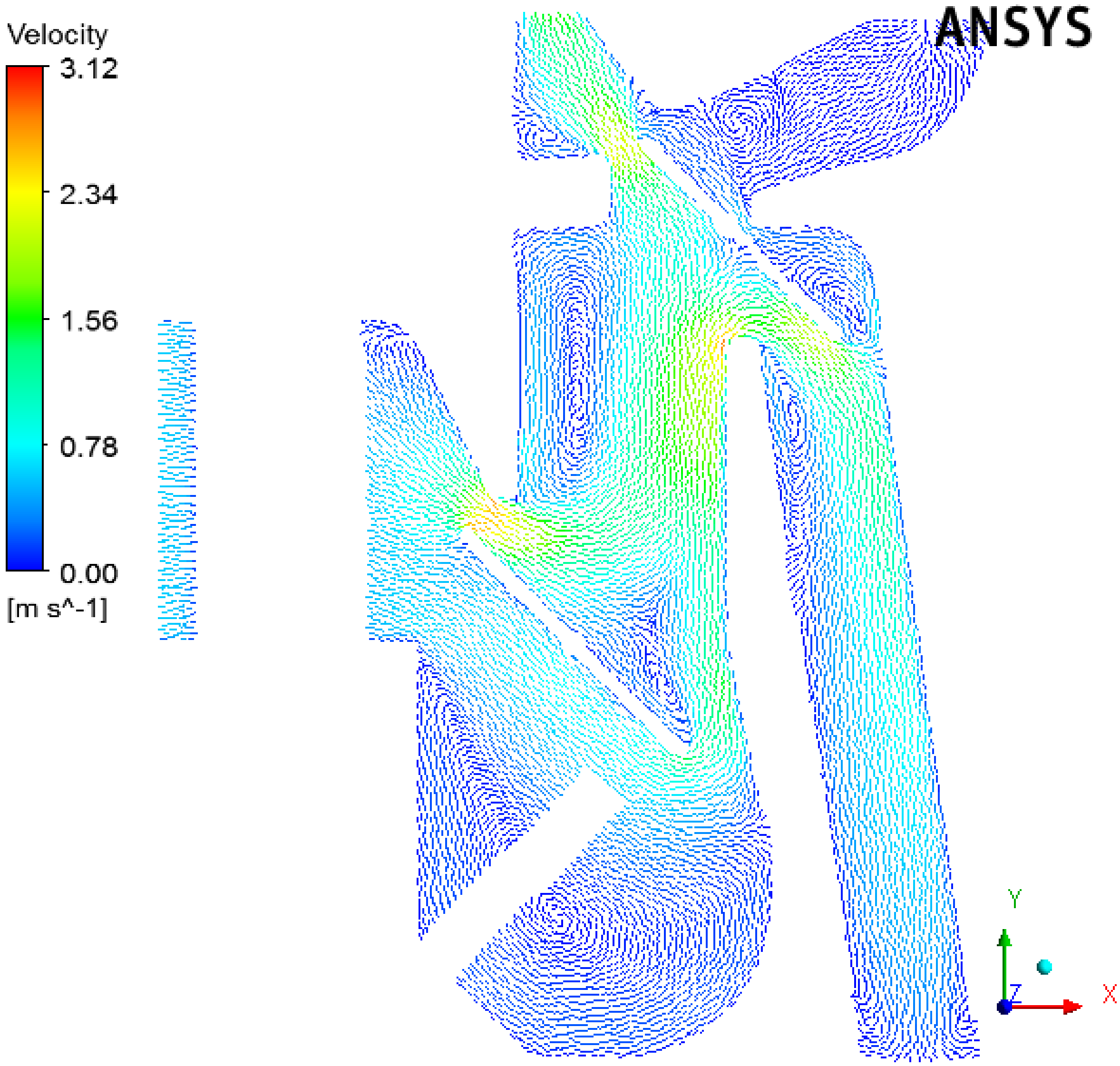
d. View the vector plot colored by pressure. Ensure that **Range** in the **Color** tab is set to **Global**.

Figure 2.24: Vectors Colored by Pressure (DP1)



e. View the vector plot colored by velocity. Ensure that **Range** in the **Color** tab is set to **Global**.

Figure 2.25: Vectors Colored by Velocity (DP1)



f. When you are finished viewing results of the design point DP1 in ANSYS CFD-Post, select **File** → **Close CFD-Post** to quit ANSYS CFD-Post and return to the ANSYS Workbench **Project Schematic**, and then select **File** → **Exit** to exit from ANSYS Workbench.

2. Study the results of the second design point (DP2).

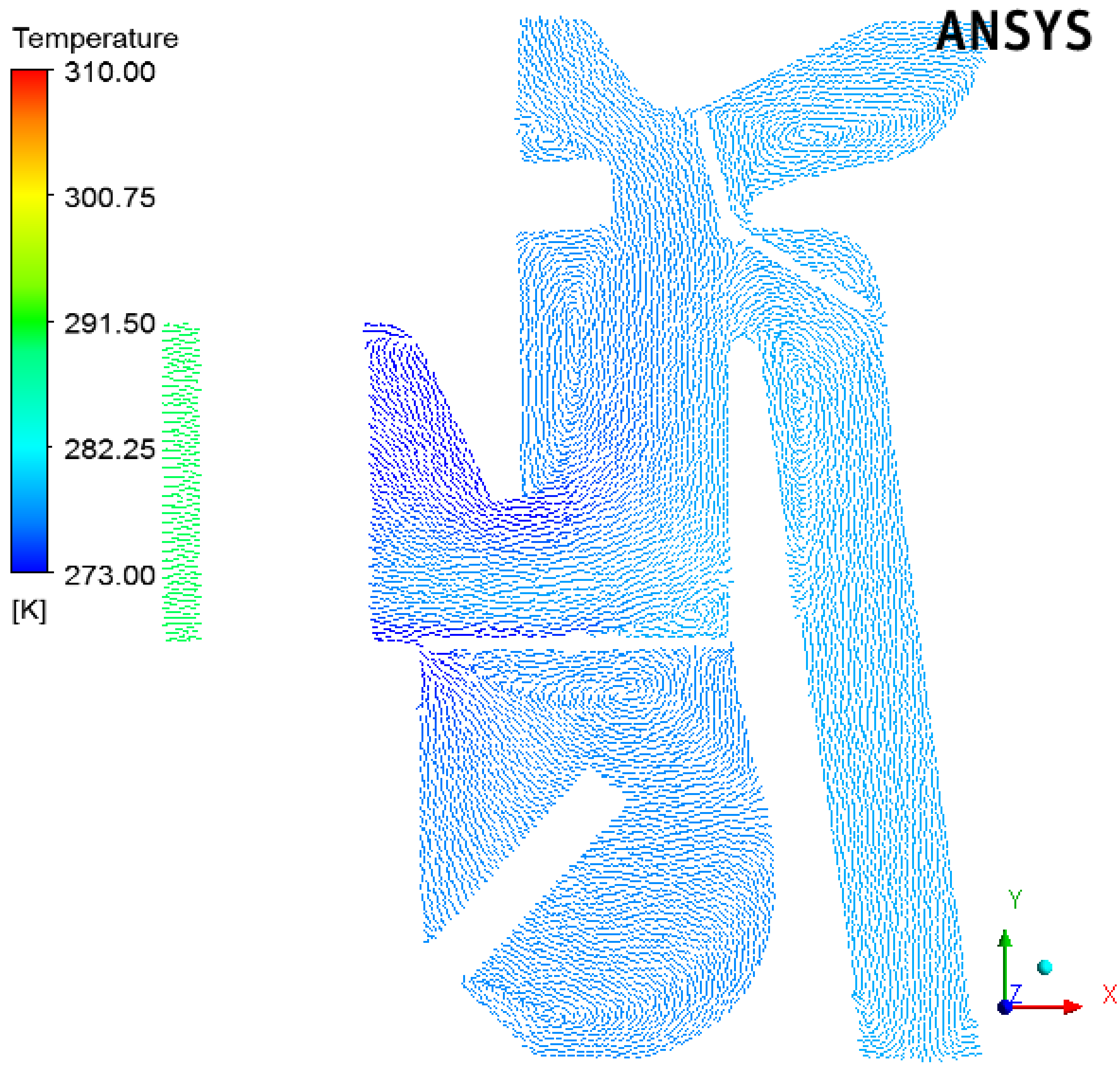
   a. Open the ANSYS Workbench project for the second design point (DP2).

      In your current working folder, double-click the `fluent-workbench-param_dp2.wbpj` file to open ANSYS Workbench.

   b. Open CFD-Post by double-clicking the **Results** cell in the Project Schematic for the Fluid Flow (Fluent) analysis system.

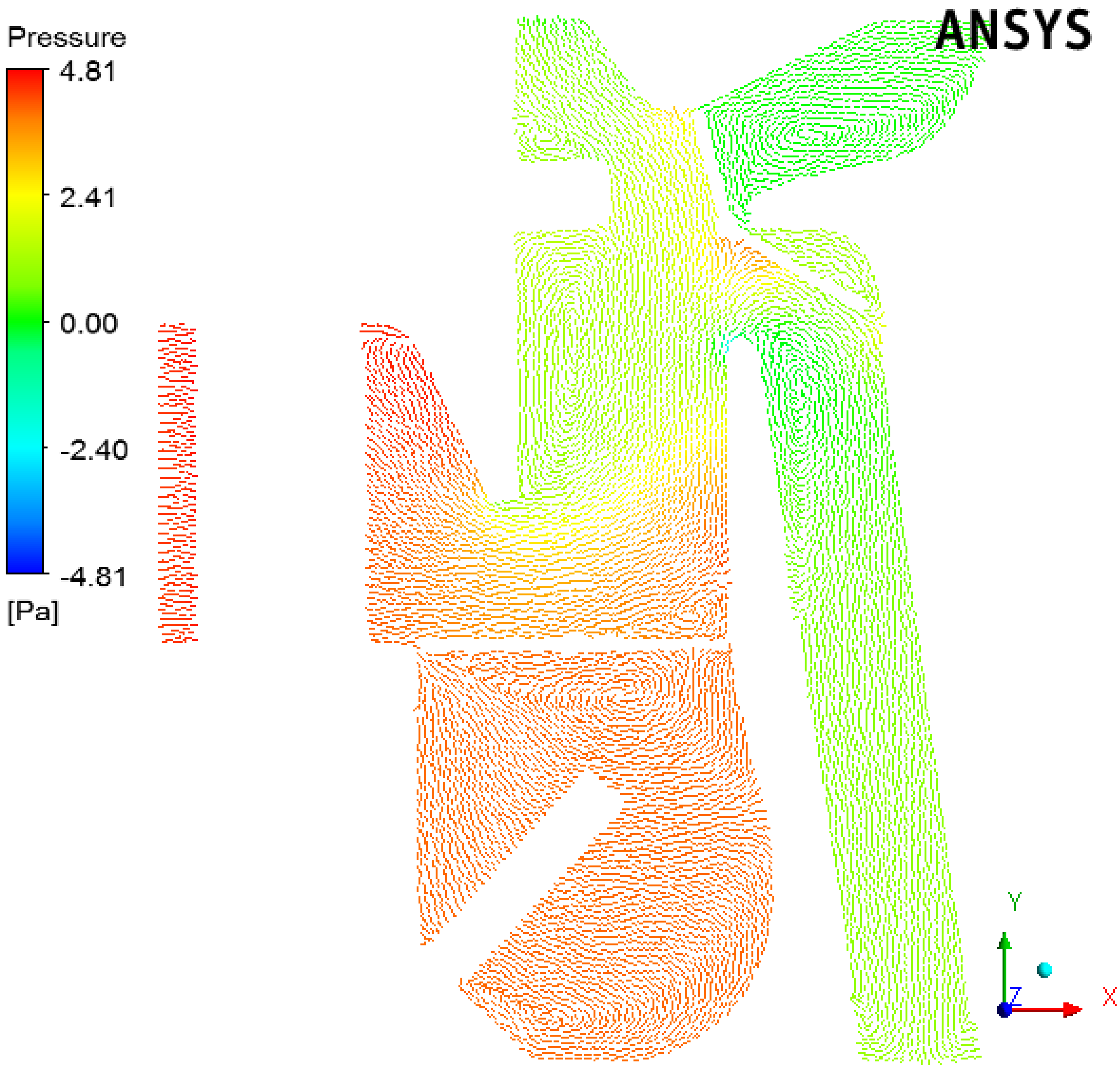
c. View the vector plot colored by temperature. Ensure that **Range** in the **Color** tab is set to **User Specified** and the **Min** and **Max** temperature values are set and the **Min** and **Max** temperature values are set to `273` K and `310` K, respectively.

Figure 2.26: Vectors Colored by Temperature (DP2)



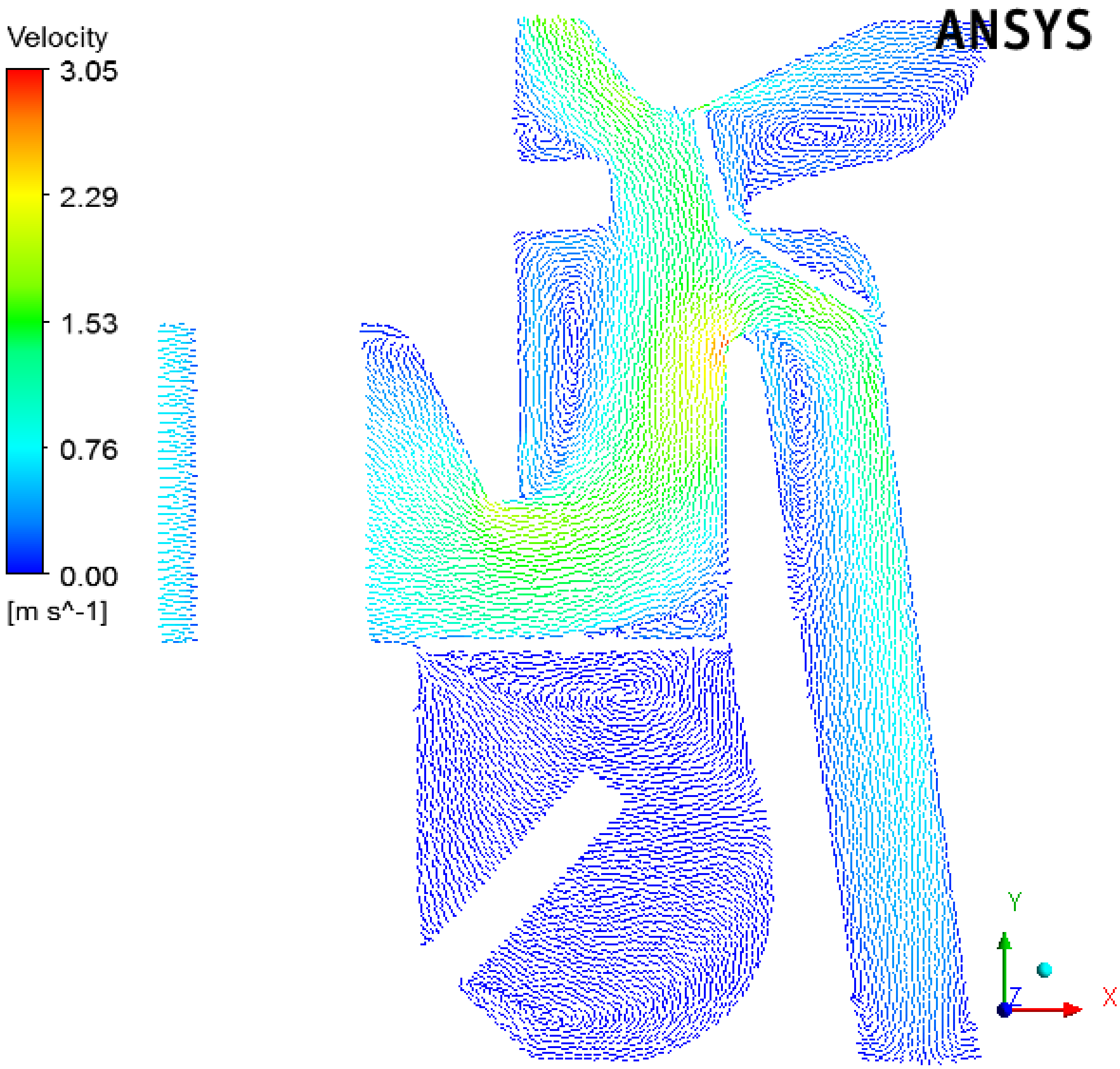
d. View the vector plot colored by pressure. Ensure that **Range** in the **Color** tab is set to **Global**.

Figure 2.27: Vectors Colored by Pressure (DP2)



e. View the vector plot colored by velocity. Ensure that **Range** in the **Color** tab is set to **Global**.

Figure 2.28: Vectors Colored by Velocity (DP2)



3. When you are finished viewing results in ANSYS CFD-Post, select **File** → **Close CFD-Post** to quit ANSYS CFD-Post and return to the ANSYS Workbench **Project Schematic**, and then select **File** → **Exit** to exit from ANSYS Workbench.

2.4.9. Summary

In this tutorial, input and output parameters were created within ANSYS Workbench, ANSYS Fluent, and ANSYS CFD-Post in order to study the airflow in an automotive HVAC system. ANSYS Fluent was used to calculate the fluid flow throughout the geometry using the computational mesh, and ANSYS CFD-Post was used to analyze the results. ANSYS Workbench was used to create additional design points based on the original settings, and the corresponding simulations were run to create separate projects where parameterized analysis could be performed to study the effects of variable angles of the inlet valves, velocities, and temperatures. Also, note that simplified solution settings were used in this tutorial to speed up the solution time. For more improved solution accuracy, you would typically use denser mesh and higher order discretization for all flow equations.

Chapter 3: Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow

This tutorial is divided into the following sections:

3.1. Introduction

This tutorial illustrates the setup and solution of a three-dimensional turbulent fluid flow and heat transfer problem in a mixing elbow. The mixing elbow configuration is encountered in piping systems in power plants and process industries. It is often important to predict the flow field and temperature field in the area of the mixing region in order to properly design the junction.

This tutorial demonstrates how to do the following:

- Launch the serial version of ANSYS Fluent.
- Read an existing mesh file into ANSYS Fluent.
- Use mixed units to define the geometry and fluid properties.
- Set material properties and boundary conditions for a turbulent forced-convection problem.
- Create a surface report definition and use it as a convergence criterion.
- Calculate a solution using the pressure-based solver.
- Visually examine the flow and temperature fields using the postprocessing tools available in ANSYS Fluent.
- Change the solver method to coupled in order to increase the convergence speed.
- Adapt the mesh based on the temperature gradient to further improve the prediction of the temperature field.
- Run the ANSYS Fluent solver in parallel.

3.2. Prerequisites

This tutorial assumes that you have little or no experience with ANSYS Fluent, and so each step will be explicitly described.

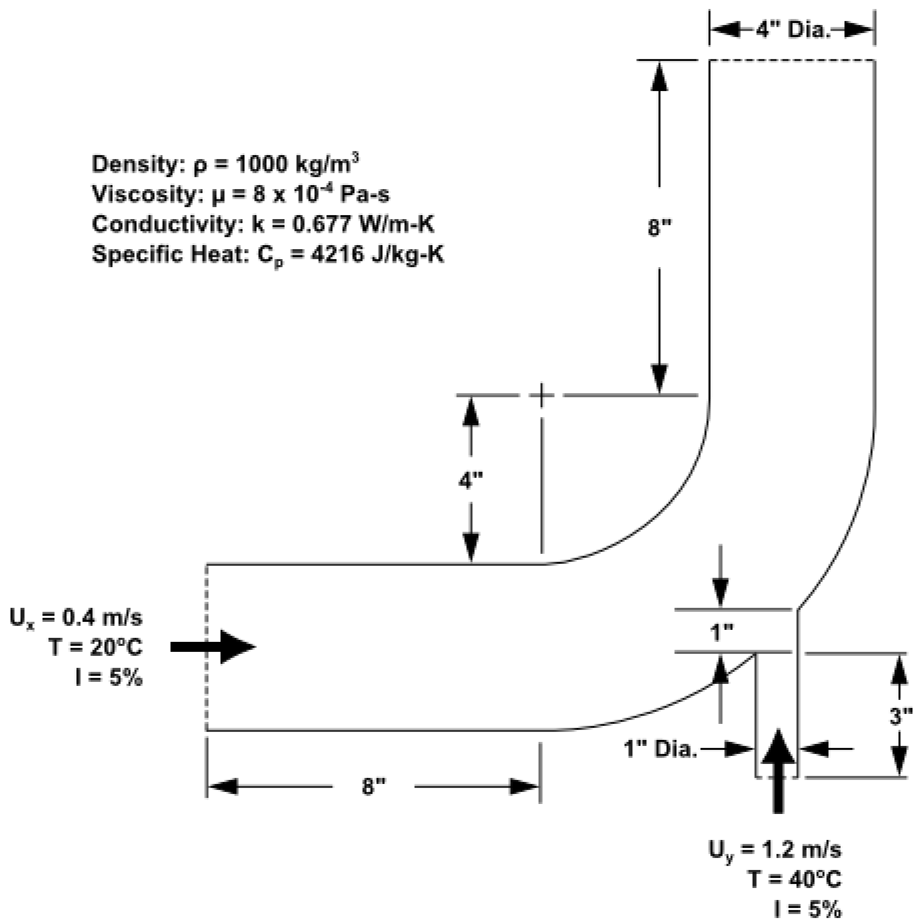
3.3. Problem Description

The problem to be considered is shown schematically in Figure 3.1: Problem Specification (p. 122). A cold fluid at 20° C flows into the pipe through a large inlet, and mixes with a warmer fluid at 40° C that enters through a smaller inlet located at the elbow. The pipe dimensions are in inches and the fluid properties and boundary conditions are given in SI units. The Reynolds number for the flow at the larger inlet is 50,800, so a turbulent flow model will be required.

Note

Since the geometry of the mixing elbow is symmetric, only half of the elbow must be modeled in ANSYS Fluent.

Figure 3.1: Problem Specification



3.4. Setup and Solution in Serial

To help you quickly identify graphical user interface items at a glance and guide you through the steps of setting up and running your simulation, the ANSYS Fluent Tutorial Guide uses several type styles and mini flow charts. See Typographical Conventions Used In This Manual (p. xvi) for detailed information.

The following sections describe the setup and solution steps for running this tutorial in serial:

3.4.1. Preparation

1. Set up a working folder on the computer you will be using.

2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

   Note

   If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

3. Enter the name of this tutorial into the search bar.

4. Narrow the results by using the filter on the left side of the page.

   a. Click **ANSYS Fluent** under **Product**.

   b. Click **18.0** under **Version**.

5. Select this tutorial from the list.

6. Click the **introduction_R180.zip** link to download the input files.

7. Unzip the `introduction_R180.zip` file you downloaded to your working folder. This file contains a folder, `introduction`, that holds the file `elbow.msh` that you will use in this tutorial.

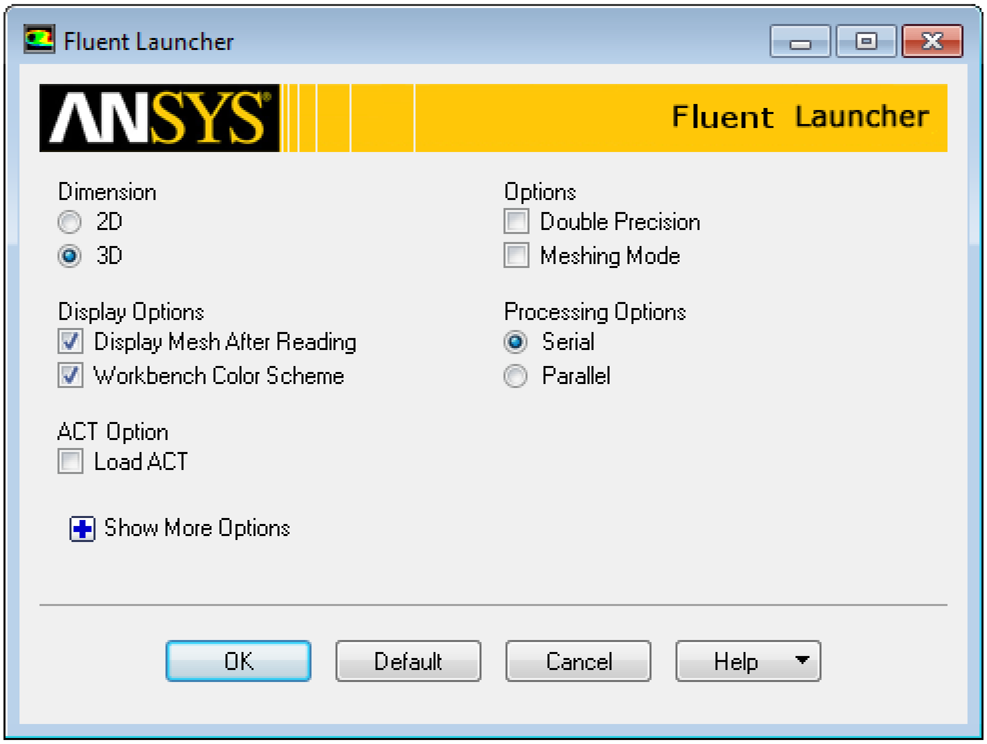
Note

ANSYS Fluent tutorials are prepared using ANSYS Fluent on a Windows system. The screen shots and graphic images in the tutorials may be slightly different than the appearance on your system, depending on the operating system and/or graphics card.

3.4.2. Launching ANSYS Fluent

1. From the Windows **Start** menu, select **Start > All Programs > ANSYS 18.0 > Fluid Dynamics > Fluent 18.0** to start Fluent Launcher.

   Fluent Launcher allows you to decide which version of ANSYS Fluent you will use, based on your geometry and on your processing capabilities.



2. Ensure that the proper options are enabled.

   a. Select **3D** from the **Dimension** list by clicking the radio button or the text.

   b. Select **Serial** from the **Processing Options** list.

   c. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.

      Note

      An option is enabled when there is a check mark in the check box, and disabled when the check box is empty. To change an option from disabled to enabled (or vice versa), click the check box or the text.

   d. Ensure that the **Double Precision** option is disabled.

   Note

   Fluent will retain your preferences for future sessions.

   Extra

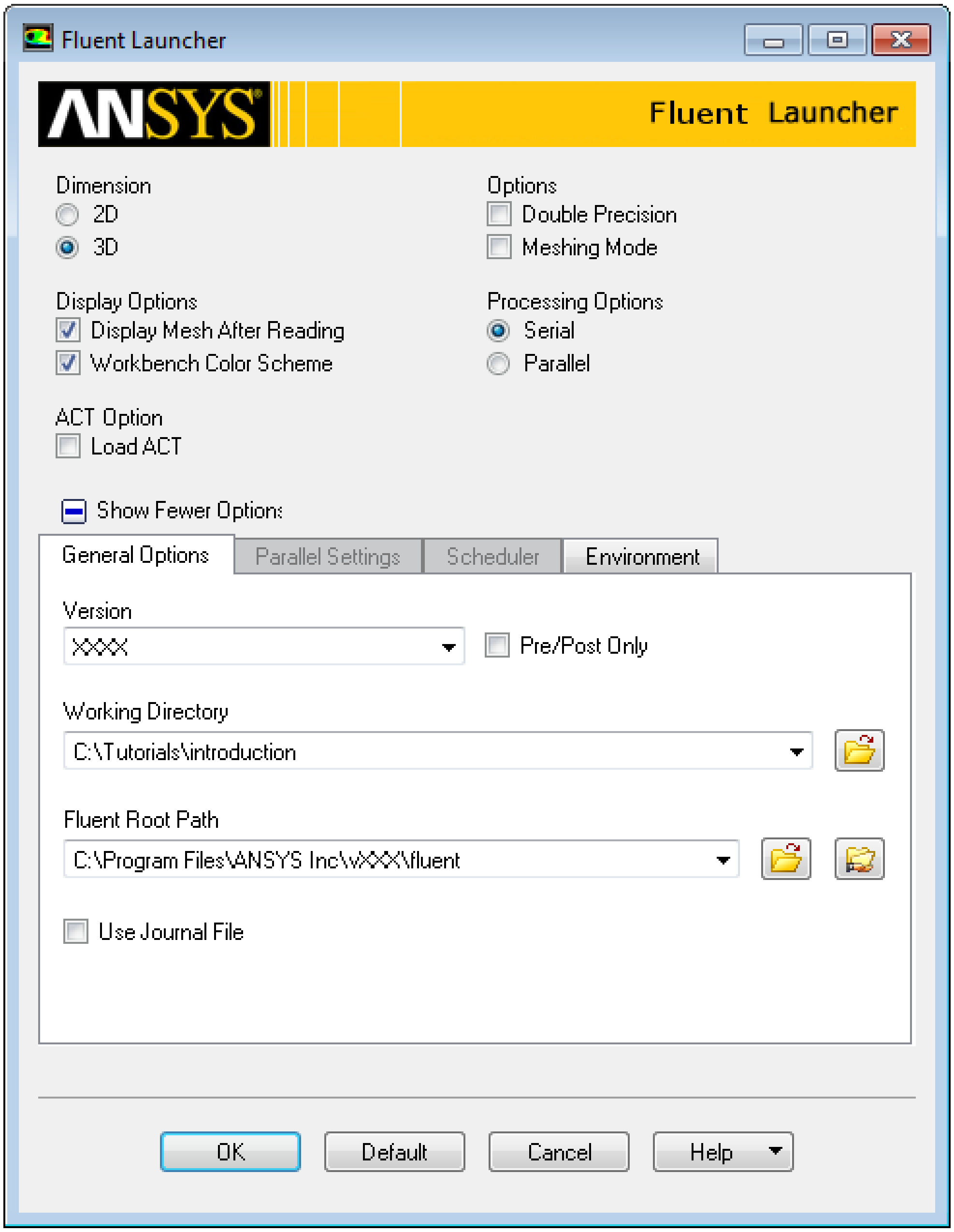
   You can also restore the default settings by clicking the **Default** button.

3. Set the working folder to the one created when you unzipped `introduction_R180.zip`.

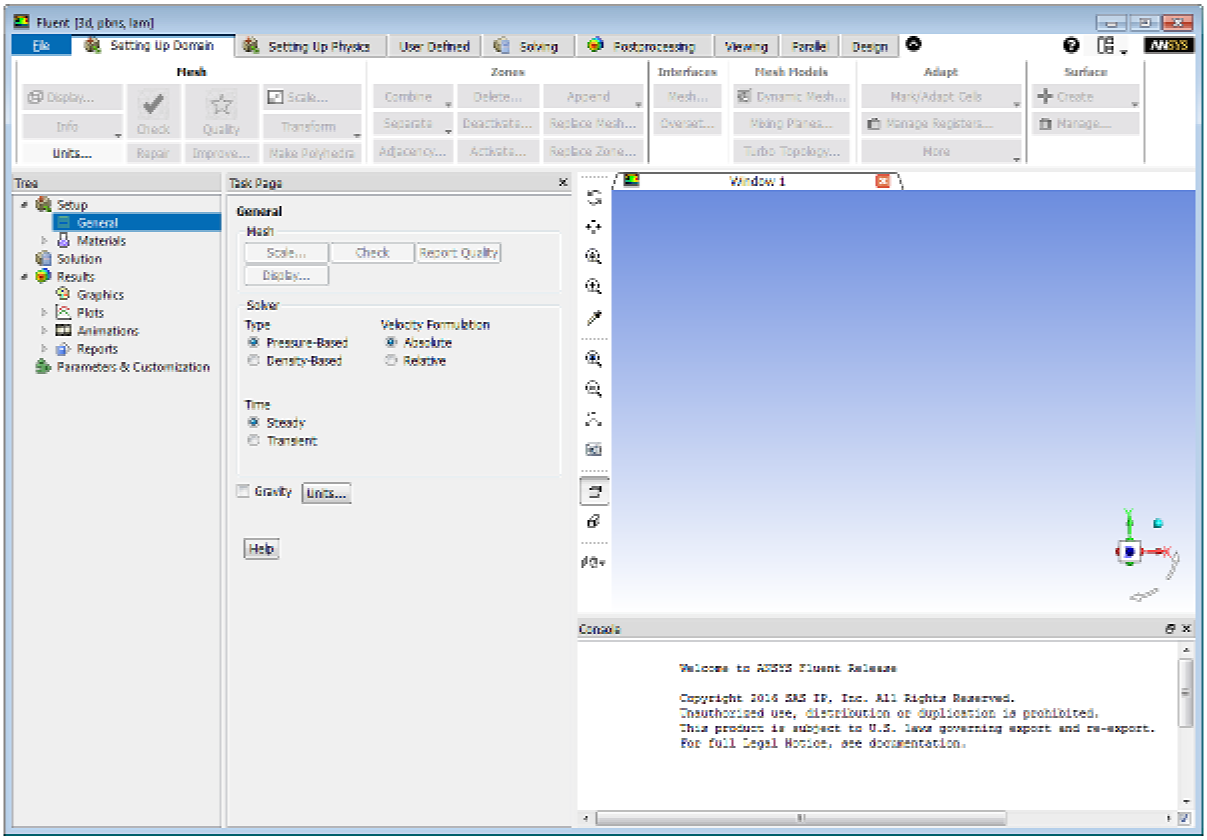
a. Click the **Show More Options** button to reveal additional options.

b. Enter the path to your working folder for **Working Directory** by double-clicking the text box and typing.

Alternatively, you can click the browse button () next to the **Working Directory** text box and browse to the directory, using the **Browse For Folder** dialog box.



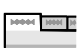
4. Click **OK** to launch ANSYS Fluent.

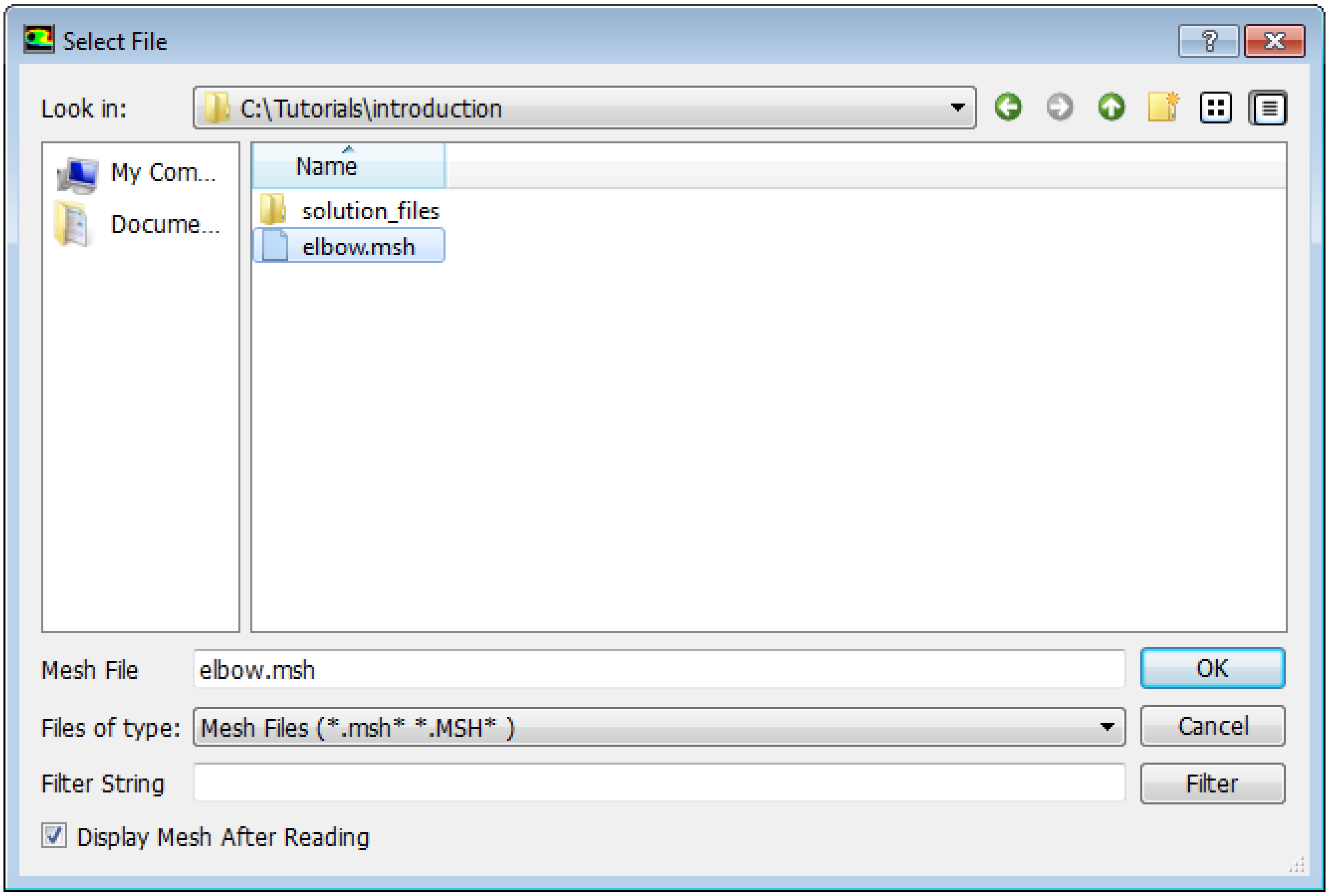


3.4.3. Reading the Mesh

1. Read the mesh file `elbow.msh`.

   Click the **File** ribbon tab, then click **Read** and **Mesh...** in the menus that open in order to open the **Select File** dialog box.

    **File → Read → Mesh...**



a. Select the mesh file by clicking **elbow.msh** in the **introduction** folder created when you unzipped the original file.

b. Click **OK** to read the file and close the **Select File** dialog box.

As the mesh file is read by ANSYS Fluent, messages will appear in the console reporting the progress of the conversion. ANSYS Fluent will report that 13,852 hexahedral fluid cells have been read, along with a number of boundary faces with different zone identifiers.

After having completed reading mesh, ANSYS Fluent displays the mesh in the graphics window.

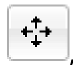
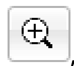
Extra

You can use the mouse to probe for mesh information in the graphics window. If you click the right mouse button with the pointer on any node in the mesh, information about the associated zone will be displayed in the console, including the name of the zone.

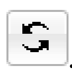
Alternatively, you can click the probe button () in the graphics toolbar and click the left mouse button on any node. This feature is especially useful when you have several zones of the same type and you want to distinguish between them quickly.

For this 3D problem, you can make it easier to probe particular nodes by changing the view. The following table describes how to manipulate objects in the graphics window:

Table 3.1: View Manipulation Instructions

Action	Using Graphics Toolbar Buttons and the Mouse
Rotate view (vertical, horizontal)	After clicking the **Rotate View** icon, , press and hold the left mouse button and drag the mouse. Dragging side to side rotates the view about the vertical axis, and dragging up and down rotates the view about the horizontal axis.
Translate or pan view	After clicking the **Pan** icon, , press and hold the left mouse button and drag the object with the mouse until the view is satisfactory.
Zoom in and out of view	After clicking the **Zoom In/Out** icon, , press and hold the left mouse button and drag the mouse up and down to zoom in and out of the view.
Zoom to selected area	After clicking the **Zoom to Area** icon, , press and hold the left mouse button and drag the mouse diagonally to the right. This action will cause a rectangle to appear in the display. When you release the mouse button, a new view will be displayed that consists entirely of the contents of the rectangle. Note that to zoom in, you must drag the mouse to the right, and to zoom out, you must drag the mouse to the left.

Clicking the **Fit to Window** icon, , will cause the object to fit exactly and be centered in the window.

After you have clicked a button in the graphics toolbar, you can return to the default mouse button settings by clicking .

To judge the scale of your 3D geometry, you can click the **Orthographic Projection** icon, . This will display the length scale ruler near the bottom of the graphics window.

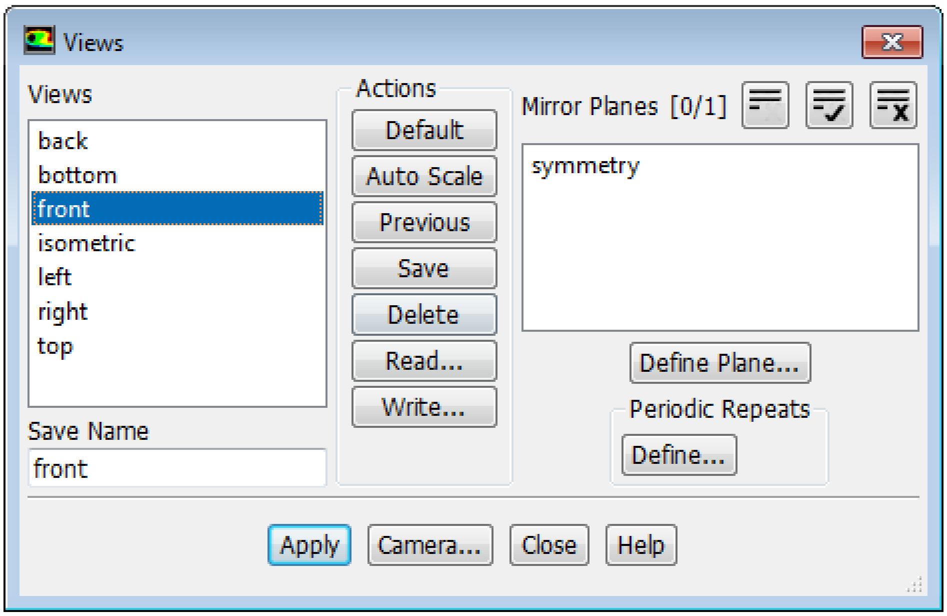
Note that you can change the default mouse button actions in the **Viewing** tab (in the **Mouse** group box). For more information, see the Fluent User's Guide.

2. Manipulate the mesh display using the **Views** dialog box to obtain a front view as shown in Figure 3.2: The Hexahedral Mesh for the Mixing Elbow (p. 130).

   In the **Viewing** ribbon tab, locate the **Display** group box.

Then click the **Views...** button to open the **Views** dialog box.

Viewing → **Display** → **Views...**



Note

You can also open the **Views** dialog box from the tree, by expanding **Results**, right-clicking **Graphics**, and selecting **Views...**.

Results → **Graphics** **Views...**

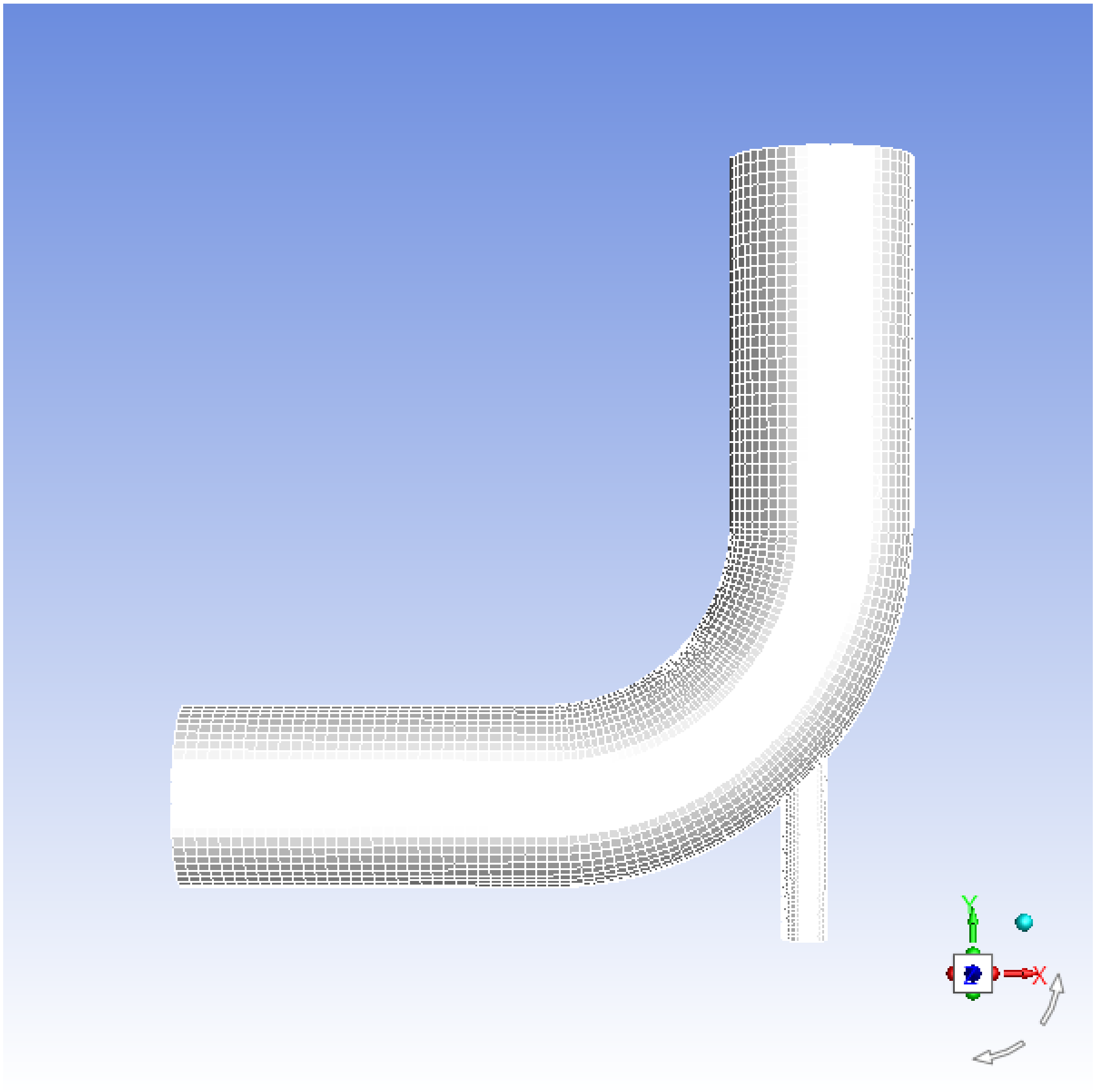
a. Select **front** from the **Views** selection list.

Note

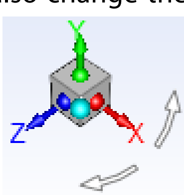
A list item is selected if it is highlighted, and deselected if it is not highlighted.

b. Click **Apply** and close the **Views** dialog box.

Figure 3.2: The Hexahedral Mesh for the Mixing Elbow



Extra

You can also change the orientation of the objects in the graphics window using the axis triad 

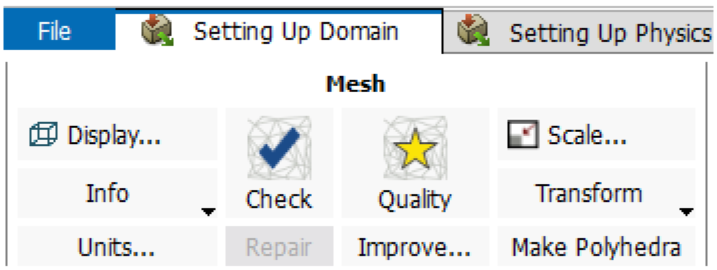
as follows:

- To orient the model in the positive/negative direction, click an axis/semi-sphere.
- To orient the model in the negative/positive direction, right-click an axis/semi-sphere.
- To set the isometric view, click the cyan iso-ball.

- To perform in-plane clockwise or counterclockwise 90° rotations, click the white rotational arrows .
- To perform free rotations in any direction, click and hold—in the vicinity of the triad—and use the mouse. Release the left mouse button to stop rotating.

3.4.4. Setting Up Domain

In this step, you will perform the mesh-related activities using the **Setting Up Domain** ribbon tab (**Mesh** group box).



1. Check the mesh.

   Setting Up Domain → **Mesh** → **Check**

   ANSYS Fluent will report the results of the mesh check in the console.

```
  Domain Extents:
    x-coordinate: min (m) = -8.000000e+00, max (m) = 8.000000e+00
    y-coordinate: min (m) = -9.134633e+00, max (m) = 8.000000e+00
    z-coordinate: min (m) = 0.000000e+00, max (m) = 2.000000e+00
  Volume statistics:
    minimum volume (m3): 5.098298e-04
    maximum volume (m3): 2.330736e-02
      total volume (m3): 1.607154e+02
  Face area statistics:
    minimum face area (m2): 4.865882e-03
    maximum face area (m2): 1.017924e-01
  Checking mesh.........................
 Done.
```

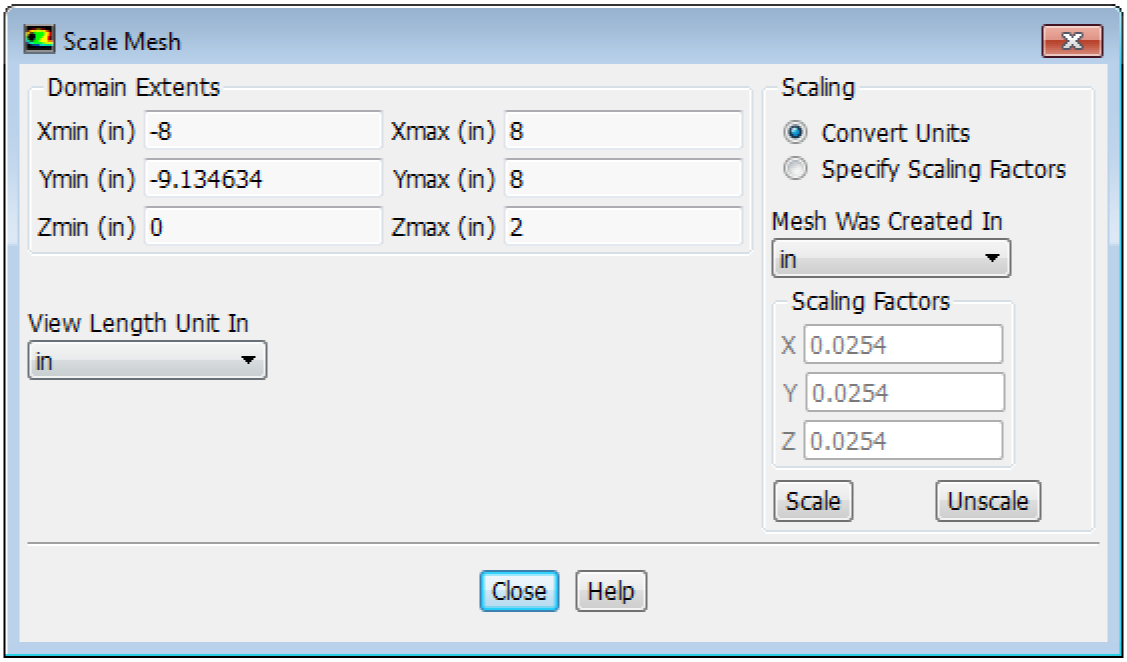
   The mesh check will list the minimum and maximum x, y, and z values from the mesh in the default SI unit of meters. It will also report a number of other mesh features that are checked. Any errors in the mesh will be reported at this time. Ensure that the minimum volume is not negative, since ANSYS Fluent cannot begin a calculation when this is the case.

   Note

   The minimum and maximum values may vary slightly when running on different platforms.

2. Scale the mesh.

   Setting Up Domain → **Mesh** → **Scale...**



a. Ensure that **Convert Units** is selected in the **Scaling** group box.

b. From the **Mesh Was Created In** drop-down list, select **in** by first clicking the down-arrow button and then clicking the **in** item from the list that appears.

c. Click **Scale** to scale the mesh.

> **Warning**
>
> Be sure to click the **Scale** button only once.

Domain Extents *will continue to be reported in the default SI unit of meters.*

d. Select **in** from the **View Length Unit In** drop-down list to set inches as the working unit for length.

e. Confirm that the domain extents are as shown in the previous dialog box.

f. Close the **Scale Mesh** dialog box.

The mesh is now sized correctly and the working unit for length has been set to inches.

Note

Because the default SI units will be used for everything except length, there is no need to change any other units in this problem. The choice of inches for the unit of length has been made by the actions you have just taken. If you want a different working unit for length, other than inches (for example, millimeters), click **Units...** in the **Setting Up Domain** ribbon tab (**Mesh** group box) and make the appropriate change in the **Set Units** dialog box.

---

3. Check the mesh.

Setting Up Domain → Mesh → Check

Note

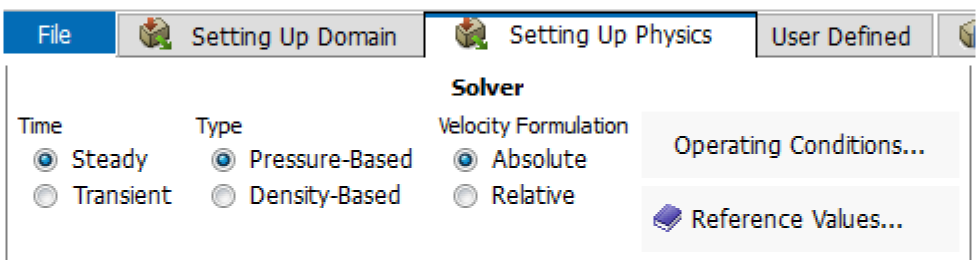
It is a good idea to check the mesh after you manipulate it (that is, scale, convert to polyhedra, merge, separate, fuse, add zones, or smooth and swap). This will ensure that the quality of the mesh has not been compromised.

3.4.5. Setting Up Physics

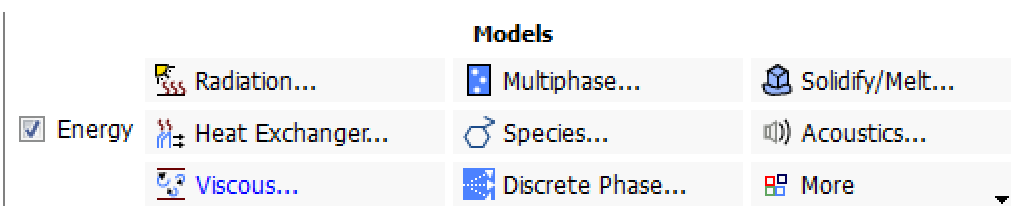
In the steps that follow, you will select a solver and specify physical models, material properties, and zone conditions for your simulation using the **Setting Up Physics** ribbon tab.

1. In the **Solver** group box of the **Setting Up Physics** ribbon tab, retain the default selection of the steady pressure-based solver.

   Setting Up Physics → Solver



2. Set up your models for the CFD simulation using the **Models** group box of the **Setting Up Physics** ribbon tab.



Note

You can also use the **Models** task page, which can be accessed from the tree by expanding **Setup** and double-clicking the **Models** tree item.

---

   a. Enable heat transfer by activating the energy equation.

      In the **Setting Up Physics** ribbon tab, enable **Energy** (**Models** group box).

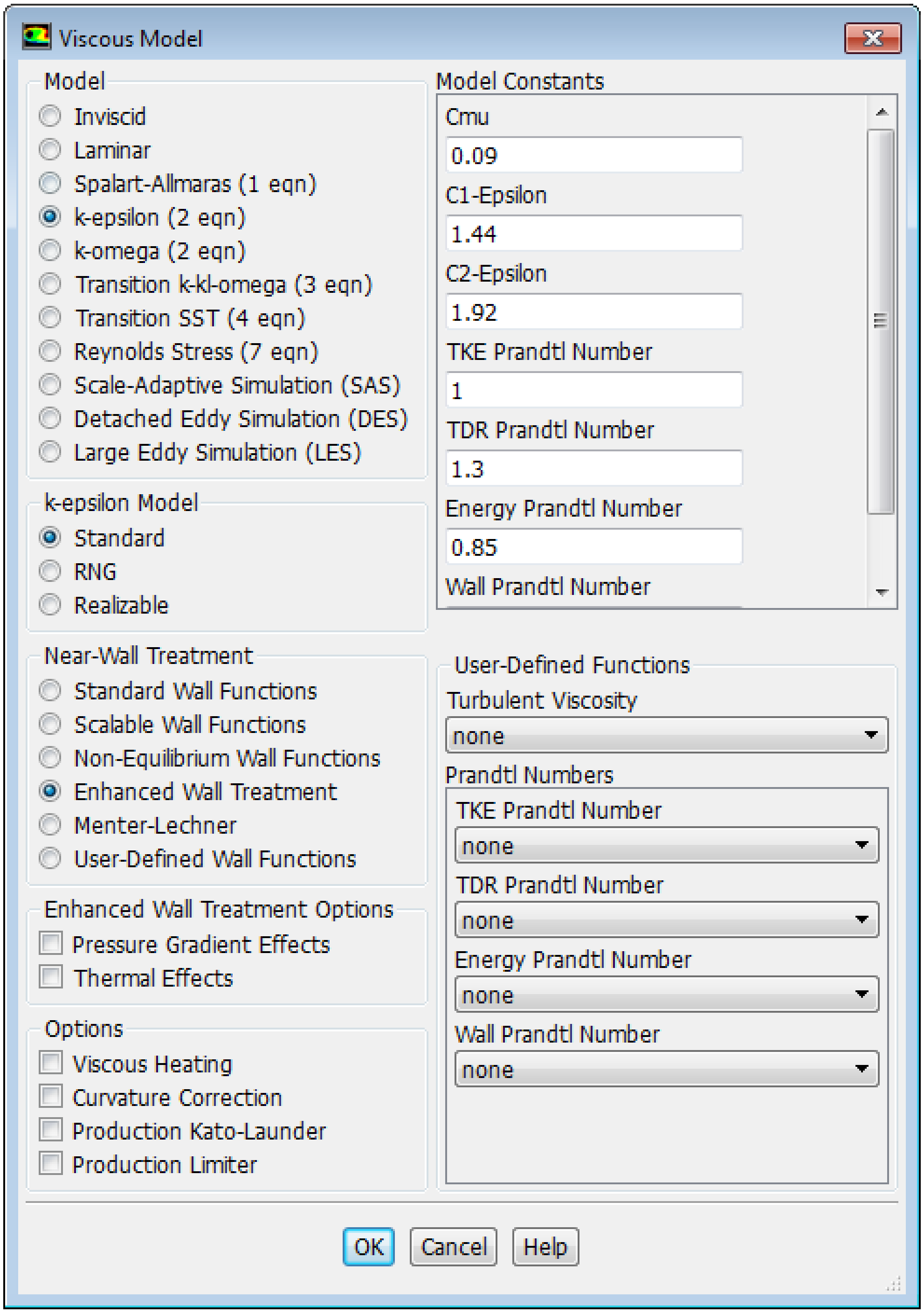
Setting Up Physics → Models → Energy

Note

You can also double-click the **Setup/Models/Energy** tree item and enable the energy equation in the **Energy** dialog box.

b. Enable the k- ε turbulence model.

Setting Up Physics → Models → Viscous...



i. Select **k-epsilon** from the **Model** list.

*The **Viscous Model** dialog box will expand.*

ii. Retain the default selection of **Standard** in the **k-epsilon Model** group box.

iii. Select **Enhanced Wall Treatment** in the **Near-Wall Treatment** group box.

Note

The default Standard Wall Functions are generally applicable if the first cell center adjacent to the wall has a y+ larger than 30. In contrast, the Enhanced Wall Treatment option provides consistent solutions for all y+ values. Enhanced Wall Treatment is recommended when using the k-epsilon model for general single-phase fluid flow problems. For more information about Near Wall Treatments in the k-epsilon model, refer to the Fluent User's Guide.

---

iv. Click **OK** to accept all the other default settings and close the **Viscous Model** dialog box.

Note that the **Viscous...** label in the ribbon is displayed in blue to indicate that the Viscous model is enabled. Also **Energy** and **Viscous** appear as enabled under the **Setup/Models** tree branch.

Note

While the ribbon is the primary tool for setting up and solving your problem, the tree is a dynamic representation of your case. The models, materials, conditions, and other settings that you have specified in your problem will appear in the tree. Many of the frequently used ribbon items are also available via the right-click functionality of the tree.

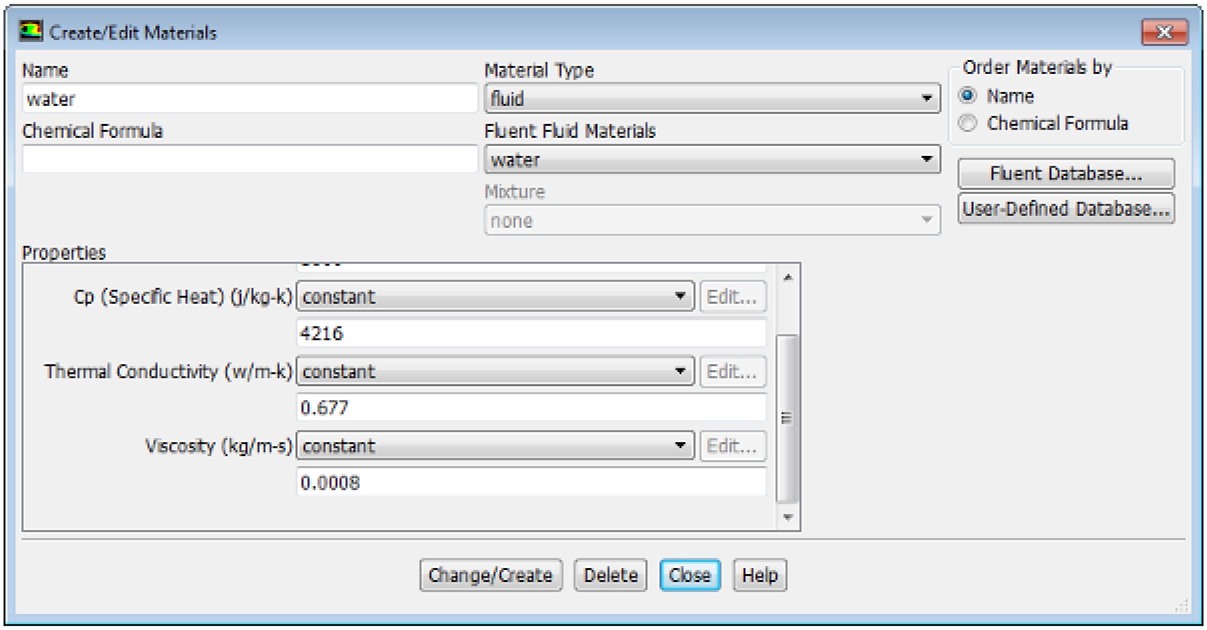
---

3. Set up the materials for the CFD simulation using the **Materials** group box of the **Setting Up Physics** ribbon tab.

Create a new material called **water** using the **Create/Edit Materials** dialog box.

a. In the **Setting Up Physics** ribbon tab, click **Create/Edit...** (**Materials** group box).

Setting Up Physics → Materials → Create/Edit...



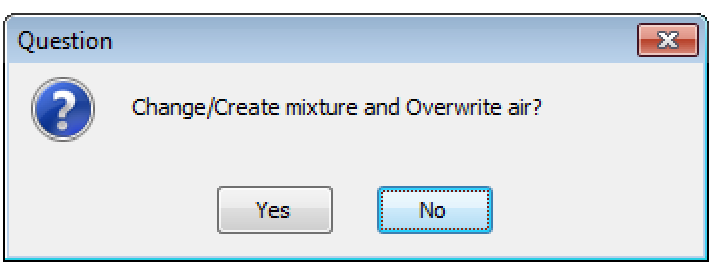
b. Enter `water` for **Name**.

c. Enter the following values in the **Properties** group box:

Property	Value
Density	`1000` [kg/m^3]
c_p (Specific Heat)	`4216` [J/kg-K]
Thermal Conductivity	`0.677` [W/m-K]
Viscosity	`8e-04` [kg/m-s]

d. Click **Change/Create**.

*A **Question** dialog box will open, asking if you want to overwrite air. Click **No** so that the new material **water** is added to the **Fluent Fluid Materials** list of materials that originally contained only **air**.*



Extra

You could have copied the material **water-liquid (h2o < l >)** from the materials database (accessed by clicking the **Fluent Database...** button). If the properties in the database are different from those you want to use, you can edit the values in the

Properties group box in the **Create/Edit Materials** dialog box and click **Change/Create** to update your local copy. The original copy will not be affected.

---

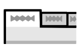
e. Ensure that there are now two materials (water and air) defined locally by examining the **Fluent Fluid Materials** drop-down list.

   *Both the materials will also be listed under **Fluid** in the **Materials** task page and under the **Materials** tree branch.*

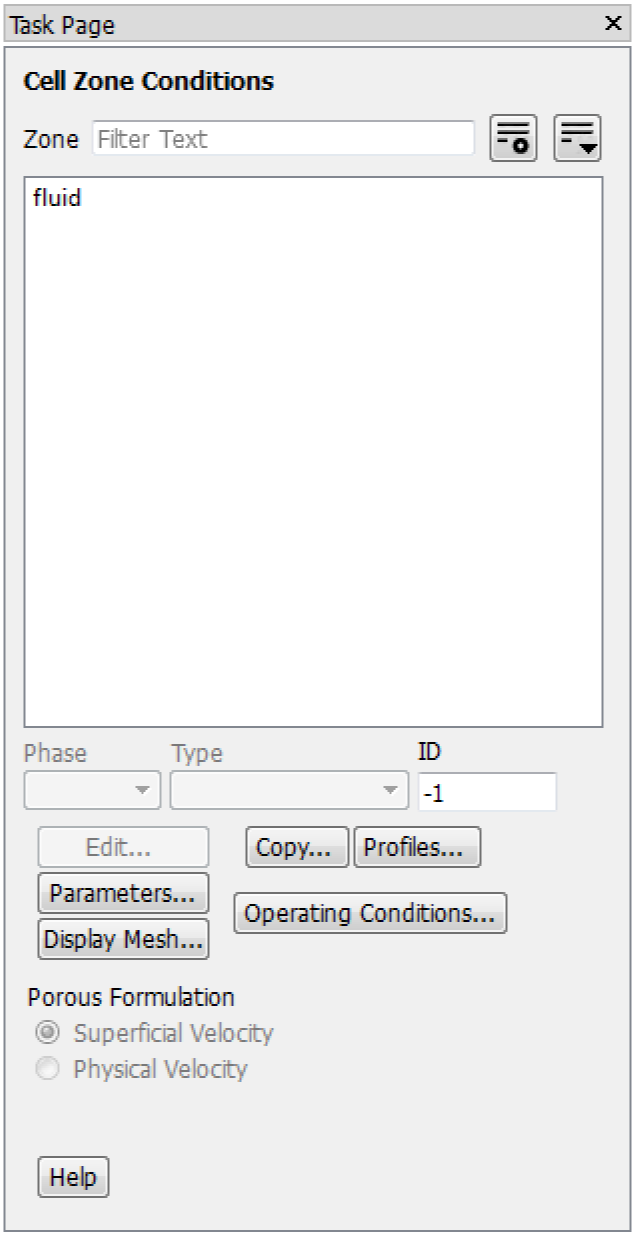
f. Close the **Create/Edit Materials** dialog box.

4. Set up the cell zone conditions for the fluid zone (**fluid**) using the **Zones** group box of the **Setting Up Physics** ribbon tab.

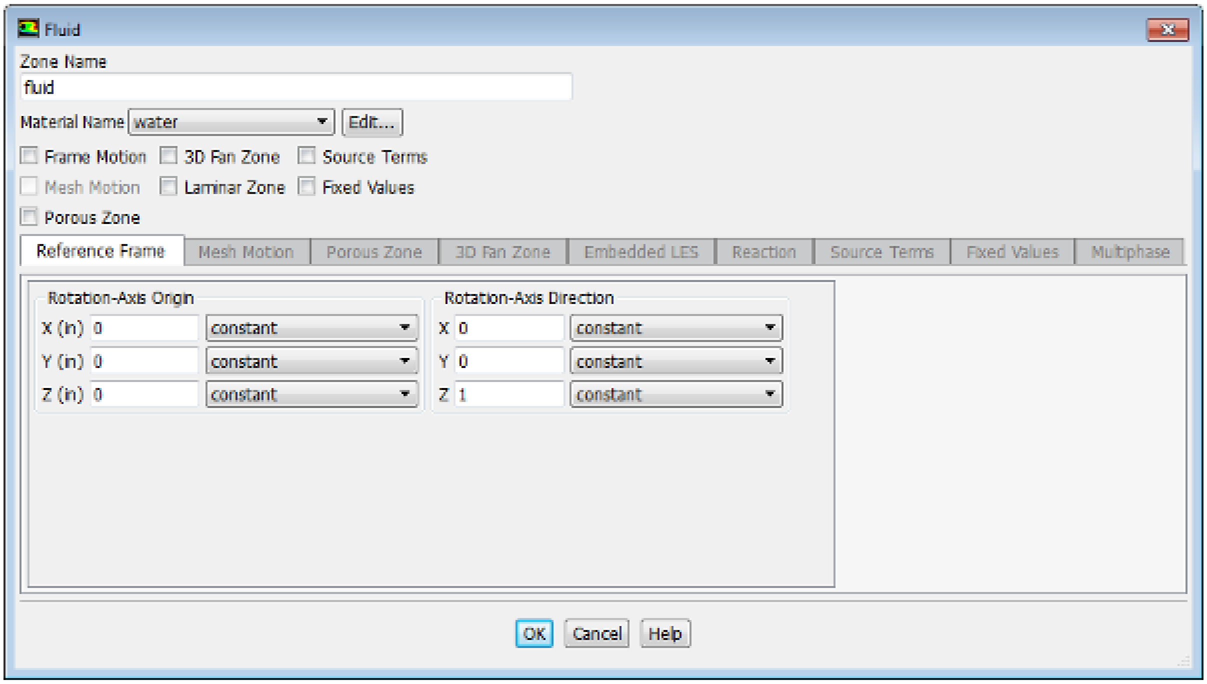
a. In the **Setting Up Physics** tab, click **Cell Zones** (**Zones** group box).

    **Setting Up Physics** → **Zones** → **Cell Zones**

   This opens the **Cell Zone Conditions** task page.



b. Double-click **fluid** in the **Zone** list to open the **Fluid** dialog box.



Note

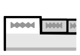
You can also double-click the **Setup/Cell Zone Conditions/fluid** tree item in order to open the corresponding dialog box.

c. Select **water** from the **Material Name** drop-down list.

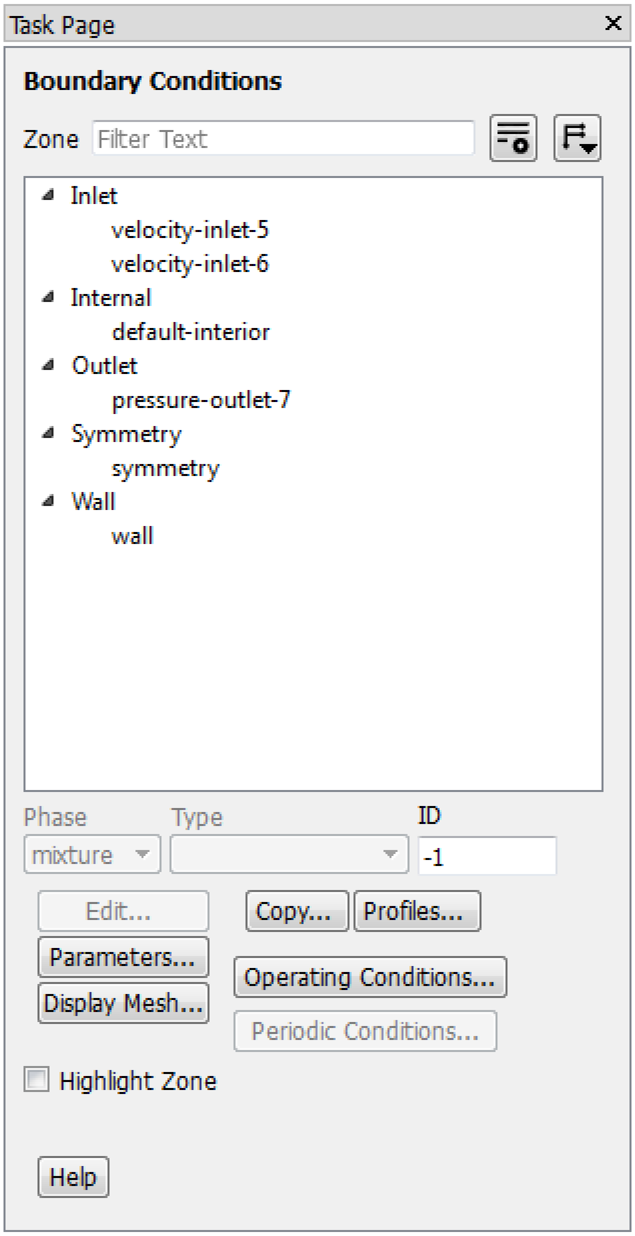
d. Click **OK** to close the **Fluid** dialog box.

5. Set up the boundary conditions for the inlets, outlet, and walls for your CFD analysis using the **Zones** group box of the **Setting Up Physics** ribbon tab.

a. In the **Setting Up Physics** tab, click **Boundaries** (**Zones** group box).

 **Setting Up Physics** → **Zones** → **Boundaries**

This opens the **Boundary Conditions** task page where the boundaries defined in your simulation are displayed in the **Zone** selection list.



Note

To display boundary zones grouped by zone type (as shown previously), click the **Toggle Tree View** button () in the upper right corner of the **Boundary Conditions** task page and select **Zone Type** under **Group By**.

Here the zones have names with numerical identifying tags. It is good practice to give boundaries meaningful names in a meshing application to help when you set up the model. You can also change boundary names in Fluent by simply editing the boundary and making revisions in the **Zone Name** text box.

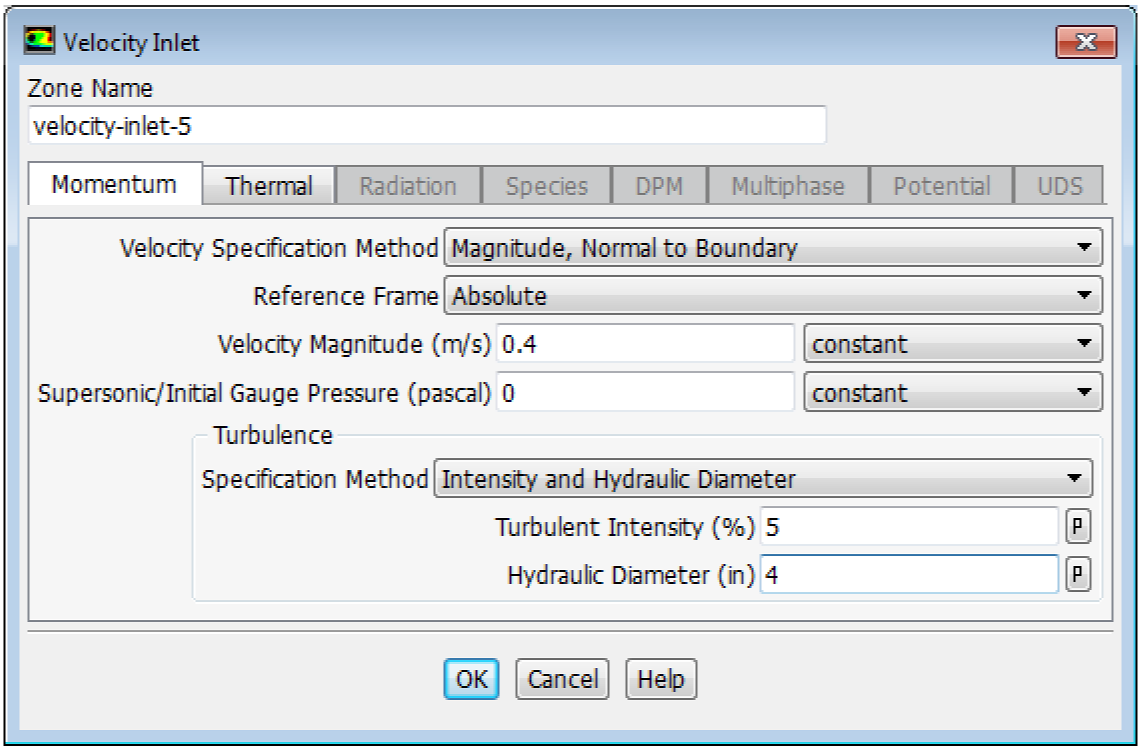
b. Set the boundary conditions at the cold inlet (**velocity-inlet-5**).

Tip

If you are unsure of which inlet zone corresponds to the cold inlet, you can probe the mesh display using the right mouse button or the probe toolbar button () as described previously in this tutorial. The information will be displayed in the ANSYS Fluent console, and the zone you probed will be automatically selected from the **Zone** selection list in the **Boundary Conditions** task page.

---

i. Double-click **velocity-inlet-5** to open the **Velocity Inlet** dialog box.



ii. Retain the default selection of **Magnitude, Normal to Boundary** from the **Velocity Specification Method** drop-down list.

iii. Enter `0.4` [m/s] for **Velocity Magnitude**.

iv. In the **Turbulence** group box, select **Intensity and Hydraulic Diameter** from the **Specification Method** drop-down list.

v. Retain the default value of 5 [%] for **Turbulent Intensity**.

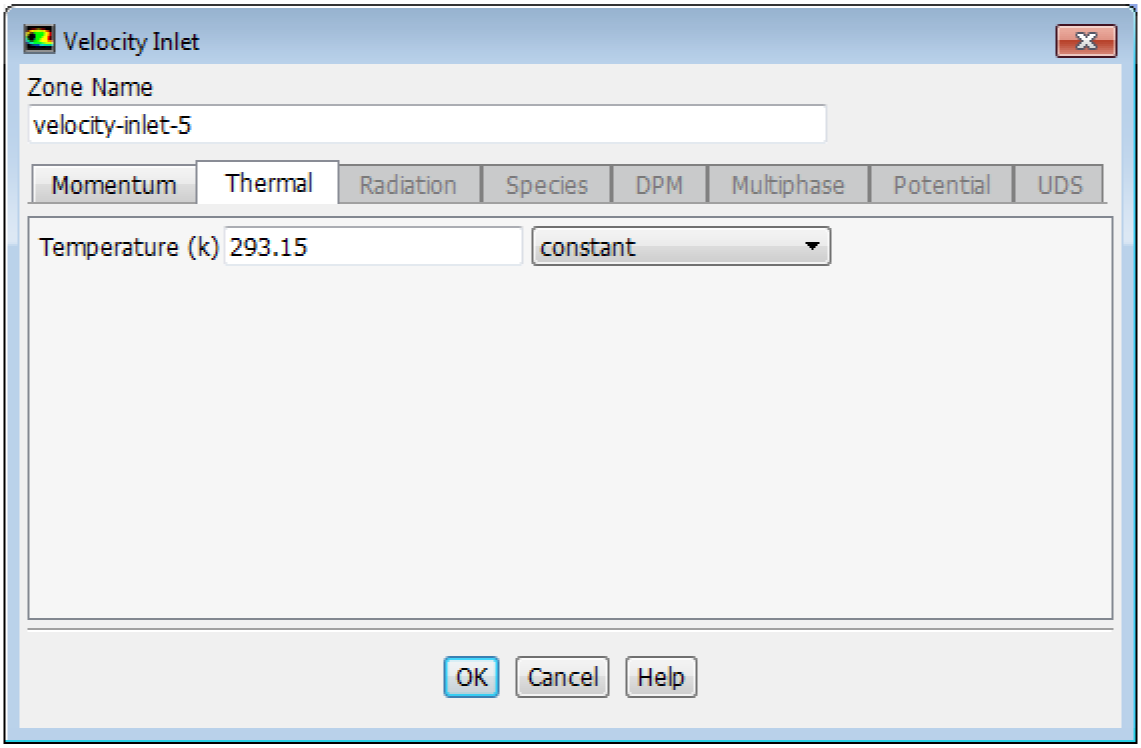
vi. Enter 4 [inches] for **Hydraulic Diameter**.

The hydraulic diameter D_h *is defined as:*

$$D_h = \frac{4A}{P_w}$$

where A is the cross-sectional area and P_w is the wetted perimeter.

vii. Click the **Thermal** tab.



viii. Enter `293.15` [K] for **Temperature**.

ix. Click **OK** to close the **Velocity Inlet** dialog box.

Note

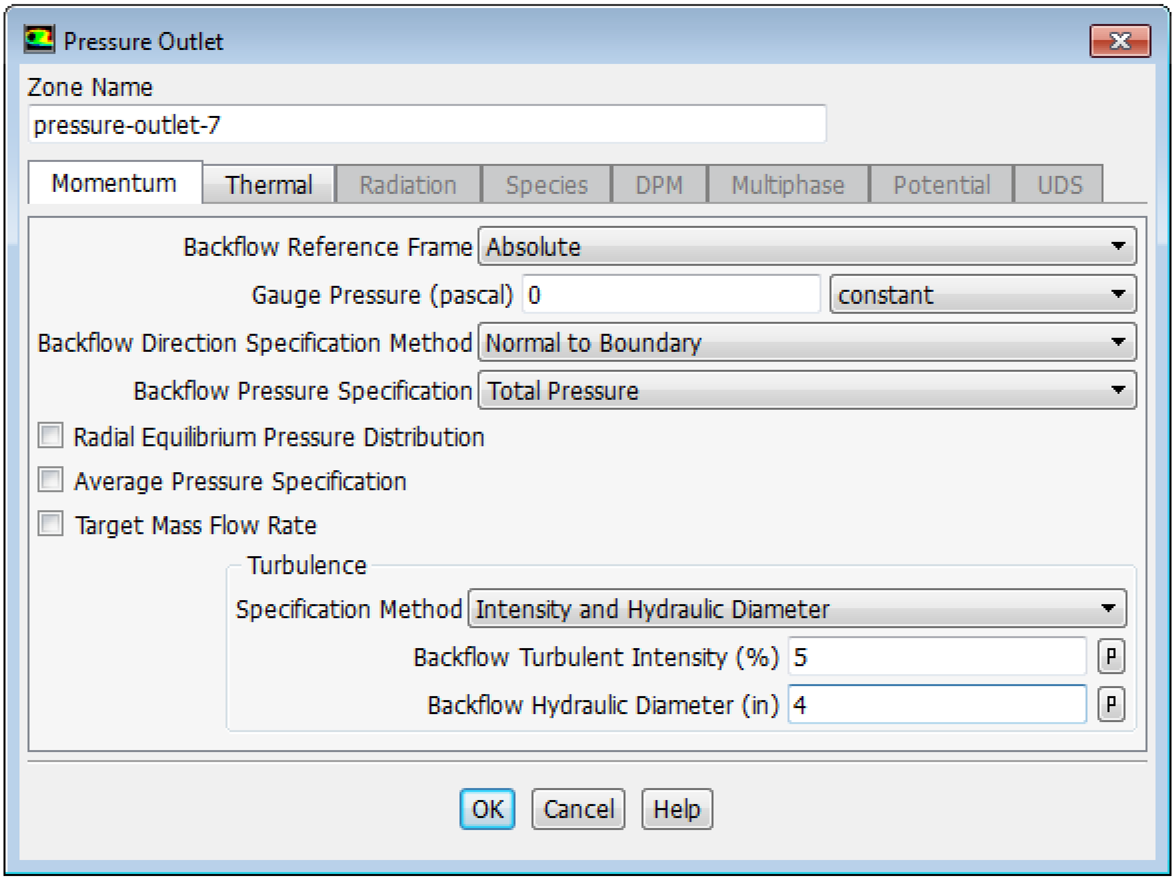
You can also access the **Velocity Inlet** dialog box by double-clicking the **Setup/Boundary Conditions/velocity-inlet-5** tree item.

---

c. In a similar manner, set the boundary conditions at the hot inlet (**velocity-inlet-6**), using the values in the following table:

Setting	Value
Velocity Specification Method	Magnitude, Normal to Boundary
Velocity Magnitude	`1.2` [m/s]
Specification Method	Intensity and Hydraulic Diameter
Turbulent Intensity	5 [%]
Hydraulic Diameter	1 [inch]
Temperature	`313.15` [K]

d. Double-click **pressure-outlet-7** in the **Zone** selection list and set the boundary conditions at the outlet, as shown in the following figure.

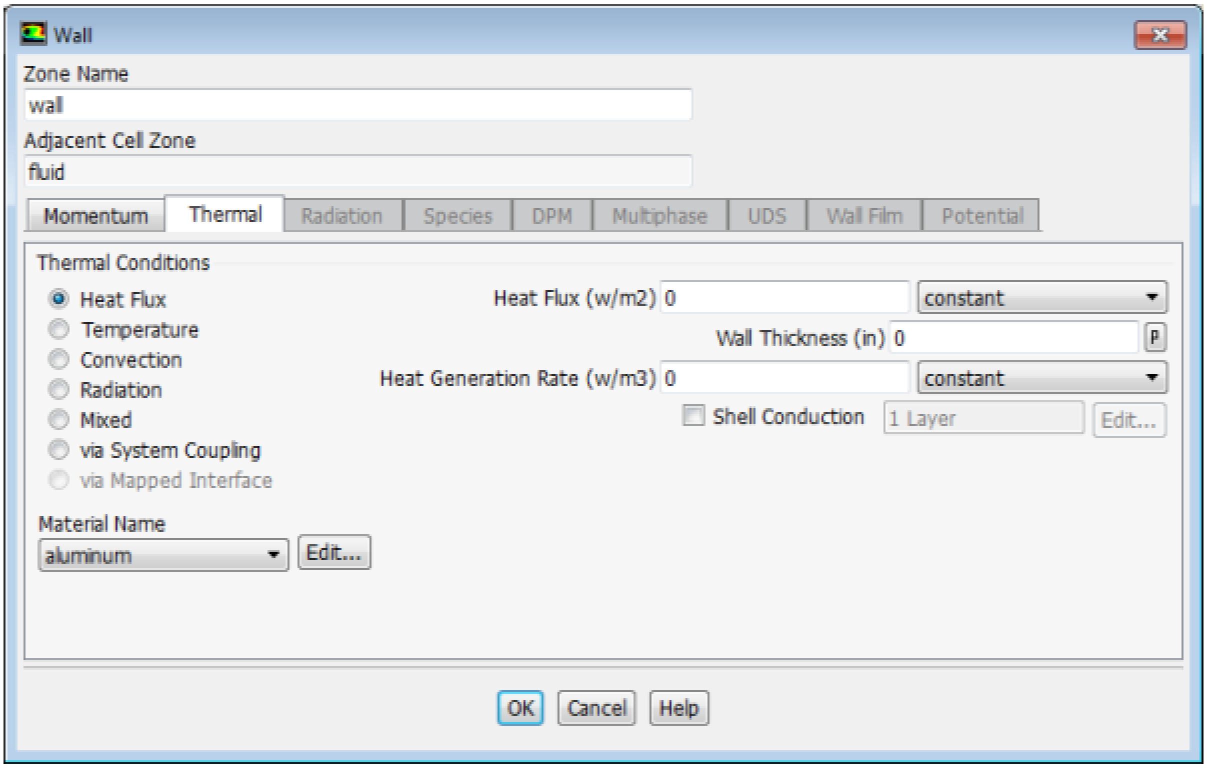


Note

- You do not need to set a backflow temperature in this case (in the **Thermal** tab) because the material properties are not functions of temperature. If they were, a flow-weighted average of the inlet conditions would be a good starting value.
- ANSYS Fluent will use the backflow conditions only if the fluid is flowing into the computational domain through the outlet. Since backflow might occur at some point during the solution procedure, you should set reasonable backflow conditions to prevent convergence from being adversely affected.

---

e. For the wall of the pipe (**wall**), retain the default value of `0` W/m^2 for **Heat Flux** in the **Thermal** tab.



3.4.6. Solving

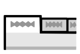
In the steps that follow, you will set up and run the calculation using the **Solving** ribbon tab.

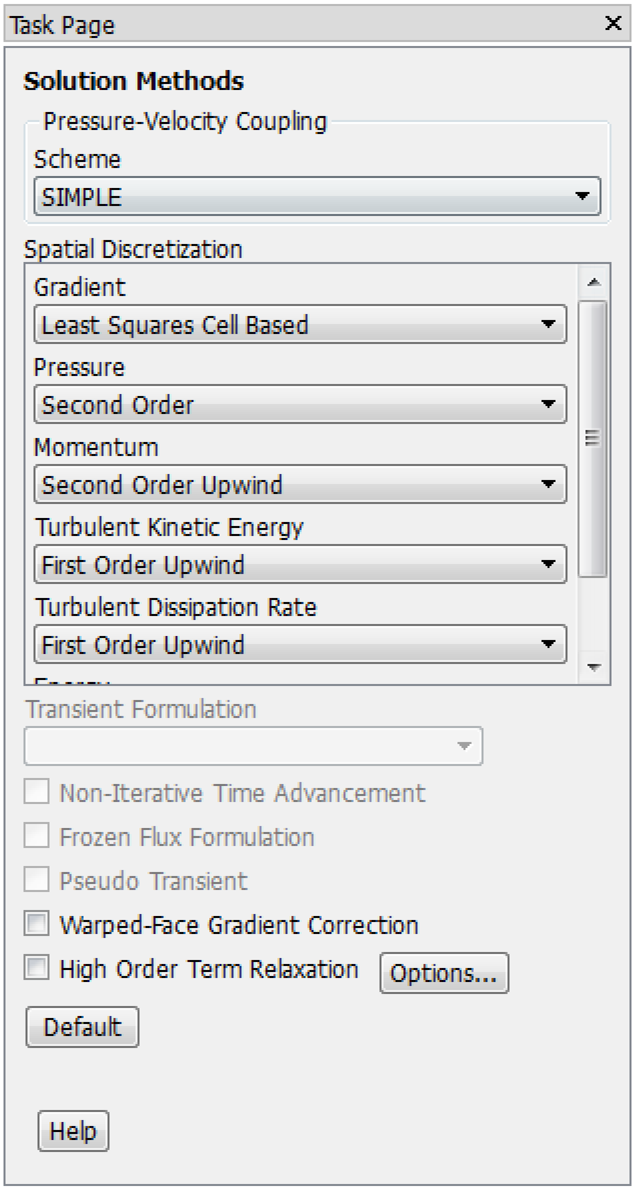
Note

You can also use the task pages listed under the **Solution** tree branch to perform solution-related activities.

1. Select a solver scheme.

   a. In the **Solving** ribbon tab, click **Methods...** (**Solution** group box).

 **Solving** → **Solution** → **Methods...**

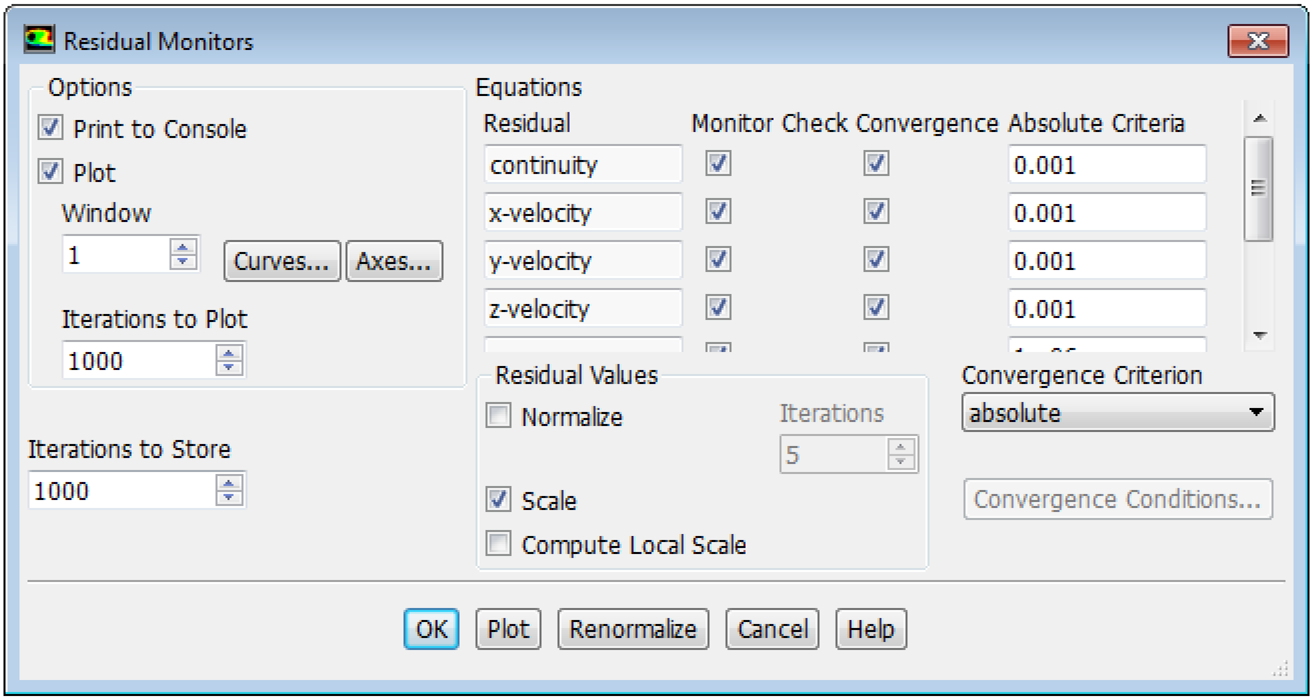


b. In the **Solution Methods** task page, retain the default selections for the **Scheme** and **Spatial Discretization**.

2. Enable the plotting of residuals during the calculation.

a. In the **Solving** ribbon tab, click **Residuals...** (**Reports** group box).

Solving → **Reports** → **Residuals...**



Note

You can also access the **Residual Monitors** dialog box by double-clicking the **Solution/Monitors/Residual** tree item.

---

b. Ensure that **Plot** is enabled in the **Options** group box.

c. Retain the default value of `0.001` for the **Absolute Criteria** of **continuity**.

d. Click **OK** to close the **Residual Monitors** dialog box.

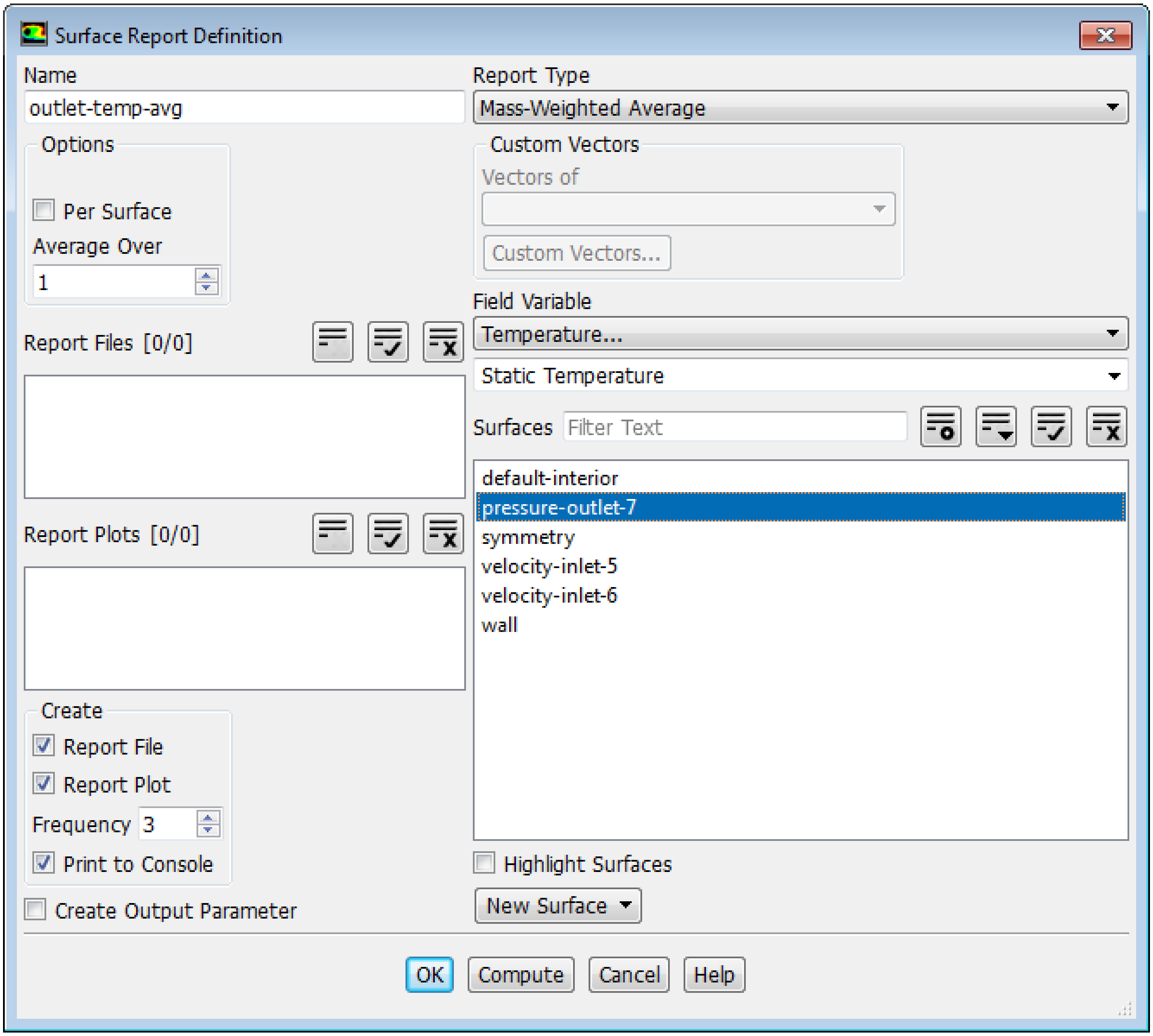
Note

By default, the residuals of all of the equations solved for the physical models enabled for your case will be monitored and checked by ANSYS Fluent as a means to determine the convergence of the solution. It is a good practice to also create and plot a surface report definition that can help evaluate whether the solution is truly converged. You will do this in the next step.

---

3. Create a surface report definition of average temperature at the outlet (**pressure-outlet-7**).

Solving → **Reports** → **Definitions** → **New** → **Surface Report** → **Mass-Weighted Average...**



Note

You can also access the **Surface Report Definition** dialog box by right-clicking **Report Definitions** in the tree (under **Solution**) and selecting **New/Surface Report/Mass-Weighted Average...** from the menu that opens.

a. Enter `outlet-temp-avg` for the **Name** of the report definition.

b. Enable **Report File**, **Report Plot**, and **Print to Console** in the **Create** group box.

During a solution run, ANSYS Fluent will write solution convergence data in a report file, plot the solution convergence history in a graphics window, and print the value of the report definition to the console.

c. Set **Frequency** to 3 by clicking the up-arrow button.

This setting instructs ANSYS Fluent to update the plot of the surface report, write data to a file, and print data in the console after every 3 iterations during the solution.

d. Select **Temperature...** and **Static Temperature** from the **Field Variable** drop-down lists.

e. Select **pressure-outlet-7** from the **Surfaces** selection list.

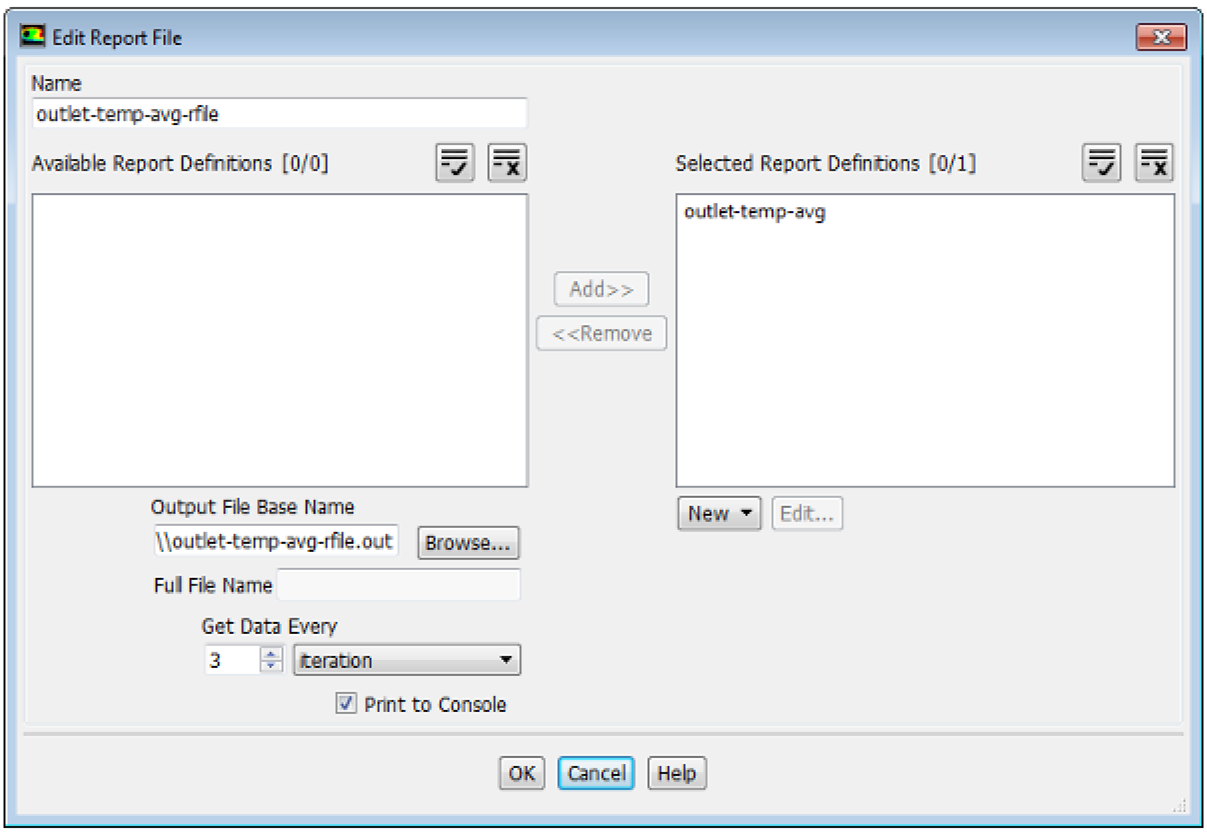
f. Click **OK** to save the surface report definition and close the **Surface Report Definition** dialog box.

The new surface report definition **outlet-temp-avg** will appear under the **Solution/Report Definitions** tree item. ANSYS Fluent also automatically creates the following items:

- **outlet-temp-avg-rfile** (under the **Solution/Monitors/Report Files** tree branch)

- **outlet-temp-avg-rplot** (under the **Solution/Monitors/Report Plots** tree branch)

4. In the tree, double-click **outlet-temp-avg-rfile** (under **Solution/Monitors/Report Files**) and examine the report file settings in the **Edit Report File** dialog box.



The dialog box is automatically populated with data from the **outlet-temp-avg** report definition.

a. Verify that **outlet-temp-avg** is in the **Selected Report Definitions** list.

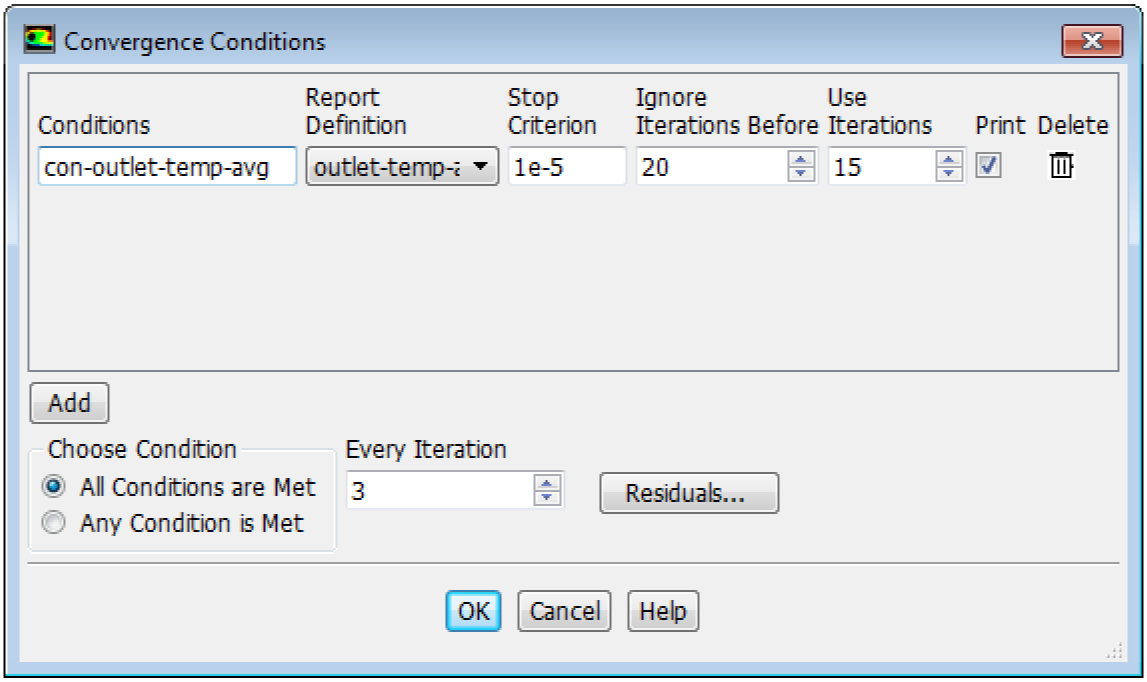
If you had created multiple report definitions, the additional ones would be listed under **Available Report Definitions***, and you could use the* **Add>>** *and* **<<Remove** *buttons to manage which were written in this particular report definition file.*

b. (optional) Edit the name and location of the resulting file as necessary using the **Output File Base Name** field or **Browse...** button.

c. Click **OK** to close the **Edit Report File** dialog box.

5. Create a convergence condition for **outlet-temp-avg**.

Solving → **Reports** → **Convergence...**



a. Click the **Add** button.

b. Enter `con-outlet-temp-avg` for **Conditions**.

c. Select **outlet-temp-avg** from the **Report Definition** drop-down list.

d. Enter `1e-5` for **Stop Criterion**.

e. Enter `20` for **Ignore Iterations Before**.

f. Enter `15` for **Use Iterations**.

g. Enable **Print**.

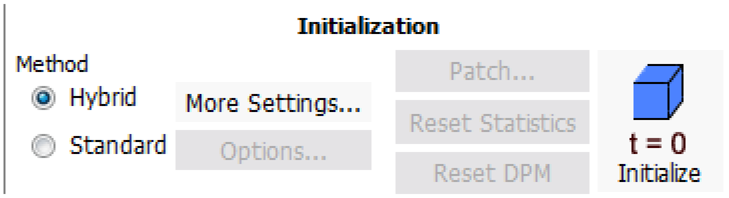
h. Set **Every Iteration** to 3.

i. Click **OK** to save the convergence condition settings and close the **Convergence Conditions** dialog box.

These settings will cause Fluent to consider the solution converged when the surface report definition value for each of the previous 15 iterations is within 0.001% of the current value. Convergence of the values will be checked every 3 iterations. The first 20 iterations will be ignored, allowing for any initial solution dynamics to settle out. Note that the value printed to the console is the deviation between the current and previous iteration values only.

6. Initialize the flow field using the **Initialization** group box of the **Solving** ribbon tab.

Solving → Initialization

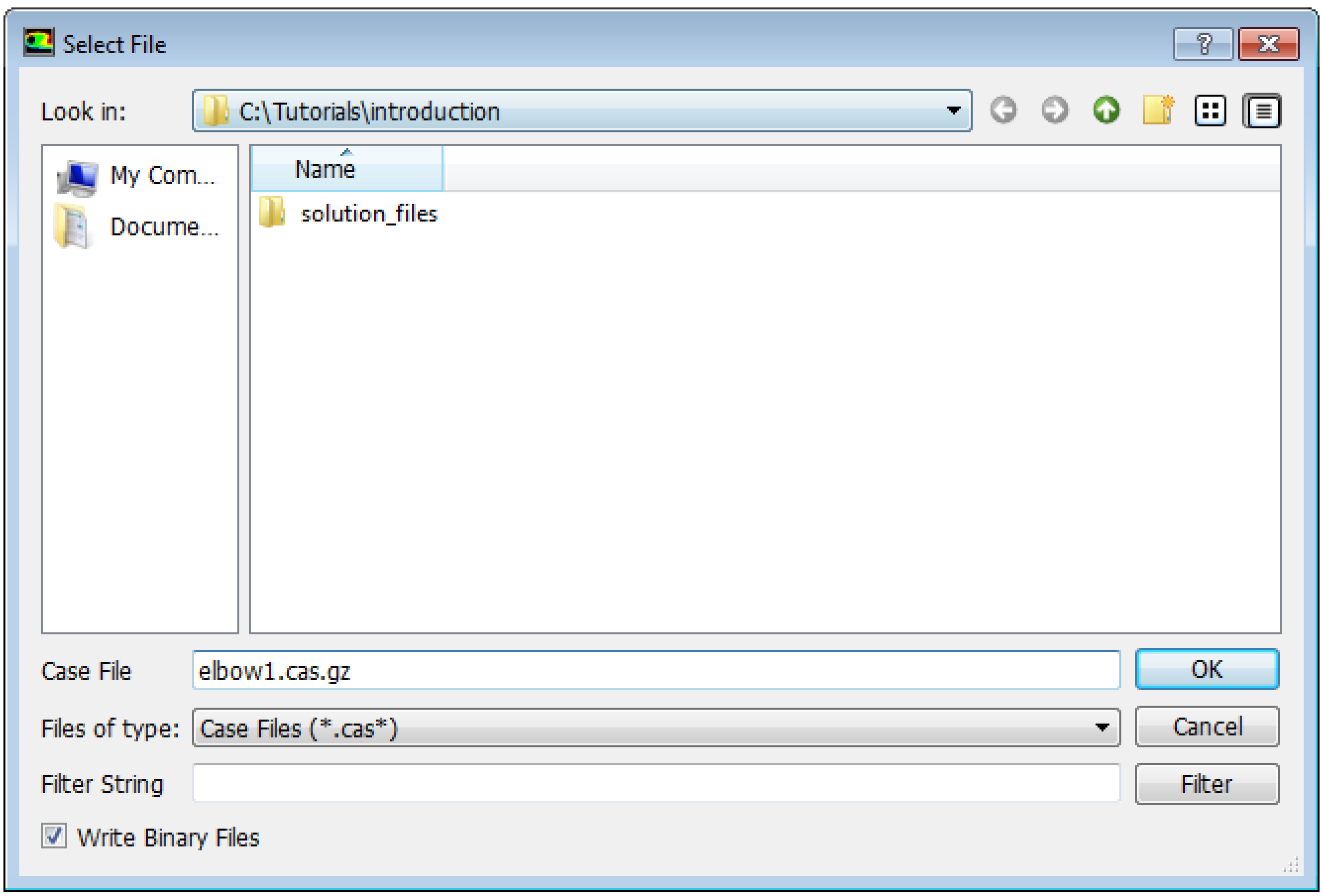


a. Retain the default selection of **Hybrid** from the **Method** list.

b. Click **Initialize**.

7. Save the case file (`elbow1.cas.gz`).

File → Write → Case...



a. (optional) Indicate the folder in which you would like the file to be saved.

By default, the file will be saved in the folder from which you read in `elbow.msh` *(that is, the* `introduction` *folder). You can indicate a different folder by browsing to it or by creating a new folder.*

b. Enter `elbow1.cas.gz` for **Case File**.

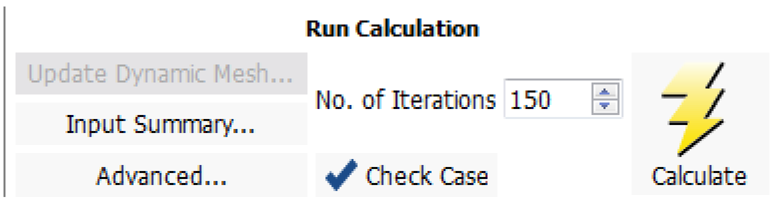
Adding the extension `.gz` to the end of the file name extension instructs ANSYS Fluent to save the file in a compressed format. You do not have to include `.cas` in the extension (for example, if you enter `elbow1.gz`, ANSYS Fluent will automatically save the file as `elbow1.cas.gz`). The `.gz` extension can also be used to save data files in a compressed format.

c. Ensure that the default **Write Binary Files** option is enabled, so that a binary file will be written.

d. Click **OK** to save the case file and close the **Select File** dialog box.

8. Start the calculation by requesting 150 iterations in the **Solving** ribbon tab (**Run Calculation** group box).

Solving → **Run Calculation**



a. Enter `150` for **No. of Iterations**.

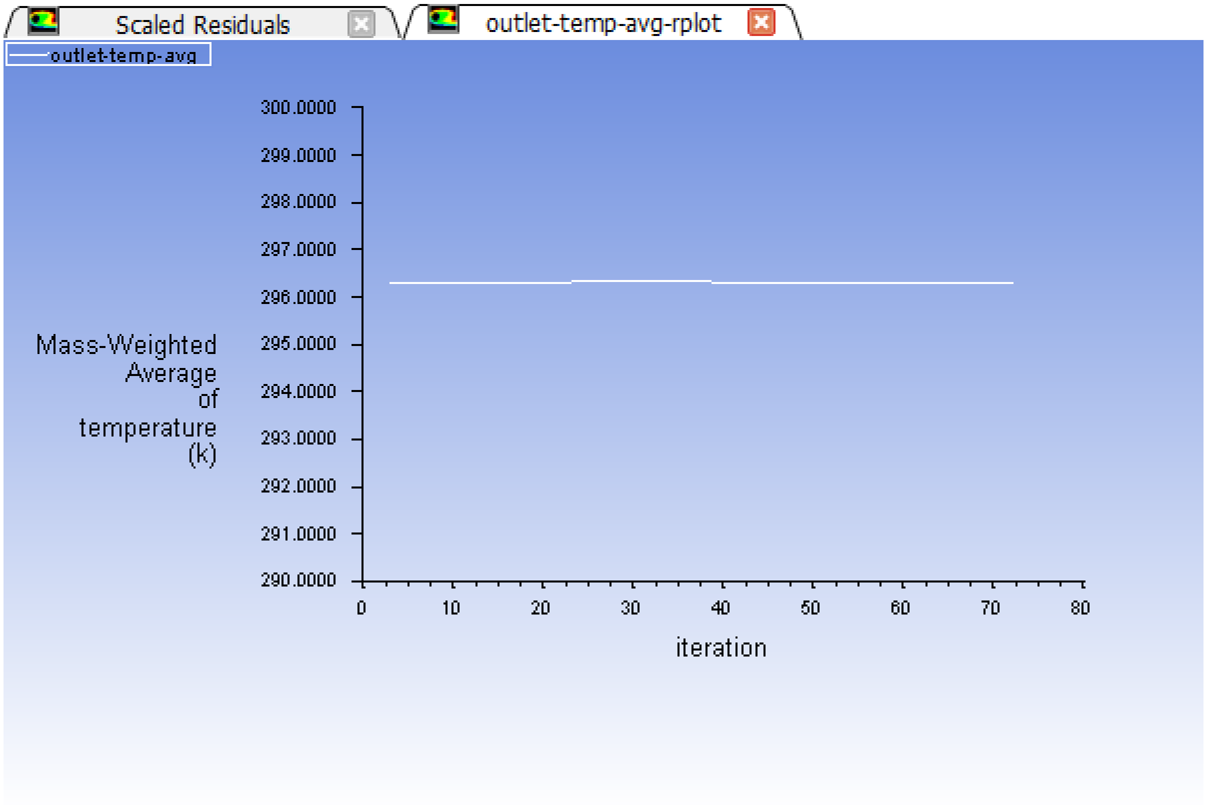
b. Click **Calculate**.

Note

By starting the calculation, you are also starting to save the surface report data at the rate specified in the **Surface Report Definition** dialog box. If a file already exists in your working directory with the name you specified in the **Edit Report File** dialog box, then a **Question** dialog box will open, asking if you would like to append the new data to the existing file. Click **No** in the **Question** dialog box, and then click **OK** in the **Warning** dialog box that follows to overwrite the existing file.

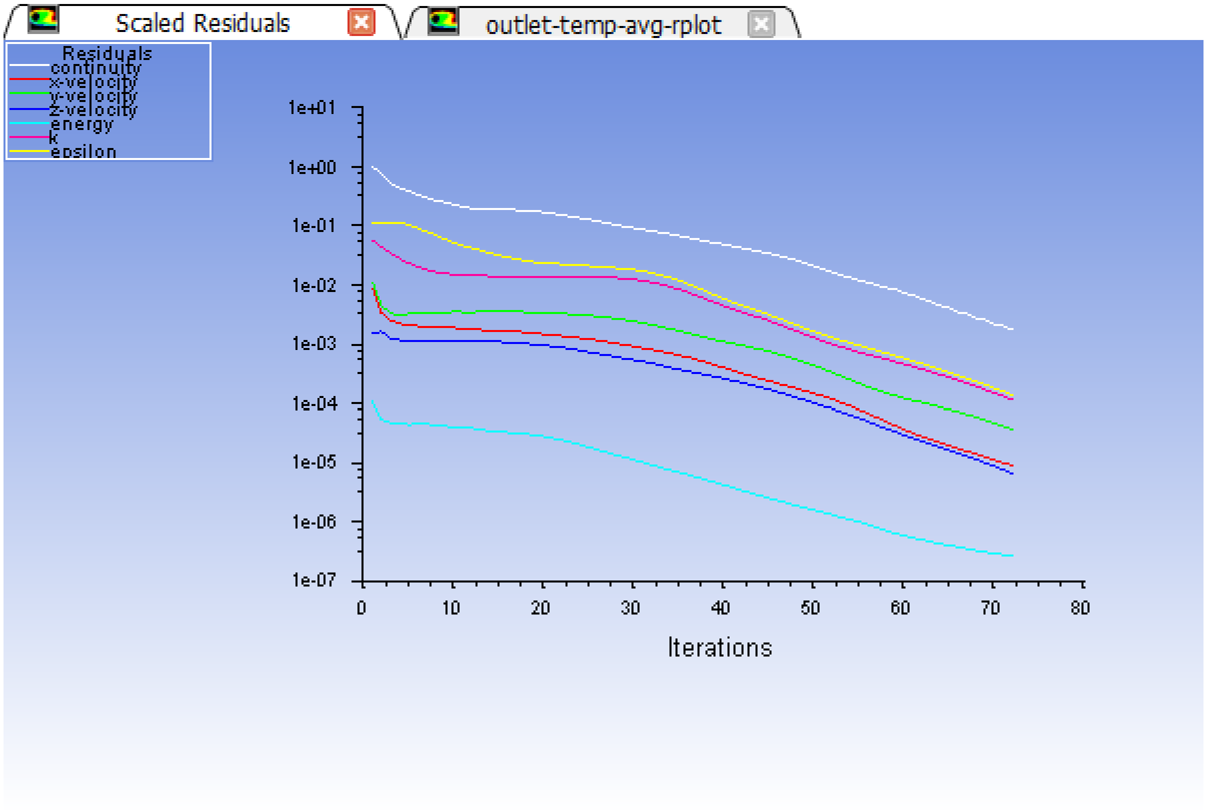
As the calculation progresses, the surface report history will be plotted in the **outlet-temp-avg-rplot** tab in the graphics window (Figure 3.3: Convergence History of the Mass-Weighted Average Temperature (p. 153)).

Figure 3.3: Convergence History of the Mass-Weighted Average Temperature



Similarly, the residuals history will be plotted in the **Scaled Residuals** tab in the graphics window (Figure 3.4: Residuals (p. 154)).

Figure 3.4: Residuals



Note

You can monitor the two convergence plots simultaneously by right-clicking a tab in the graphics window and selecting **SubWindow View** from the menu that opens. To return to a tabbed graphics window view, right-click a graphics window title area and select **Tabbed View**.

Since the residual values vary slightly by platform, the plot that appears on your screen may not be exactly the same as the one shown here.

The solution will be stopped by ANSYS Fluent when any of the following occur:

- the surface report definition converges to within the tolerance specified in the **Convergence Conditions** dialog box
- the residual monitors converge to within the tolerances specified in the **Residual Monitors** dialog box
- the number of iterations you requested in the **Run Calculation** task page has been reached

In this case, the solution is stopped when the convergence criterion on outlet temperature is satisfied, after approximately 72 iterations. The exact number of iterations for convergence will vary, depending on the platform being used. An **Information** dialog box will open to alert you that the calculation is complete. Click **OK** in the **Information** dialog box to proceed.

9. Examine the plots for convergence (Figure 3.3: Convergence History of the Mass-Weighted Average Temperature (p. 153) and Figure 3.4: Residuals (p. 154)).

Note

There are no universal metrics for judging convergence. Residual definitions that are useful for one class of problem are sometimes misleading for other classes of problems. Therefore it is a good idea to judge convergence not only by examining residual levels, but also by monitoring relevant integrated quantities and checking for mass and energy balances.

There are three indicators that convergence has been reached:

- The residuals have decreased to a sufficient degree.

  The solution has converged when the **Convergence Criterion** for each variable has been reached. The default criterion is that each residual will be reduced to a value of less than 10^{-3}, except the **energy** residual, for which the default criterion is 10^{-6}.

- The solution no longer changes with more iterations.

  Sometimes the residuals may not fall below the convergence criterion set in the case setup. However, monitoring the representative flow variables through iterations may show that the residuals have stagnated and do not change with further iterations. This could also be considered as convergence.

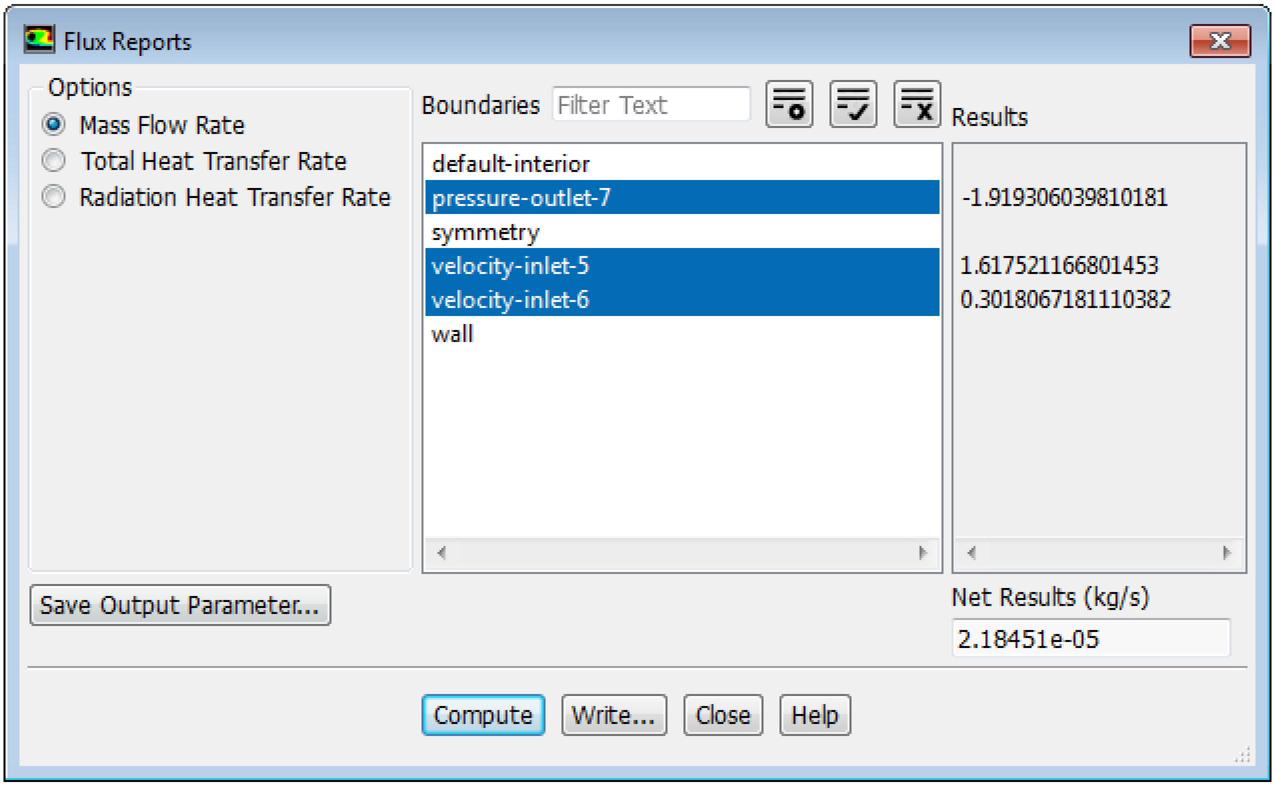
- The overall mass, momentum, energy, and scalar balances are obtained.

  You can examine the overall mass, momentum, energy and scalar balances in the **Flux Reports** dialog box. The net imbalance should be less than 0.2 % of the net flux through the domain when the solution has converged. In the next step you will check to see if the mass balance indicates convergence.

---

10. Examine the mass flux report for convergence using the **Postprocessing** ribbon tab.

Postprocessing → Reports → Fluxes...



a. Ensure that **Mass Flow Rate** is selected from the **Options** list.

b. Select **pressure-outlet-7**, **velocity-inlet-5**, and **velocity-inlet-6** from the **Boundaries** selection list.

c. Click **Compute**.

*The individual and net results of the computation will be displayed in the **Results** and **Net Results** boxes, respectively, in the **Flux Reports** dialog box, as well as in the console.*

The sum of the flux for the inlets should be very close to the sum of the flux for the outlets. The net results show that the imbalance in this case is well below the 0.2% criterion suggested previously.

d. Close the **Flux Reports** dialog box.

11. Save the data file (`elbow1.dat.gz`).

File → Write → Data...

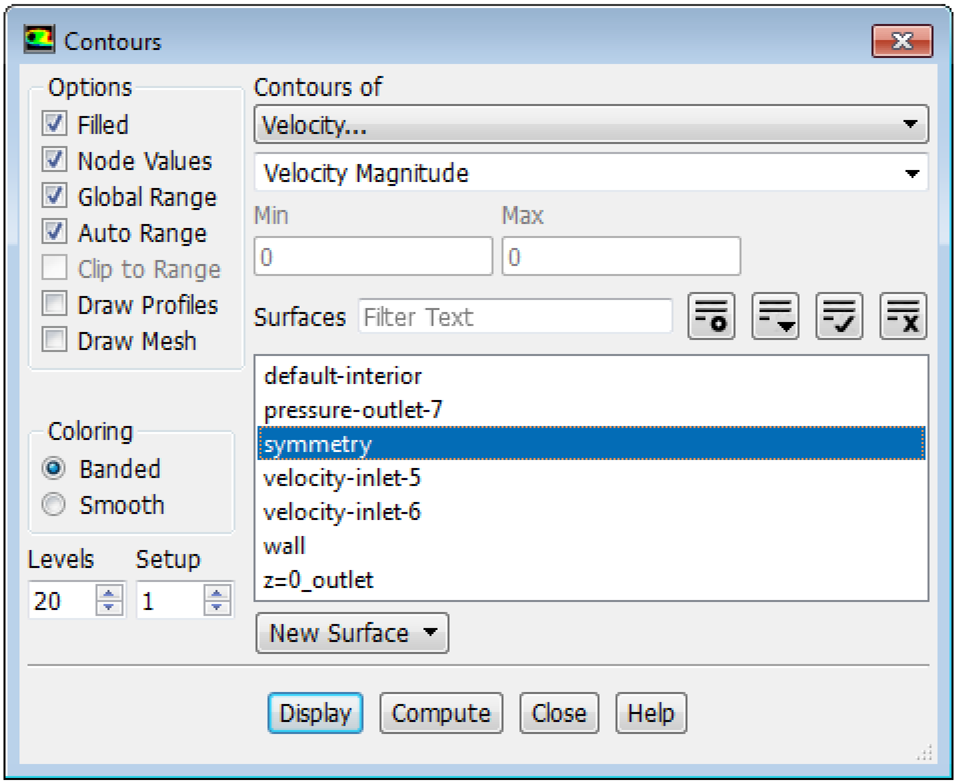
In later steps of this tutorial you will save additional case and data files with different suffixes.

3.4.7. Displaying the Preliminary Solution

In the steps that follow, you will visualize various aspects of the flow for the preliminary solution using the **Postprocessing** ribbon tab.

1. Display filled contours of velocity magnitude on the symmetry plane (Figure 3.5: Predicted Velocity Distribution after the Initial Calculation (p. 158)).

Postprocessing → Graphics → Contours → Edit...



a. Enable **Filled** in the **Options** group box.

b. Ensure that **Node Values** is enabled in the **Options** group box.

c. Select **Velocity...** and **Velocity Magnitude** from the **Contours of** drop-down lists.

d. Select **symmetry** from the **Surfaces** selection list.

e. Click **Display** to display the contours in the active graphics window. Clicking the **Fit to Window** icon () will cause the object to fit exactly and be centered in the window.

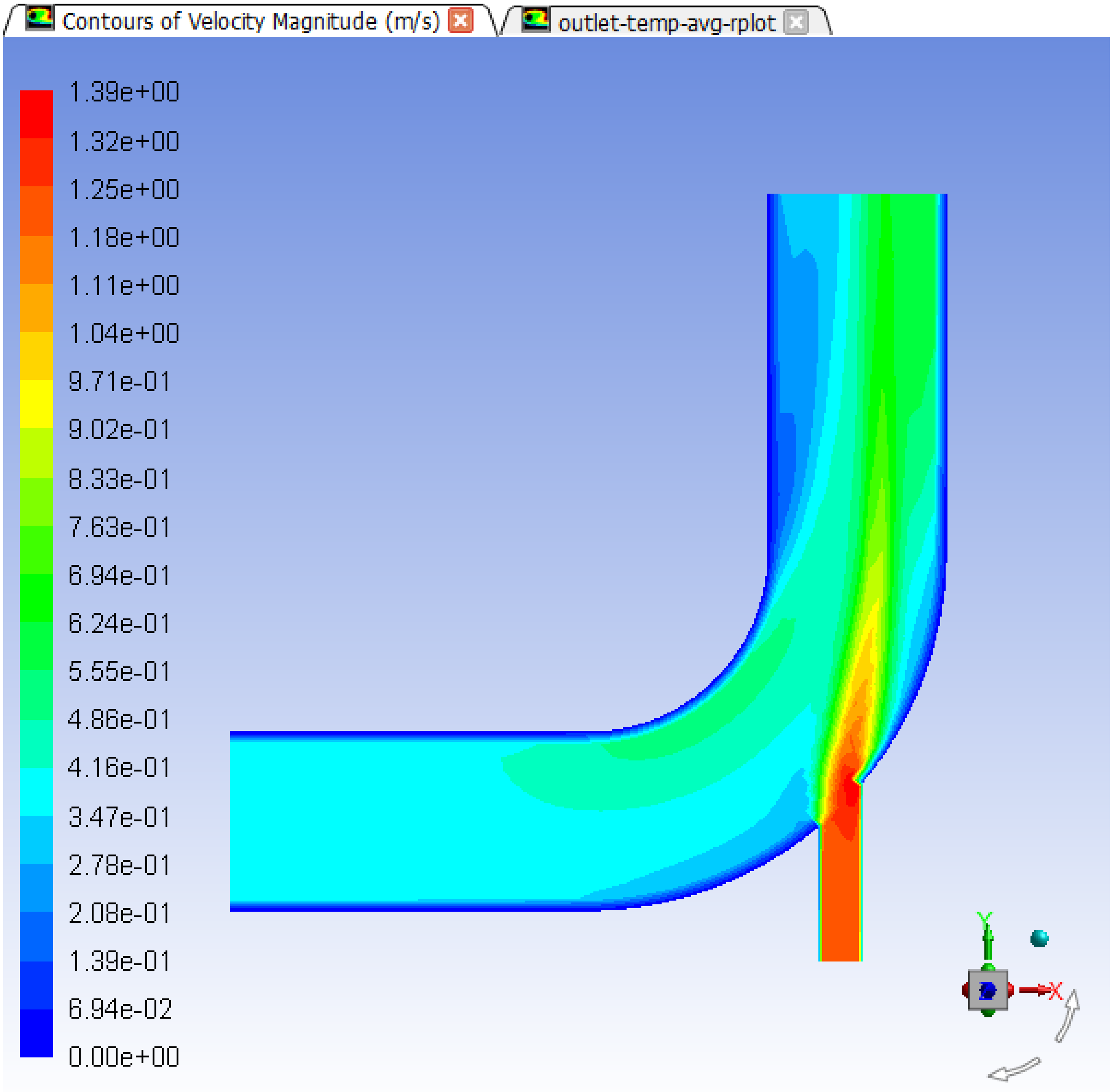
Note

If you cannot see the velocity contour display, select the appropriate tab in the graphics window.

f. Close the **Contours** dialog box.

g. Disable the **Lighting** option in the **Viewing** ribbon tab (in the **Display** group box).

Figure 3.5: Predicted Velocity Distribution after the Initial Calculation



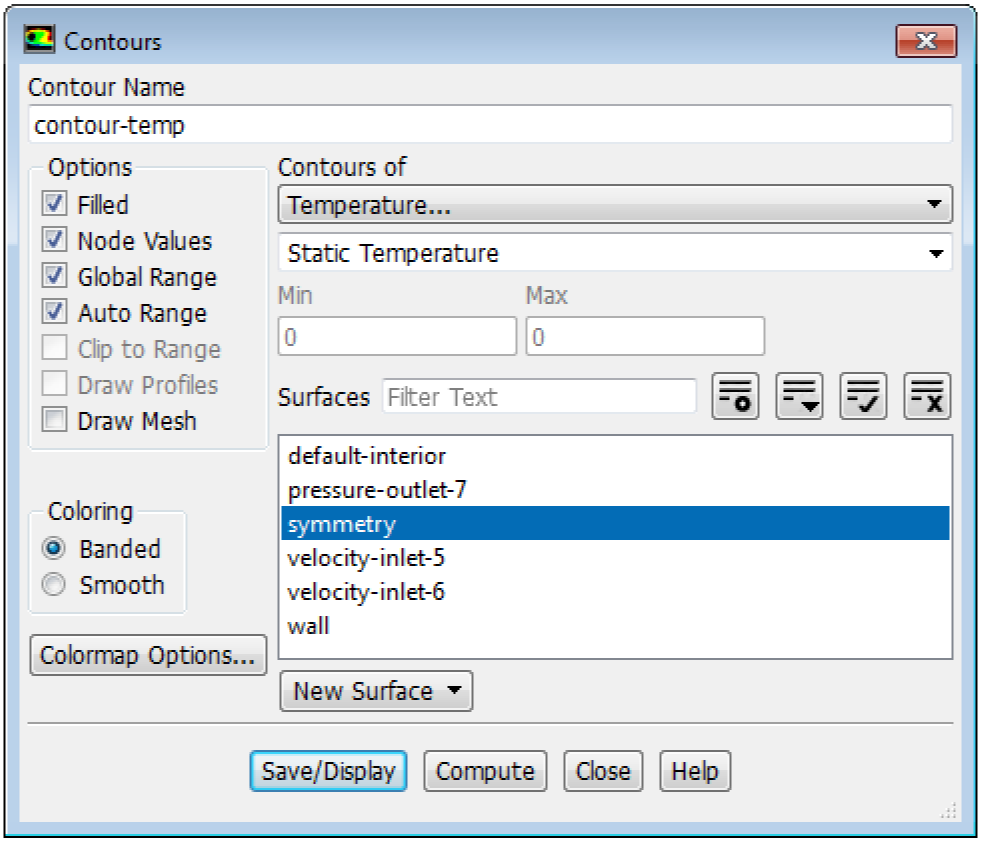
Extra

When you probe a point in the displayed domain with the right mouse button or the probe tool, the level of the corresponding contour is highlighted in the colormap in the graphics window, and is also reported in the console.

2. Create and display a definition for temperature contours on the symmetry plane (Figure 3.6: Predicted Temperature Distribution after the Initial Calculation (p. 160)).

   Postprocessing → Graphics → Contours → New...

   You can create contour definitions and save them for later use.



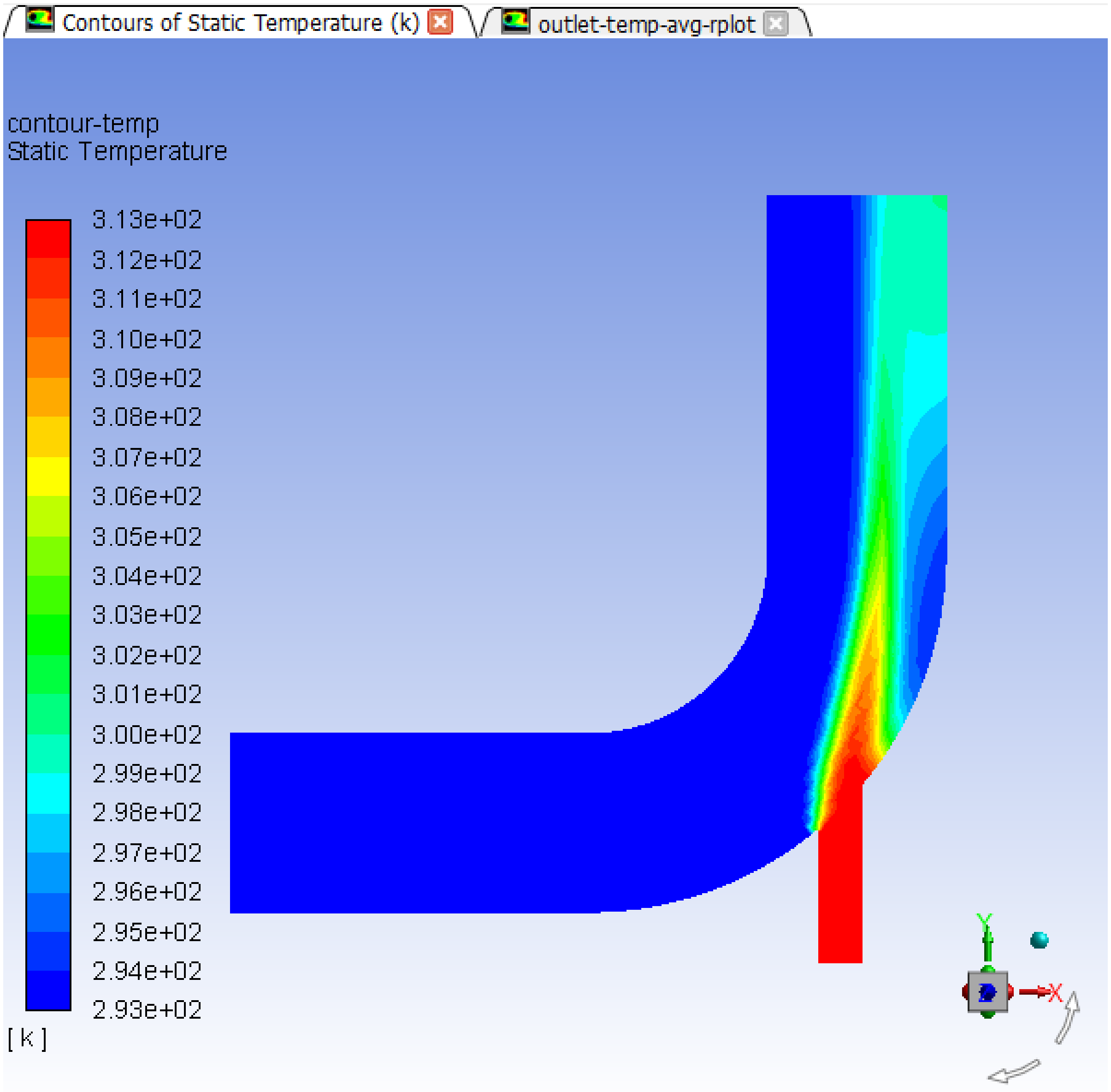
a. Enter `contour-temp` for **Contour Name**.

b. Select **Temperature...** and **Static Temperature** from the **Contours of** drop-down lists.

c. Click **Save/Display** and close the **Contours** dialog box.

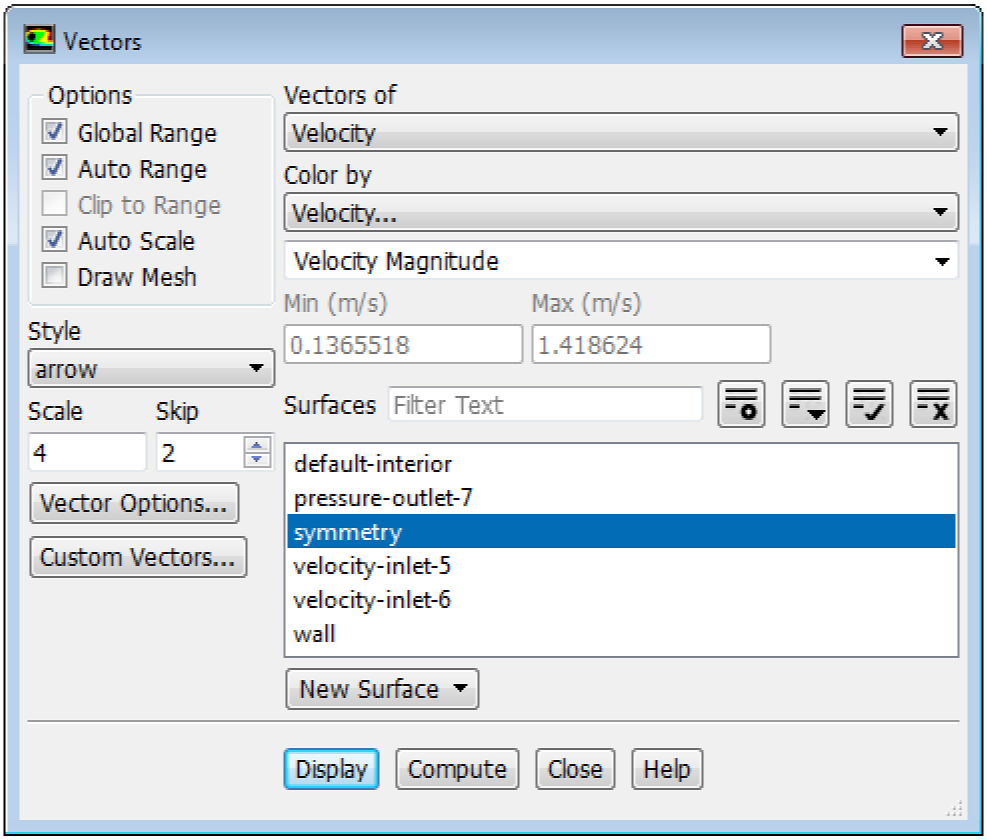
   The new **contour-temp** definition appears under the **Results/Graphics/Contours** tree branch. To edit your contour definition, right-click it and select **Edit...** from the menu that opens.

Figure 3.6: Predicted Temperature Distribution after the Initial Calculation



3. Display velocity vectors on the symmetry plane (Figure 3.9: Magnified View of Resized Velocity Vectors (p. 164)).

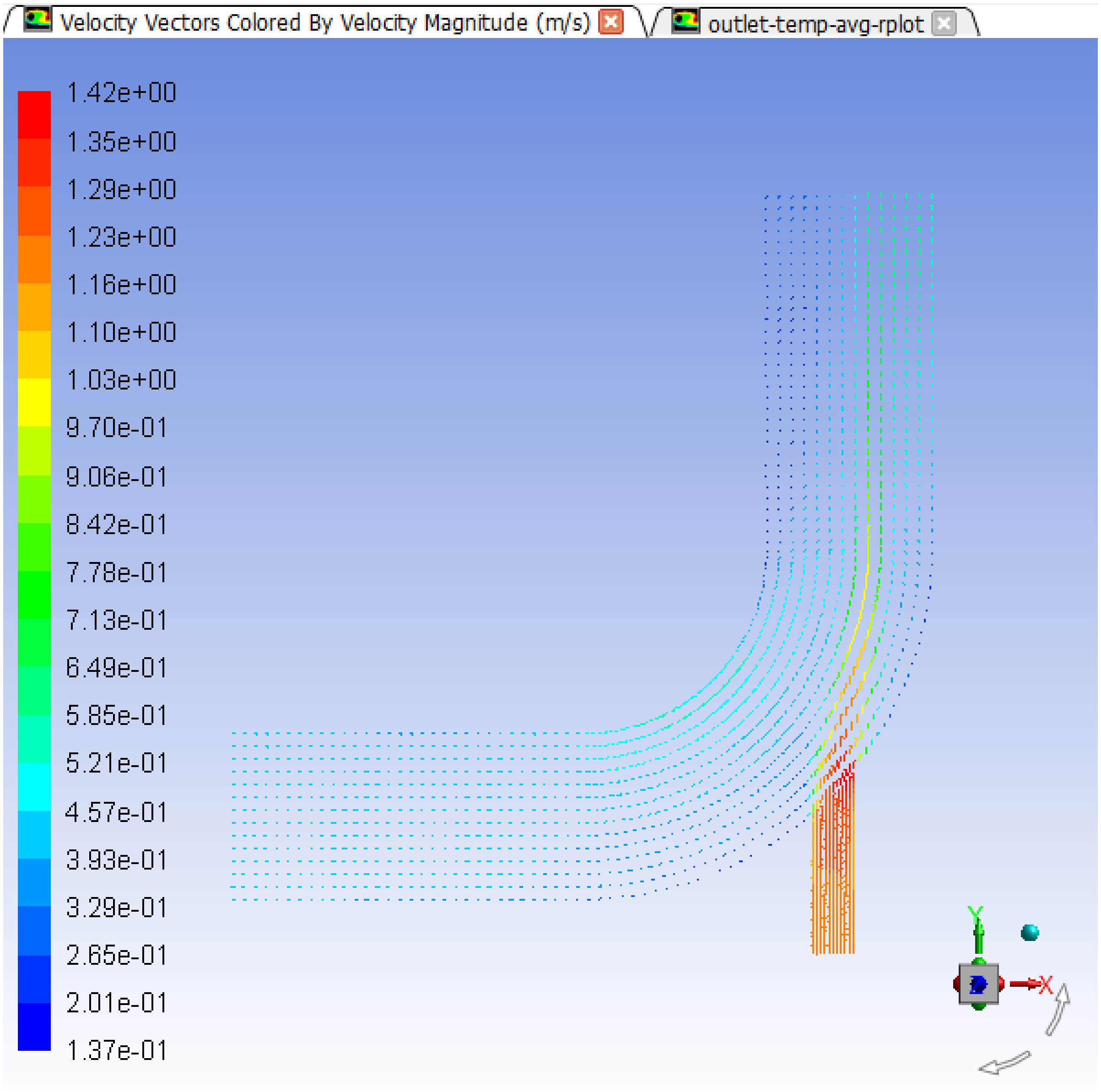
   Postprocessing → Graphics → Vectors → Edit...



a. Select **symmetry** from the **Surfaces** selection list.

b. Click **Display** to plot the velocity vectors.

Figure 3.7: Velocity Vectors Colored by Velocity Magnitude



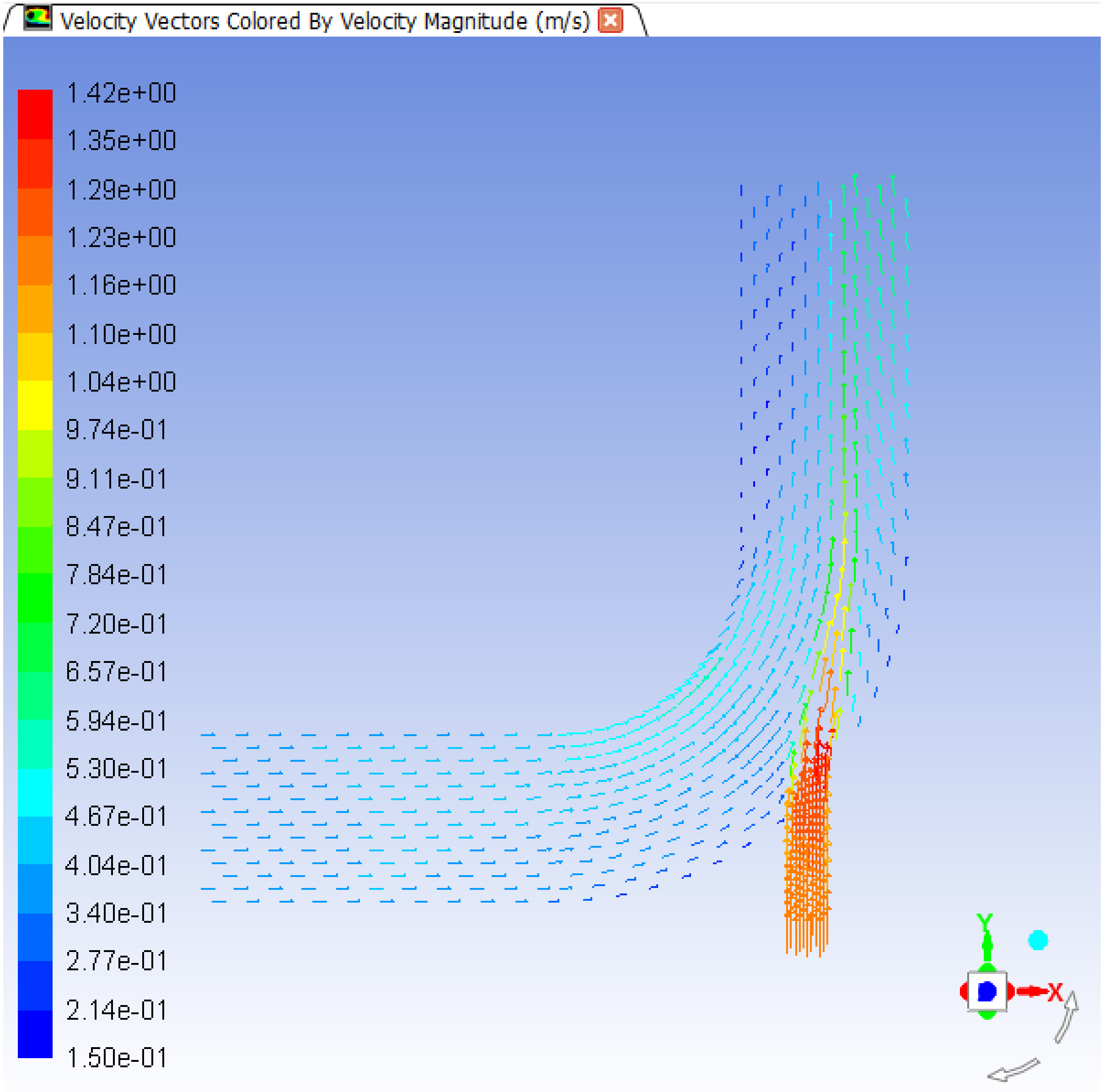
The **Auto Scale** option is enabled by default in the **Options** group box. This scaling sometimes creates vectors that are too small or too large in the majority of the domain. You can improve the clarity by adjusting the **Scale** and **Skip** settings, thereby changing the size and number of the vectors when they are displayed.

c. Enter 4 for **Scale**.

d. Set **Skip** to 2.

e. Click **Display** again to redisplay the vectors.

Figure 3.8: Resized Velocity Vectors

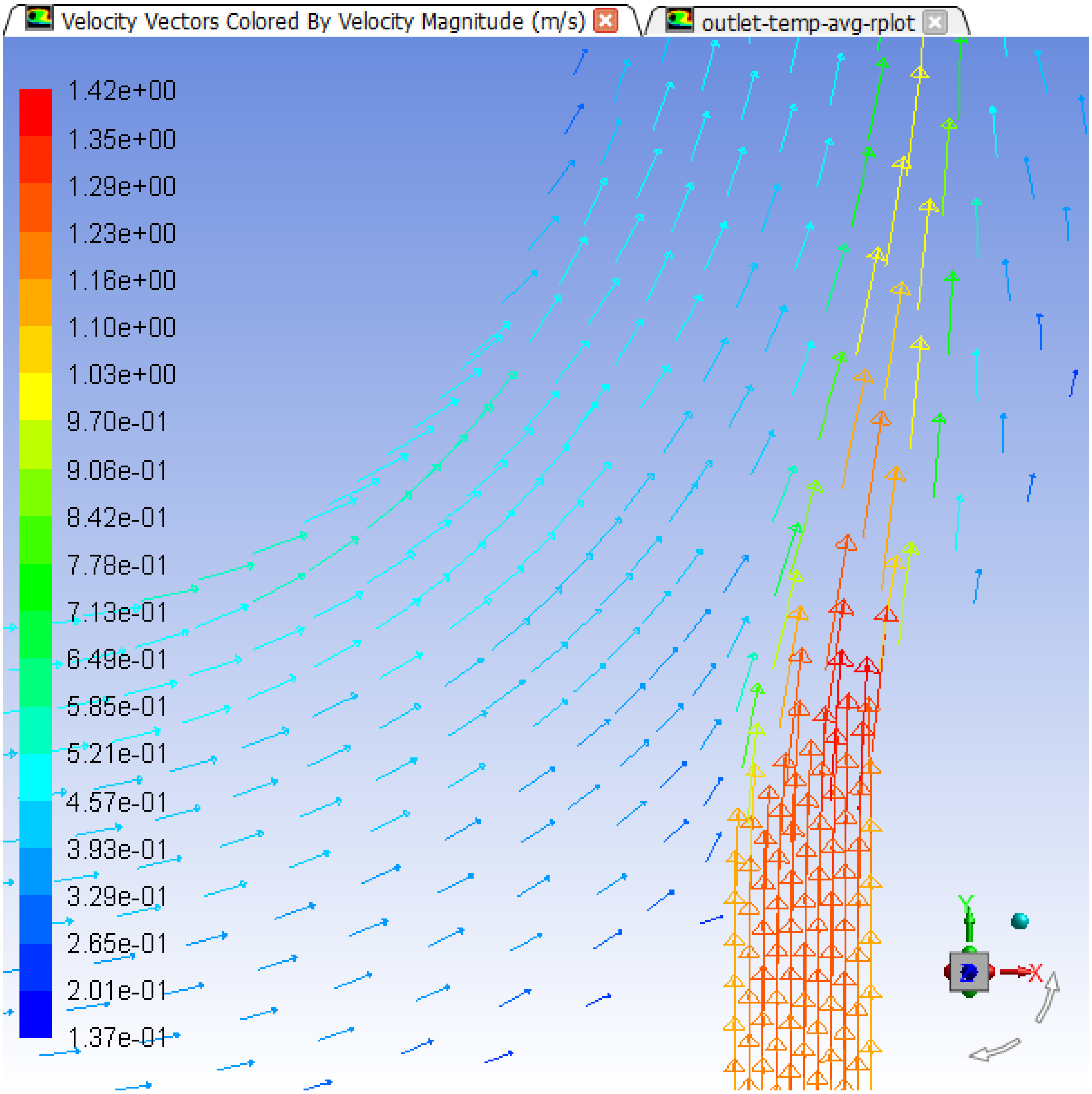


f. Close the **Vectors** dialog box.

g. Zoom in on the vectors in the display.

To manipulate the image, refer to Table 3.1: View Manipulation Instructions (p. 128). The image will be redisplayed at a higher magnification (Figure 3.9: Magnified View of Resized Velocity Vectors (p. 164)).

Figure 3.9: Magnified View of Resized Velocity Vectors

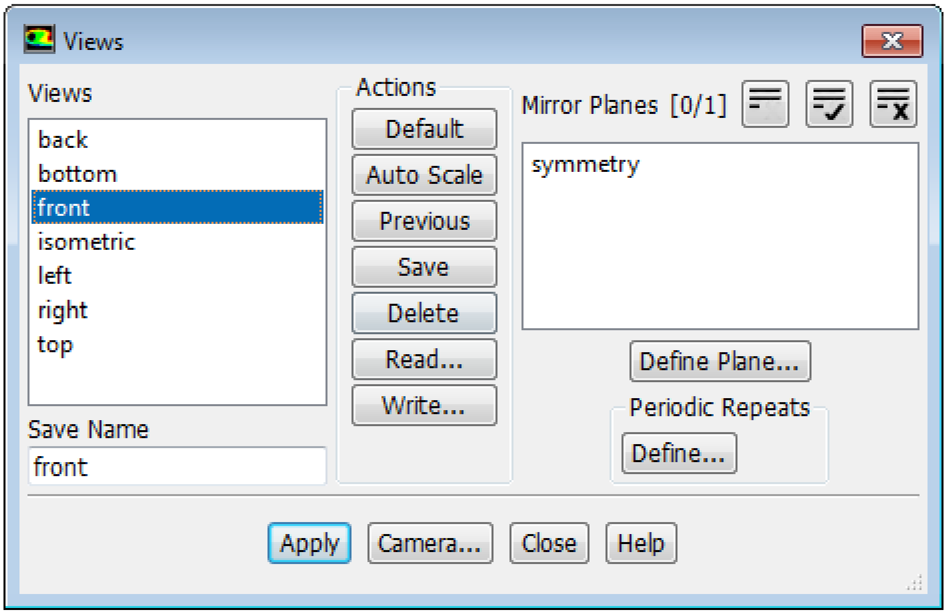


h. Zoom out to the original view.

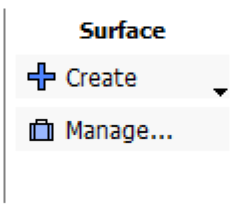
You also have the option of selecting the original view in the **Views** dialog box:

Viewing → **Display** → **Views...**

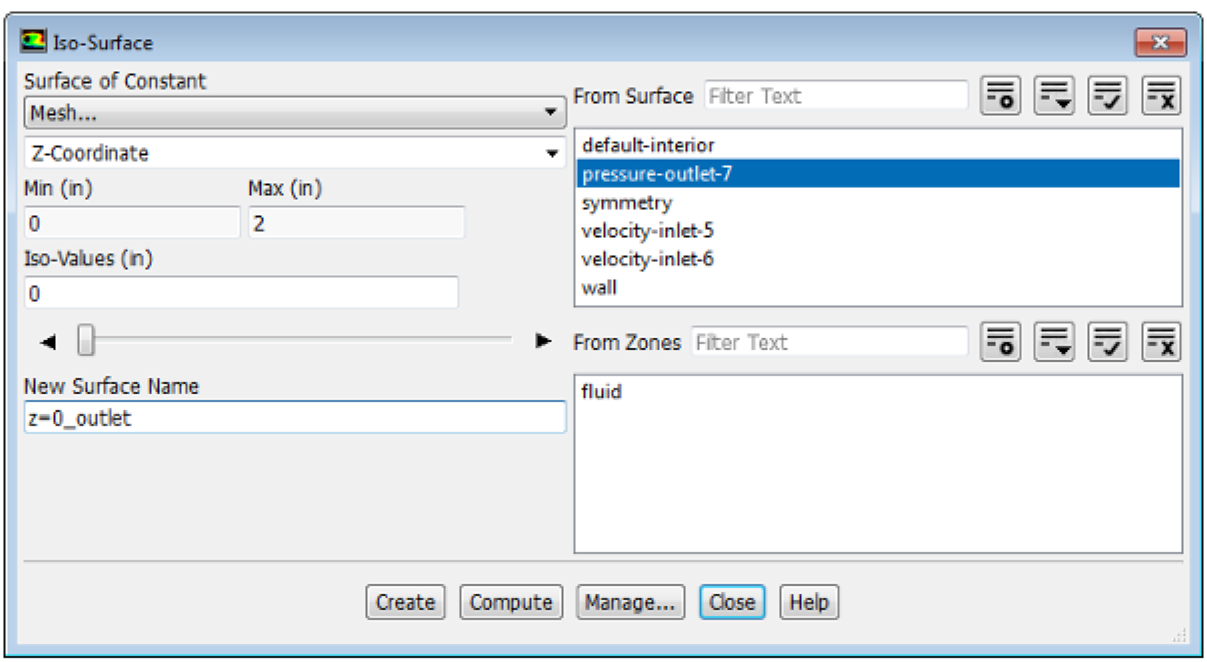
Select **front** from the **Views** selection list and click **Apply**, then close the **Views** dialog box.



4. Create a line at the centerline of the outlet. For this task, you will use the **Surface** group box of the **Post-processing** tab.



 **Postprocessing** → **Surface** → **Create** → **Iso-Surface...**



a. Select **Mesh...** and **Z-Coordinate** from the **Surface of Constant** drop-down lists.

b. Click **Compute** to obtain the extent of the mesh in the z-direction.

   The range of values in the z-direction is displayed in the **Min** and **Max** fields.

c. Retain the default value of `0` inches for **Iso-Values**.

d. Select `pressure-outlet-7` from the **From Surface** selection list.

e. Enter `z=0_outlet` for **New Surface Name**.

f. Click **Create**.

   The new line surface representing the intersection of the plane z=0 and the surface `pressure-outlet-7` is created, and its name **z=0_outlet** appears in the **From Surface** selection list.

Note

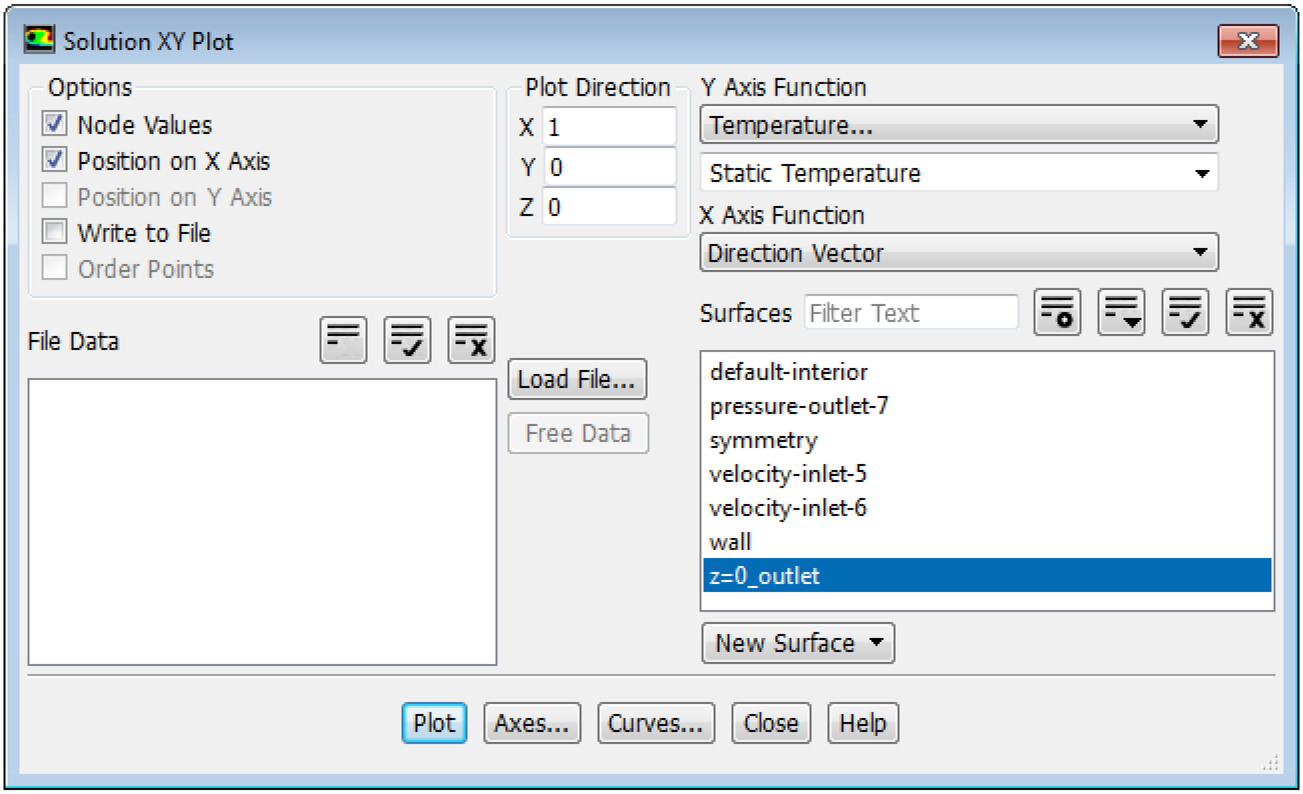
- After the line surface **z=0_outlet** is created, a new entry will automatically be generated for **New Surface Name**, in case you would like to create another surface.
- If you want to delete or otherwise manipulate any surfaces, click **Manage...** to open the **Surfaces** dialog box.

---

g. Close the **Iso-Surface** dialog box.

5. Display and save an XY plot of the temperature profile across the centerline of the outlet for the initial solution (Figure 3.10: Outlet Temperature Profile for the Initial Solution (p. 168)).

   Postprocessing → Plots → XY Plot → Edit...



a. Select **Temperature...** and **Static Temperature** from the **Y Axis Function** drop-down lists.

b. Select the **z=0_outlet** surface you just created from the **Surfaces** selection list.

c. Click **Plot**.

d. Enable **Write to File** in the **Options** group box.

   *The button that was originally labeled **Plot** will change to **Write...**.*

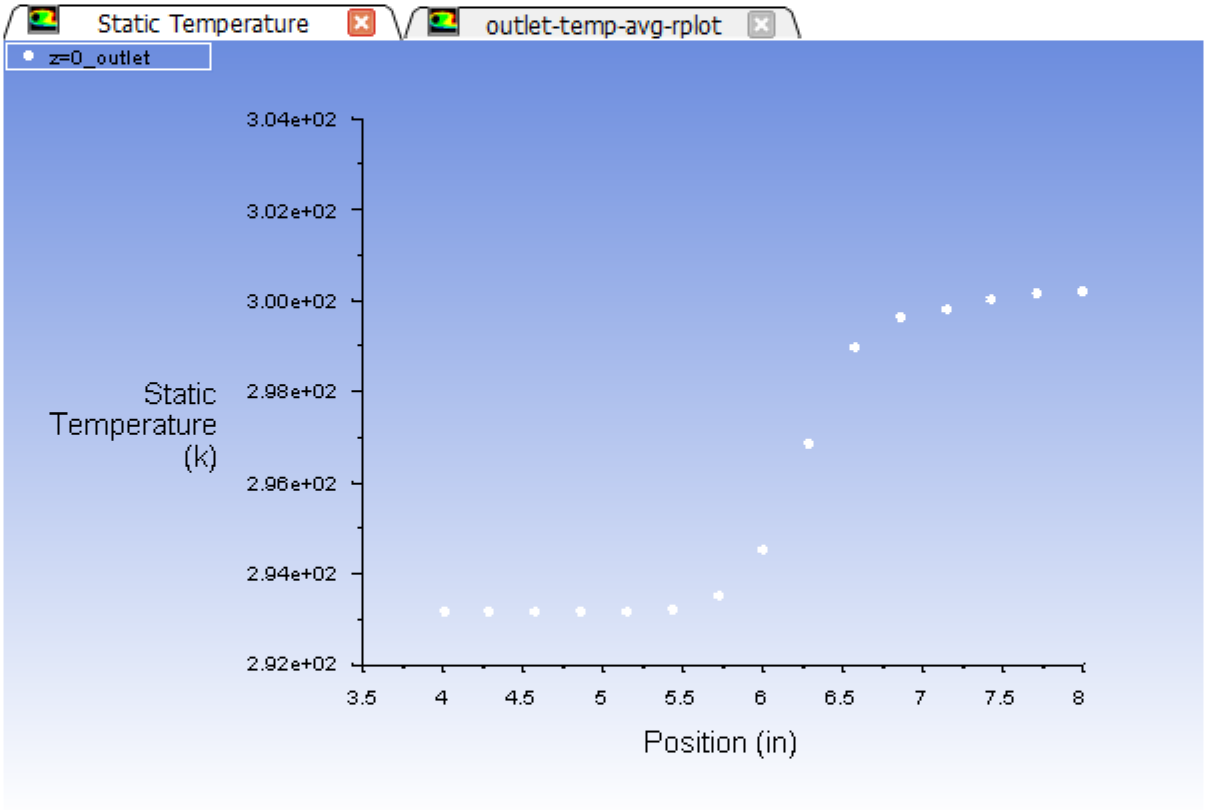
e. Click **Write...**.

   i. In the **Select File** dialog box, enter `outlet_temp1.xy` for **XY File**.

   ii. Click **OK** to save the temperature data and close the **Select File** dialog box.

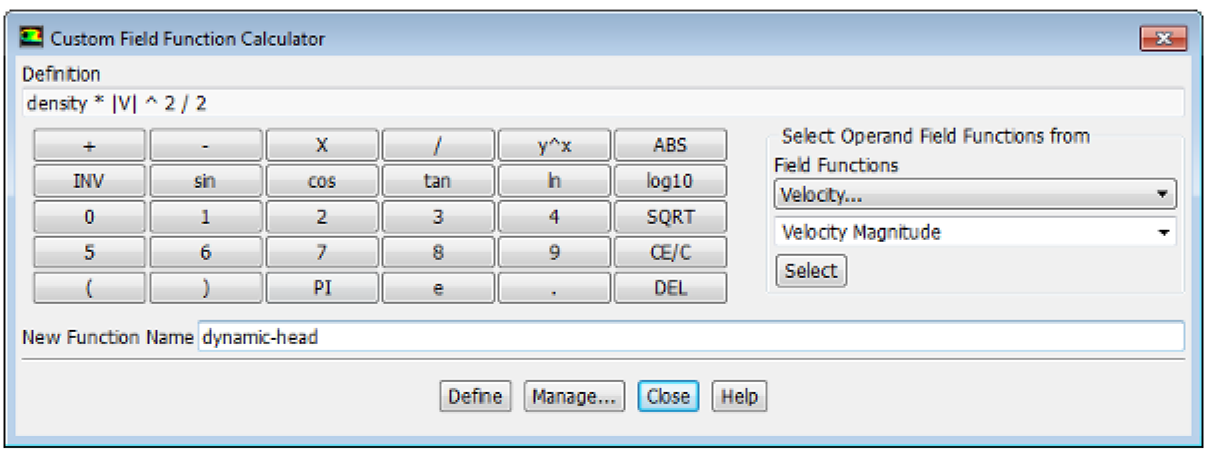
f. Close the **Solution XY Plot** dialog box.

Figure 3.10: Outlet Temperature Profile for the Initial Solution



6. Define a custom field function for the dynamic head formula ($\rho \cdot |V|^2 / 2$).

User Defined → **Field Functions** → **Custom...**



a. Select **Density...** and **Density** from the **Field Functions** drop-down lists, and click the **Select** button to add **density** to the **Definition** field.

b. Click the **X** button to add the multiplication symbol to the **Definition** field.

c. Select **Velocity...** and **Velocity Magnitude** from the **Field Functions** drop-down lists, and click the **Select** button to add **|V|** to the **Definition** field.

d. Click **y^x** to raise the last entry in the **Definition** field to a power, and click **2** for the power.

e. Click the **/** button to add the division symbol to the **Definition** field, and then click **2**.

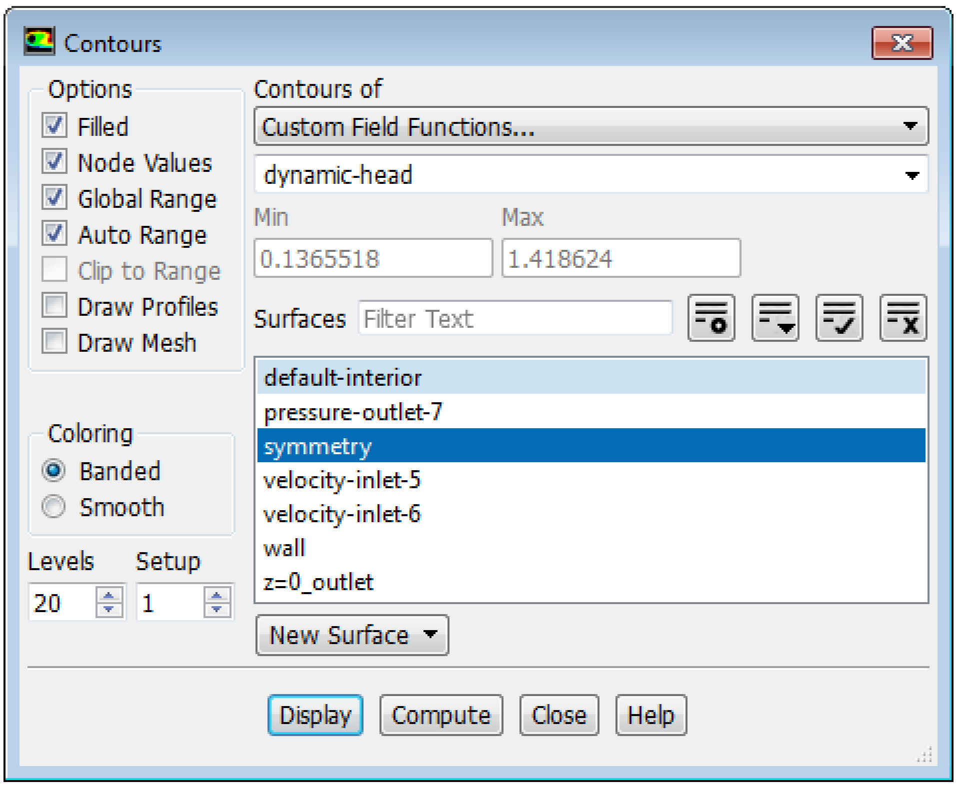
f. Enter `dynamic-head` for **New Function Name**.

g. Click **Define** and close the **Custom Field Function Calculator** dialog box.

   The **dynamic-head** tree item will appear under the **Parameters & Customization/Custom Field Functions** tree branch.

7. Display filled contours of the custom field function (Figure 3.11: Contours of the Dynamic Head Custom Field Function (p. 170)).

   Postprocessing → **Graphics** → **Contours** → **Edit...**



a. Select **Custom Field Functions...** and **dynamic-head** from the **Contours of** drop-down lists.

Tip

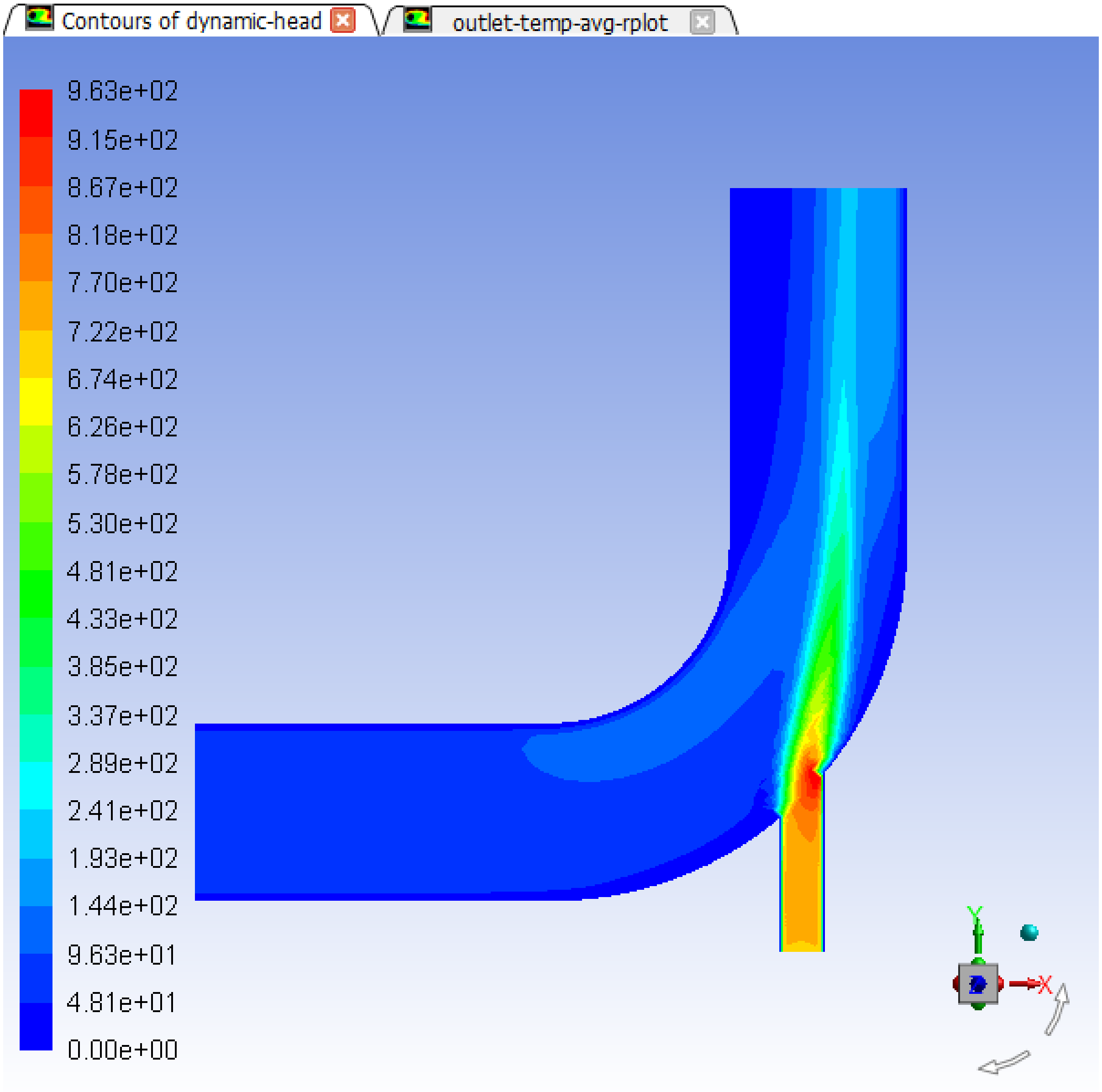
Custom Field Functions... is at the top of the upper **Contours of** drop-down list.

---

b. Ensure that **symmetry** is selected from the **Surfaces** selection list.

c. Click **Display** and close the **Contours** dialog box.

Figure 3.11: Contours of the Dynamic Head Custom Field Function



Note

You may need to change the view by zooming out after the last vector display, if you have not already done so.

8. Save the settings for the custom field function by writing the case and data files (`elbow1.cas.gz` and `elbow1.dat.gz`).

File → Write → Case & Data...

a. Ensure that `elbow1.cas.gz` is entered for **Case/Data File**.

Note

When you write the case and data file at the same time, it does not matter whether you specify the file name with a `.cas` or `.dat` extension, as both will be saved.

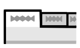
b. Click **OK** to save the files and close the **Select File** dialog box.

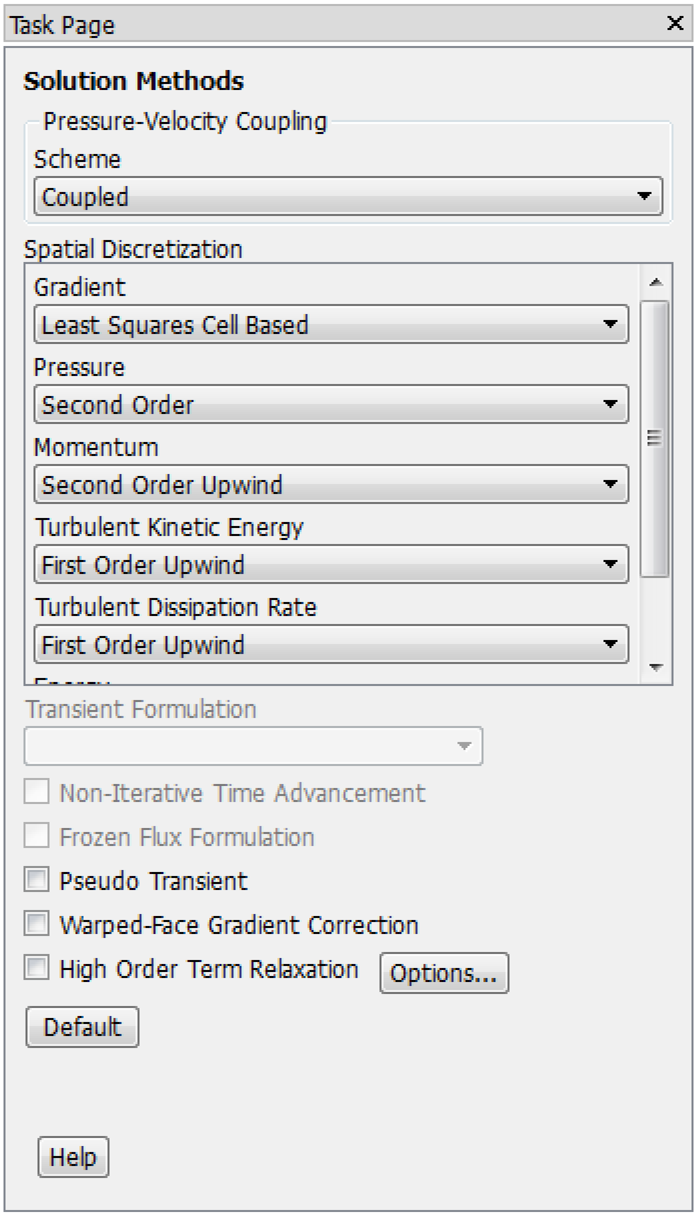
c. Click **OK** to overwrite the files that you had saved earlier.

3.4.8. Using the Coupled Solver

The elbow solution computed in the first part of this tutorial used the SIMPLE solver scheme for Pressure-Velocity coupling. For many general fluid-flow problems, convergence speed can be improved by using the Coupled solver. You will now change the ***Solution Method*** *to a coupled scheme and compare the convergence speed of the SIMPLE and Coupled solvers.*

1. Change the solver settings.

 **Solving → Solution → Methods...**



a. Select **Coupled** from the **Scheme** drop-down list.

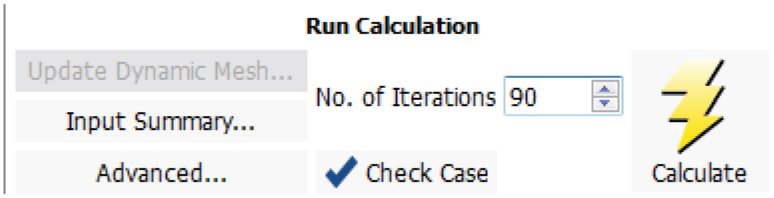
b. Leave the **Spatial Discretization** options at their default settings.

2. Re-initialize the flow field.

   Solving → **Initialization** → **Initialize**

3. Run the solution for an additional 90 iterations.

   Solving → **Run Calculation**



a. Enter `90` for **No. of Iterations**.

b. Click **Calculate**.

   A dialog box will appear stating that `outlet-temp-avg-rfile.out` already exists, and asking if you want to create a new file. Click **No**. Another dialog box will appear asking whether it is OK to overwrite the file. Click **Yes**.

   The solution will converge in approximately 36 iterations (Figure 3.12: Residuals for the Coupled Solver Calculation (p. 173)). Note that this is faster than the convergence rate using the SIMPLE pressure-velocity coupling. The convergence history is shown in Figure 3.13: Convergence History of Mass-Weighted Average Temperature (p. 174).

Figure 3.12: Residuals for the Coupled Solver Calculation

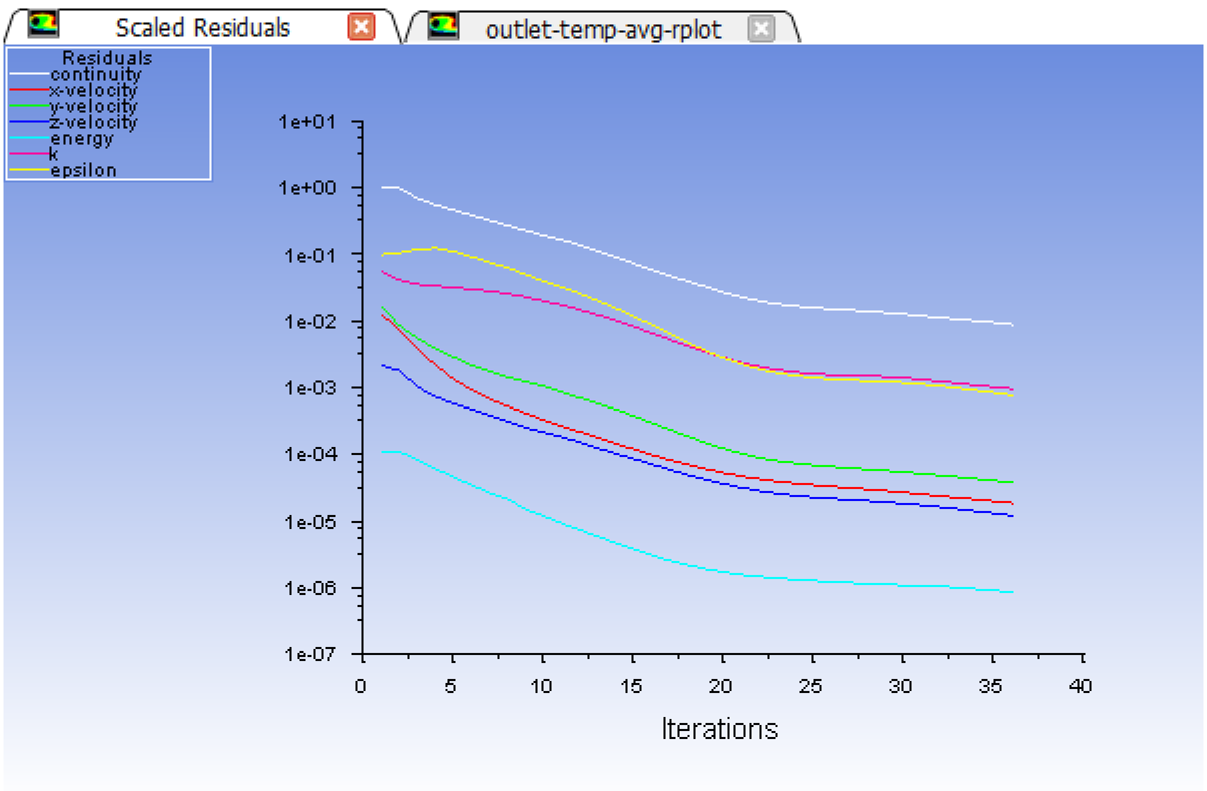
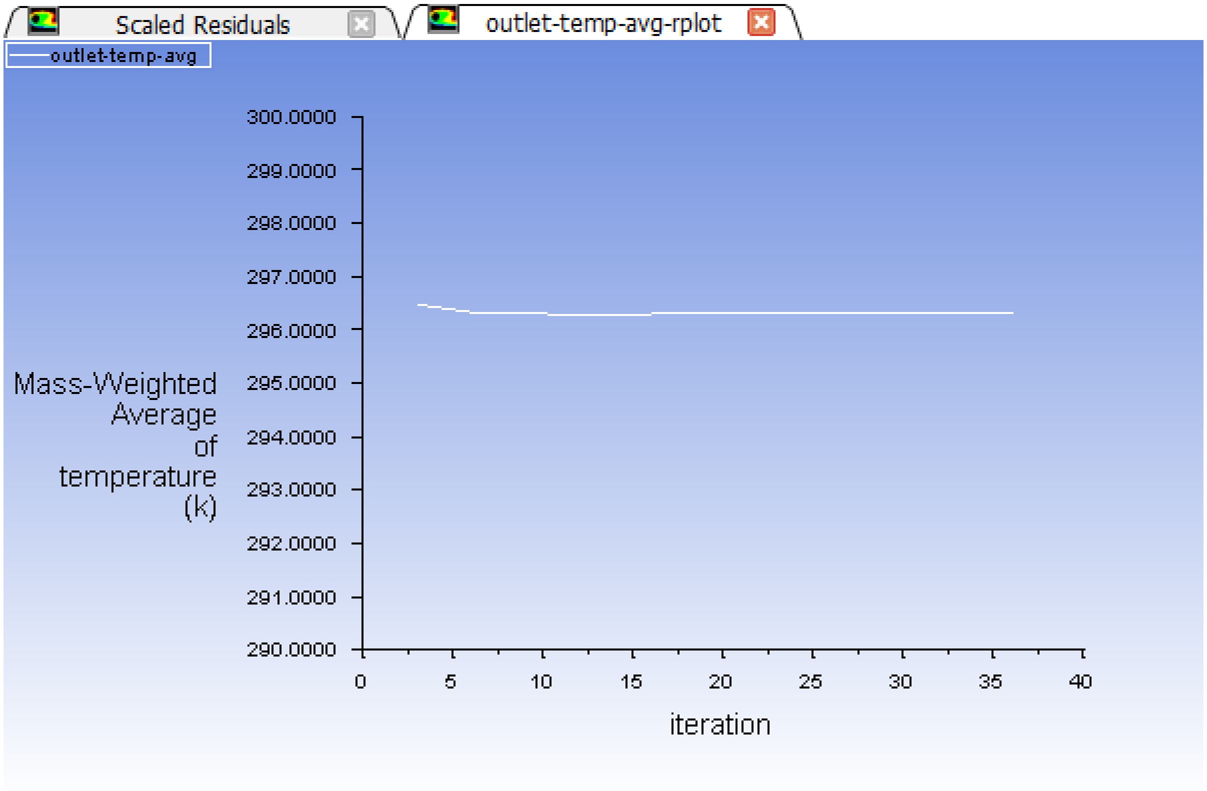


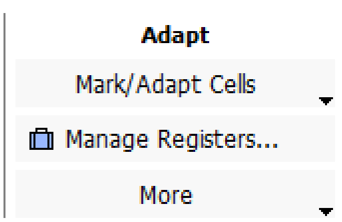
Figure 3.13: Convergence History of Mass-Weighted Average Temperature

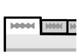


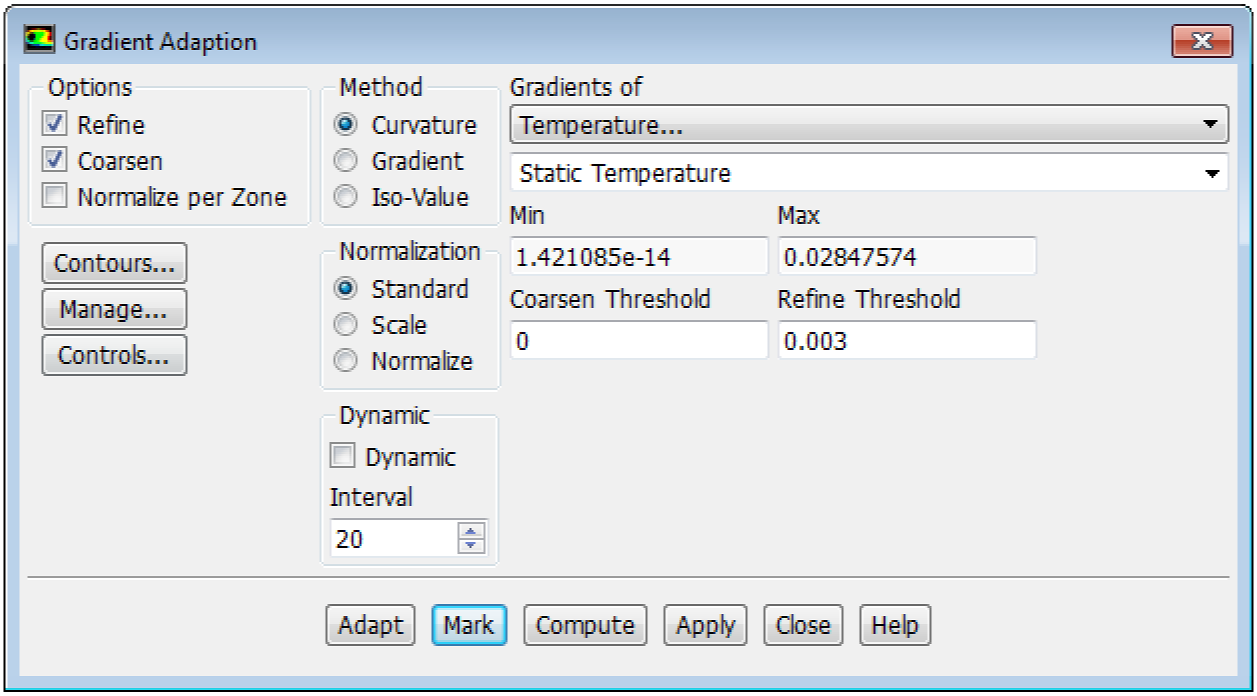
3.4.9. Adapting the Mesh

For the first two runs of this tutorial, you have solved the elbow problem using a fairly coarse mesh. The elbow solution can be improved further by refining the mesh to better resolve the flow details. ANSYS Fluent provides a built-in capability to easily adapt (locally refine) the mesh according to solution gradients. In the following steps you will adapt the mesh based on the temperature gradients in the current solution and compare the results with the previous results.

1. Adapt the mesh in the regions of high temperature gradient. For this task, you will use the **Adapt** group box in the **Setting Up Domain** ribbon tab.



 **Setting Up Domain → Adapt → Mark/Adapt Cells → Gradient...**



a. Ensure that **Refine** is enabled in the **Options** group box.

   *ANSYS Fluent will not coarsen beyond the original mesh for a 3D mesh. Hence, it is not necessary to deselect **Coarsen** in this instance.*

b. Select **Temperature...** and **Static Temperature** from the **Gradients of** drop-down lists.

c. Click **Compute**.

   *ANSYS Fluent will update the **Min** and **Max** values to show the minimum and maximum temperature gradient.*

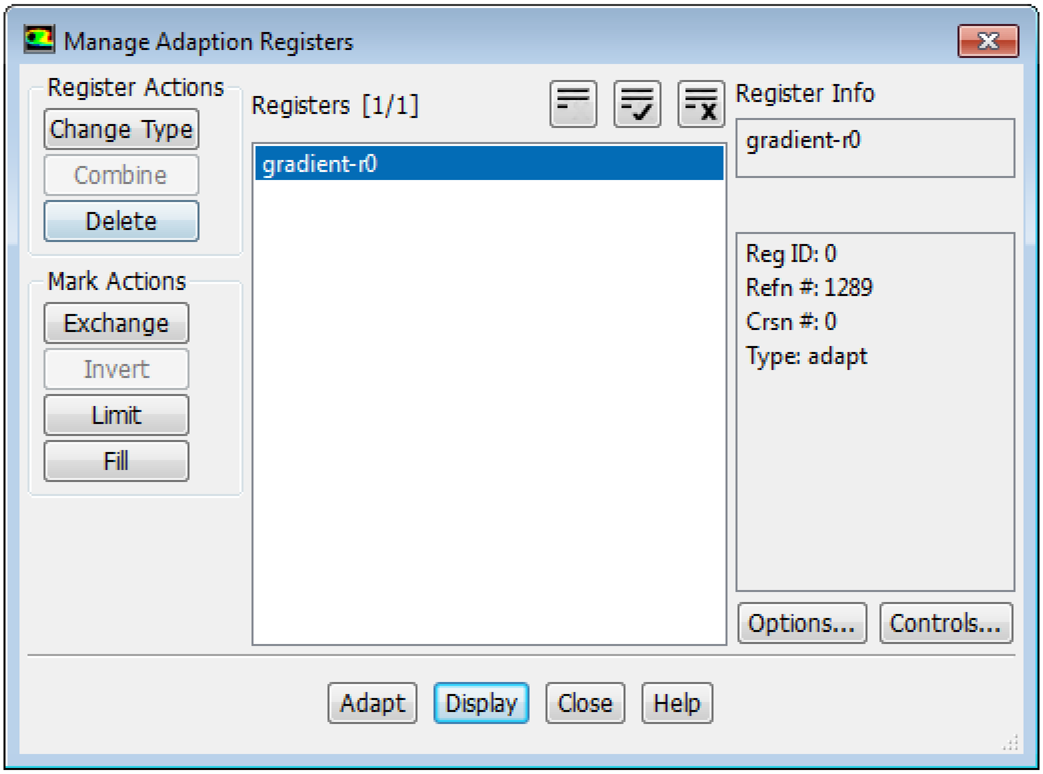
d. Enter `0.003` for **Refine Threshold**.

   *A general rule is to use 10% of the maximum gradient when setting the value for **Refine Threshold**.*

e. Click **Mark**.

   ANSYS Fluent will report in the console that approximately 1289 cells were marked for refinement.

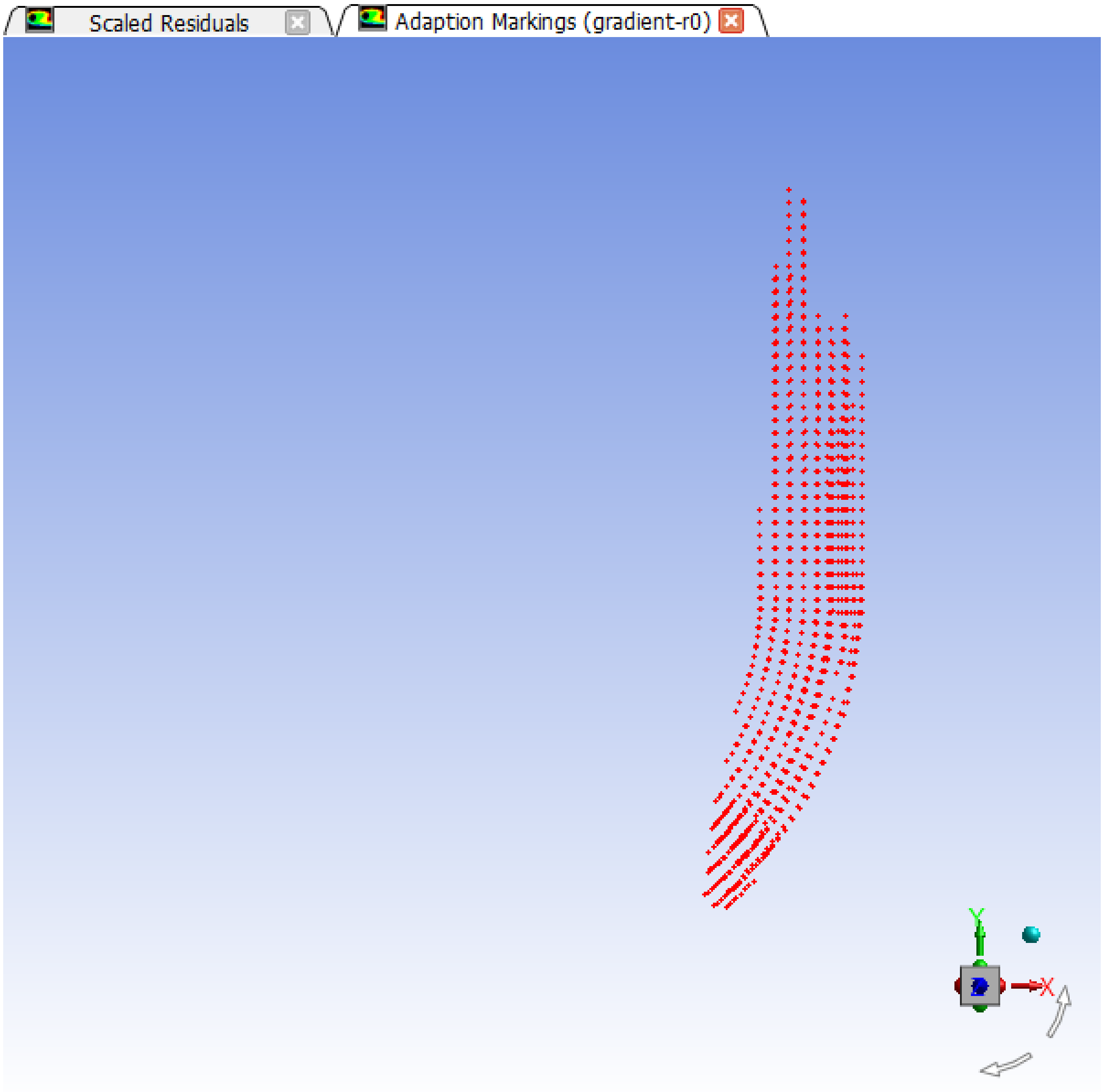
f. Click **Manage...** to open the **Manage Adaption Registers** dialog box.



i. Click **Display**.

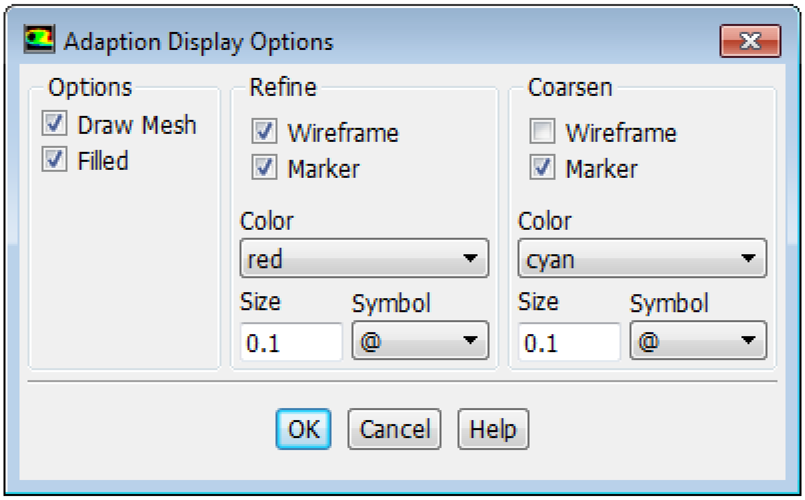
ANSYS Fluent will display the cells marked for adaption in the graphics window (Figure 3.14: Cells Marked for Adaption (p. 177)).

Figure 3.14: Cells Marked for Adaption



Extra You can change the way ANSYS Fluent displays cells marked for adaption (Figure 3.15: Alternative Display of Cells Marked for Adaption (p. 179)) by performing the following steps:

A. Click **Options...** in the **Manage Adaption Registers** dialog box to open the **Adaption Display Options** dialog box.

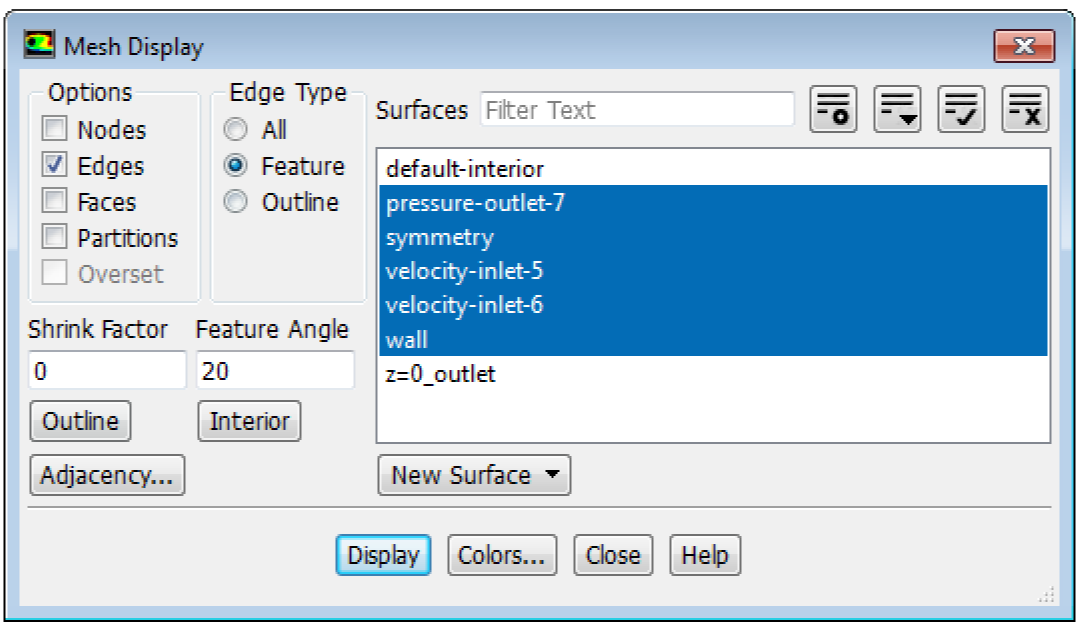


B. Enable **Wireframe** in the **Refine** group box.

C. Enable **Filled** in the **Options** group box.

D. Enable **Draw Mesh** in the **Options** group box.

*The **Mesh Display** dialog box will open.*



E. Ensure that only the **Edges** option is enabled in the **Options** group box.

F. Select **Feature** from the **Edge Type** list.

G. Select all of the items except **default-interior** and **z=0_outlet** from the **Surfaces** selection list.

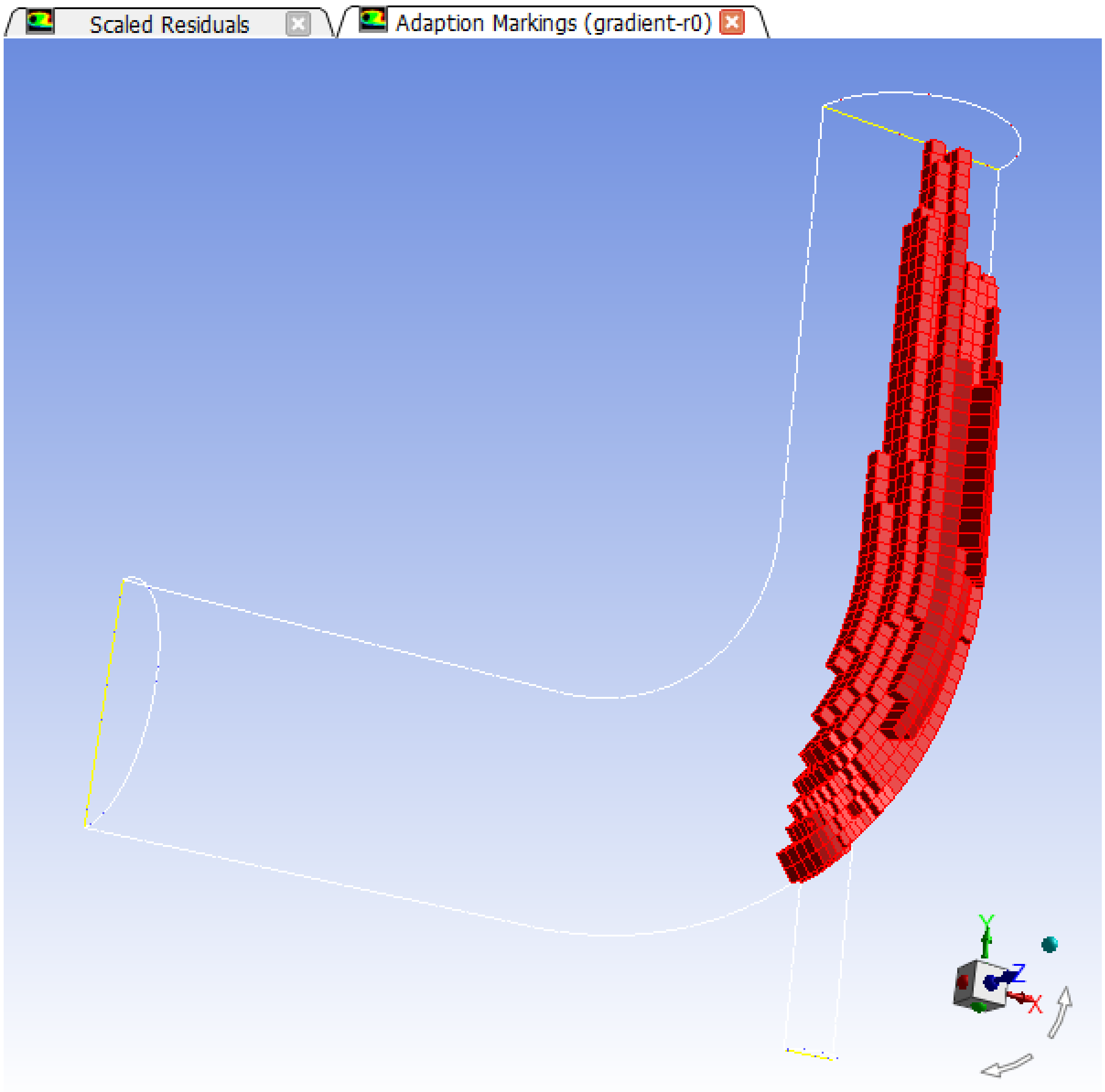
H. Click **Display** and close the **Mesh Display** dialog box.

I. Click **OK** to close the **Adaption Display Options** dialog box.

J. Click **Display** in the **Manage Adaption Registers** dialog box.

K. Rotate the view and zoom in to get the display shown in Figure 3.15: Alternative Display of Cells Marked for Adaption (p. 179).

Figure 3.15: Alternative Display of Cells Marked for Adaption



L. After viewing the marked cells, rotate the view back and zoom out again.

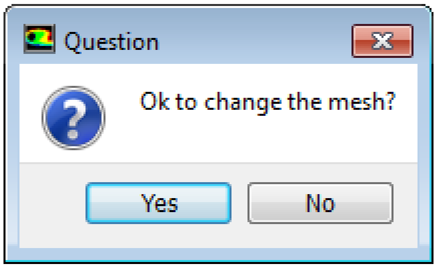
M. Click **Options...** in the **Manage Adaption Registers** dialog box to open the **Adaption Display Options** dialog box again.

N. In the **Adaption Display Options** dialog box, disable the **Draw Mesh** option and click **OK** to close it.

ii. In the **Manage Adaption Registers** dialog box, ensure that **gradient-r0** is selected from the **Registers** selection list.

iii. Click **Adapt** in the **Manage Adaption Registers** dialog box.

*A **Question** dialog box will open, confirming your intention to adapt the mesh. Click **Yes** to proceed.*

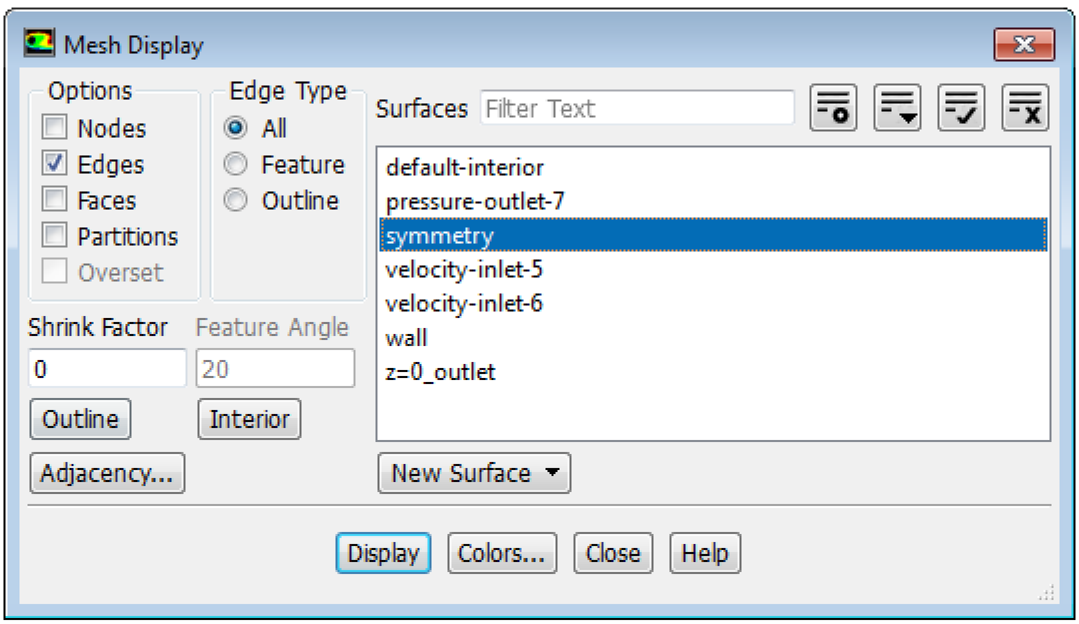


iv. Close the **Manage Adaption Registers** dialog box.

g. Close the **Gradient Adaption** dialog box.

2. Display the adapted mesh (Figure 3.16: The Adapted Mesh (p. 181)).

 **Setting Up Domain → Mesh → Display...**



a. Select **All** from the **Edge Type** list.

b. Deselect all of the highlighted items from the **Surfaces** selection list except for **symmetry**.

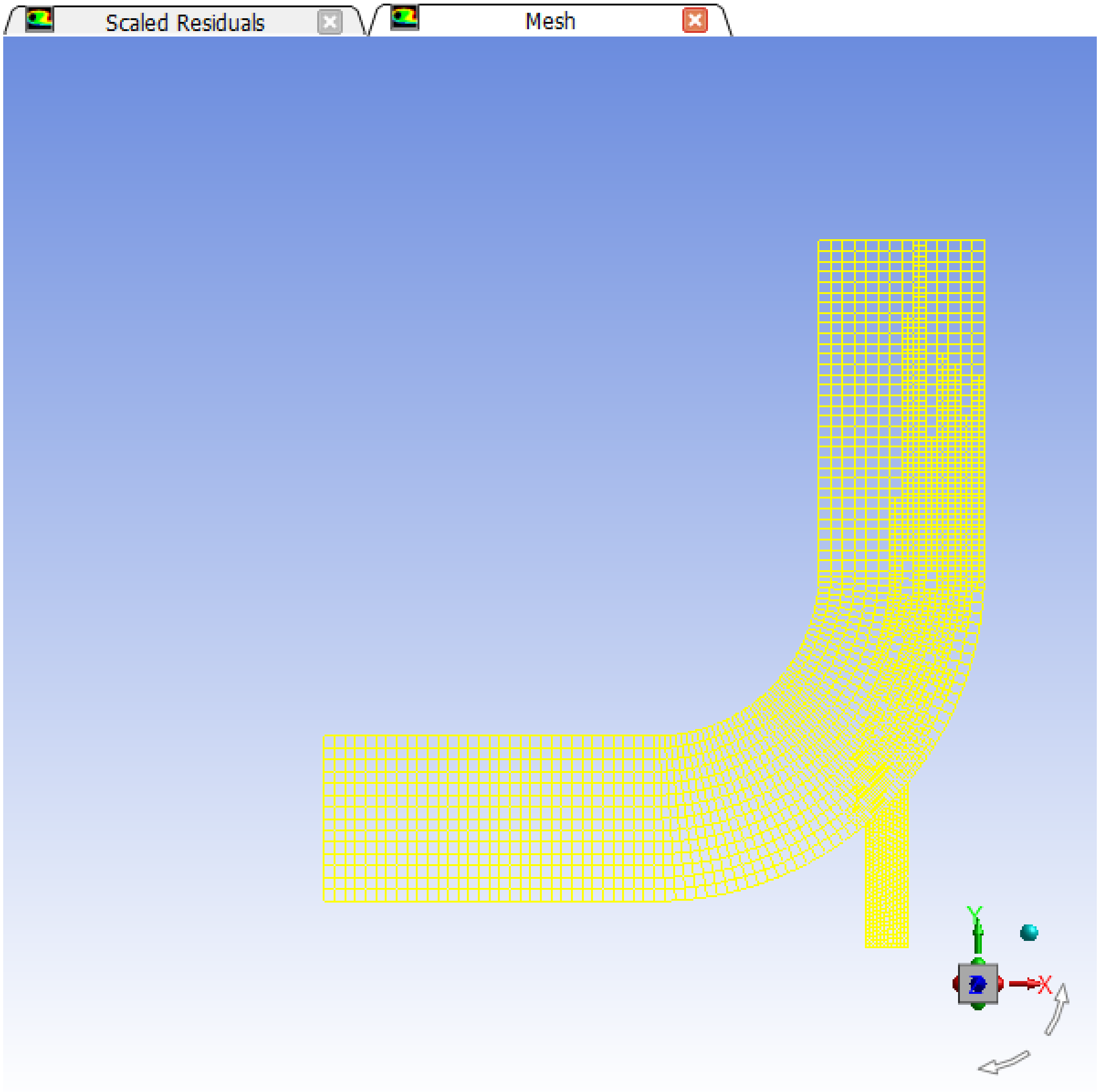
Tip

To deselect all surfaces, click the **Deselect All Shown** button () at the top of the **Surfaces** selection list. Then select the desired surface from the **Surfaces** selection list.

---

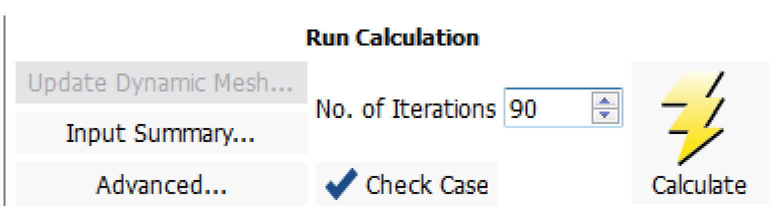
c. Click **Display** and close the **Mesh Display** dialog box.

Figure 3.16: The Adapted Mesh



3. Request an additional 90 iterations.

Solving → Run Calculation → Calculate



The solution will converge after approximately 30 additional iterations (Figure 3.17: The Complete Residual History (p. 182) and Figure 3.18: Convergence History of Mass-Weighted Average Temperature (p. 182)).

Figure 3.17: The Complete Residual History

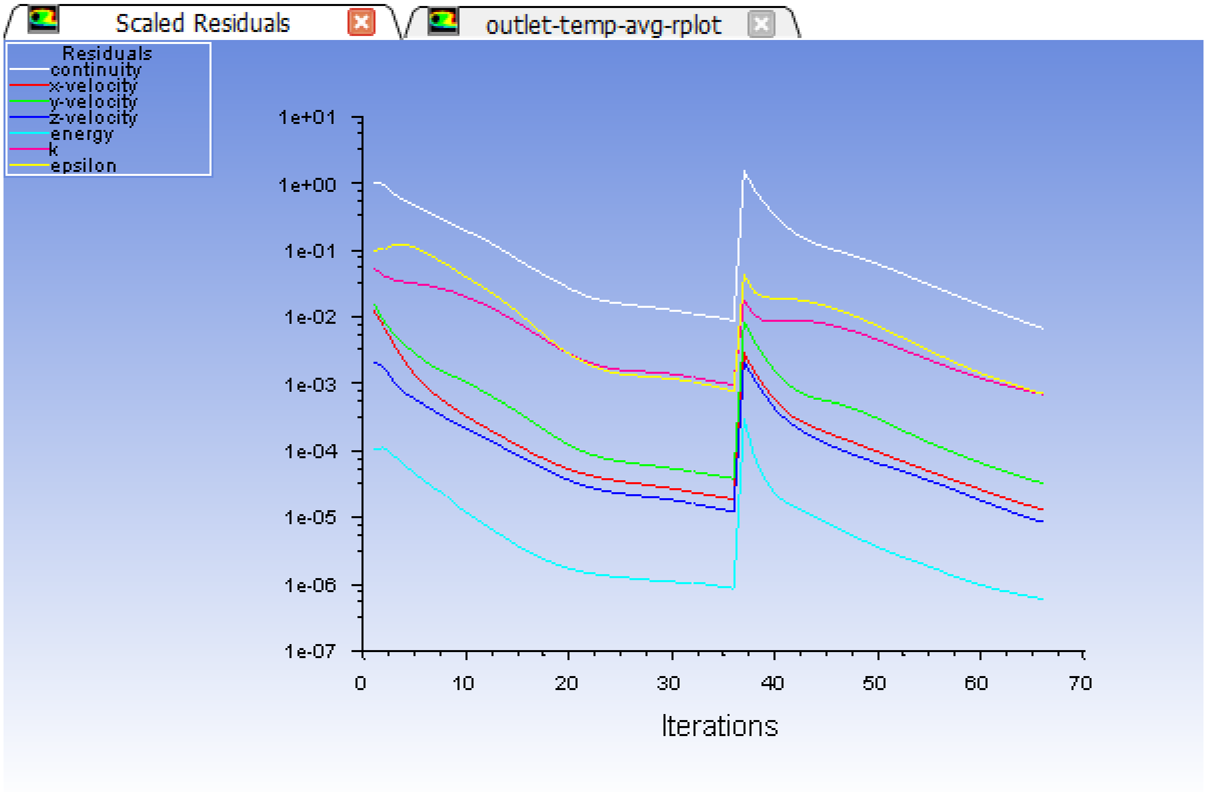
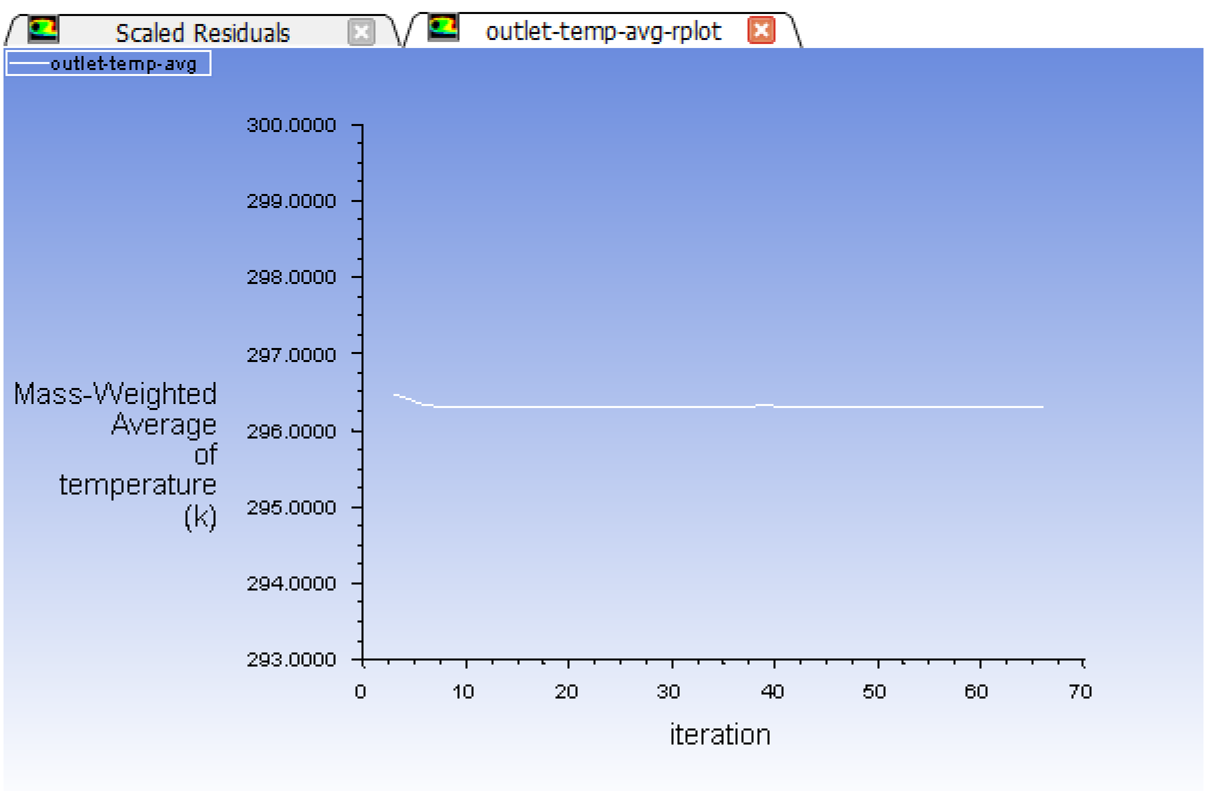


Figure 3.18: Convergence History of Mass-Weighted Average Temperature



4. Save the case and data files for the Coupled solver solution with an adapted mesh (`elbow2.cas.gz` and `elbow2.dat.gz`).

   File → Write → Case & Data...

   a. Enter `elbow2.gz` for **Case/Data File**.

   b. Click **OK** to save the files and close the **Select File** dialog box.

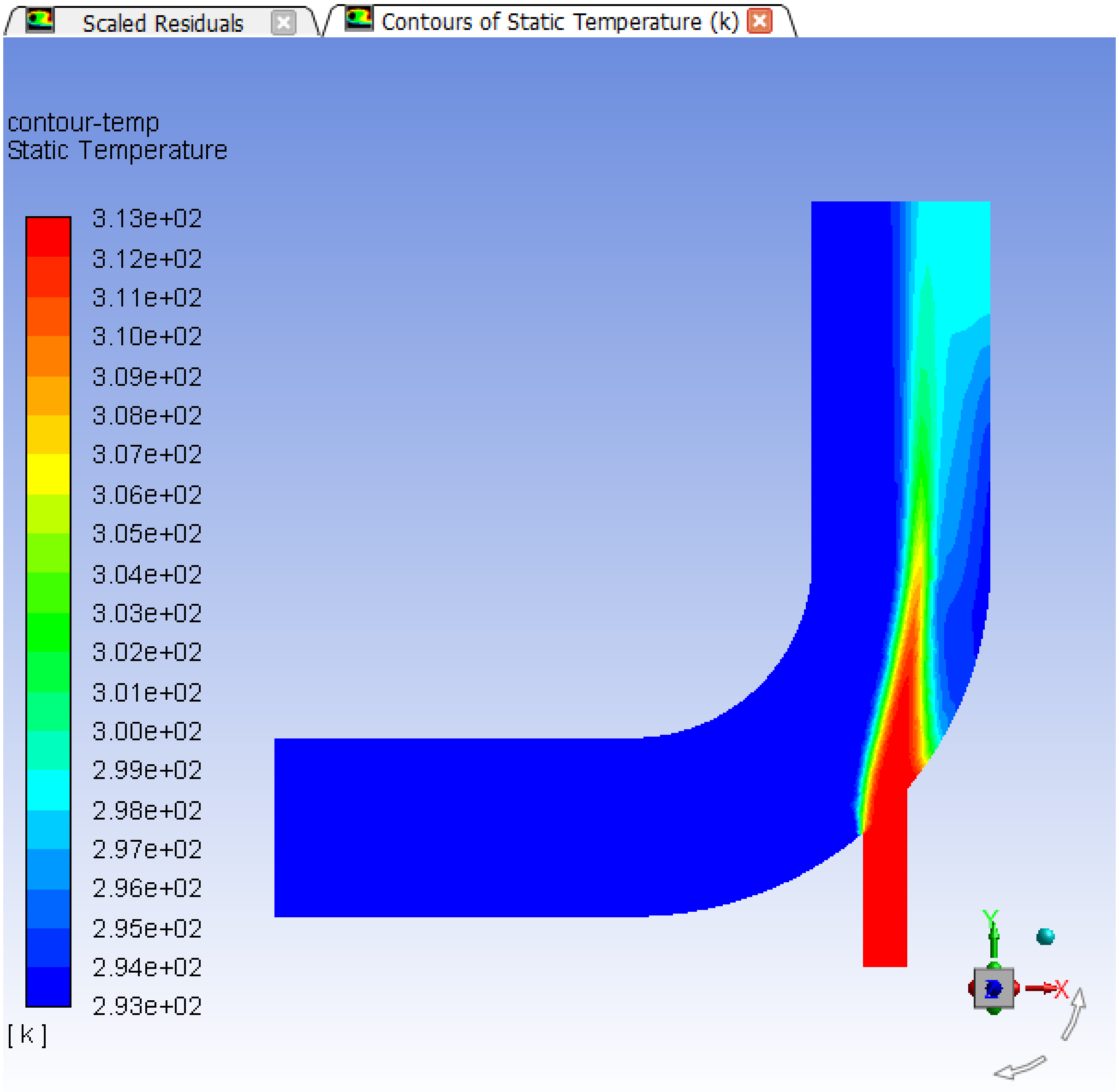
   The files `elbow2.cas.gz` and `elbow2.dat.gz` will be saved in your default folder.

5. Display the filled temperature distribution (using node values) on the revised mesh using the temperature contours definition that you created earlier (Figure 3.19: Filled Contours of Temperature Using the Adapted Mesh (p. 184)).

   Right-click the **Results/Graphics/Contours/contour-temp** tree item and select **Display** from the menu that opens.

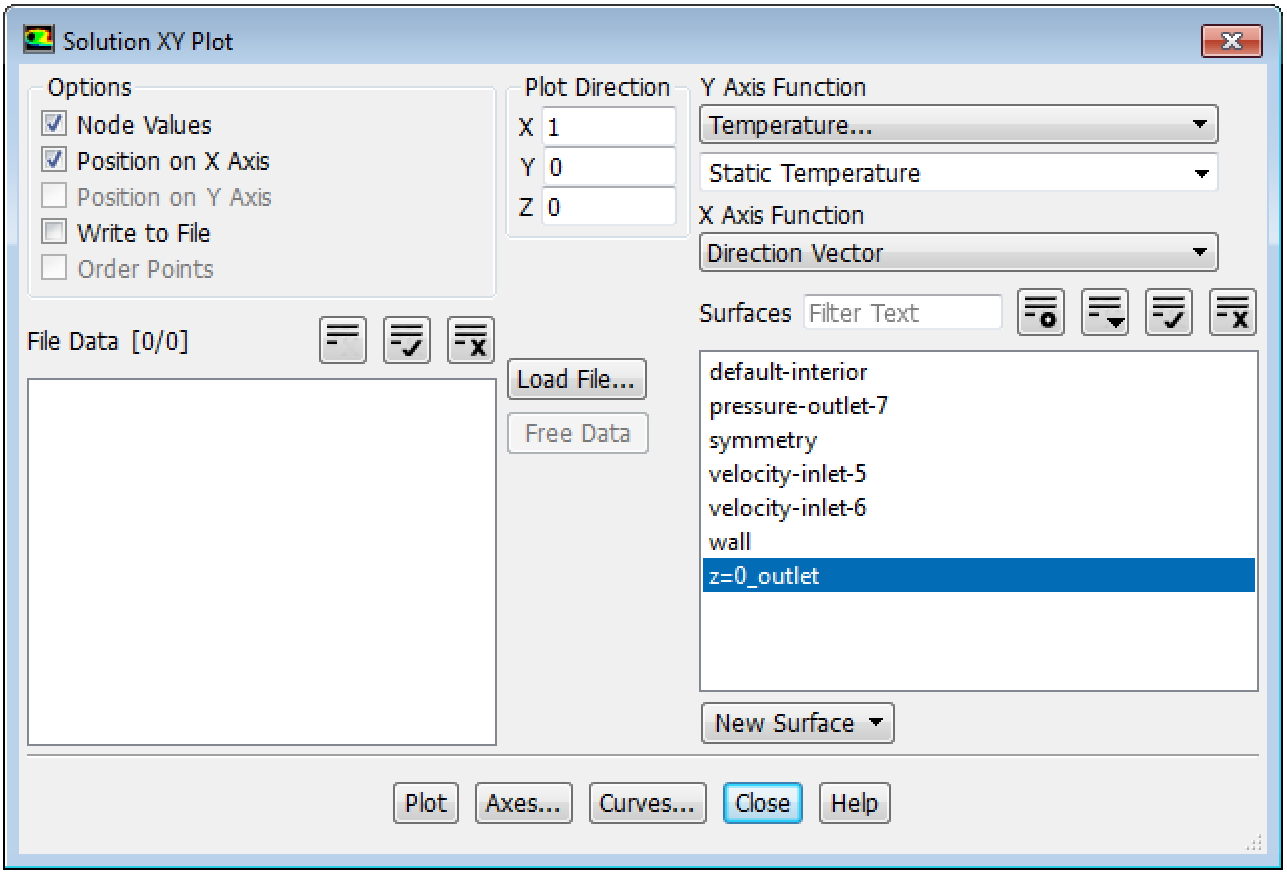
   Results → Graphics → Contours → contour-temp → Display

Figure 3.19: Filled Contours of Temperature Using the Adapted Mesh



6. Display and save an XY plot of the temperature profile across the centerline of the outlet for the adapted solution (Figure 3.20: Outlet Temperature Profile for the Adapted Coupled Solver Solution (p. 186)).

   Postprocessing → **Plots** → **XY Plot** → **Edit...**



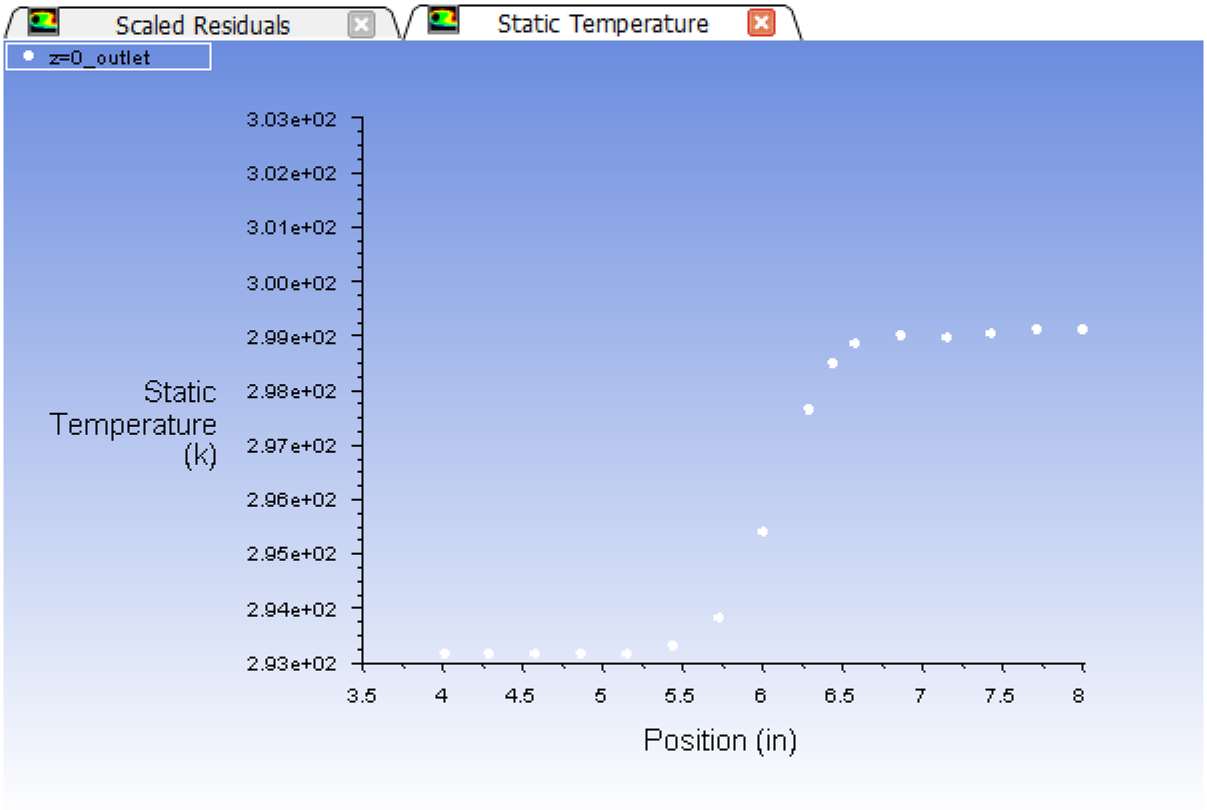
a. Ensure that the **Write to File** option is disabled in the **Options** group box.

b. Ensure that **Temperature...** and **Static Temperature** are selected from the **Y Axis Function** drop-down lists.

c. Ensure that **z=0_outlet** is selected from the **Surfaces** selection list.

d. Click **Plot**.

Figure 3.20: Outlet Temperature Profile for the Adapted Coupled Solver Solution



e. Enable **Write to File** in the **Options** group box.

   *The button that was originally labeled **Plot** will change to **Write...**.*

f. Click **Write...**.

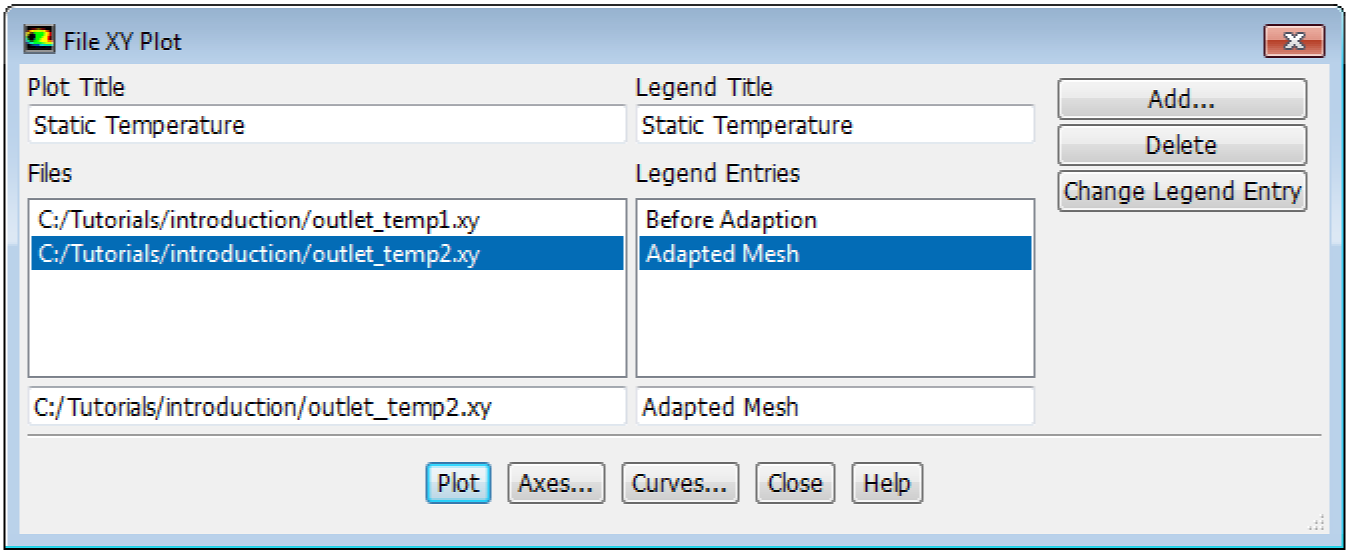
   i. In the **Select File** dialog box, enter `outlet_temp2.xy` for **XY File**.

   ii. Click **OK** to save the temperature data.

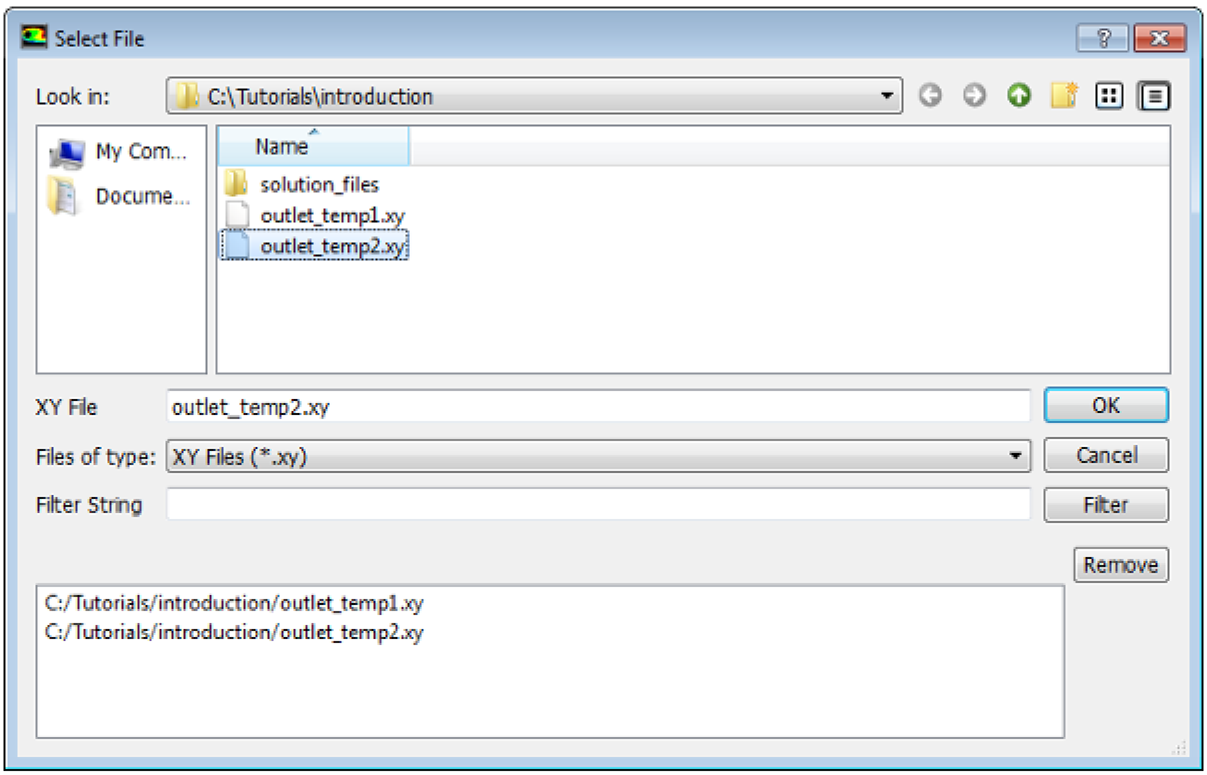
g. Close the **Solution XY Plot** dialog box.

7. Display the outlet temperature profiles for both solutions on a single plot (Figure 3.21: Outlet Temperature Profiles for the Two Solutions (p. 188)).

   Postprocessing → Plots → File...



a. Click the **Add...** button to open the **Select File** dialog box.



i. Click once on **outlet_temp1.xy** and **outlet_temp2.xy**.

Each of these files will be listed with their folder path in the bottom list to indicate that they have been selected.

Tip

If you select a file by mistake, simply click the file in the bottom list and then click **Remove**.

ii. Click **OK** to save the files and close the **Select File** dialog box.

b. Select the folder path ending in **outlet_temp1.xy** from the **Files** selection list.

c. Enter `Before Adaption` in the lower-right text-entry box.

d. Click the **Change Legend Entry** button.

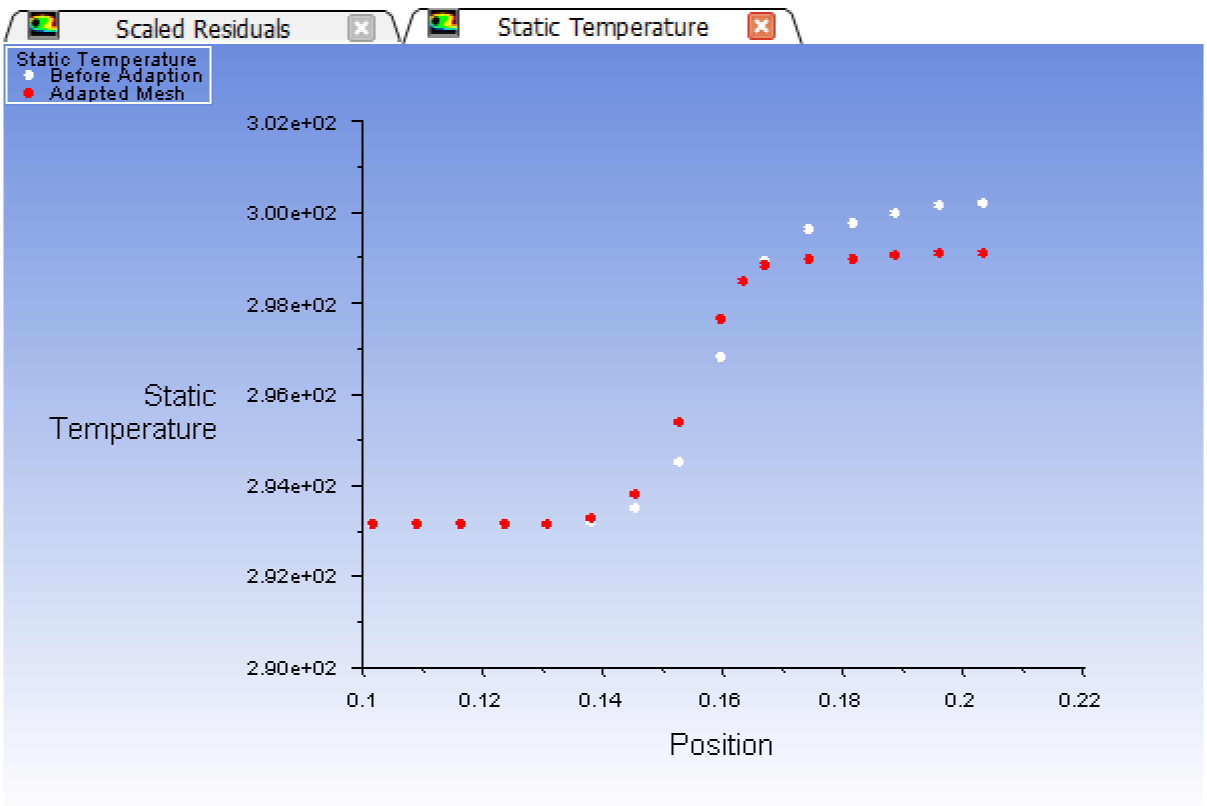
*The item in the **Legend Entries** list for **outlet_temp1.xy** will be changed to **Before Adaption**. This legend entry will be displayed in the upper-left corner of the XY plot generated in a later step.*

e. In a similar manner, change the legend entry for the folder path ending in **outlet_temp2.xy** to be `Adapted Mesh`.

f. Click **Plot** and close the **File XY Plot** dialog box.

Figure 3.21: Outlet Temperature Profiles for the Two Solutions (p. 188) shows the two temperature profiles at the centerline of the outlet. It is apparent by comparing both the shape of the profiles and the predicted outer wall temperature that the solution is highly dependent on the mesh and solution options. Specifically, further mesh adaption should be used in order to obtain a solution that is independent of the mesh.

Figure 3.21: Outlet Temperature Profiles for the Two Solutions



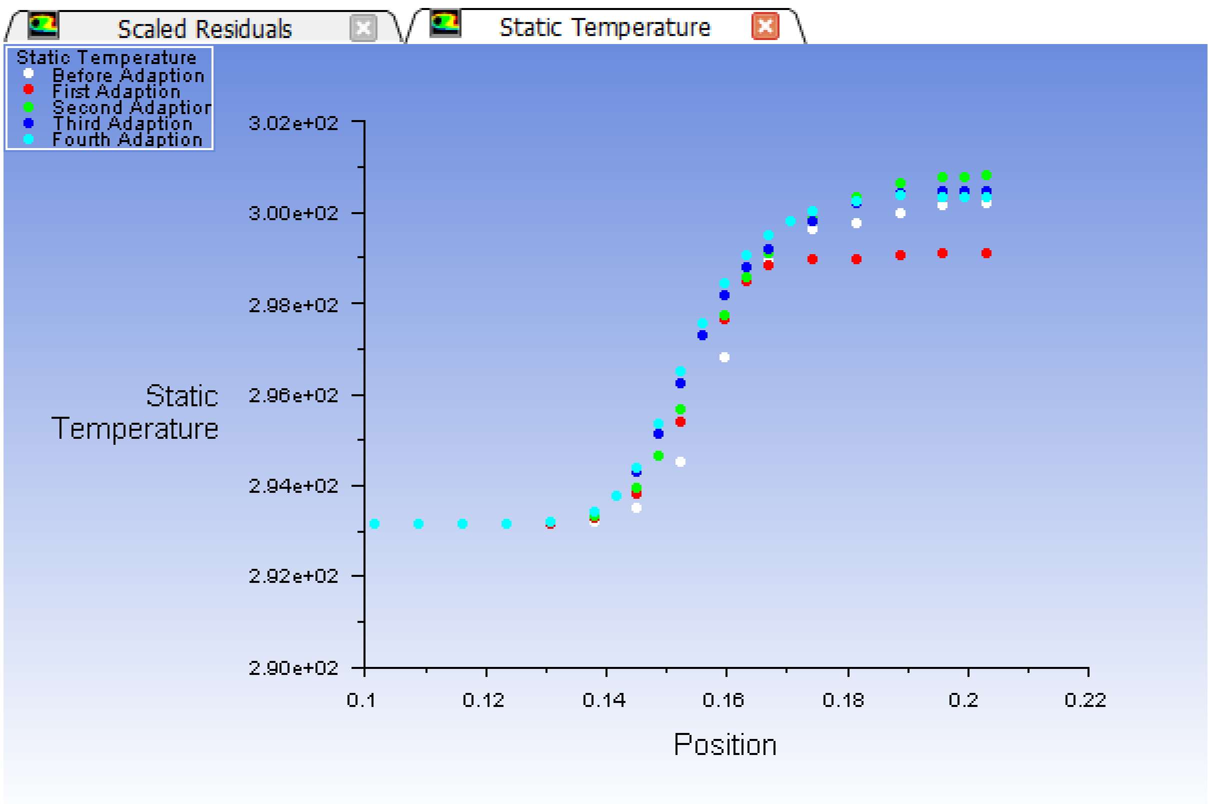
Note

When reading and writing data values, Fluent always uses SI units. Therefore, when you read in the XY data files and plot them, the position and temperature values will be plotted in SI units, regardless of the settings made in the **Scale Mesh** dialog box earlier in the tutorial.

Extra

You can perform additional rounds of mesh adaption based on temperature gradient and run the calculation to see how the temperature profile changes at the outlet. For demonstration purposes, three additional levels of mesh adaption were performed (each time using a refinement of 10% of the new maximum gradient) and the resulting temperature profiles have been plotted with `outlet_temp1.xy` and `outlet_temp2.xy` in Figure 3.22: Outlet Temperature Profiles for Subsequent Mesh Adaption Steps (p. 189).

Figure 3.22: Outlet Temperature Profiles for Subsequent Mesh Adaption Steps



It is evident from Figure 3.22: Outlet Temperature Profiles for Subsequent Mesh Adaption Steps (p. 189) that as the mesh is adapted further, the profiles converge on a mesh-independent profile. The resulting wall temperature at the outlet is predicted to be 300.4 K after mesh independence is achieved.

If computational resources allow, it is always recommended to perform successive rounds of adaption until the solution is independent of the mesh (within an acceptable tolerance). Typically, profiles of important variables are examined (in this case, temperature) and compared to determine mesh independence.

3.5. Setup and Solution in Parallel

This section explores the parallel processing capabilities of ANSYS Fluent. In order to be run in parallel, the mesh must be divided into smaller, evenly sized partitions. Each ANSYS Fluent process, called a compute node, will solve on a single partition, and information will be passed back and forth across all partition interfaces. The solver of ANSYS Fluent enables parallel processing on a dedicated parallel machine, or a network of workstations running Windows or Linux.

3.5.1. Starting the Parallel Version of ANSYS Fluent

Since the procedure for starting the parallel version of ANSYS Fluent is dependent upon the type of machine(s) you are using, two versions of this step are provided here.

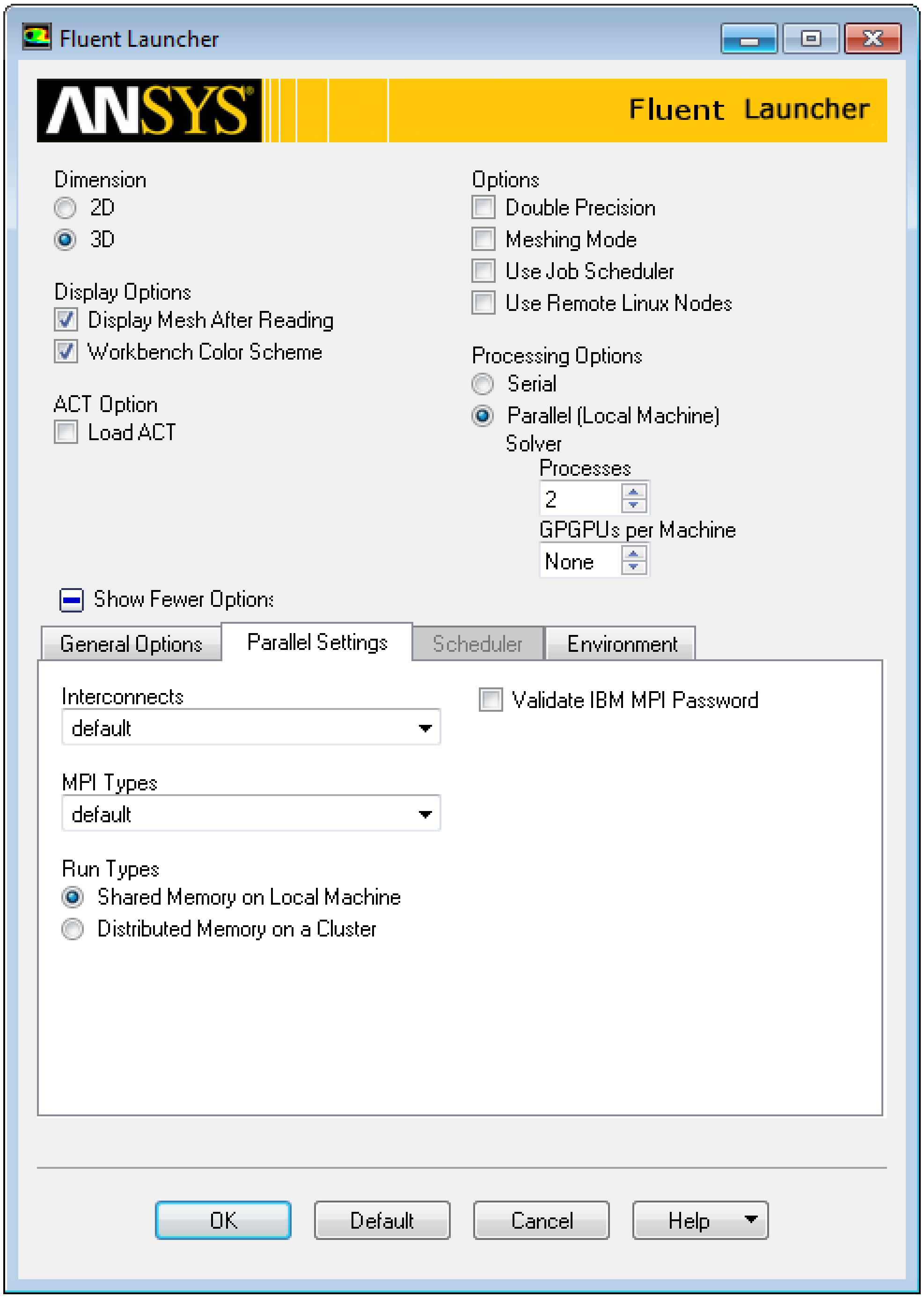
3.5.1.1. Multiprocessor Machine

Use ANSYS Fluent Launcher to start the 3D parallel version of ANSYS Fluent on a Windows or Linux machine using 2 processes.

1. Select **3D** under **Dimension**.
2. Enable **Display Mesh After Reading** under **Display Options**.
3. Make sure that **Double Precision** is disabled under **Options**.
4. Select **Parallel** under **Processing Options**.
5. Set **Processes** to 2.

   *To show details of the parallel settings, click **Show More Options**, then go to the **Parallel Settings** tab. Note that your **Run Types** will be **Shared Memory on Local Machine**.*

6. Click **OK**.



To start ANSYS Fluent on a Linux machine, type at the command prompt

```
fluent 3d -t2
```

If you type `fluent` at the command prompt, then Fluent Launcher will appear.

For additional information about parallel command line options, see the Fluent User's Guide, available in the help viewer or on the ANSYS Customer Portal (http://support.ansys.com/documentation).

3.5.1.2. Network of Computers

You can start the 3D parallel version of ANSYS Fluent on a network of Windows or Linux machines using 2 processes and check the network connectivity by performing the following steps:

1. In Fluent Launcher, restore the default settings by clicking the **Default** button.
2. Select **3D** under **Dimension**.
3. Enable **Display Mesh After Reading** under **Display Options**.
4. Make sure that **Double Precision** is disabled under **Options**.
5. Select **Parallel** under **Processing Options**.
6. Set the **Processes** to 2.
7. Click the **Show More Options** button and open the **Parallel Settings** tab.

    a. Retain the selection of **default** in the **Interconnects** and **MPI Types** drop-down lists.

    > **Note**
    >
    > On Windows platforms, **default** is the only available selection for **Interconnects**.

    b. Select **Distributed Memory on a Cluster** under **Run Types**.

    > **Note**
    >
    > The **Fluent Root Path** (in the **General Options** tab) should be specified as a network path for running Fluent across multiple machines, unless Fluent is installed in the exact same location on all of the machines that you listed for running in parallel. You can click  to convert your **Fluent Root Path** to a network path.

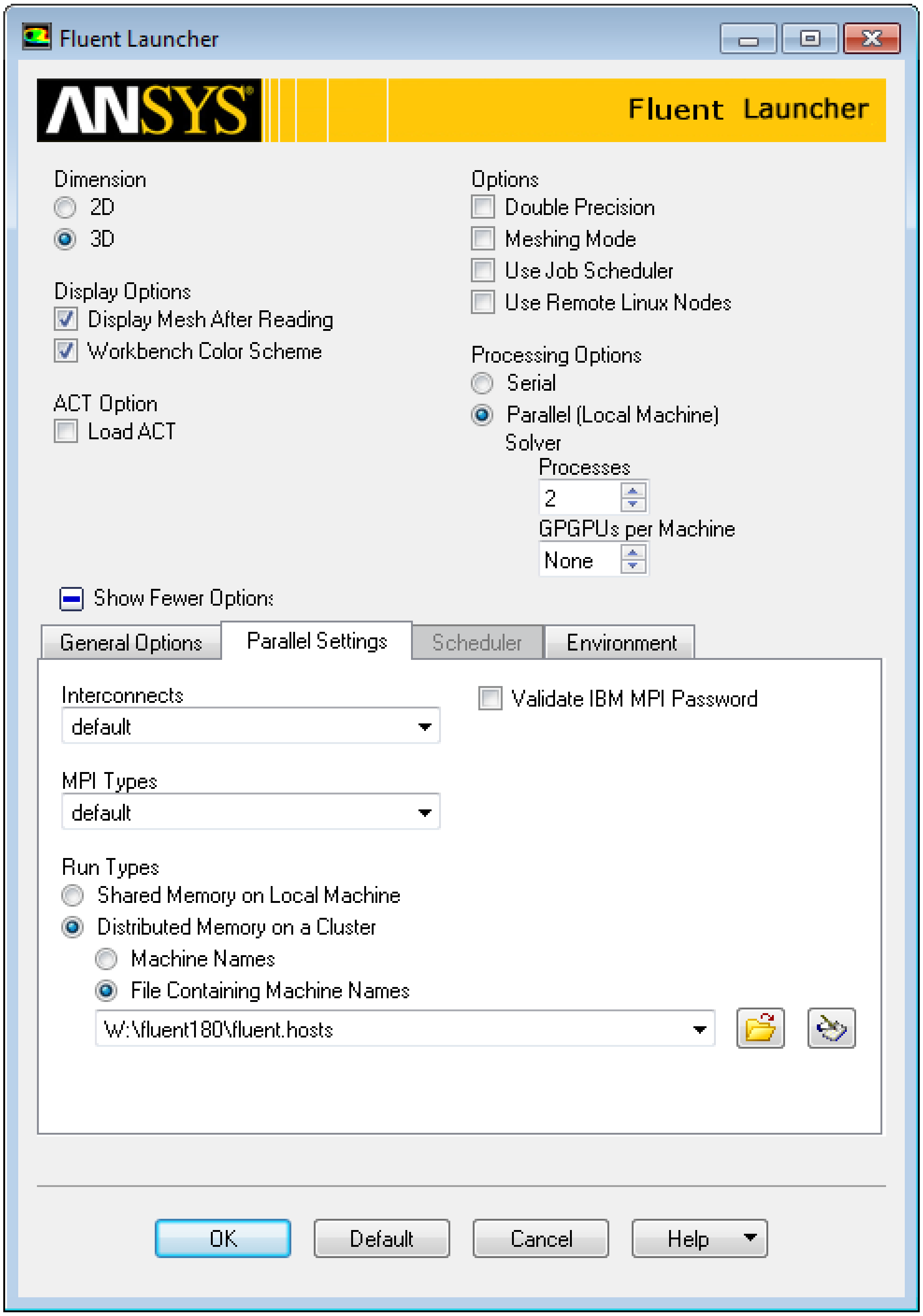
    c. Make sure that **File Containing Machine Names** is selected to specify the file.

    d. Type the name and location of the hosts text file in the text box below **File Containing Machine Names**, or browse and select it using the **Browsing Machine File** dialog box.

    *Alternatively, you can select **Machine Names** and type the names of the machines in the text box.*

8. Click **OK**.

> **Note**
>
> If you are using a Windows platform, you will need to enter your password for MPI the first time you run ANSYS Fluent using **Distributed Memory on a Cluster**.



You can also start parallel ANSYS Fluent by typing the following at the command prompt:

```
fluent 3d -t2 -cnf=fluent.hosts
```

where `-cnf` indicates the location of the hosts text file. The hosts file is a text file that contains a list of the computers on which you want to run the parallel job. If the hosts file is not located in the directory where you are typing the startup command, you will need to supply the full pathname to the file.

For example, the `fluent.hosts` file may look like the following:

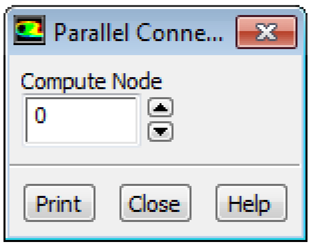
```
 my_computer
another_computer
```

For additional information about hosts files and parallel command line options, see the Fluent User's Guide, available in the help viewer or on the ANSYS Customer Portal (http://support.ansys.com/documentation).

9. Check the network connectivity information.

   Although ANSYS Fluent displays a message confirming the connection to each new compute node and summarizing the host and node processes defined, you may find it useful to review the same information at some time during your session, especially if more compute nodes are spawned to several different machines.

   Parallel → **Network** → **Connectivity...**



   a. Set **Compute Node** to `0`.

      For information about all defined compute nodes, you will select node 0, since this is the node from which all other nodes are spawned.

   b. Click **Print**.

```
--------------------------------------------------------------------------------
ID    Hostname        Core  O.S.         PID    Vendor
--------------------------------------------------------------------------------
n1    my_computer     2/32  Windows-x64  31588  Intel(R) Xeon(R) E5-2687W
n0*   my_computer     1/32  Windows-x64  31736  Intel(R) Xeon(R) E5-2687W
host  my_computer           Windows-x64  21192  Intel(R) Xeon(R) E5-2687W

MPI Option Selected: ibmmpi
Selected system interconnect: default
--------------------------------------------------------------------------------
```

      ID is the sequential denomination of each compute node (the host process is always host), Hostname is the name of the machine hosting the compute node (or the host process), Core is the core number / total number of cores available, O.S. is the architecture, PID is the process ID number, and Vendor is the vendor of the processor. Information about the selected MPI option and system interconnect is provided at the bottom of the report.

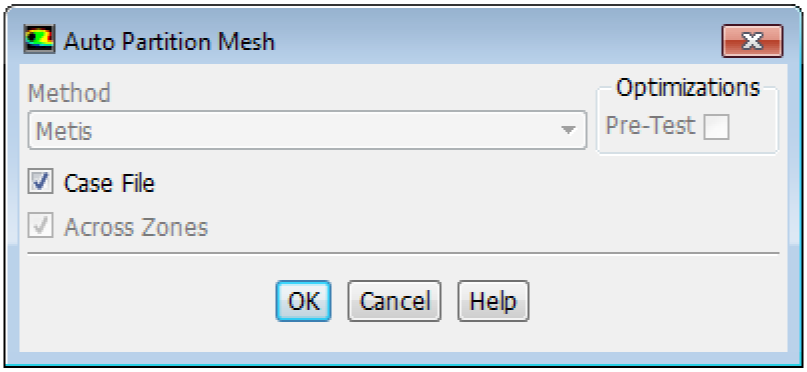
   c. Close the **Parallel Connectivity** dialog box.

3.5.2. Reading and Partitioning the Mesh

When you use the parallel solver, you need to subdivide (or partition) the mesh into groups of cells that can be solved on separate processors. If you read an unpartitioned mesh into the parallel solver, ANSYS Fluent will automatically partition it using the default partition settings. You can then check the partitions to see if you need to modify the settings and repartition the mesh.

1. Inspect the automatic partitioning settings.

Parallel → General → Auto Partition...



*If the **Case File** option is enabled (the default setting), and there exists a valid partition section in the case file (that is, one where the number of partitions in the case file divides evenly into the number of compute nodes), then that partition information will be used rather than repartitioning the mesh. You need to disable the **Case File** option only if you want to change other parameters in the **Auto Partition Mesh** dialog box.*

a. Make sure that **Case File** is enabled.

   *When the **Case File** option is enabled, ANSYS Fluent will automatically select a partitioning method for you. This is the preferred initial approach for most problems. In the next step, you will inspect the partitions created and be able to change them, if required.*

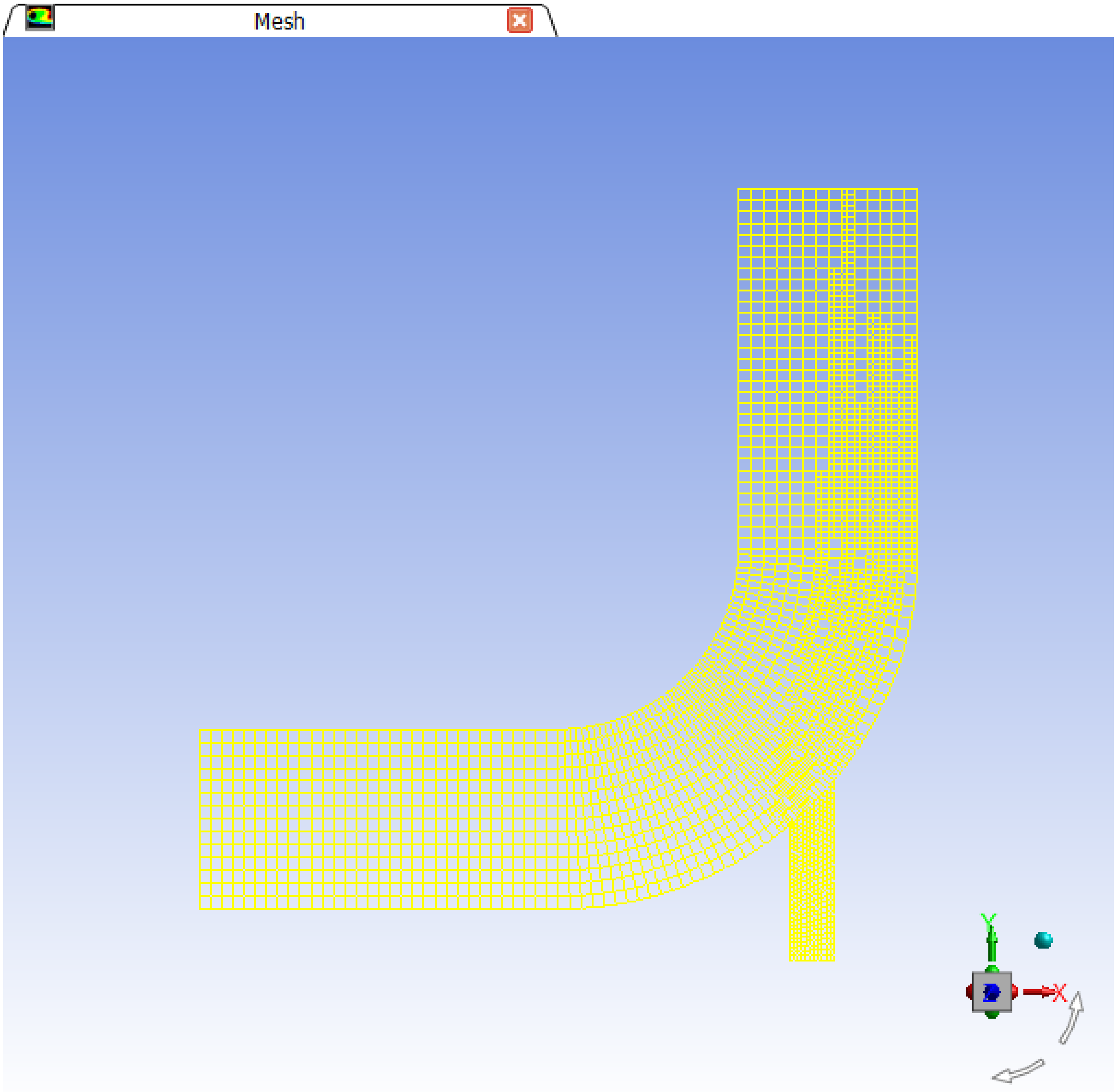
b. Click **OK** to close the **Auto Partition Mesh** dialog box.

2. Read the case file `elbow2.cas.gz`.

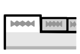
   File → Read → Case...

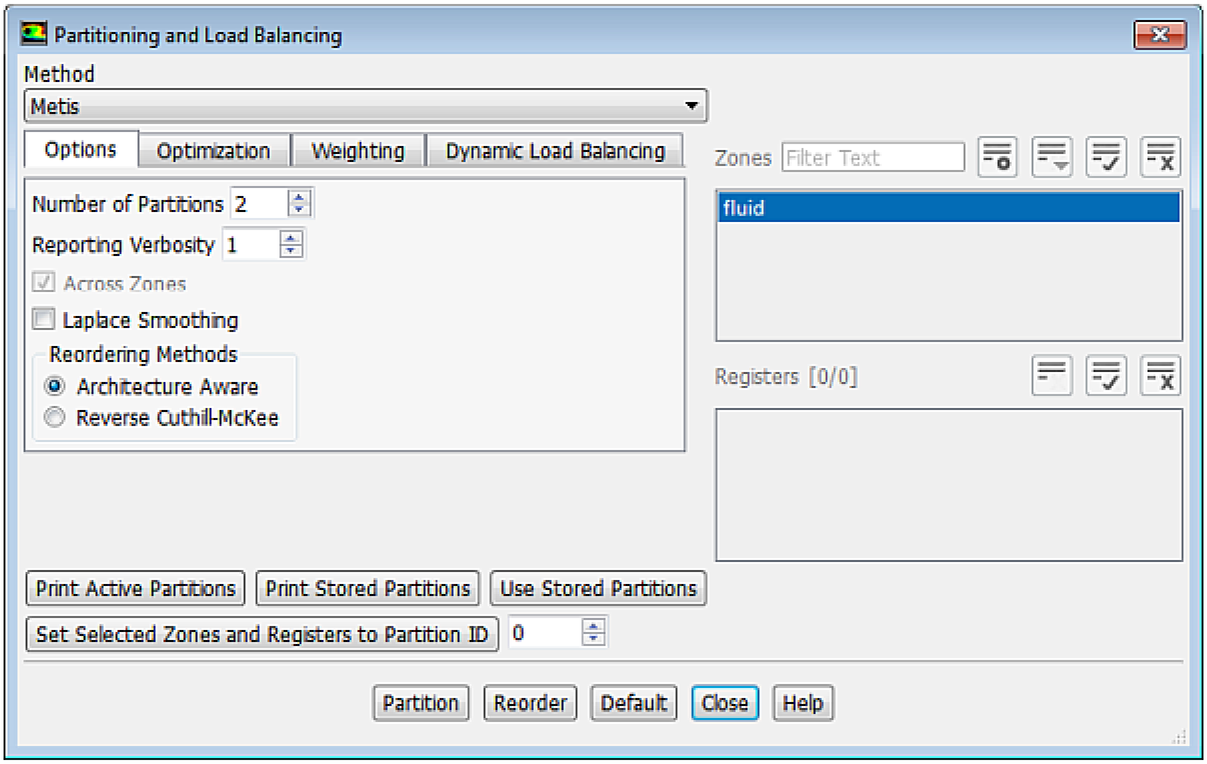
3. Examine the front view of the **symmetry** mesh zone (Figure 3.23: Mesh Along the Symmetry Plane for the Mixing Elbow (p. 196)).

Figure 3.23: Mesh Along the Symmetry Plane for the Mixing Elbow



4. Check the partition information.

 **Parallel → General → Partition/Load Balance...**



a. Click **Print Active Partitions**.

ANSYS Fluent will print the active partition statistics in the console.

```
2 Active Partitions:
    P    Cells I-Cells Cell Ratio   Faces I-Faces Face Ratio Neighbors Load   Ext Cells
    0    11570     439      0.038   37868     633      0.017         1    1         410
    1    11305     410      0.036   37856     633      0.017         1    1         439

   ----------------------------------------------------------------------
   Collective Partition Statistics:        Minimum   Maximum   Total
   ----------------------------------------------------------------------
   Cell count                              11305     11570     22875
   Mean cell count deviation               -1.2%     1.2%
   Partition boundary cell count           410       439       849
   Partition boundary cell count ratio     3.6%      3.8%      3.7%

   Face count                              37856     37868     75091
   Mean face count deviation               -0.0%     0.0%
   Partition boundary face count           633       633       633
   Partition boundary face count ratio     1.7%      1.7%      0.8%

   Partition neighbor count                1         1
   ----------------------------------------------------------------------
   Partition Method                        Metis
   Stored Partition Count                  2
```

Note

ANSYS Fluent distinguishes between two cell partition schemes within a parallel problem: the active cell partition, and the stored cell partition. Here, both are set to the cell partition that was created upon reading the case file. If you repartition the mesh using the **Partitioning and Load Balancing** dialog box, the new partition will be referred to as the stored cell partition. To make it the active cell partition, you need to click the **Use Stored Partitions** button in the **Partitioning and Load Balan-**

cing dialog box. The active cell partition is used for the current calculation, while the stored cell partition (the last partition performed) is used when you save a case file. This distinction is made mainly to allow you to partition a case on one machine or network of machines and solve it on a different one.

For details, see the chapter on parallel processing in the Fluent User's Guide, available in the help viewer or on the ANSYS Customer Portal (http://support.ansys.com/documentation).

b. Review the partition statistics.

An optimal partition should produce an equal number of cells in each partition for load balancing, a minimum number of partition interfaces to reduce interpartition communication, and a minimum number of partition neighbors to reduce the startup time for communication. In the displayed partition statistics, you will be looking for relatively small values of mean cell and face count deviation, and total partition boundary cell and face count ratio. Values less than 5% are considered reasonable in most cases. However, with very large core counts and/or especially complex cases, larger values may be unavoidable.

c. Close the **Partitioning and Load Balancing** dialog box.

5. Examine the partitions graphically.

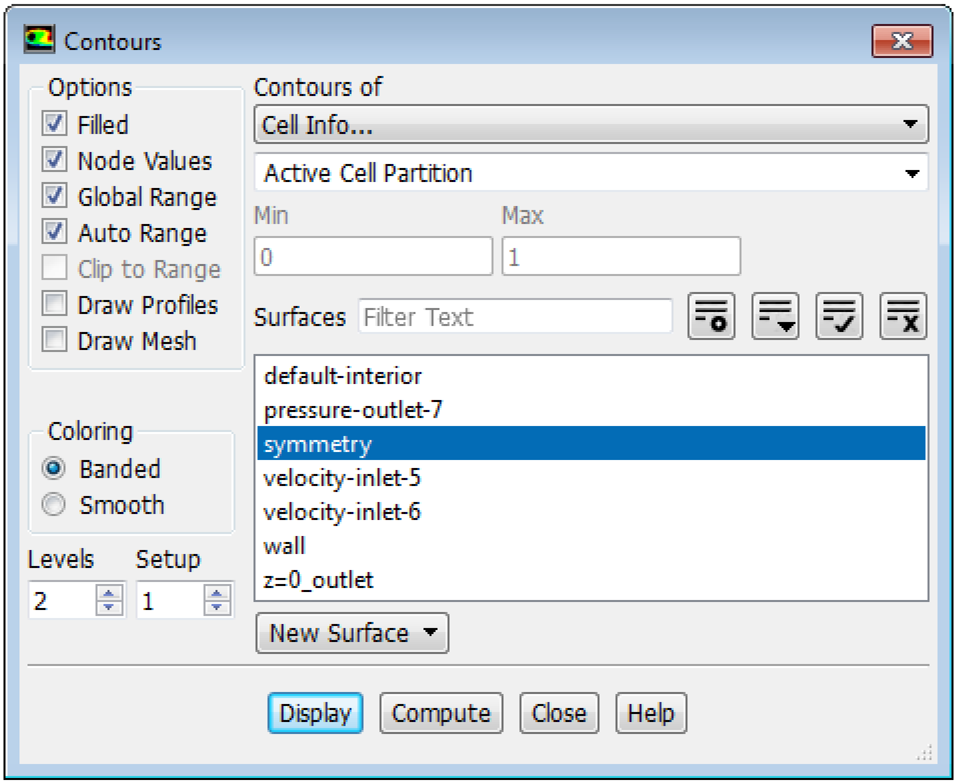
a. Initialize the solution using the default values.

Solving → **Initialization** → **Initialize**

In order to use the ***Contours*** *dialog box to inspect the partition you just created, you have to initialize the solution, even though you are not going to solve the problem at this point. The default values are sufficient for this initialization.*

b. Display the cell partitions (Figure 3.24: Cell Partitions (p. 200)).

Postprocessing → **Graphics** → **Contours** → **Edit...**



i. Make sure that **Filled** is selected in the **Options** group box.

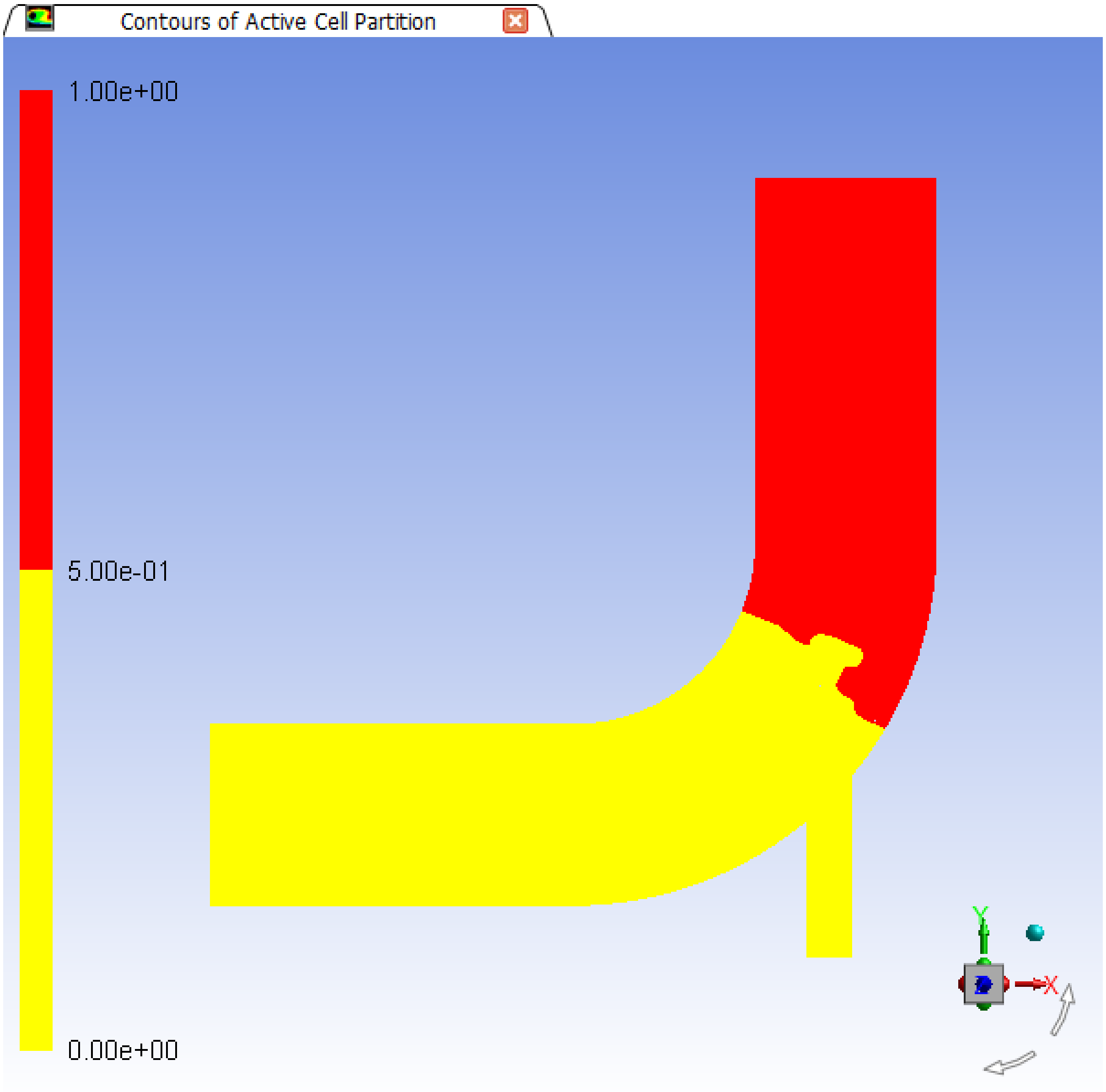
ii. Select **Cell Info...** and **Active Cell Partition** from the **Contours of** drop-down lists.

iii. Select **symmetry** from the **Surfaces** selection list.

iv. Set **Levels** to 2, which is the number of compute nodes.

v. Click **Display** and close the **Contours** dialog box.

Figure 3.24: Cell Partitions



As shown in Figure 3.24: Cell Partitions (p. 200), the cell partitions are acceptable for this problem. The position of the interface reveals that the criteria mentioned earlier will be matched. If you are dissatisfied with the partitions, you can use the ***Partitioning and Load Balancing*** *dialog box to repartition the mesh. Recall that, if you want to use the modified partitions for a calculation, you will need to make the stored cell partition the active cell partition by either clicking the* ***Use Stored Partitions*** *button in the* ***Partitioning and Load Balancing*** *dialog box or saving the case file and reading it back into ANSYS Fluent.*

For details about the procedure and options for manually partitioning a mesh, see the Fluent User's Guide, available in the help viewer or on the ANSYS Customer Portal (http://support.ansys.com/documentation).

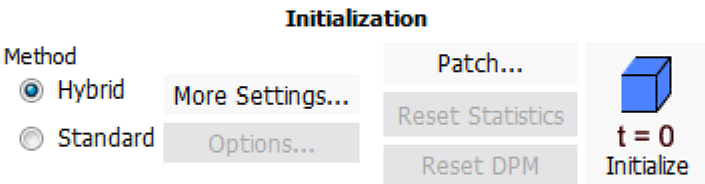
6. Save the case file with the partitioned mesh (`elbow4.cas.gz`).

   File → Write → Case...

3.5.3. Solution

1. Initialize the flow field.

   Solving → Initialization

   a. Make sure that **Hybrid** is selected under **Method** (this is the default).

   b. Click **Initialize**.

      *A **Question** dialog box will open, asking if you want to discard the data generated during the first initialization, which was used to inspect the cell partitions.*

   c. Click **OK** in the **Question** dialog box to discard the data.

2. Enable the plotting of residuals during the calculation.

   Solving → Reports → Residuals...

3. Start the calculation by requesting 90 iterations.

   Solving → Run Calculation

   *A **Question** dialog box will open, warning that `outlet-temp-avg-rfile.out` already exists and asking if you want to create a new file. Click **Yes**.*

   The solution will converge in approximately 36 iterations.

4. Save the data file (`elbow4.dat.gz`).

   File → Write → Data...

3.5.4. Checking Parallel Performance

Generally, you will use the parallel solver for large, computationally intensive problems, and you will want to check the parallel performance to determine if any optimization is required. Although the example in this tutorial is a simple 3D case, you will check the parallel performance as an exercise.

For details, see the chapter on parallel processing in the Fluent User's Guide, available in the help viewer or on the ANSYS Customer Portal (http://support.ansys.com/documentation).

Parallel → Timer → Usage

```
Performance Timer for 36 iterations on 2 compute nodes
  Average wall-clock time per iteration:              0.097 sec
  Global reductions per iteration:                       55 ops
  Global reductions time per iteration:               0.000 sec (0.0%)
  Message count per iteration:                          637 messages
```

```
Data transfer per iteration:                     0.550 MB
LE solves per iteration:                             4 solves
LE wall-clock time per iteration:                0.046 sec (47.7%)
LE global solves per iteration:                      4 solves
LE global wall-clock time per iteration:         0.000 sec (0.1%)
LE global matrix maximum size:                    30
AMG cycles per iteration:                        6.889 cycles
Relaxation sweeps per iteration:                   655 sweeps
Relaxation exchanges per iteration:                  0 exchanges

Total wall-clock time:                           3.499 sec
```

The most accurate way to evaluate parallel performance is by running the same parallel problem on 1 CPU and on n CPUs, and comparing the Total wall-clock time (elapsed time for the iterations) in both cases. Ideally you would want to have the Total wall-clock time with n CPUs be $1/n$ times the Total wall-clock time with 1 CPU. In practice, this improvement will be reduced by the performance of the communication subsystem of your hardware, and the overhead of the parallel process itself.

Note

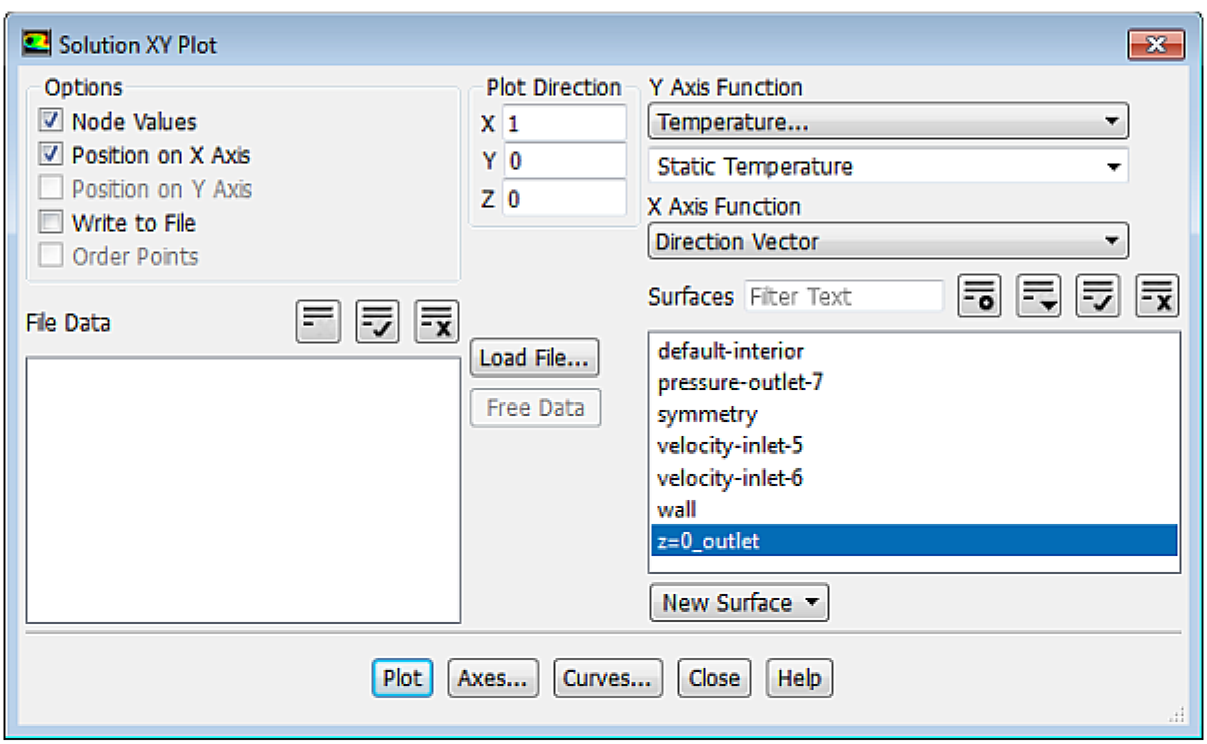
The wall clock time and the ratio of iterations to convergence time may differ depending on the type of computer you are running (for example, Windows 64, Linux 64, and so on).

3.5.5. Postprocessing

Here, two plots are generated so that you can confirm that the results obtained with the parallel solver are the same as those obtained previously with the serial solver.

1. Display an XY plot of temperature along the centerline of the outlet (Figure 3.25: Temperature Distribution at the Outlet (p. 203)).

   Postprocessing → Plots → XY Plot → Edit...

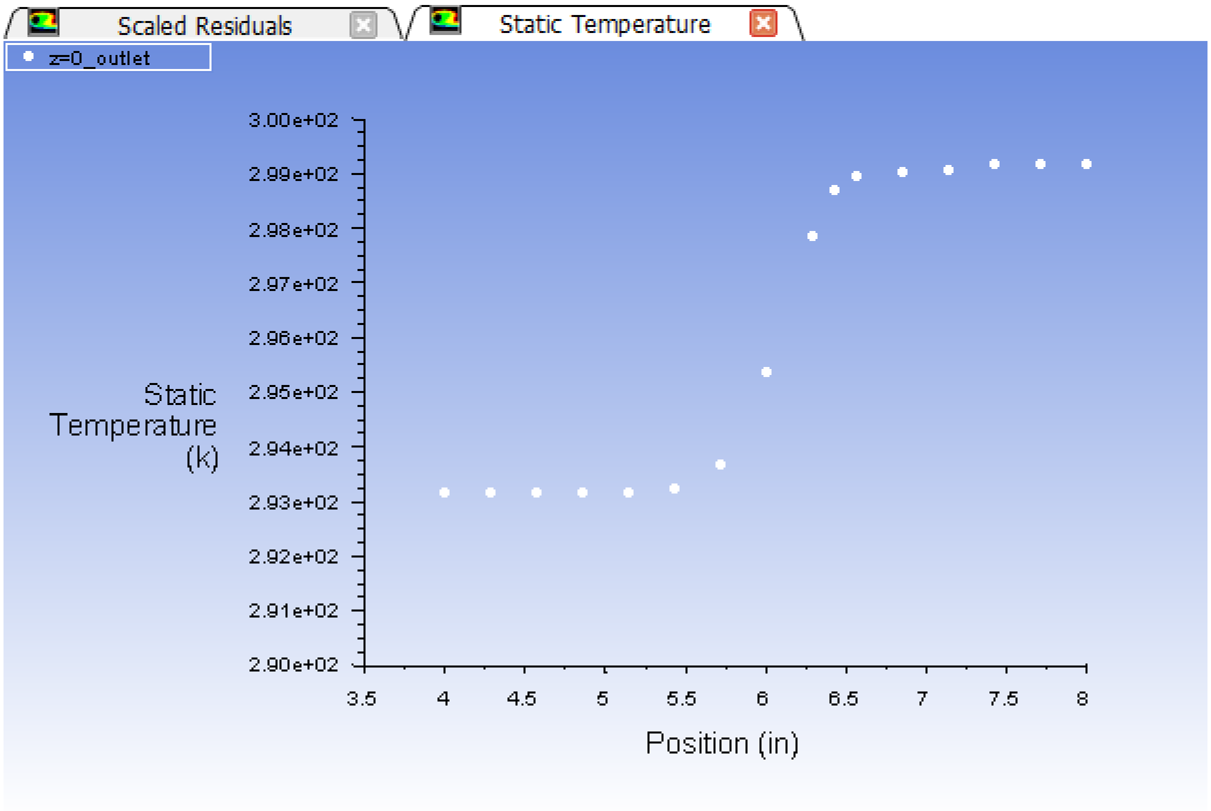


a. Select **Temperature...** and **Static Temperature** from the **Y Axis Function** drop-down lists.

b. Select **z=0_outlet** from the **Surfaces** selection list.

c. Click **Plot** and close the **Solution XY Plot** dialog box.

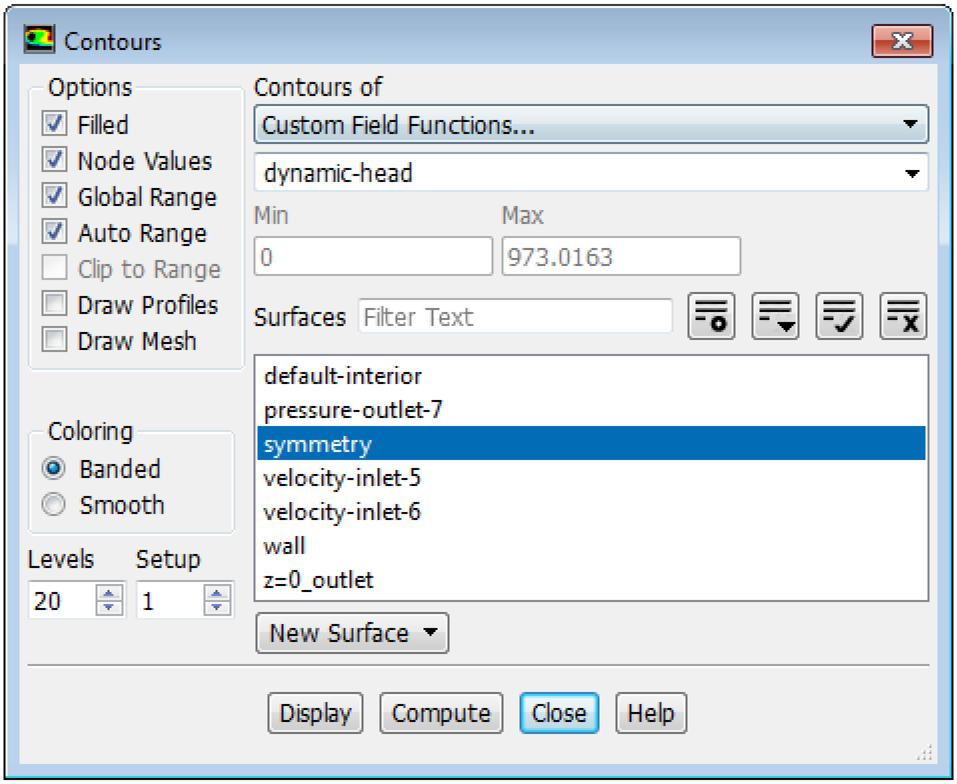
Figure 3.25: Temperature Distribution at the Outlet



Compare the plot of Temperature at the Outlet with the serial solution shown in Figure 3.20: Outlet Temperature Profile for the Adapted Coupled Solver Solution (p. 186).

2. Display filled contours of the custom field function **dynamic-head** (Figure 3.26: Contours of the Custom Field Function, Dynamic Head (p. 205)).

Postprocessing → Graphics → Contours → Edit...



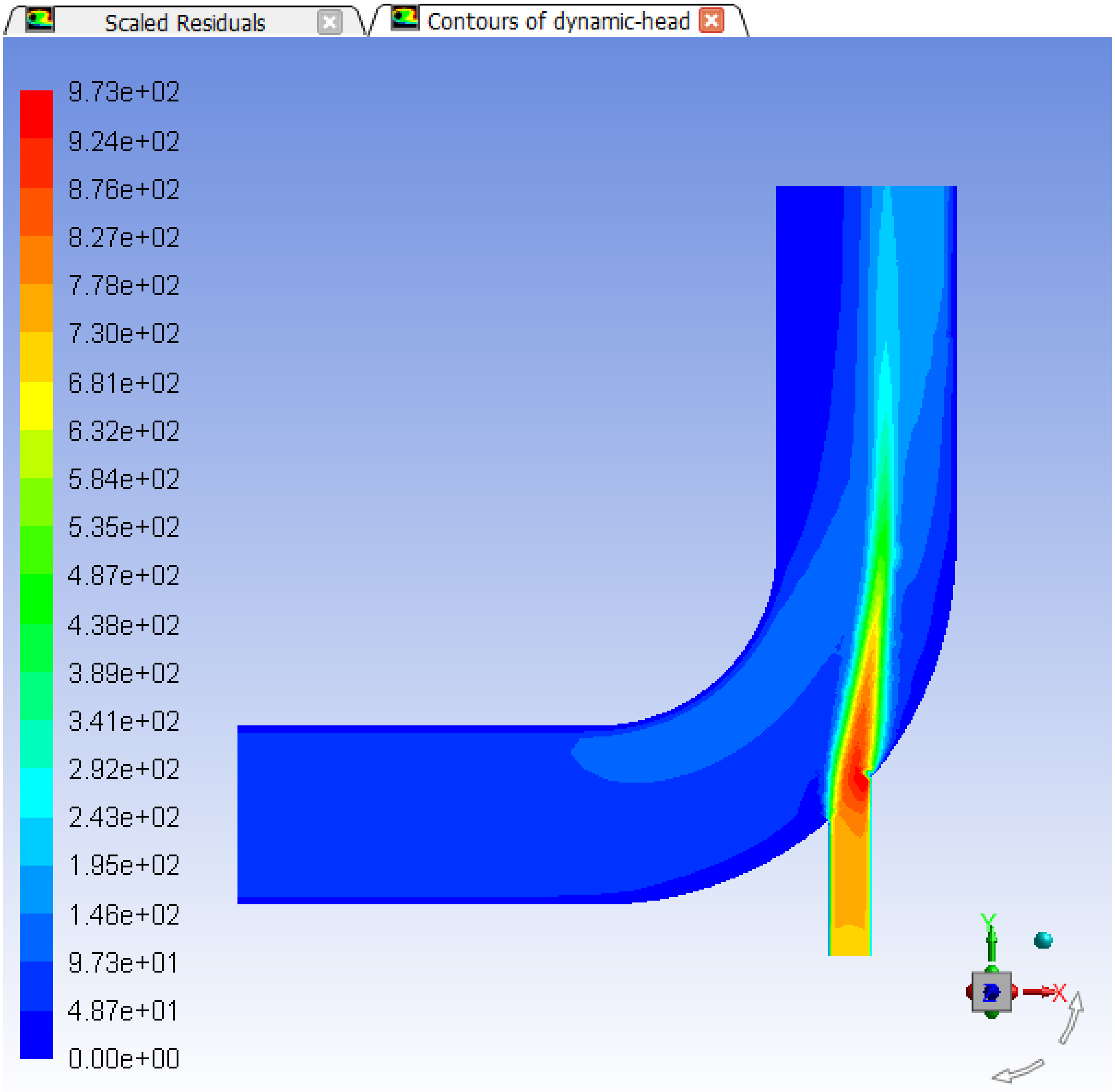
a. Select **Custom Field Functions...** and **dynamic-head** from the **Contours of** drop-down lists.

b. Enter `20` for **Levels**.

c. Select **symmetry** from the **Surfaces** selection list.

d. Click **Display** and close the **Contours** dialog box.

Figure 3.26: Contours of the Custom Field Function, Dynamic Head



3.6. Summary

A comparison of the convergence speed for the SIMPLE and Coupled pressure-velocity coupling schemes indicates that the latter converges much faster. With more complex meshes, the difference in speed between the two schemes can be significant.

Chapter 4: Modeling Periodic Flow and Heat Transfer

This tutorial is divided into the following sections:

4.1. Introduction

Many industrial applications, such as steam generation in a boiler or air cooling in the coil of an air conditioner, can be modeled as two-dimensional periodic heat flow. This tutorial illustrates how to set up and solve a periodic heat transfer problem, given a pre-generated mesh.

The system that is modeled is a bank of tubes containing a flowing fluid at one temperature that is immersed in a second fluid in cross flow at a different temperature. Both fluids are water, and the flow is classified as laminar and steady, with a Reynolds number of approximately 100. The mass flow rate of the cross flow is known and the model is used to predict the flow and temperature fields that result from convective heat transfer.

Due to symmetry of the tube bank and the periodicity of the flow inherent in the tube bank geometry, only a portion of the geometry will be modeled in ANSYS Fluent, with symmetry applied to the outer boundaries. The resulting mesh consists of a periodic module with symmetry. In the tutorial, the inlet boundary will be redefined as a periodic zone, and the outflow boundary defined as its shadow.

This tutorial demonstrates how to do the following:

- Create periodic zones.
- Define a specified periodic mass flow rate.
- Model periodic heat transfer with specified temperature boundary conditions.
- Calculate a solution using the pressure-based, pseudo-transient, coupled solver.
- Plot temperature profiles on specified isosurfaces.

4.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)

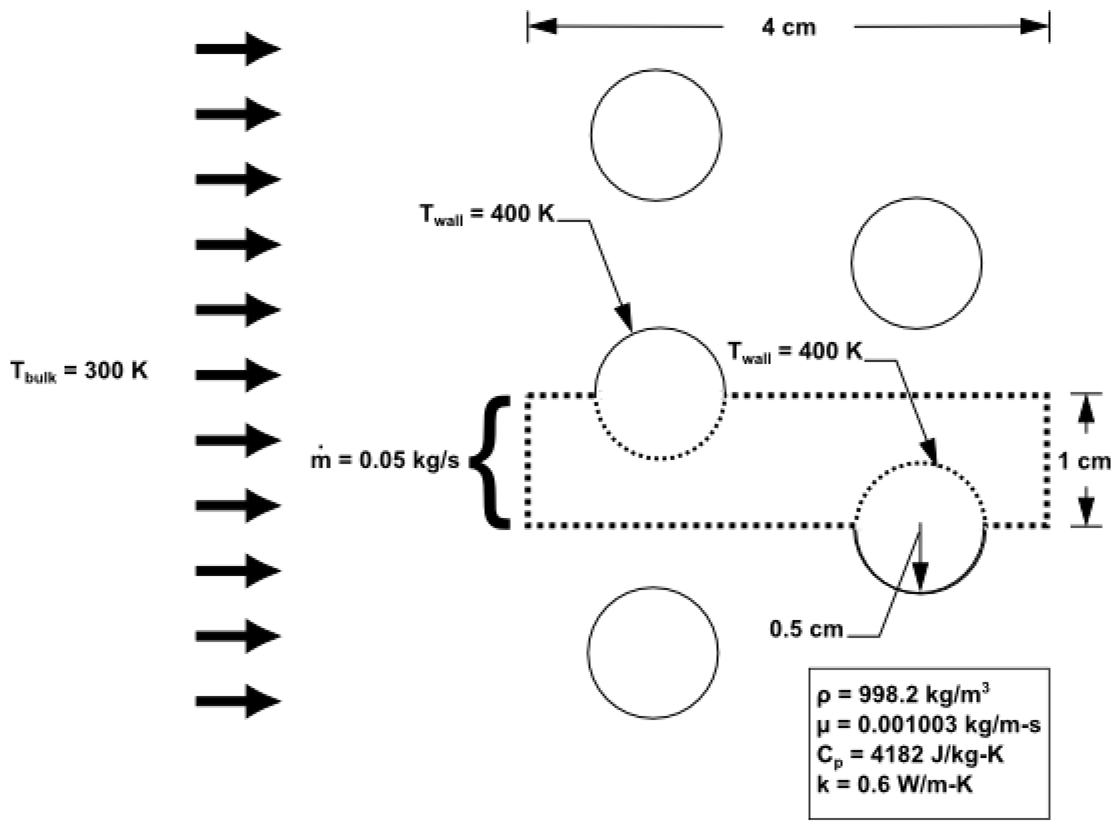
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

and that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

4.3. Problem Description

This problem considers a 2D section of a tube bank. A schematic of the problem is shown in Figure 4.1: Schematic of the Problem (p. 208). The bank consists of uniformly-spaced tubes with a diameter of 1 cm, which are staggered across the cross-fluid flow. Their centers are separated by a distance of 2 cm in the x direction, and 1 cm in the y direction. The bank has a depth of 1 m.

Figure 4.1: Schematic of the Problem



Because of the symmetry of the tube bank geometry, only a portion of the domain must be modeled. The computational domain is shown in outline in Figure 4.1: Schematic of the Problem (p. 208). A mass flow rate of 0.05 kg/s is applied to the inlet boundary of the periodic module. The temperature of the tube wall (T_{wall}) is 400 K and the bulk temperature of the cross flow water (T_{bulk}) is 300 K. The properties of water that are used in the model are shown in Figure 4.1: Schematic of the Problem (p. 208).

4.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

4.4.1. Preparation

To prepare for running this tutorial:

1. Set up a working folder on the computer you will be using.
2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

   Note

   If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

3. Enter the name of this tutorial into the search bar.
4. Narrow the results by using the filter on the left side of the page.
   a. Click **ANSYS Fluent** under **Product**.
   b. Click **18.0** under **Version**.
5. Select this tutorial from the list.
6. Click the **periodic_flow_heat_R180.zip** link to download the input files.
7. Unzip `periodic_flow_heat_R180.zip` to your working folder.

   The file `tubebank.msh` can be found in the `periodic_flow_heat` folder created after unzipping the file.

8. Use Fluent Launcher to start the **2D** version of ANSYS Fluent.

   Fluent Launcher displays your **Display Options** preferences from the previous session.

   For more information about the Fluent Launcher, see starting ANSYS Fluent using the Fluent Launcher in the Fluent Getting Started Guide.

9. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.
10. Ensure that you are running in single precision (disable **Double Precision**).
11. Select **Serial** under **Processing Options**.

4.4.2. Mesh

1. Read the mesh file `tubebank.msh`.

File → Read → Mesh...

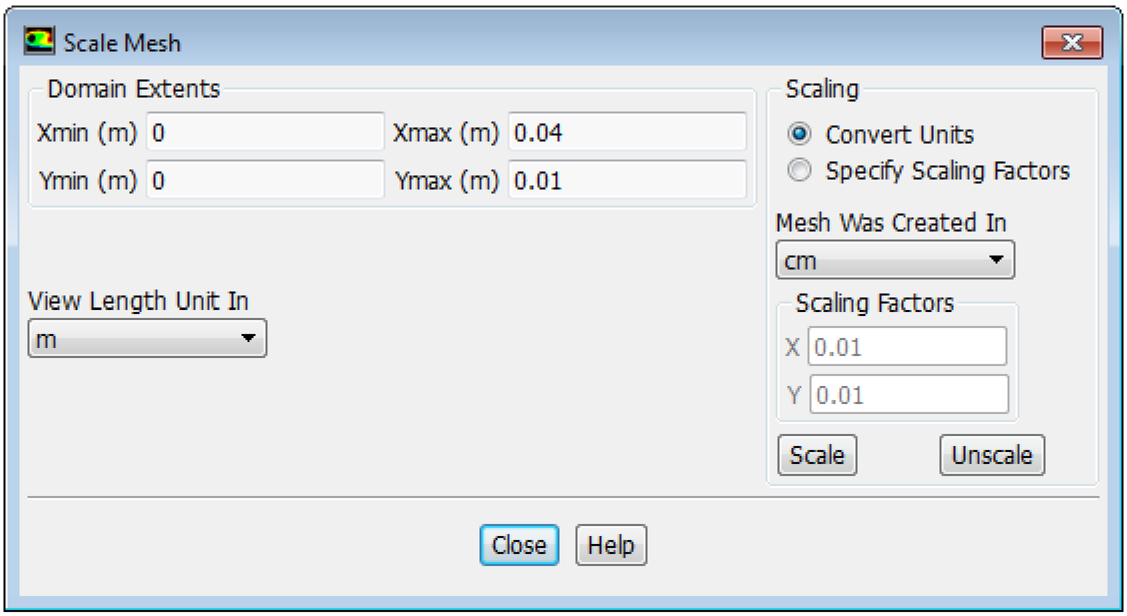
2. Check the mesh.

   Setting Up Domain → Mesh → Check

   ANSYS Fluent will perform various checks on the mesh and report the progress in the ANSYS Fluent console window. Ensure that the minimum volume reported is a positive number.

3. Scale the mesh.

   Setting Up Domain → Mesh → Scale...



   a. Select **cm** (centimeters) from the **Mesh Was Created In** drop-down list in the **Scaling** group box.

   b. Click **Scale** to scale the mesh.

   c. Close the **Scale Mesh** dialog box.

4. Check the mesh.

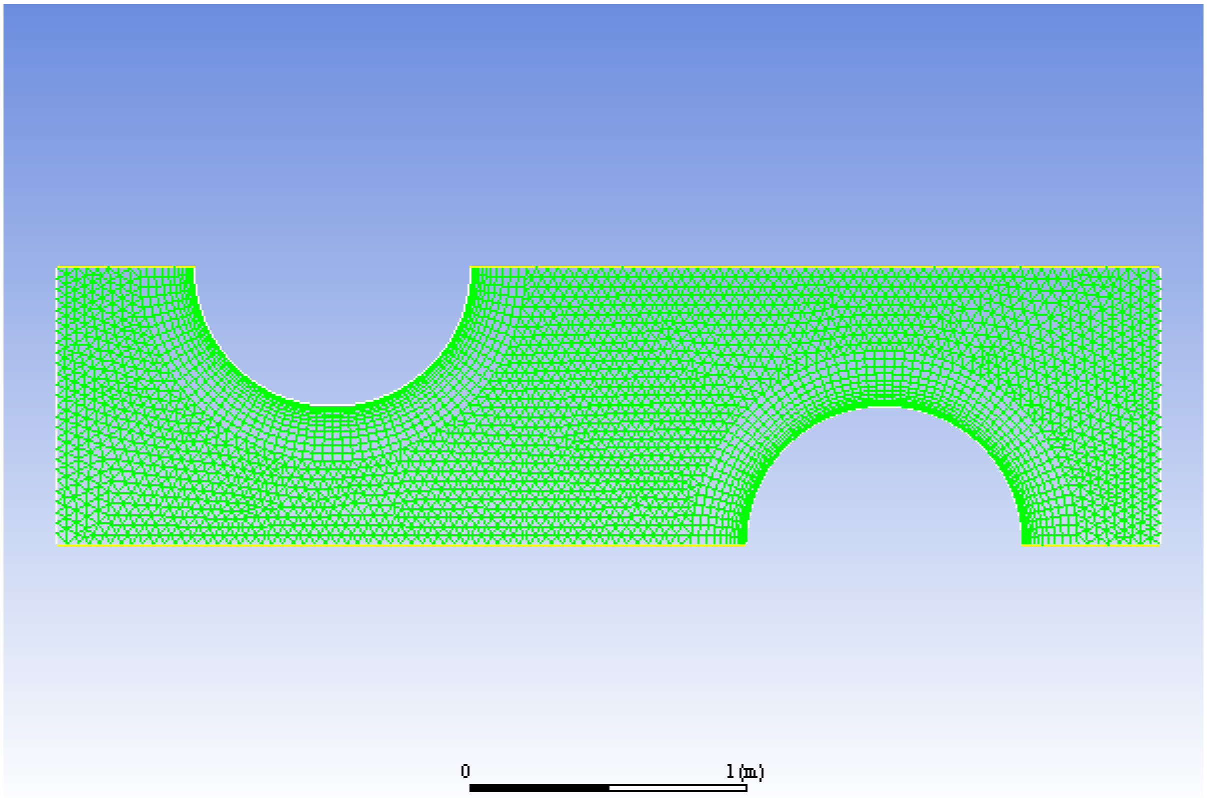
   Setting Up Domain → Mesh → Check

   Note

   It is a good idea to check the mesh after you manipulate it (scale, convert to polyhedra, merge, separate, fuse, add zones, or smooth and swap.) This will ensure that the quality of the mesh has not been compromised.

5. Examine the mesh (Figure 4.2: Mesh for the Periodic Tube Bank (p. 211)).

Figure 4.2: Mesh for the Periodic Tube Bank



Quadrilateral cells are used in the regions surrounding the tube walls and triangular cells are used for the rest of the domain, resulting in a hybrid mesh (see Figure 4.2: Mesh for the Periodic Tube Bank (p. 211)). The quadrilateral cells provide better resolution of the viscous gradients near the tube walls. The remainder of the computational domain is filled with triangular cells for the sake of convenience.

Tip

You can use the right mouse button to probe for mesh information in the graphics window. If you click the right mouse button on any node in the mesh, information will be displayed in the ANSYS Fluent console about the associated zone, including the name of the zone. This feature is especially useful when you have several zones of the same type and you want to distinguish between them quickly.

---

6. Create the periodic zone.

   *The inlet (**wall-9**) and outflow (**wall-12**) boundaries currently defined as wall zones need to be redefined as periodic using the text user interface. The **wall-9** boundary will be redefined as a translationally periodic zone and **wall-12** as a periodic shadow of **wall-9**.*

   a. Press **Enter** in the console to get the command prompt (>).

   b. Enter the text command and input the responses to the prompts as shown. Press **Enter** after each entry.

   ```
   > mesh/modify-zones/make-periodic Enter
   ```

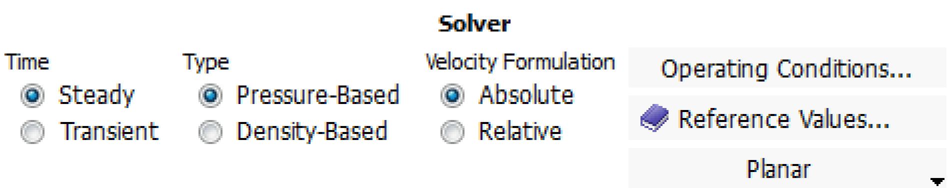
   You will be prompted to enter the zones corresponding to the periodic boundaries and specify the configuration of the periodicity.

```
Periodic zone [()] 9 Enter

Shadow zone [()] 12 Enter

Rotational periodic? (if no, translational) [yes] no Enter
Create periodic zones? [yes] yes Enter

Auto detect translation vector? [yes] yes Enter

 zone 12 deleted

 created periodic zones.
```

4.4.3. General Settings

1. Retain the default settings for the solver.

   Setting Up Physics → **Solver**



4.4.4. Models

1. Enable heat transfer.

   Setting Up Physics → **Models** → **Energy**

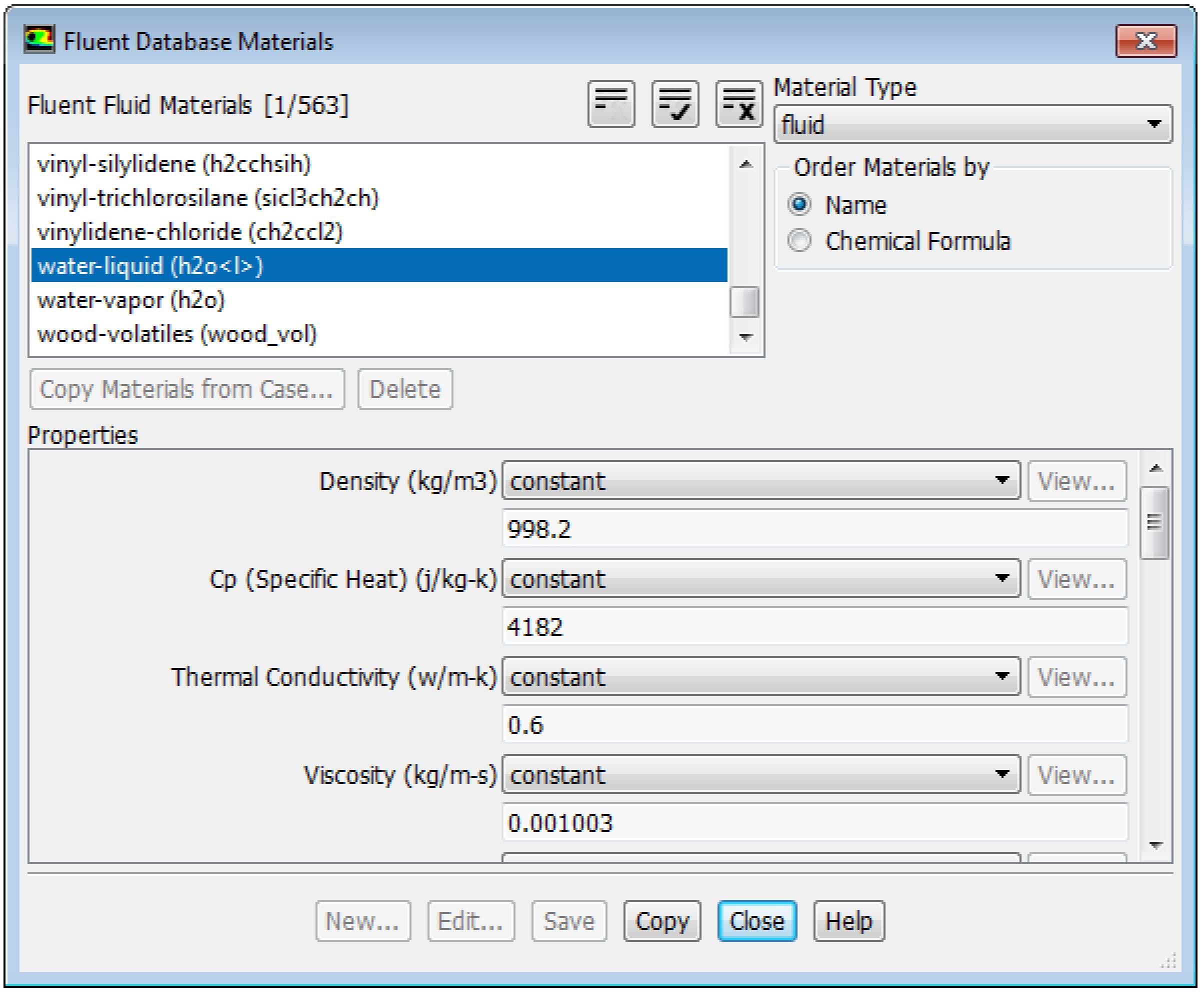
4.4.5. Materials

The default properties for water defined in ANSYS Fluent are suitable for this problem. In this step, you will make sure that this material is available for selecting in future steps.

1. Add water to the list of fluid materials by copying it from the ANSYS Fluent materials database.

   Setup → **Materials** → **Fluid** → **Create/Edit...**

   a. Click **Fluent Database...** in the **Create/Edit Materials** dialog box to open the **Fluent Database Materials** dialog box.

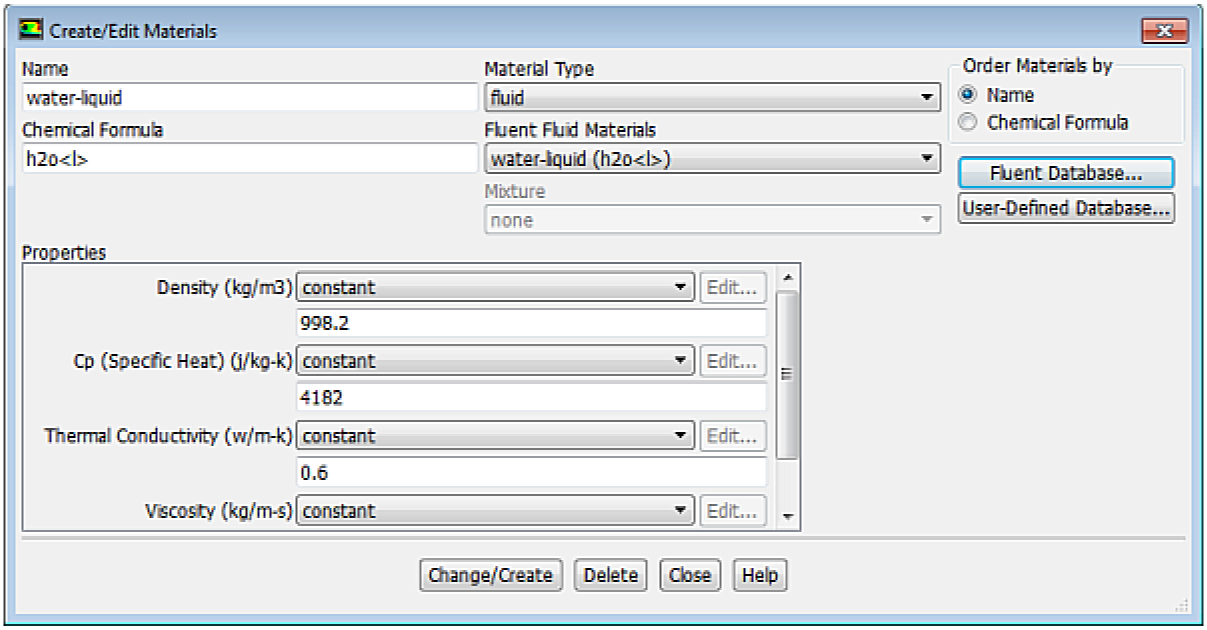


i. Select **water-liquid (h2o<l>)** in the **Fluent Fluid Materials** selection list.

   *Scroll down the list to find **water-liquid (h2o<l>)**. Selecting this item will display the default properties in the dialog box.*

ii. Click **Copy** and close the **Fluent Database Materials** dialog box.

*The **Create/Edit Materials** dialog box will now display the copied properties for **water-liquid**.*

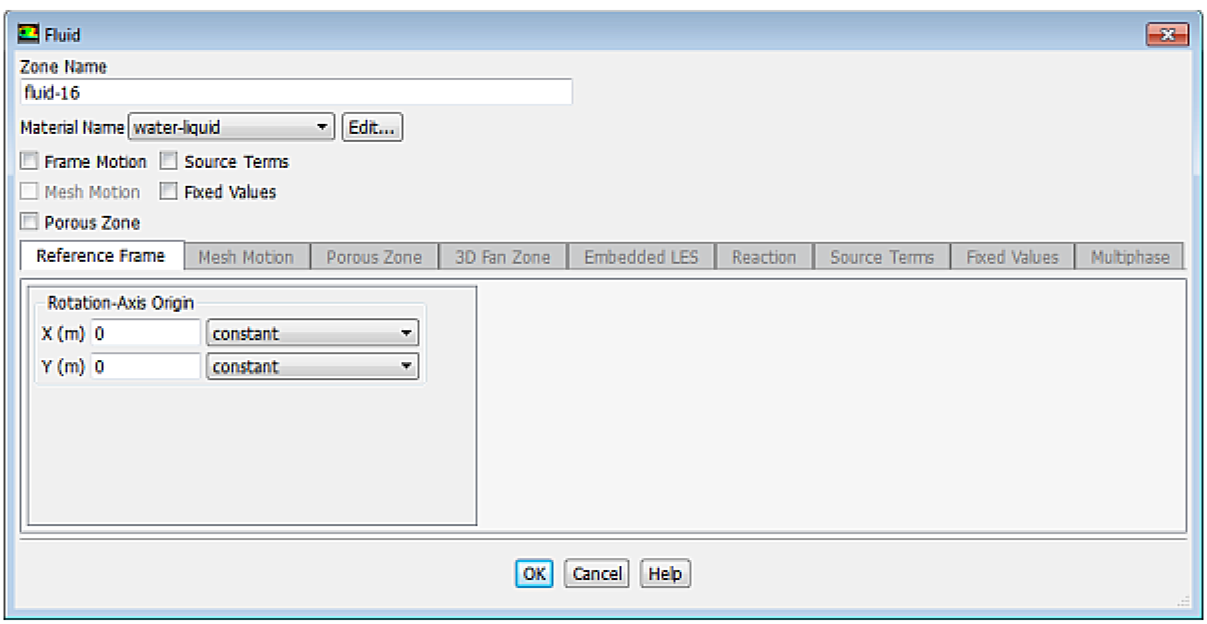


b. Click **Change/Create** and close the **Create/Edit Materials** dialog box.

4.4.6. Cell Zone Conditions

1. Set the cell zone conditions for the continuum fluid zone (**fluid-16**).

   Setup → **Cell Zone Conditions** → **fluid-16** → **Edit...**



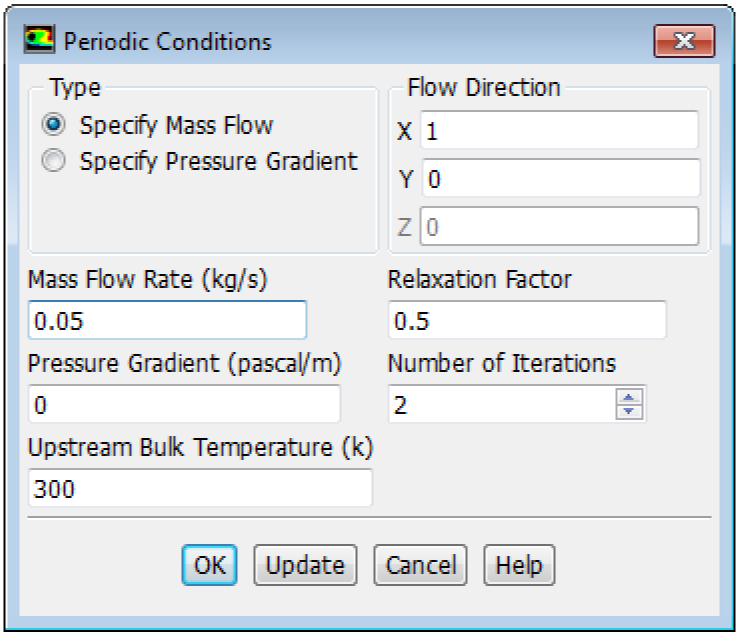
a. Select **water-liquid** from the **Material Name** drop-down list.

b. Click **OK** to close the **Fluid** dialog box.

4.4.7. Periodic Conditions

1. Define the periodic flow conditions.

   Setup → **Boundary Conditions** → **periodic-9** → **Periodic Conditions...**



   a. Select **Specify Mass Flow** in the **Type** list.

      *This will allow you to specify the **Mass Flow Rate**.*

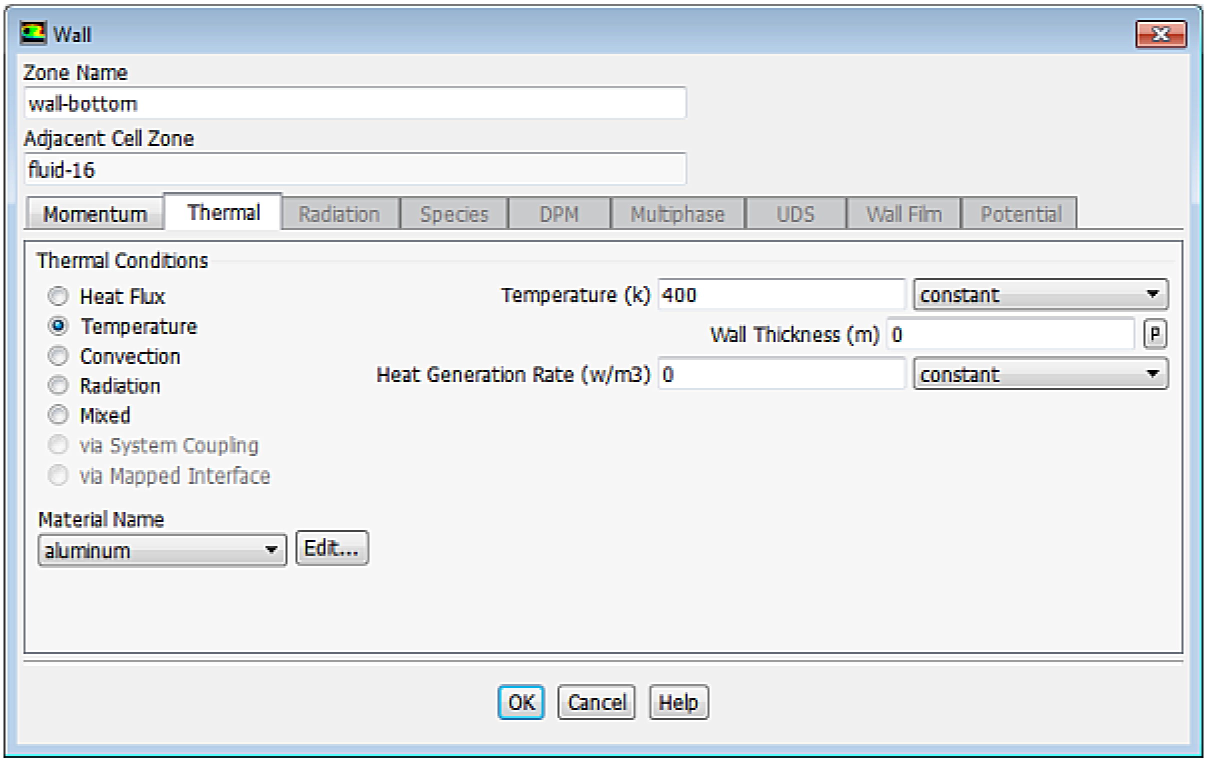
   b. Enter `0.05` kg/s for **Mass Flow Rate**.

   c. Click **OK** to close the **Periodic Conditions** dialog box.

4.4.8. Boundary Conditions

1. Set the boundary conditions for the bottom wall of the left tube (**wall-21**).

   Setup → **Boundary Conditions** → **wall-21** → **Edit...**



a. Enter `wall-bottom` for **Zone Name**.

b. Click the **Thermal** tab.

i. Select **Temperature** in the **Thermal Conditions** list.

ii. Enter `400` K for **Temperature**.

These settings will specify a constant wall temperature of 400 K.

c. Click **OK** to close the **Wall** dialog box.

2. Set the boundary conditions for the top wall of the right tube (**wall-3**).

Setup → **Boundary Conditions** → **wall-3** → **Edit...**

a. Enter `wall-top` for **Zone Name**.

b. Click the **Thermal** tab.

i. Select **Temperature** from the **Thermal Conditions** list.

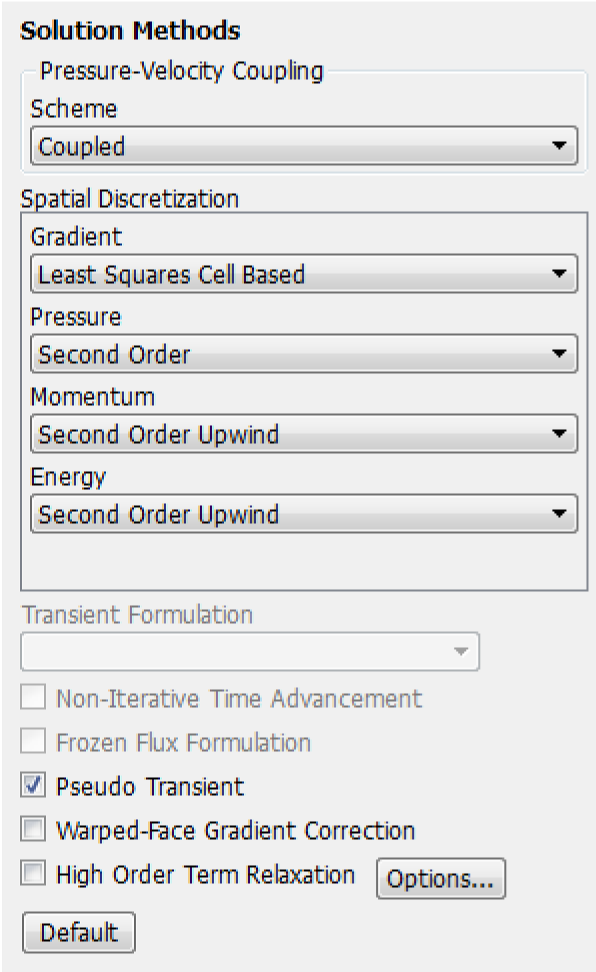
ii. Enter `400` K for **Temperature**.

c. Click **OK** to close the **Wall** dialog box.

4.4.9. Solution

1. Set the solution parameters.

Solving → **Solution** → **Methods...**



a. Select **Coupled** from the **Scheme** drop-down list in the **Pressure-Velocity Coupling** group box.

b. Retain the default setting of **Least Squares Cell Based** for the **Gradient** in the **Spatial Discretization** group box.

c. Retain the default setting of **Second Order** for the **Pressure** drop-down list.

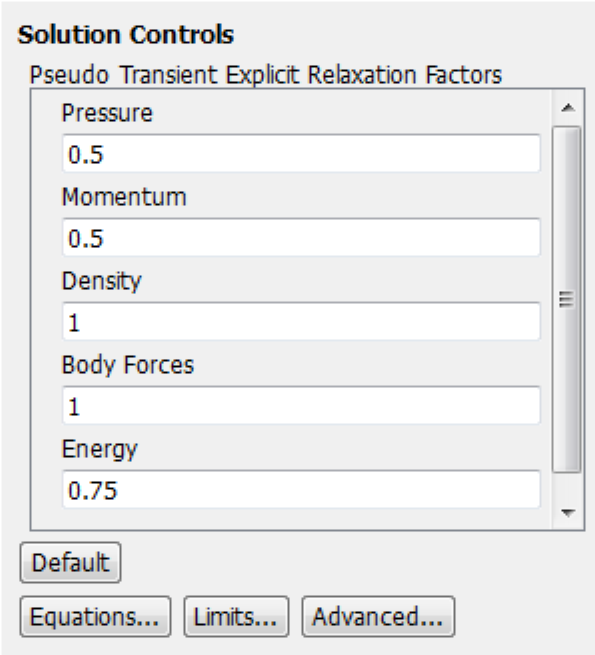
d. Retain the default setting of **Second Order Upwind** in the **Momentum** and **Energy** drop-down lists.

e. Enable **Pseudo Transient**.

   The Pseudo Transient option enables the pseudo transient algorithm in the coupled pressure-based solver. This algorithm effectively adds an unsteady term to the solution equations in order to improve stability and convergence behavior.

2. Set the solution controls.

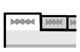
   Solving → **Controls** → **Controls...**

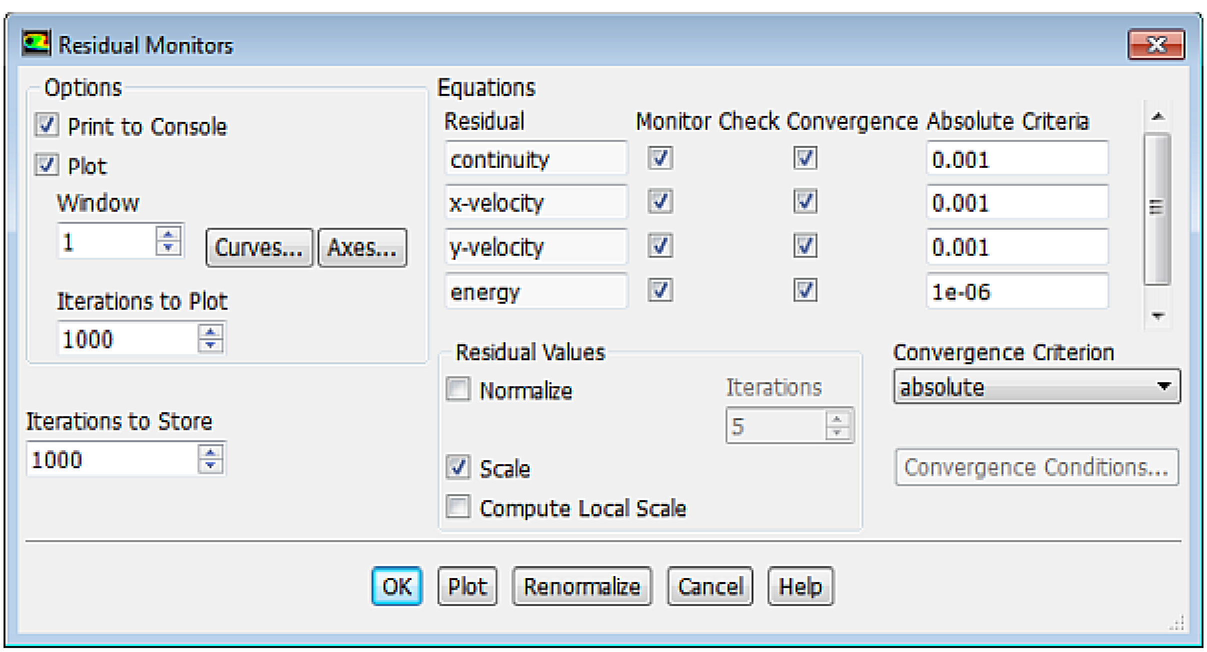


a. Retain the default values in the **Pseudo Transient Explicit Relaxation Factors** group box.

In some cases, the default Pseudo Transient Explicit Relaxation Factors may need to be reduced in order to prevent oscillation of residual values or stabilization of residual values above the convergence criteria. For additional information about setting Pseudo Transient Explicit Relaxation Factors, see the Fluent User's Guide.

3. Enable the plotting of residuals during the calculation.

 **Solving** → **Reports** → **Residuals...**



a. Ensure **Plot** is enabled in the **Options** group box.

b. Click **OK** to close the **Residual Monitors** dialog box.

4. Initialize the solution.

   You will first use the default Hybrid initialization method and then patch the fluid zone with the upstream temperature value.

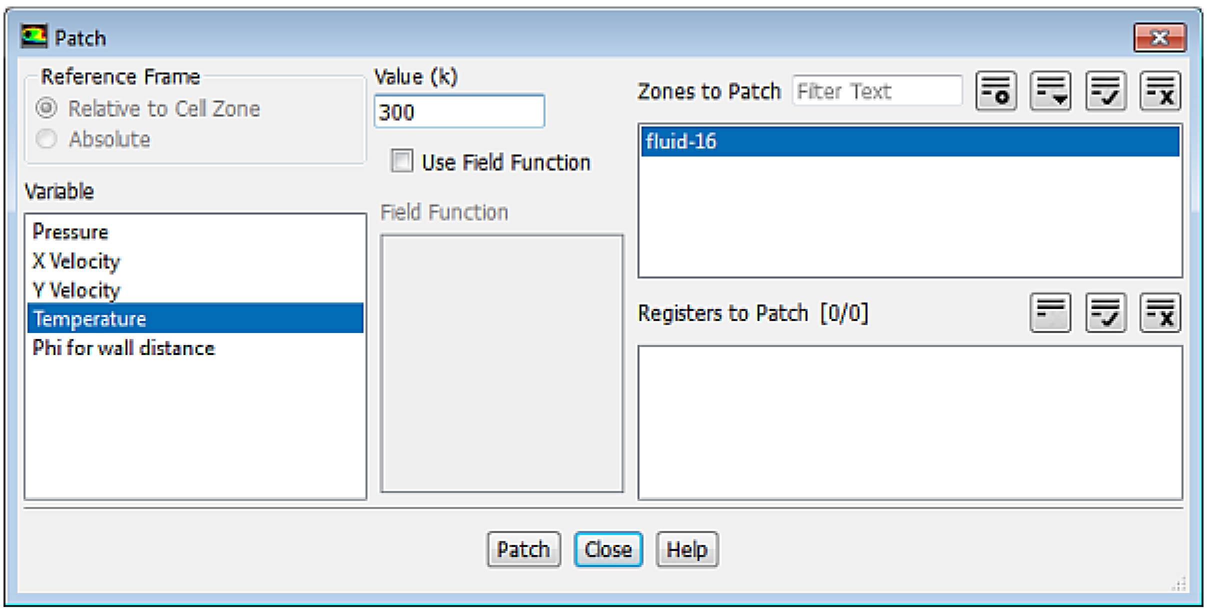
   a. Initialize using the default settings.

      Solving → **Initialization** → **Initialize**

   b. Patch the fluid zone with the bulk upstream temperature value.

      The Hybrid Initialization method computes the initial flow field based on inlet and outlet boundary conditions. In this case we have periodic boundary conditions with a specified upstream bulk temperature. You will patch the initialized solution with this temperature value in order to improve convergence.

      Solving → **Initialization** → **Patch...**



      i. Select **Temperature** in the **Variable** selection list.

      ii. Enter `300` for **Value (k)**.

         Recall that the upstream bulk temperature, T_{bulk}, is specified as 300 K.

      iii. Select **fluid-16** in the **Zones to Patch** selection list.

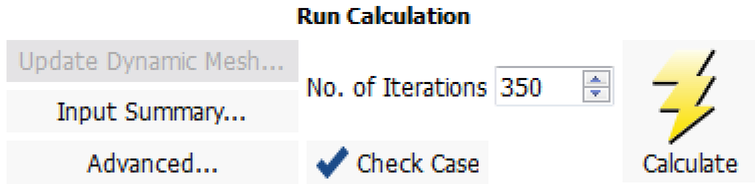
      iv. Click **Patch** and close the **Patch** dialog box.

5. Save the case file (`tubebank.cas.gz`).

   File → **Write** → **Case...**

6. Start the calculation by requesting 350 iterations.

   Solving → **Run Calculation**



a. Enter `350` for **No. of Iterations**.

b. Click **Calculate**.

The solution will converge in approximately 111 iterations.

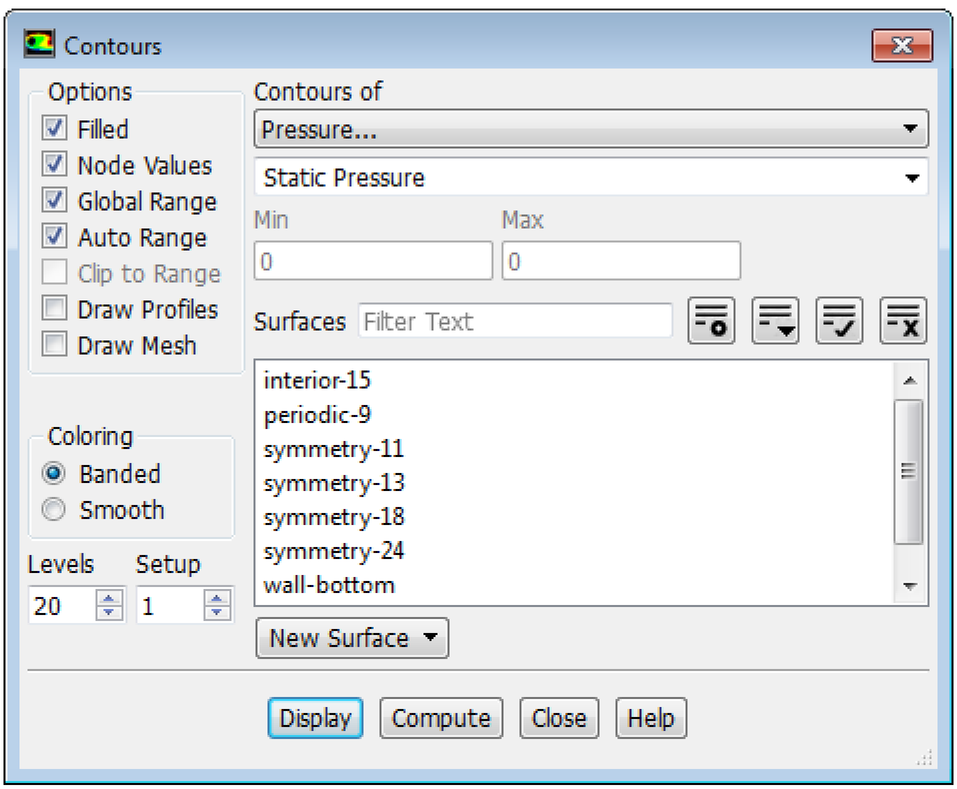
7. Save the case and data files (`tubebank.cas.gz` and `tubebank.dat.gz`).

File → **Write** → **Case & Data...**

4.4.10. Postprocessing

1. Display filled contours of static pressure (Figure 4.3: Contours of Static Pressure (p. 221)).

Postprocessing → **Graphics** → **Contours** → **Edit...**

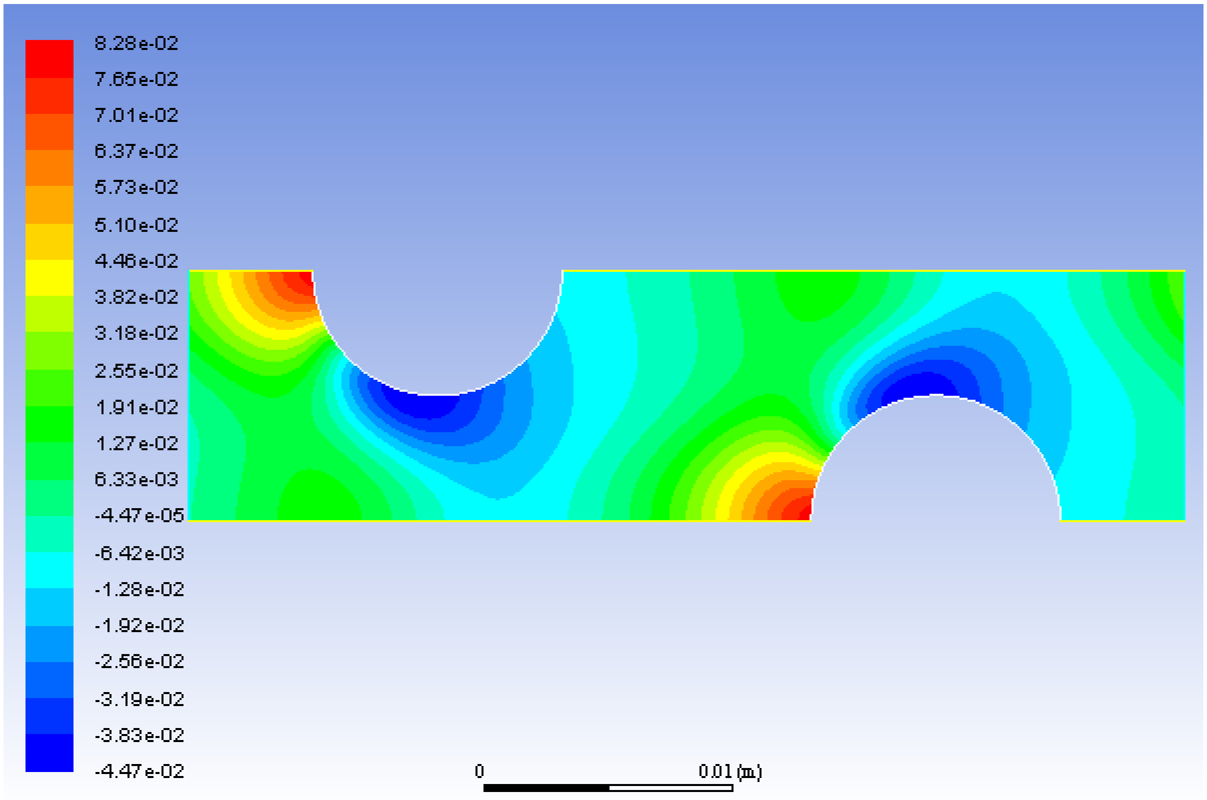


a. Enable **Filled** in the **Options** group box.

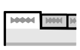
b. Retain the default selection of **Pressure...** and **Static Pressure** from the **Contours of** drop-down lists.

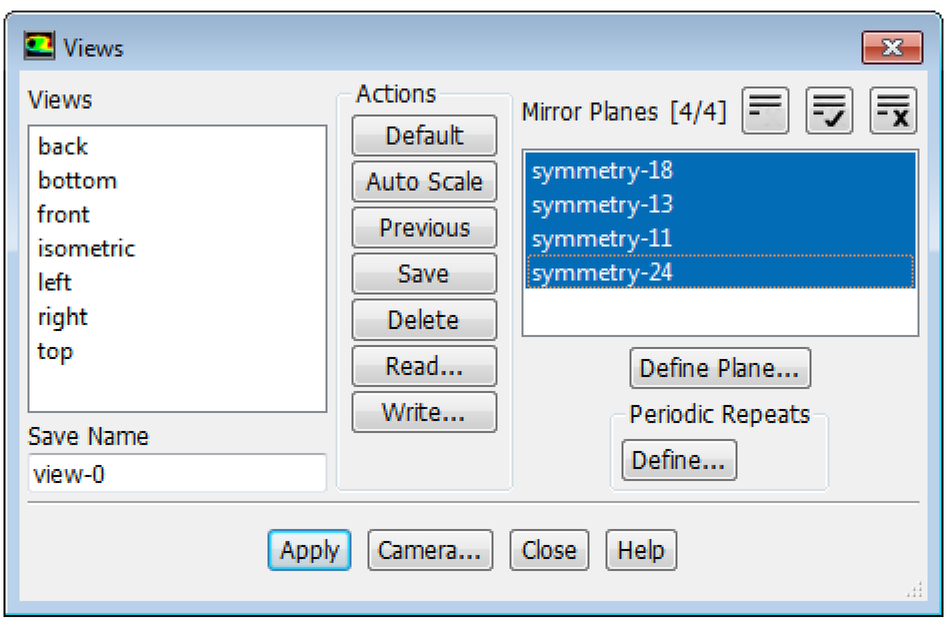
c. Click **Display**.

Figure 4.3: Contours of Static Pressure



d. Change the view to mirror the display across the symmetry planes (Figure 4.4: Contours of Static Pressure with Symmetry (p. 222)).

 **Viewing → Display → Views...**



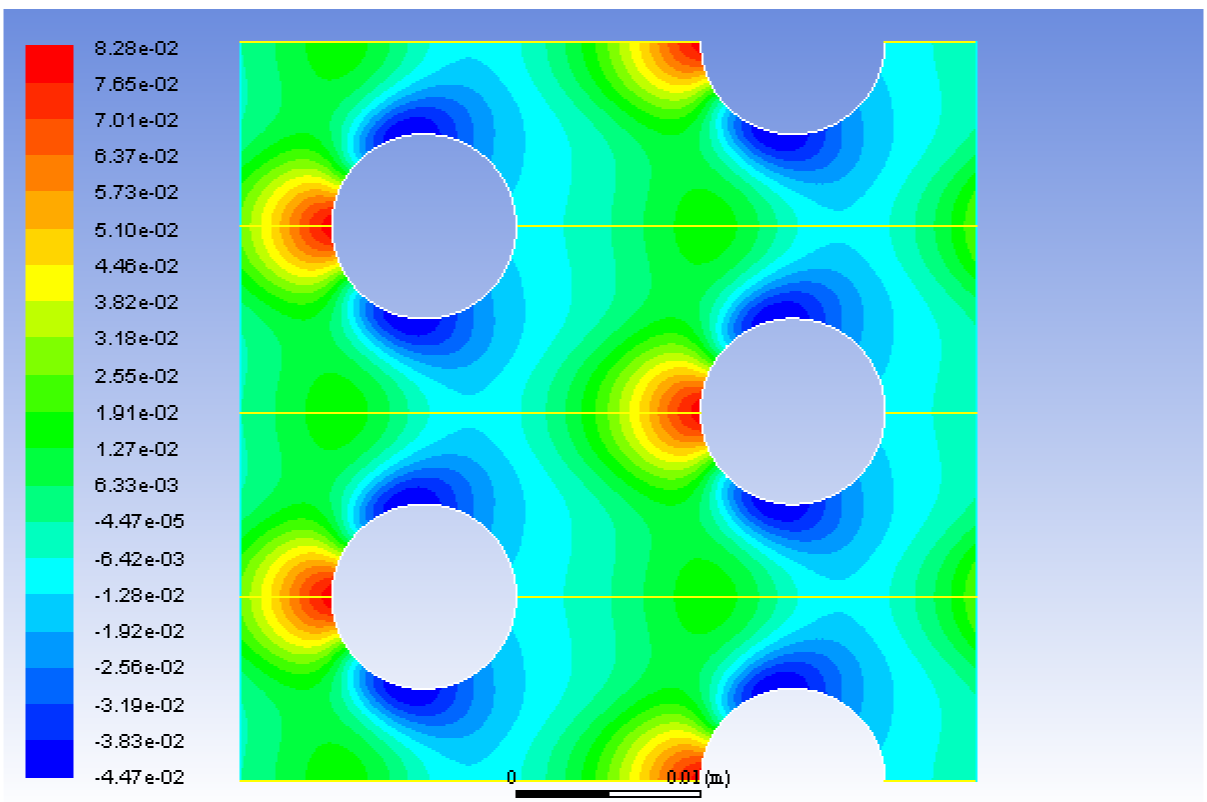
i. Select all of the symmetry zones (**symmetry-18**, **symmetry-13**, **symmetry-11**, and **symmetry-24**) in the **Mirror Planes** selection list by clicking in the upper right corner.

> **Note**
>
> There are four symmetry zones in the **Mirror Planes** selection list because the top and bottom symmetry planes in the domain are each composed of two symmetry zones, one on each side of the tube centered on the plane. It is also possible to generate the same display shown in Figure 4.4: Contours of Static Pressure with Symmetry (p. 222) by selecting just one of the symmetry zones on the top symmetry plane, and one on the bottom.

ii. Click **Apply** and close the **Views** dialog box.

iii. Translate the display of symmetry contours so that it is centered in the graphics window by using the left mouse button (Figure 4.4: Contours of Static Pressure with Symmetry (p. 222)).

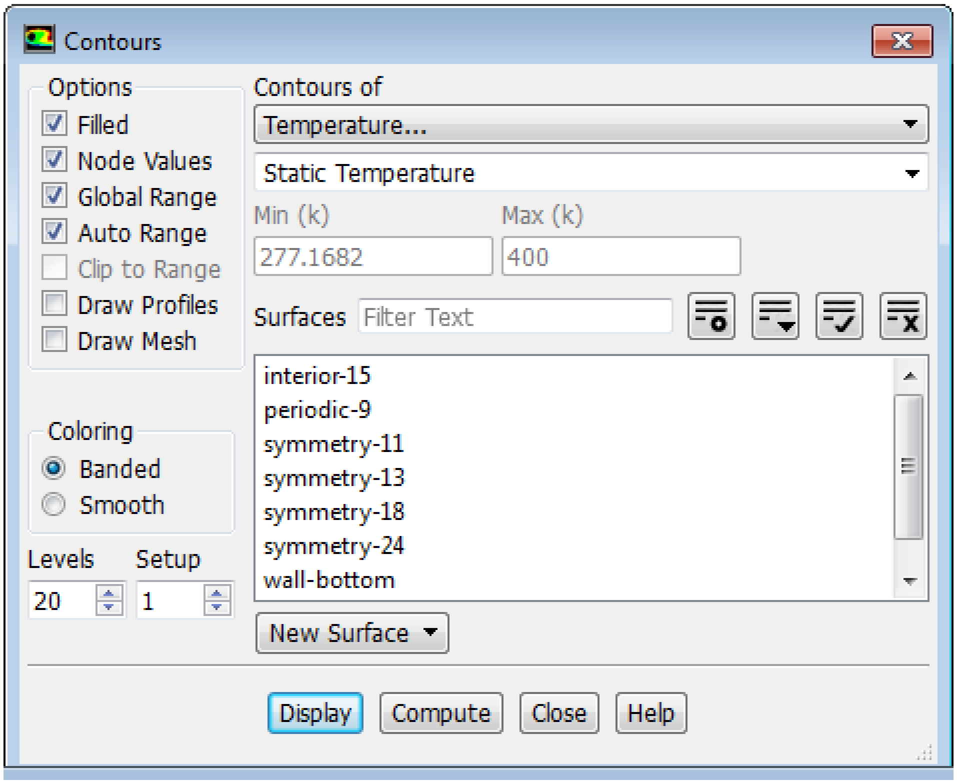
Figure 4.4: Contours of Static Pressure with Symmetry



The pressure contours displayed in Figure 4.4: Contours of Static Pressure with Symmetry (p. 222) do not include the linear pressure gradient computed by the solver. Thus, the contours are periodic at the inlet and outflow boundaries.

2. Display filled contours of static temperature (Figure 4.5: Contours of Static Temperature (p. 223)).

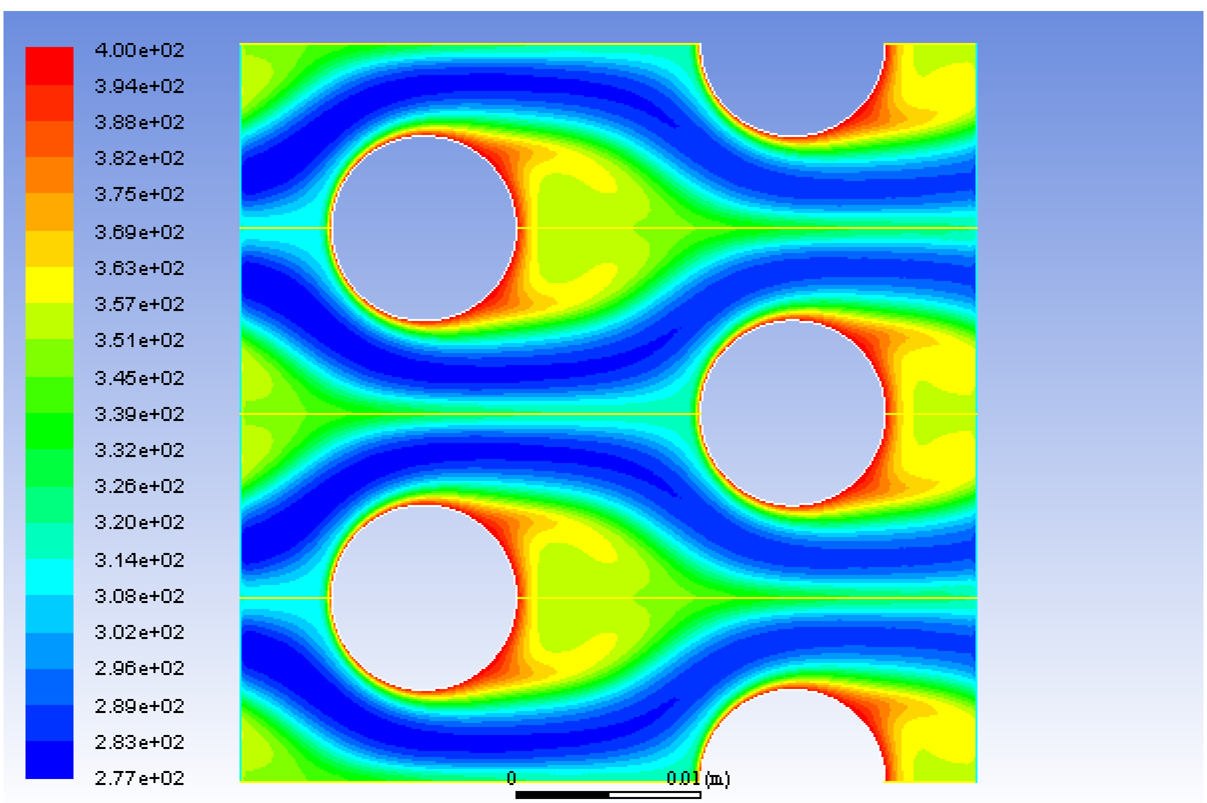
Postprocessing → **Graphics** → **Contours** → **Edit...**



a. Select **Temperature...** and **Static Temperature** from the **Contours of** drop-down lists.

b. Click **Display** and close the **Contours** dialog box.

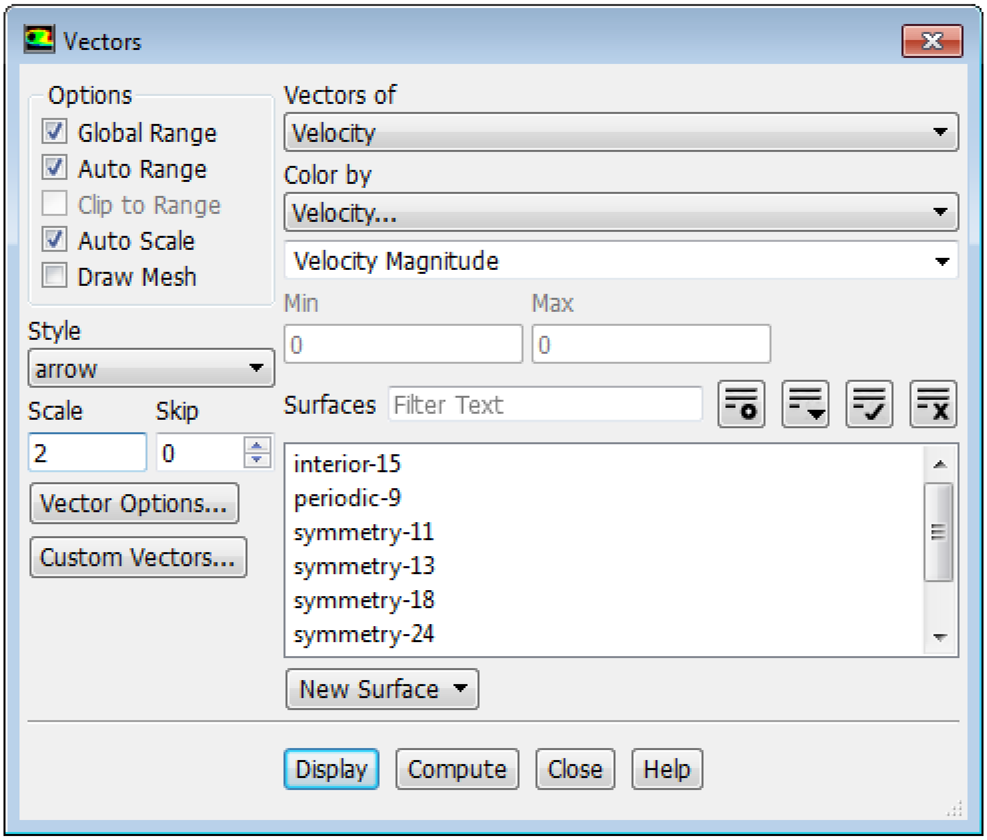
Figure 4.5: Contours of Static Temperature



The contours in Figure 4.5: Contours of Static Temperature (p. 223) reveal the temperature increase in the fluid due to heat transfer from the tubes. The hotter fluid is confined to the near-wall and wake regions, while a narrow stream of cooler fluid is convected through the tube bank.

3. Display the velocity vectors (Figure 4.6: Velocity Vectors (p. 225)).

   Postprocessing → **Graphics** → **Vectors** → **Edit...**



   a. Enter 2 for **Scale**.

      This will increase the size of the displayed vectors, making it easier to view the flow patterns.

   b. Retain the default selection of **Velocity** from the **Vectors of** drop-down list.

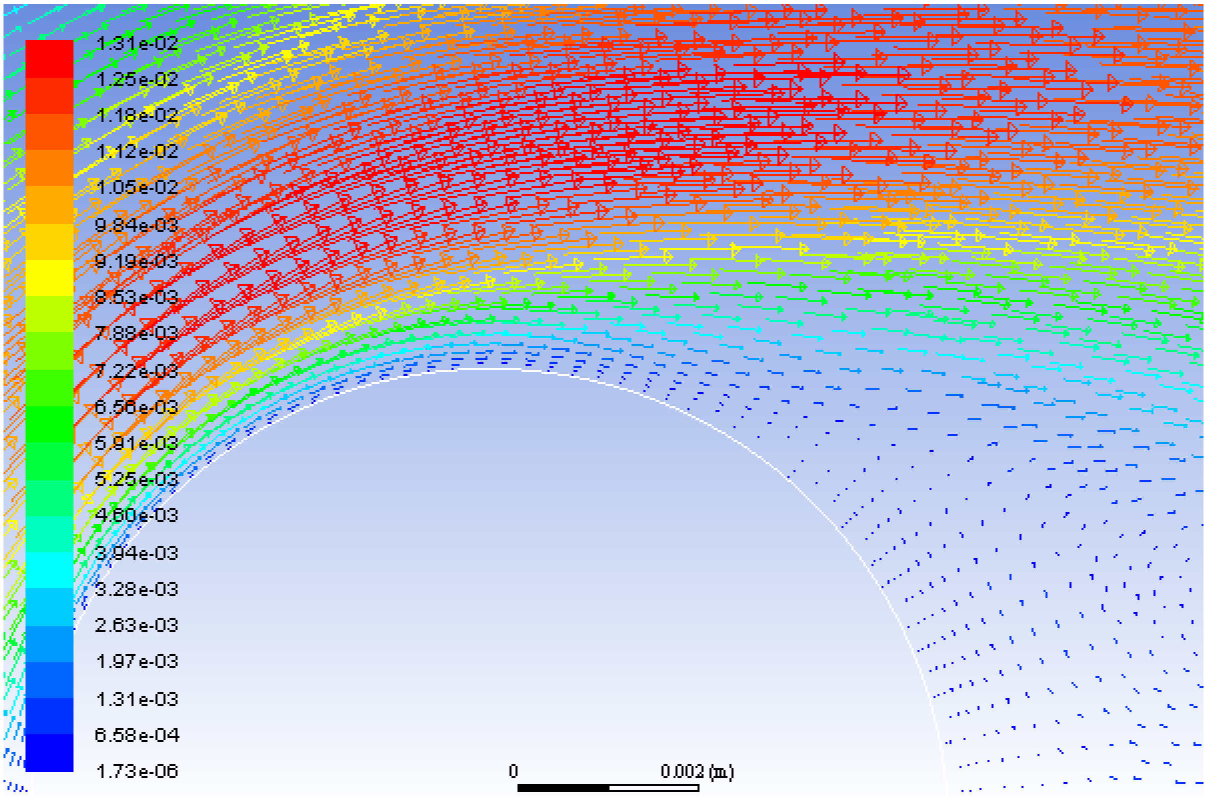
   c. Retain the default selection of **Velocity...** and **Velocity Magnitude** from the **Color by** drop-down lists.

   d. Click **Display** and close the **Vectors** dialog box.

   e. Zoom in on the upper right portion of one of the left tubes to get the display shown in (Figure 4.6: Velocity Vectors (p. 225)), by using the middle mouse button in the graphics window.

The magnified view of the velocity vector plot in Figure 4.6: Velocity Vectors (p. 225) clearly shows the recirculating flow behind the tube and the boundary layer development along the tube surface.

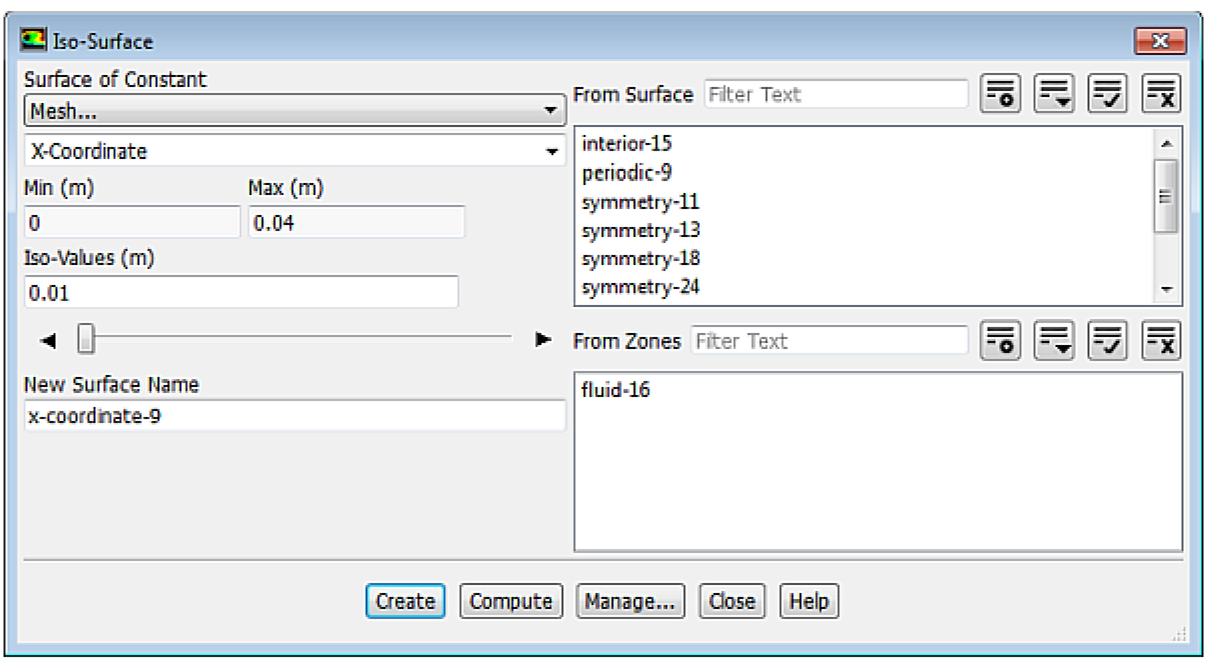
Figure 4.6: Velocity Vectors



4. Create an isosurface on the periodic tube bank at x = 0.01 m (through the first column of tubes).

   This isosurface and the ones created in the steps that follow will be used for the plotting of temperature profiles.

   Setting Up Domain → Surface → Create → Iso-Surface...



a. Select **Mesh...** and **X-Coordinate** from the **Surface of Constant** drop-down lists.

b. Enter `0.01` for **Iso-Values**.

c. Enter `x=0.01m` for **New Surface Name**.

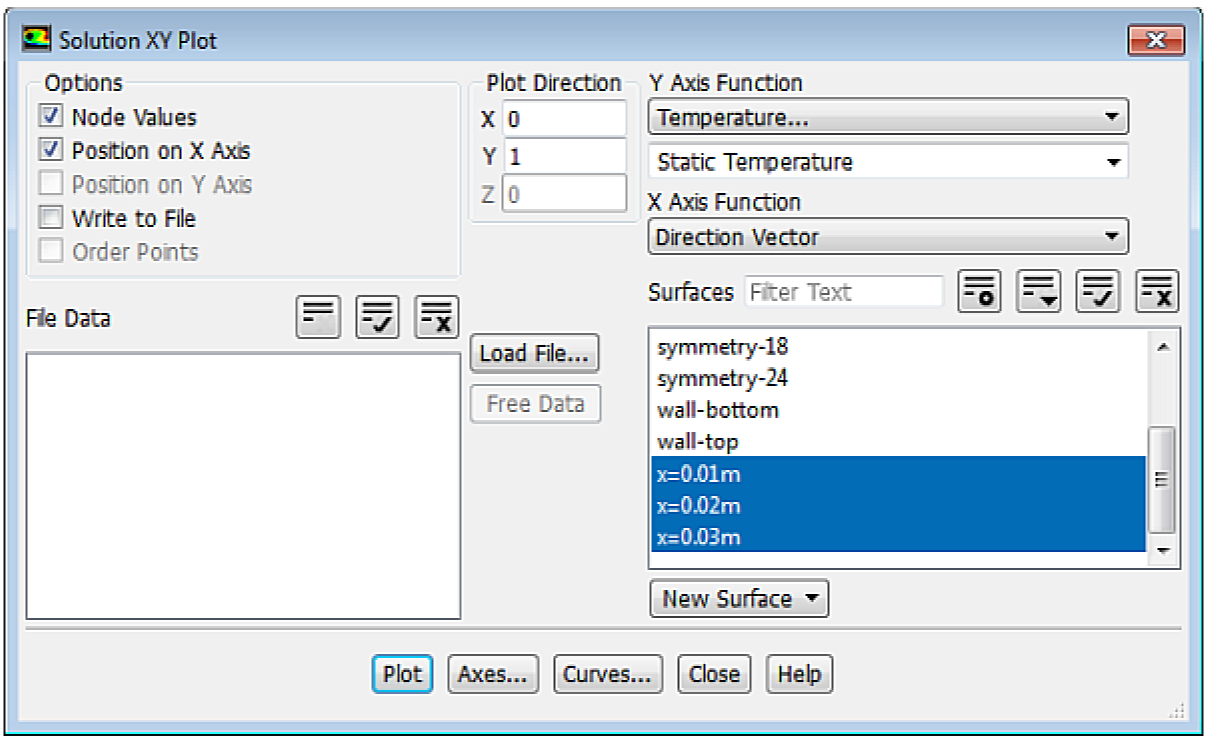
d. Click **Create**.

5. In a similar manner, create an isosurface on the periodic tube bank at x = 0.02 m (halfway between the two columns of tubes) named `x=0.02m`.

6. In a similar manner, create an isosurface on the periodic tube bank at x = 0.03 m (through the middle of the second column of tubes) named `x=0.03m`, and close the **Iso-Surface** dialog box.

7. Create an XY plot of static temperature on the three isosurfaces (Figure 4.7: Static Temperature at x=0.01, 0.02, and 0.03 m (p. 228)).

Postprocessing → **Plots** → **XY Plot** → **Edit...**



a. Enter `0` for **X** and `1` for **Y** in the **Plot Direction** group box.

*With a **Plot Direction** vector of `(0,1)`, ANSYS Fluent will plot the selected variable as a function of y. Since you are plotting the temperature profile on cross sections of constant x, the temperature varies with the y direction.*

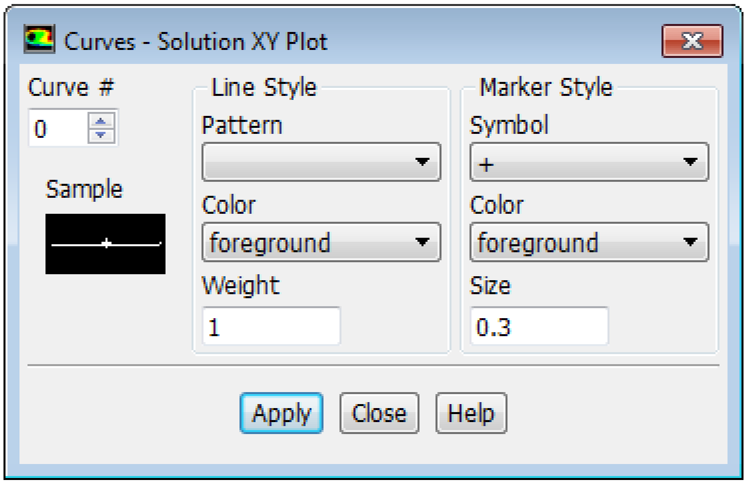
b. Select **Temperature...** and **Static Temperature** from the **Y-Axis Function** drop-down lists.

c. Select **x=0.01m**, **x=0.02m**, and **x=0.03m** in the **Surfaces** selection list.

*Scroll down to find the **x=0.01m**, **x=0.02m**, and **x=0.03m** surfaces.*

d. Click the **Curves...** button to open the **Curves - Solution XY Plot** dialog box.

This dialog box is used to define plot styles for the different plot curves.



i. Select + from the **Symbol** drop-down list.

   Scroll up to find the + item.

ii. Click **Apply** to assign the + symbol to the x = 0.01 m curve.

iii. Set the **Curve #** to `1` to define the style for the x = 0.02 m curve.

iv. Select **x** from the **Symbol** drop-down list.

   *Scroll up to find the **x** item.*

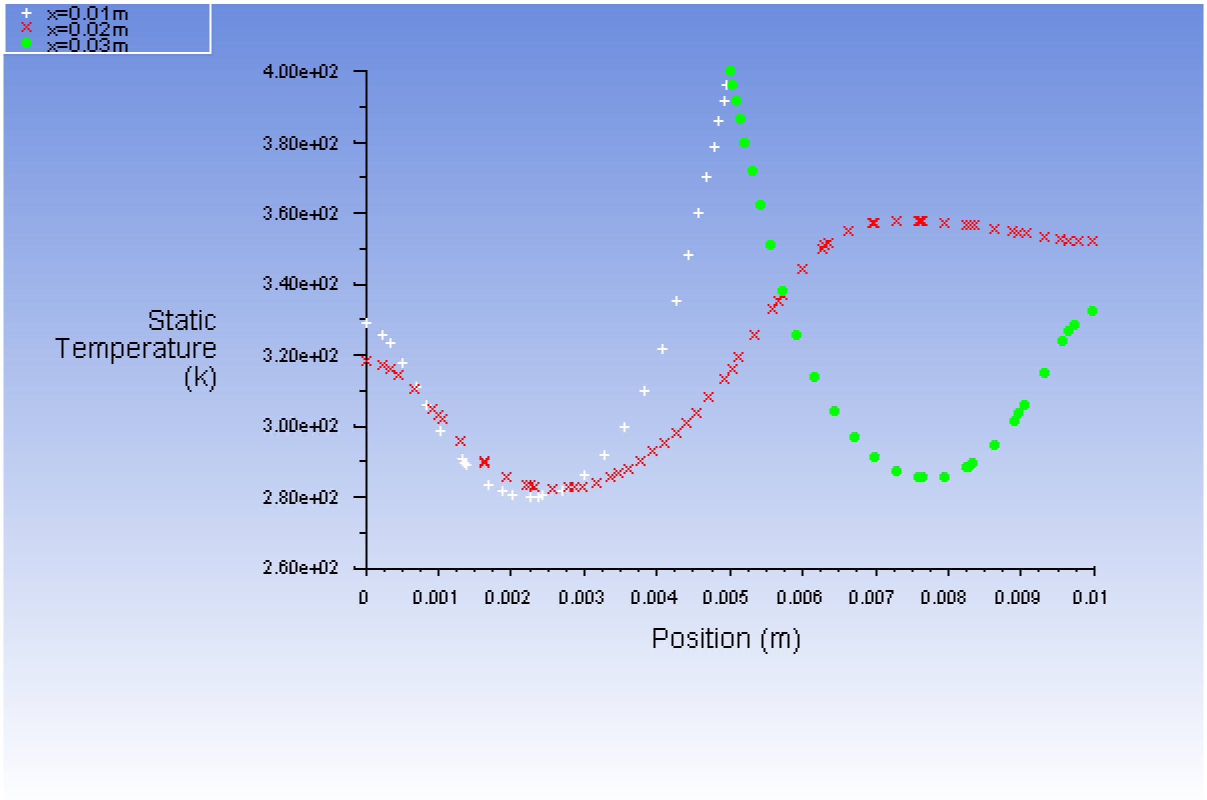
v. Enter `0.5` for **Size**.

vi. Click **Apply** and close the **Curves - Solution XY Plot** dialog box.

   Since you did not change the curve style for the x = 0.03 m curve, the default symbol will be used.

e. Click **Plot** and close the **Solution XY Plot** dialog box.

Figure 4.7: Static Temperature at x=0.01, 0.02, and 0.03 m



4.5. Summary

In this tutorial, periodic flow and heat transfer in a staggered tube bank were modeled in ANSYS Fluent. The model was set up assuming a known mass flow through the tube bank and constant wall temperatures. Due to the periodic nature of the flow and symmetry of the geometry, only a small piece of the full geometry was modeled. In addition, the tube bank configuration lent itself to the use of a hybrid mesh with quadrilateral cells around the tubes and triangles elsewhere.

The **Periodic Conditions** dialog box makes it easy to run this type of model with a variety of operating conditions. For example, different flow rates (and hence different Reynolds numbers) can be studied, or a different inlet bulk temperature can be imposed. The resulting solution can then be examined to extract the pressure drop per tube row and overall Nusselt number for a range of Reynolds numbers.

For additional details about modeling periodic heat transfer, see the Fluent User's Guide.

4.6. Further Improvements

This tutorial guides you through the steps to reach an initial solution. You may be able to obtain a more accurate solution by adapting the mesh. Mesh adaption can also ensure that the solution is independent of the mesh. These steps are demonstrated in Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121).

Chapter 5: Modeling External Compressible Flow

This tutorial is divided into the following sections:

5.1. Introduction

The purpose of this tutorial is to compute the turbulent flow past a transonic airfoil at a nonzero angle of attack. You will use the Spalart-Allmaras turbulence model.

This tutorial demonstrates how to do the following:

- Model compressible flow (using the ideal gas law for density).
- Set boundary conditions for external aerodynamics.
- Use the Spalart-Allmaras turbulence model.
- Use Full Multigrid (FMG) initialization to obtain better initial field values.
- Calculate a solution using the pressure-based coupled solver with the pseudo transient option.
- Use force and surface reports to check solution convergence.
- Check the near-wall mesh resolution by plotting the distribution of y^+.

5.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

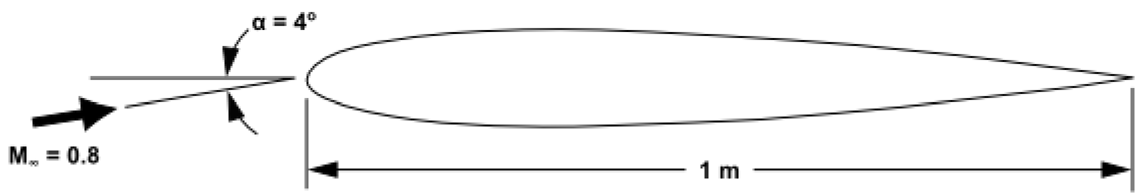
- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

and that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

5.3. Problem Description

The problem considers the flow around an airfoil at an angle of attack $\alpha=4^{\circ}$ and a free stream Mach number of 0.8 ($M_{\infty}=0.8$). The flow is transonic, and has a fairly strong shock near the mid-chord ($x/c=0.45$) on the upper (suction) side. The chord length is 1 m. The geometry of the airfoil is shown in Figure 5.1: Problem Specification (p. 230).

Figure 5.1: Problem Specification



5.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

5.4.1. Preparation

To prepare for running this tutorial:

1. Set up a working folder on the computer you will be using.

2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

> **Note**
>
> If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

3. Enter the name of this tutorial into the search bar.

4. Narrow the results by using the filter on the left side of the page.

    a. Click **ANSYS Fluent** under **Product**.

    b. Click **18.0** under **Version**.

5. Select this tutorial from the list.

6. Click the **external_compressible_R180.zip** link to download the input files.

7. Unzip `external_compressible_R180.zip` to your working folder.

   The file `airfoil.msh` *can be found in the* `external_compressible` *folder created after unzipping the file.*

8. Use Fluent Launcher to start the **2D** version of ANSYS Fluent.

   Fluent Launcher displays your **Display Options** preferences from the previous session.

   For more information about Fluent Launcher, see starting ANSYS Fluent using the Fluent Launcher in the Fluent Getting Started Guide.

9. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.

10. Enable **Double Precision**.

11. Ensure **Serial** is selected under **Processing Options**.

5.4.2. Mesh

1. Read the mesh file `airfoil.msh`.

   File → **Read** → **Mesh...**

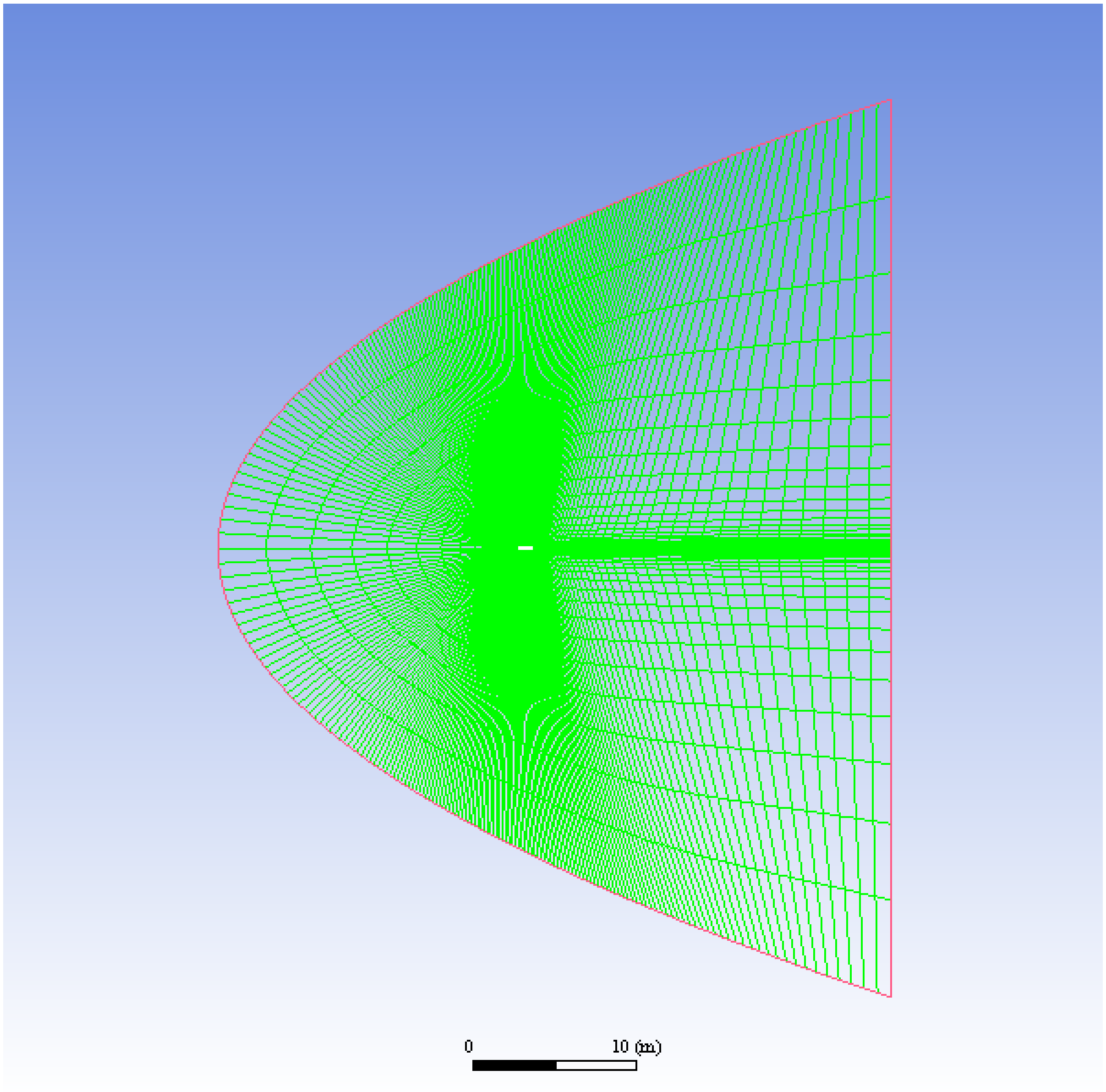
2. Check the mesh.

   Setting Up Domain → **Mesh** → **Check**

   ANSYS Fluent will perform various checks on the mesh and will report the progress in the console. Make sure that the reported minimum volume is a positive number.

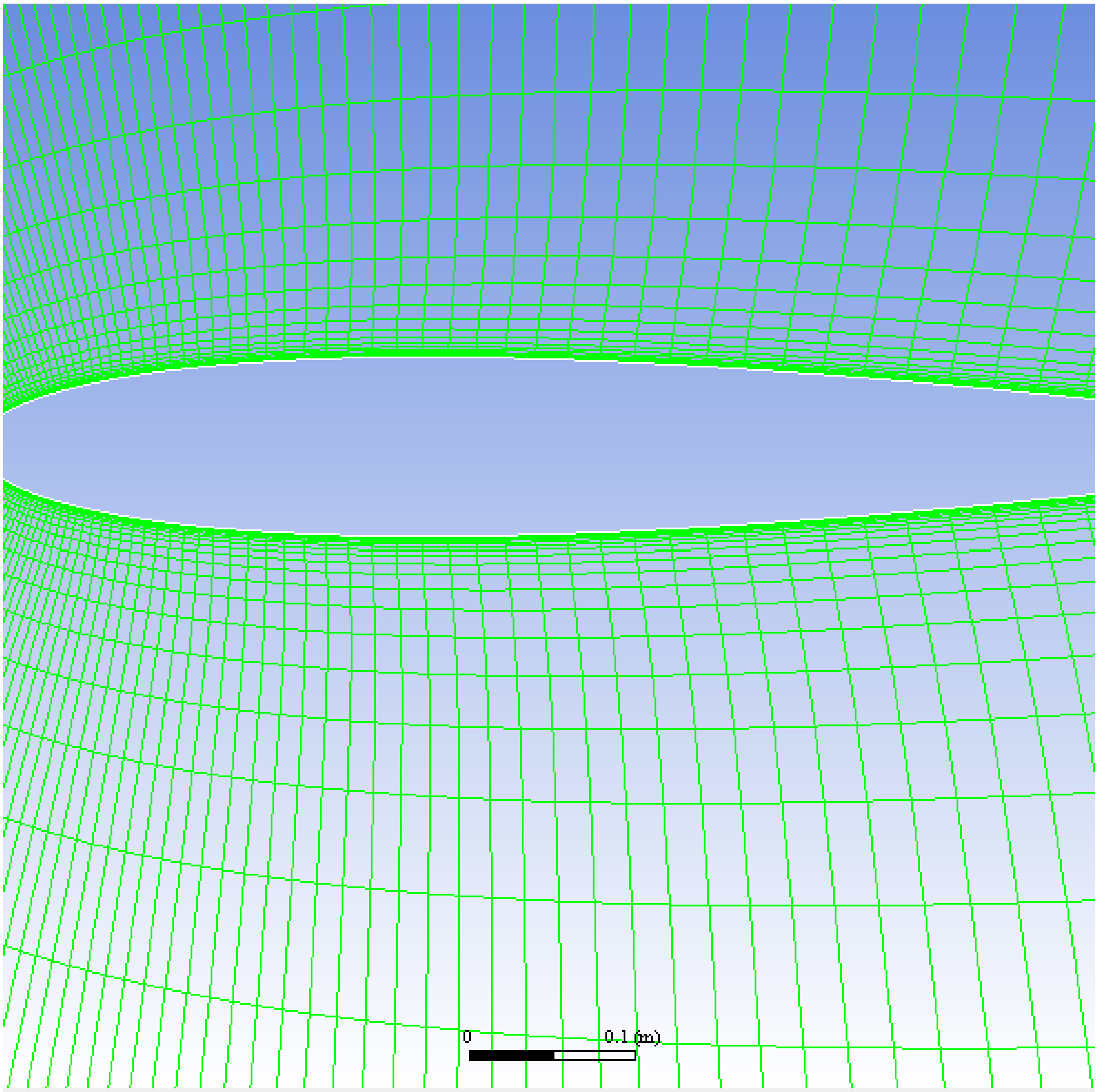
3. Examine the mesh (Figure 5.2: The Entire Mesh (p. 232) and Figure 5.3: Magnified View of the Mesh Around the Airfoil (p. 233)).

Figure 5.2: The Entire Mesh



Quadrilateral cells were used for this simple geometry because they can be stretched easily to account for different flow gradients in different directions. In the present case, the gradients normal to the airfoil wall are much greater than those tangent to the airfoil. Consequently, the cells near the surface have high aspect ratios. For geometries that are more difficult to mesh, it may be easier to create a hybrid mesh composed of quadrilateral and triangular cells.

Figure 5.3: Magnified View of the Mesh Around the Airfoil



A parabola was chosen to represent the far-field boundary because it has no discontinuities in slope, enabling the construction of a smooth mesh in the interior of the domain.

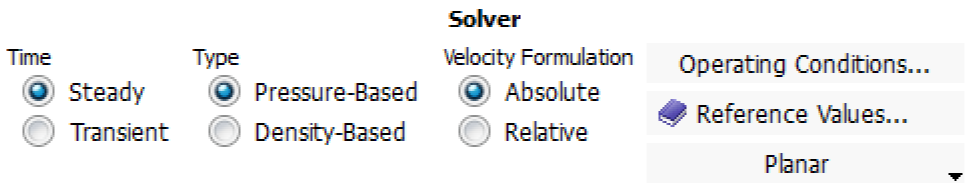
Extra

You can use the right mouse button to probe for mesh information in the graphics window. If you click the right mouse button on any node in the mesh, information will be displayed in the ANSYS Fluent console about the associated zone, including the name of the zone. This feature is especially useful when you have several zones of the same type and you want to distinguish between them quickly.

5.4.3. Solver

1. Set the solver settings.

Setting Up Physics → Solver



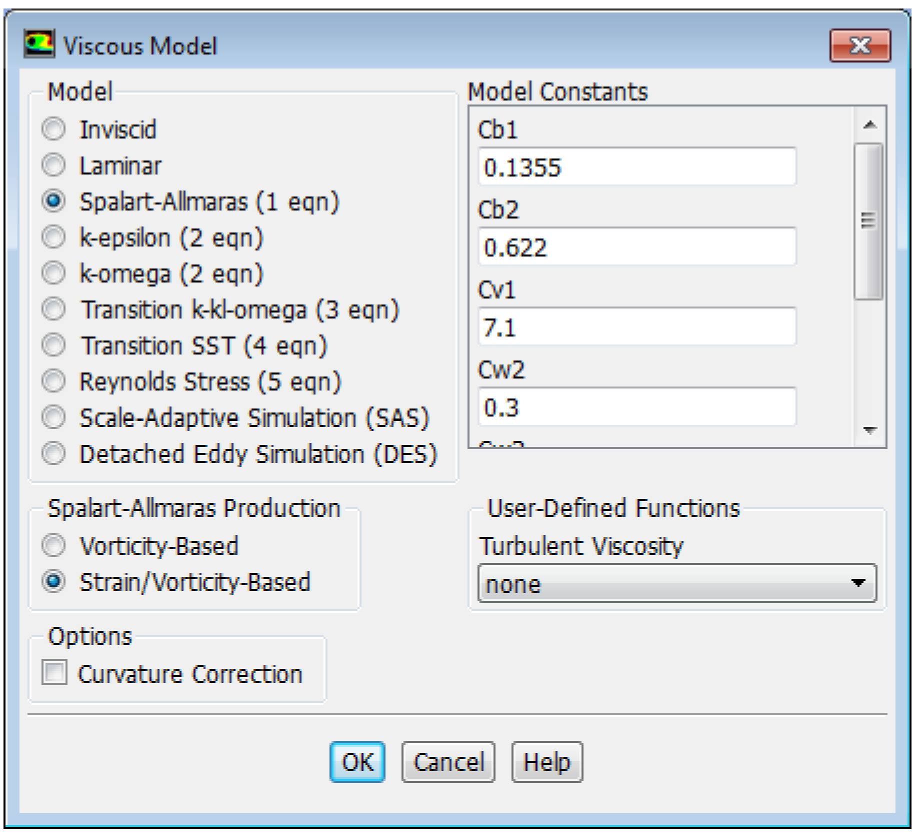
a. Retain the default selection of **Pressure-Based** from the **Type** list.

*The pressure-based solver with the **Coupled** option for the pressure-velocity coupling is a good alternative to density-based solvers of ANSYS Fluent when dealing with applications involving high-speed aerodynamics with shocks. Selection of the coupled algorithm is made in the **Solution Methods** task page in the Solution step.*

5.4.4. Models

1. Select the Spalart-Allmaras turbulence model.

Setting Up Physics → Models → Viscous...



a. Select **Spalart-Allmaras (1eqn)** in the **Model** list.

b. Select **Strain/Vorticity-Based** in the **Spalart-Allmaras Production** list.

c. Retain the default settings in the **Model Constants** group box.

d. Click **OK** to close the **Viscous Model** dialog box.

Note

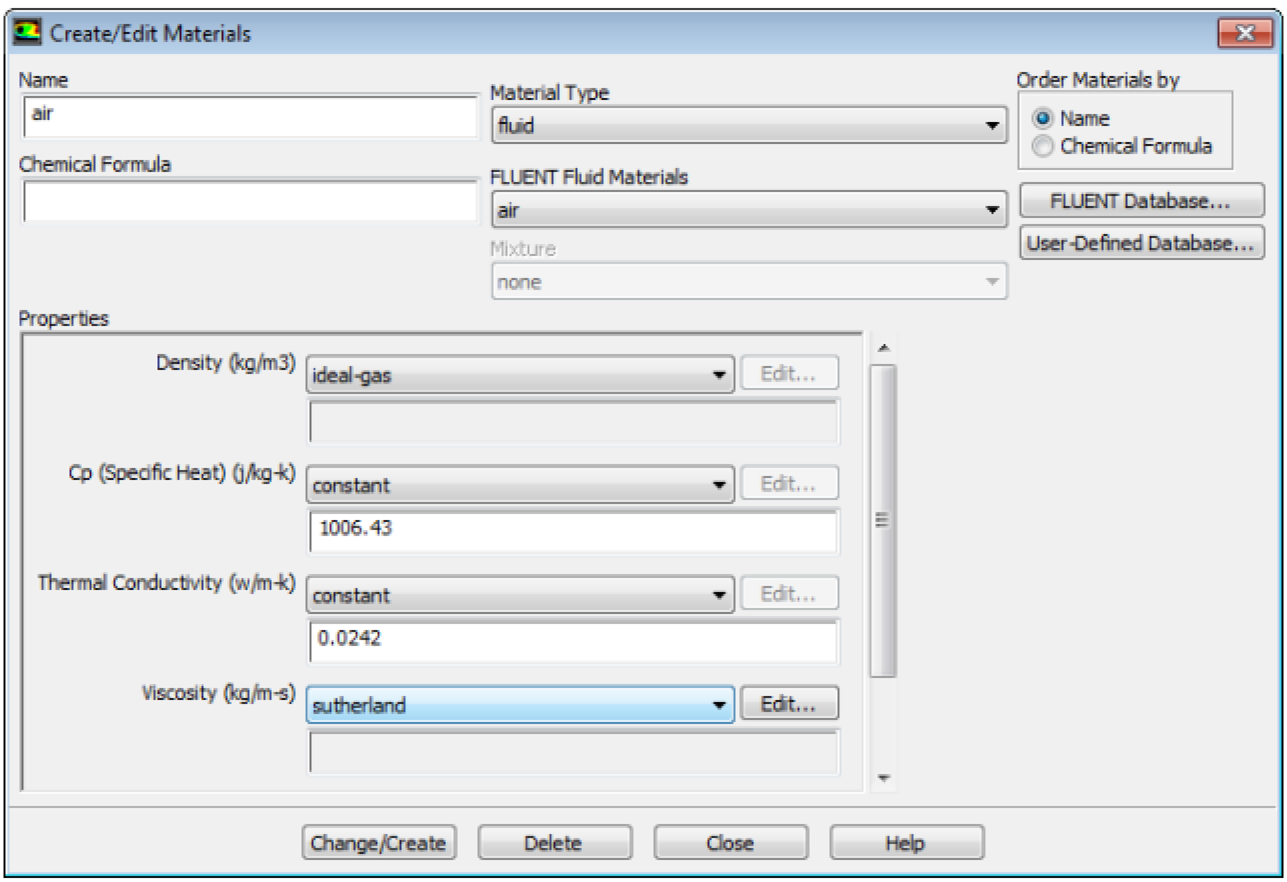
The Spalart-Allmaras model is a relatively simple one-equation model that solves a modeled transport equation for the kinematic eddy (turbulent) viscosity. This embodies a relatively new class of one-equation models in which it is not necessary to calculate a length scale related to the local shear layer thickness. The Spalart-Allmaras model was designed specifically for aerospace applications involving wall-bounded flows and has been shown to give good results for boundary layers subjected to adverse pressure gradients.

5.4.5. Materials

*The default **Fluid Material** is air, which is the working fluid in this problem. The default settings need to be modified to account for compressibility and variations of the thermophysical properties with temperature.*

1. Set the properties for **air**, the default fluid material.

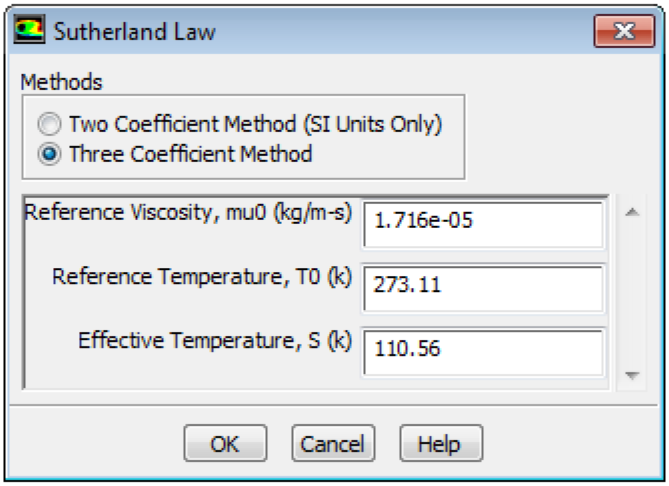
   Setup → **Materials** → **Fluids** → **air** **Edit...**



a. Select **ideal-gas** from the **Density** drop-down list.

   *The **Energy Equation** will be enabled.*

b. Select **sutherland** from the **Viscosity** drop-down list to open the **Sutherland Law** dialog box.



*Scroll down the **Viscosity** drop-down list to find **sutherland**.*

i. Retain the default selection of **Three Coefficient Method** in the **Methods** list.

ii. Click **OK** to close the **Sutherland Law** dialog box.

The Sutherland law for viscosity is well suited for high-speed compressible flows.

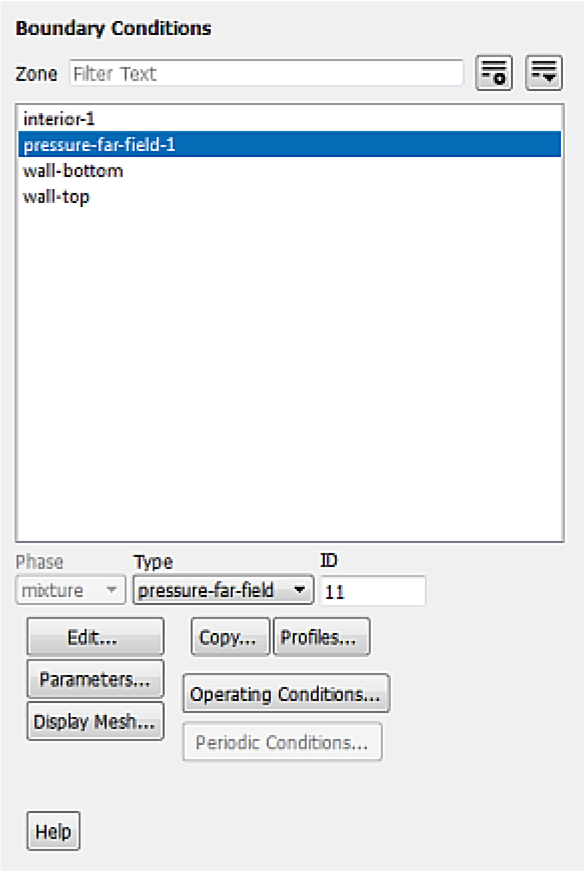
c. Click **Change/Create** to save these settings.

d. Close the **Create/Edit Materials** dialog box.

*While **Density** and **Viscosity** have been made temperature-dependent, **Cp (Specific Heat)** and **Thermal Conductivity** have been left constant. For high-speed compressible flows, thermal dependency of the physical properties is generally recommended. For simplicity, **Thermal Conductivity** and **Cp (Specific Heat)** are assumed to be constant in this tutorial.*

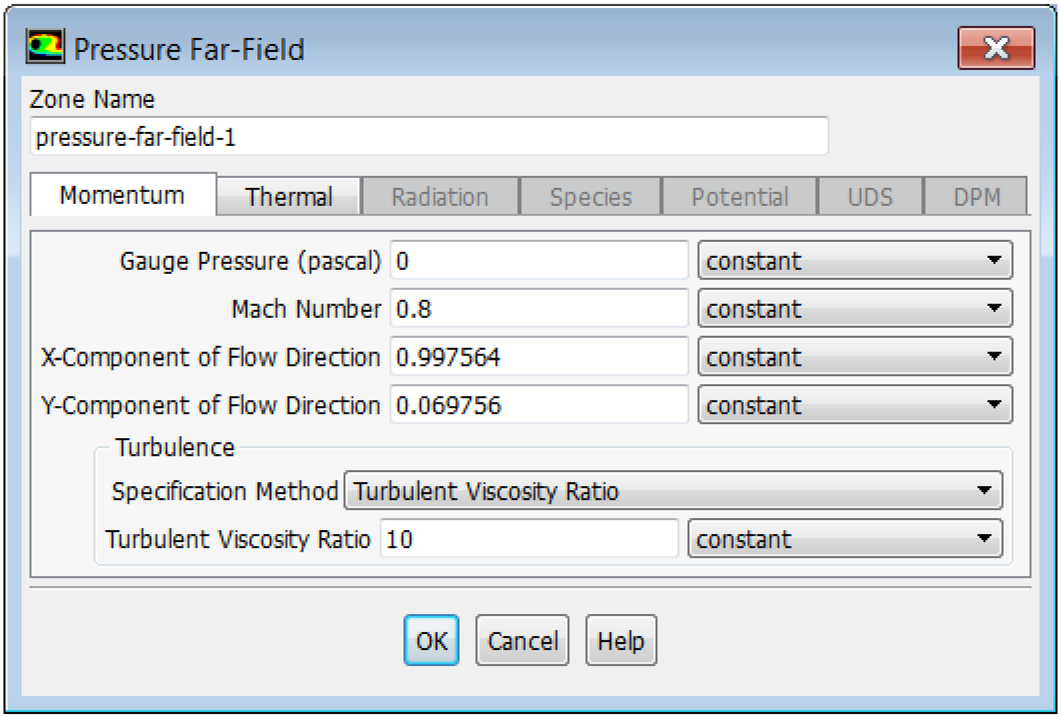
5.4.6. Boundary Conditions

Setup → **Boundary Conditions**



1. Set the boundary conditions for **pressure-far-field-1**.

Setup → **Boundary Conditions** → **pressure-far-field-1** → **Edit...**



a. Retain the default value of `0` Pa for **Gauge Pressure**.

> **Note**
>
> The gauge pressure in ANSYS Fluent is always relative to the operating pressure, which is defined in a separate input (see below).

b. Enter `0.8` for **Mach Number**.

c. Enter `0.997564` and `0.069756` for the **X-Component of Flow Direction** and **Y-Component of Flow Direction**, respectively.

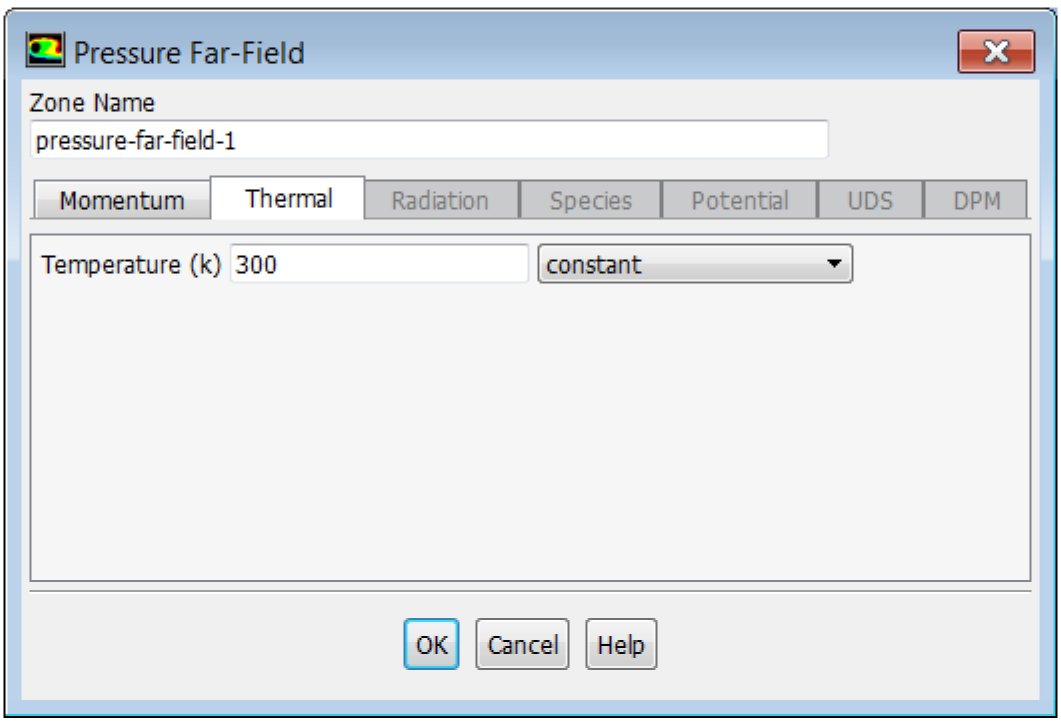
These values are determined by the 4° *angle of attack:* $\cos 4^{\circ} \approx 0.997564$ *and* $\sin 4^{\circ} \approx 0.069756$.

d. Retain **Turbulent Viscosity Ratio** from the **Specification Method** drop-down list in the **Turbulence** group box.

e. Retain the default value of `10` for **Turbulent Viscosity Ratio**.

The viscosity ratio should be between 1 and 10 for external flows.

f. Click the **Thermal** tab and retain the default value of `300` K for **Temperature**.

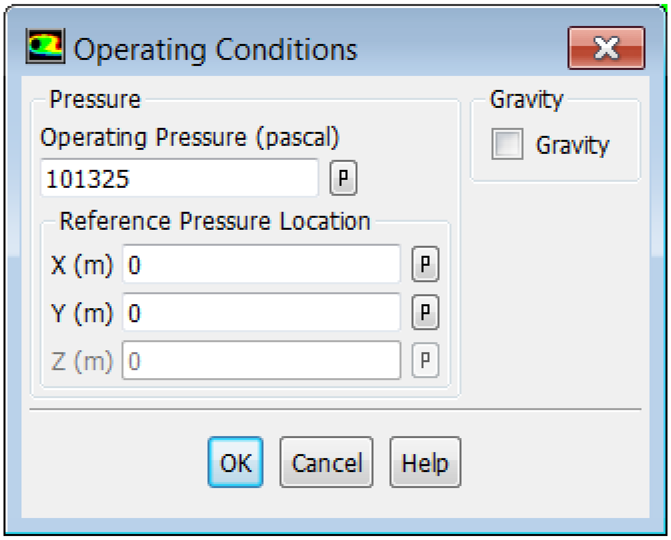


i. A. Click **OK** to close the **Pressure Far-Field** dialog box.

5.4.7. Operating Conditions

1. Set the operating pressure.

Setup → **Boundary Conditions** → **Operating Conditions...**



*The **Operating Conditions** dialog box can also be accessed from the **Cell Zone Conditions** task page.*

a. Retain the default value of `101325` Pa for **Operating Pressure**.

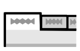
   The operating pressure should be set to a meaningful mean value in order to avoid round-off errors. The absolute pressure must be greater than zero for compressible flows. If you want to specify boundary conditions in terms of absolute pressure, you can make the operating pressure zero.

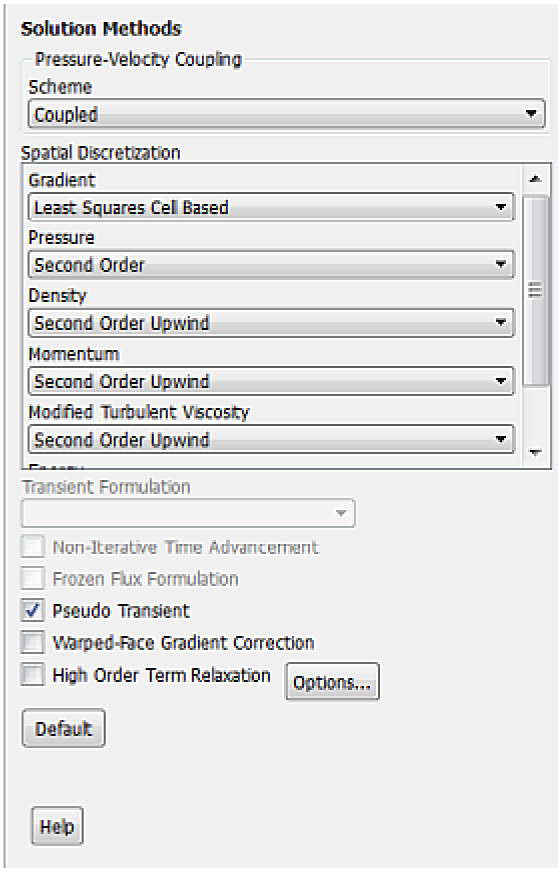
b. Click **OK** to close the **Operating Conditions** dialog box.

For information about setting the operating pressure, see the Fluent User's Guide.

5.4.8. Solution

1. Set the solution parameters.

    **Solving → Solution → Methods...**



a. Select **Coupled** from the **Scheme** drop-down list in the **Pressure-Velocity Coupling** group box.

b. Retain the default selection of **Least Squares Cell Based** from the **Gradient** drop-down list in the **Spatial Discretization** group box.

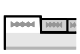
c. Retain the default selection of **Second Order** from the **Pressure** drop-down list.

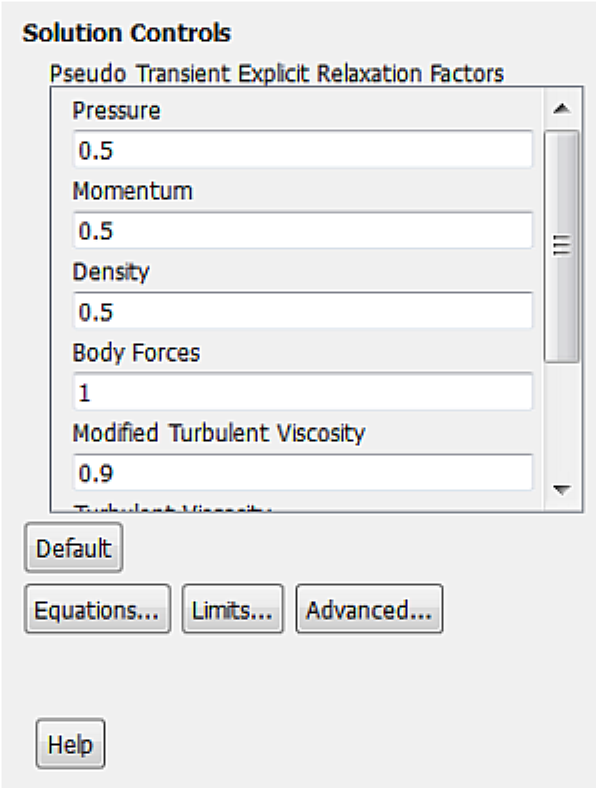
d. Select **Second Order Upwind** from the **Modified Turbulent Viscosity** drop-down list.

e. Enable **Pseudo Transient**.

The Pseudo Transient option enables the pseudo transient algorithm in the coupled pressure-based solver. This algorithm effectively adds an unsteady term to the solution equations in order to improve stability and convergence behavior. Use of this option is recommended for general fluid flow problems.

2. Set the solution controls.

 **Solving** → **Controls** → **Controls...**



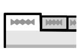
a. Enter `0.5` for **Density** in the **Pseudo Transient Explicit Relaxation Factors** group box.

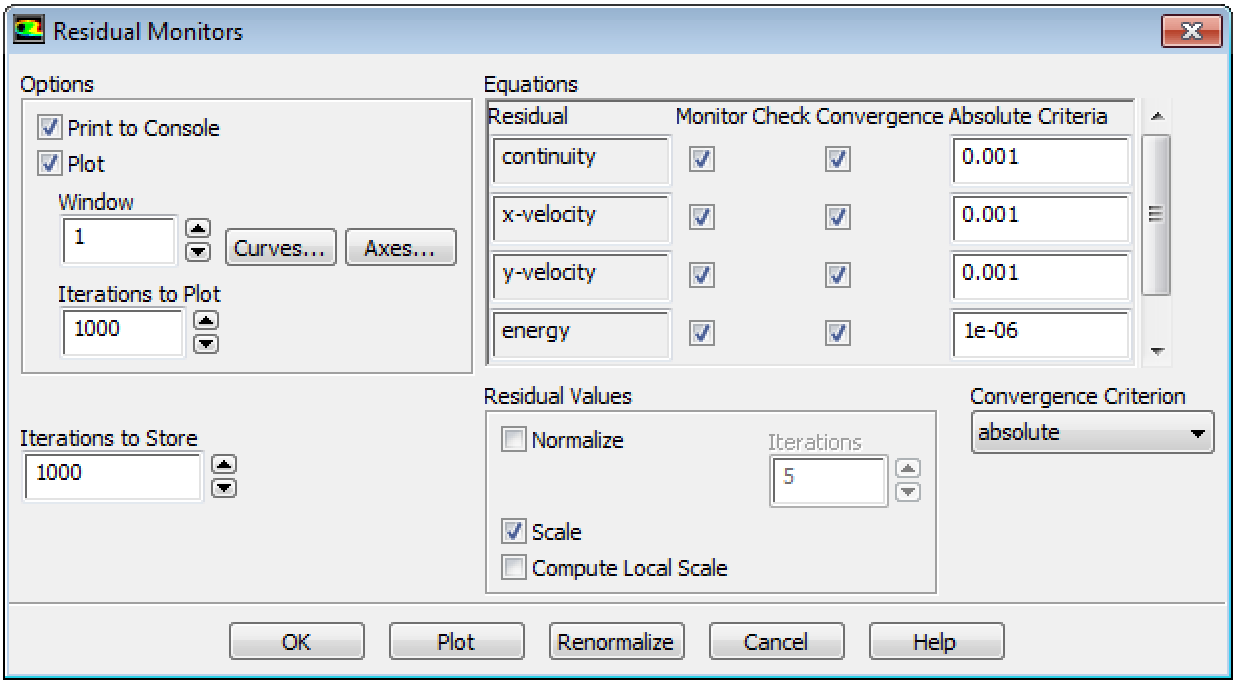
   Under-relaxing the density factor is recommended for high-speed compressible flows.

b. Enter `0.9` for **Modified Turbulent Viscosity**.

   Larger under-relaxation factors (that is, closer to 1) will generally result in faster convergence. However, instability can arise that may need to be eliminated by decreasing the under-relaxation factors.

3. Enable residual plotting during the calculation.

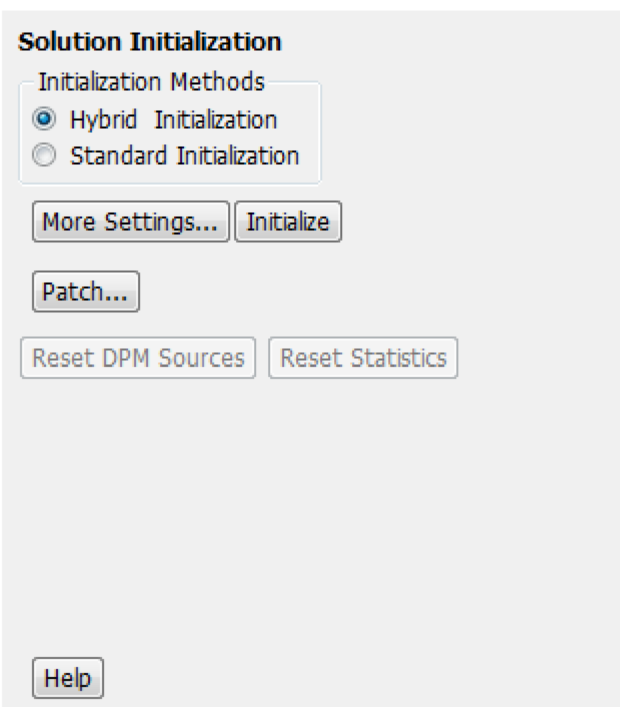
    **Solving → Reports → Residuals...**



a. Ensure that **Plot** is enabled in the **Options** group box and click **OK** to close the **Residual Monitors** dialog box.

4. Initialize the solution.

Solution → **Solution Initialization**



a. Retain the default selection of **Hybrid Initialization** from the **Initialization Methods** group box.

b. Click **Initialize** to initialize the solution.

c. Run the Full Multigrid (FMG) initialization.

FMG initialization often facilitates an easier start-up, where no CFL (Courant Friedrichs Lewy) ramping is necessary, thereby reducing the number of iterations for convergence.

i. Press **`Enter`** in the console to get the command prompt (>).

ii. Enter the text commands and input responses as shown in the boxes. Accept the default values by pressing **`Enter`** when no input response is given:

```
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

    residual reduction on level 3 is:  [0.001]
    number of cycles on level 3 is:  [100]

    residual reduction on level 4 is:  [0.001]
    number of cycles on level 4 is:  [500]

    residual reduction on level 5 [coarsest grid] is:  [0.001]
    number of cycles on level 5 is:  [500]

 * FMG customization complete

   set FMG courant-number [0.75]

   enable FMG verbose? [no] yes

  solve/initialize/fmg-initialization
Enable FMG initialization? [no] yes
```

Note

Whenever FMG initialization is performed, it is important to inspect the FMG initialized flow field using the postprocessing tools of ANSYS Fluent. Monitoring the normalized residuals, which are plotted in the console window, will give you an idea of the convergence of the FMG solver. You should notice that the value of the normalized residuals decreases. For information about FMG initialization, including convergence strategies, see the Fluent User's Guide.

5. Save the case and data files (`airfoil.cas.gz` and `airfoil.dat.gz`).

File → Write → Case & Data...

It is good practice to save the case and data files during several stages of your case setup.

6. Start the calculation by requesting 50 iterations.

Solving → Run Calculation

a. Enter `50` for **No. of Iterations**.

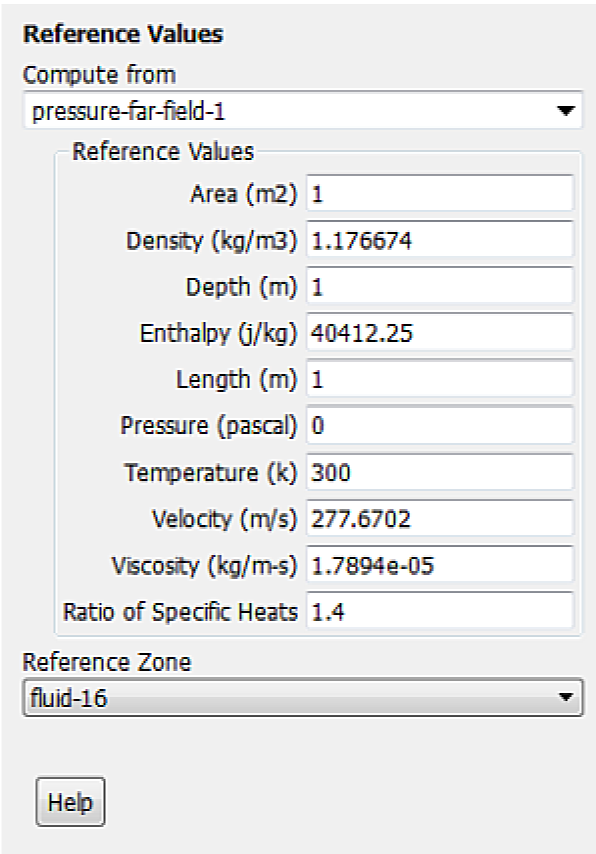
b. Click **Calculate**.

By performing some iterations before setting up the force reports, you will avoid large initial transients in the report plots. This will reduce the axes range and make it easier to judge the convergence.

7. Set the reference values that are used to compute the lift, drag, and moment coefficients.

Setup → **Reference Values**

The reference values are used to non-dimensionalize the forces and moments acting on the airfoil. The dimensionless forces and moments are the lift, drag, and moment coefficients.

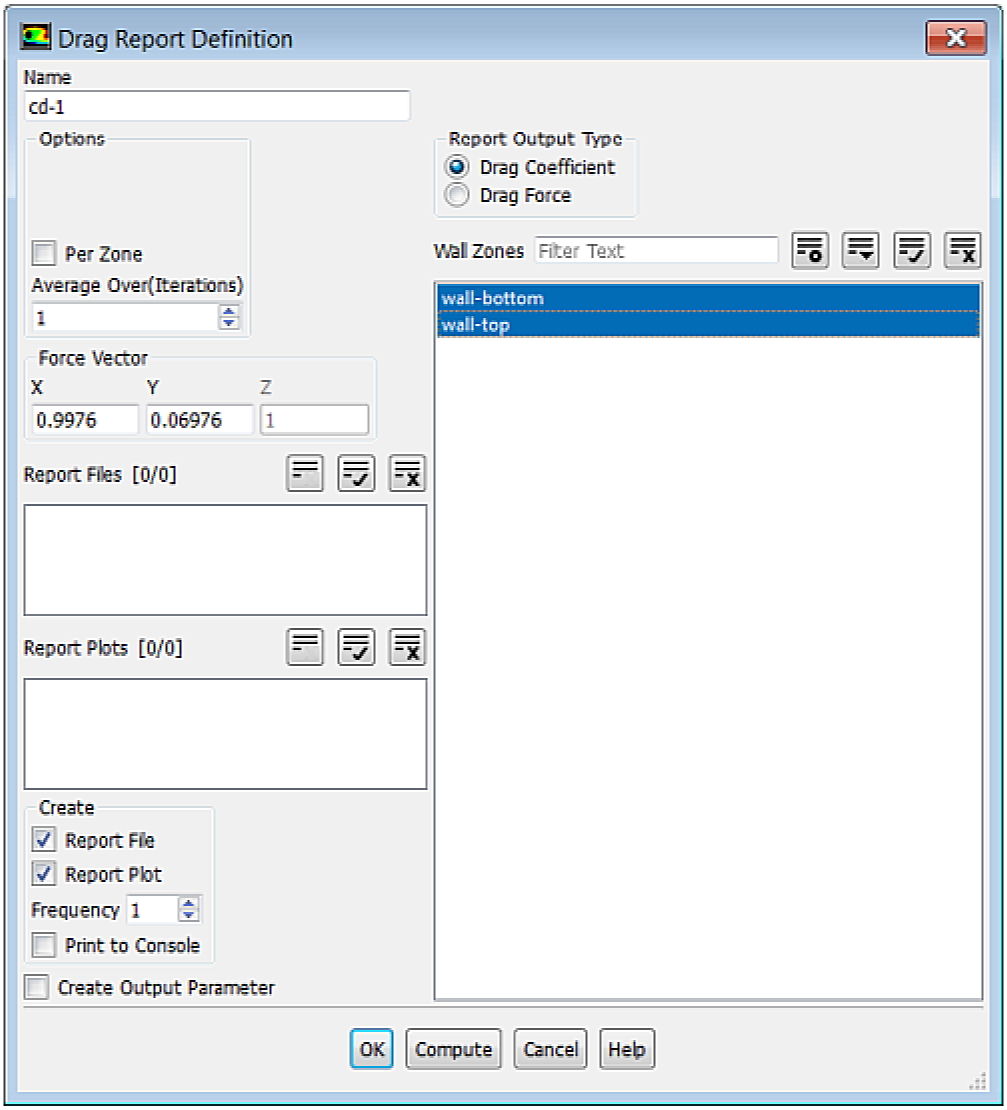


a. Select **pressure-far-field-1** from the **Compute from** drop-down list.

*ANSYS Fluent will update the **Reference Values** based on the boundary conditions at the far-field boundary.*

8. Create a force report definition to plot and write the drag coefficient for the walls of the airfoil.

Solving → **Reports** → **Definitions** → **New** → **Force Report** → **Drag...**



a. Enter **cd-1** for **Name**.

b. Make sure that **Drag Coefficient** is selected in the **Report Output Type** group box.

c. In the **Create** group box, enable **Report Plot**.

d. Enable **Report File** to save the report history to a file.

Note

If you do not enable the **Report File** option, the history information will be lost when you exit ANSYS Fluent.

---

e. Select **wall-bottom** and **wall-top** in the **Wall Zones** selection list.

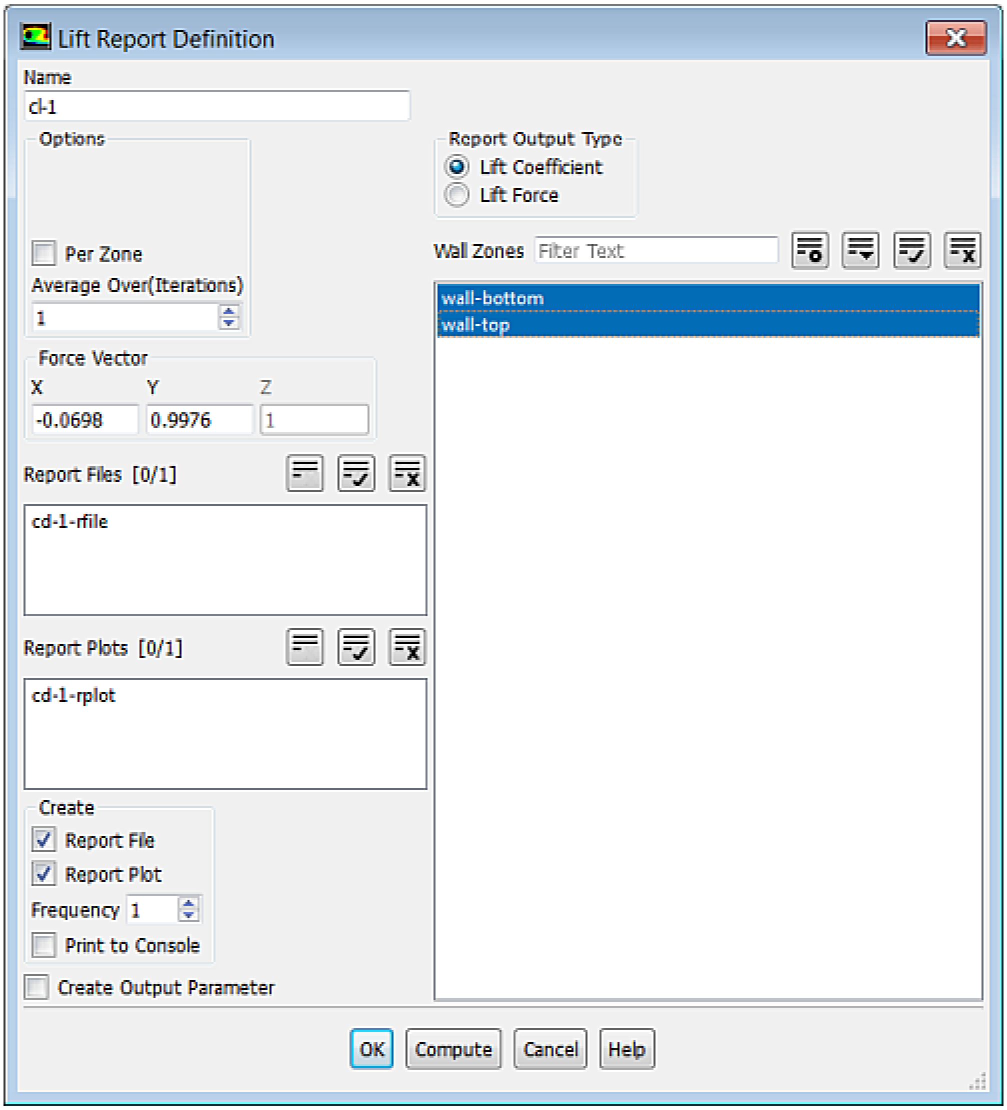
f. Enter `0.9976` for **X** and `0.06976` for **Y** in the **Force Vector** group box.

*These **X** and **Y** values ensure that the drag coefficient is calculated parallel to the free-stream flow, which is* 4° *off of the global coordinates.*

g. Click **OK** to close the **Drag Report Definition** dialog box.

9. Similarly, create a force report definition for the lift coefficient.

Solving → **Reports** → **Definitions** → **New** → **Force Report** → **Lift...**

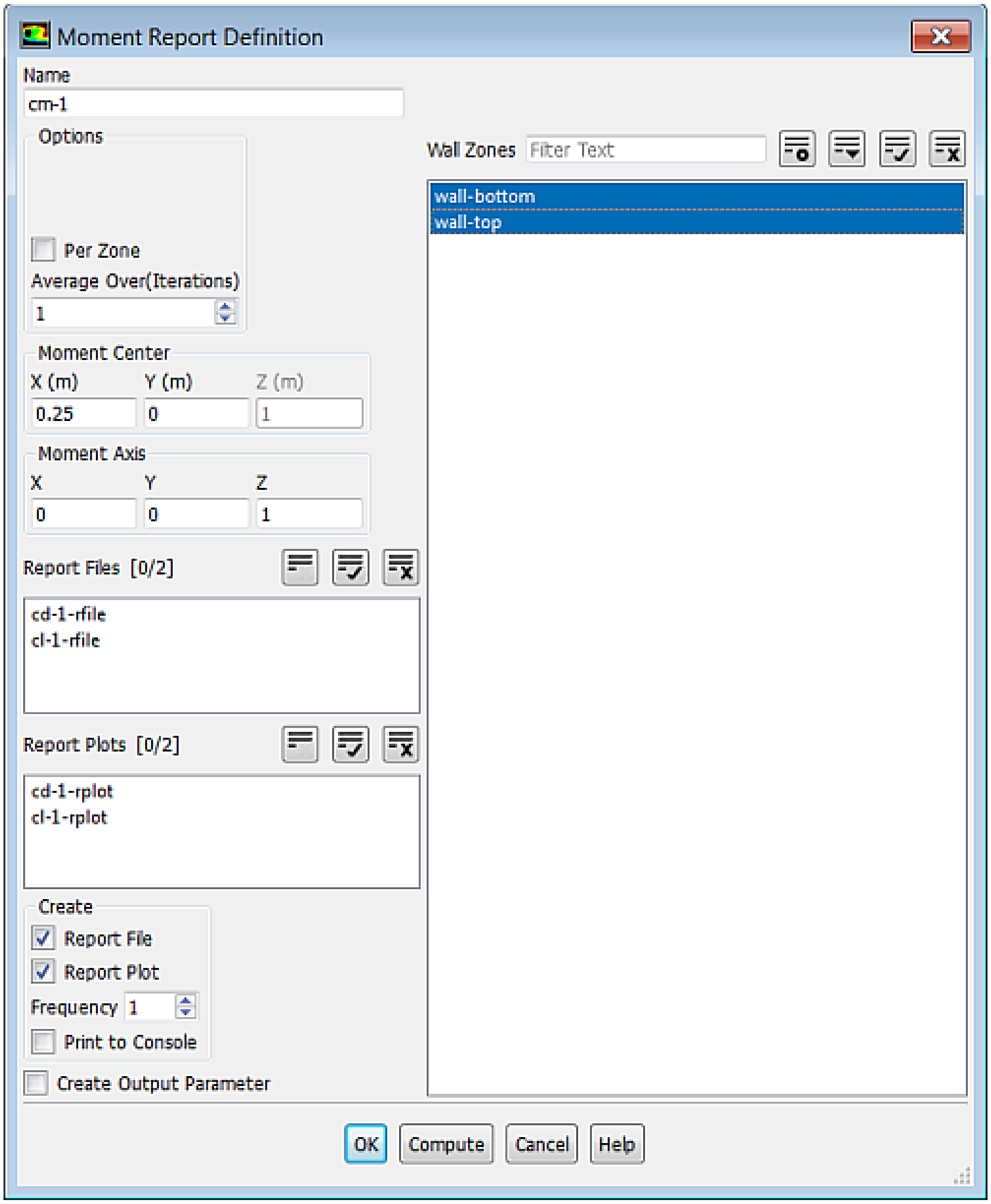


Enter the values for **X** and **Y** shown in the **Lift Report Definition** dialog box.

*The **X** and **Y** values shown ensure that the lift coefficient is calculated normal to the free-stream flow, which is* 4° *off of the global coordinates.*

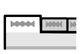
10. In a similar manner, create a force report definition for the moment coefficient.

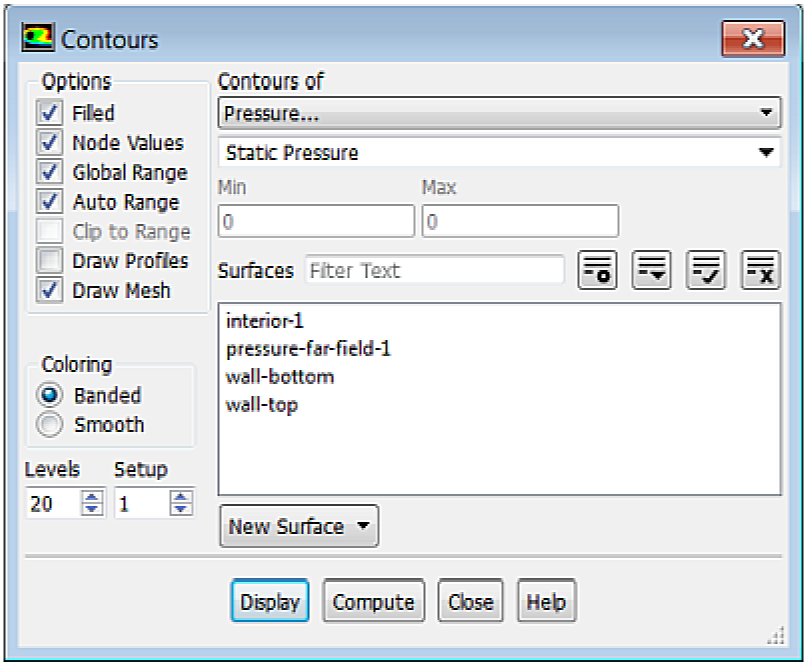
Solving → **Reports** → **Definitions** → **New** → **Force Report** → **Moment...**



Enter the values for the **Moment Center** and **Moment Axis** shown in the **Moment Report Definition** dialog box.

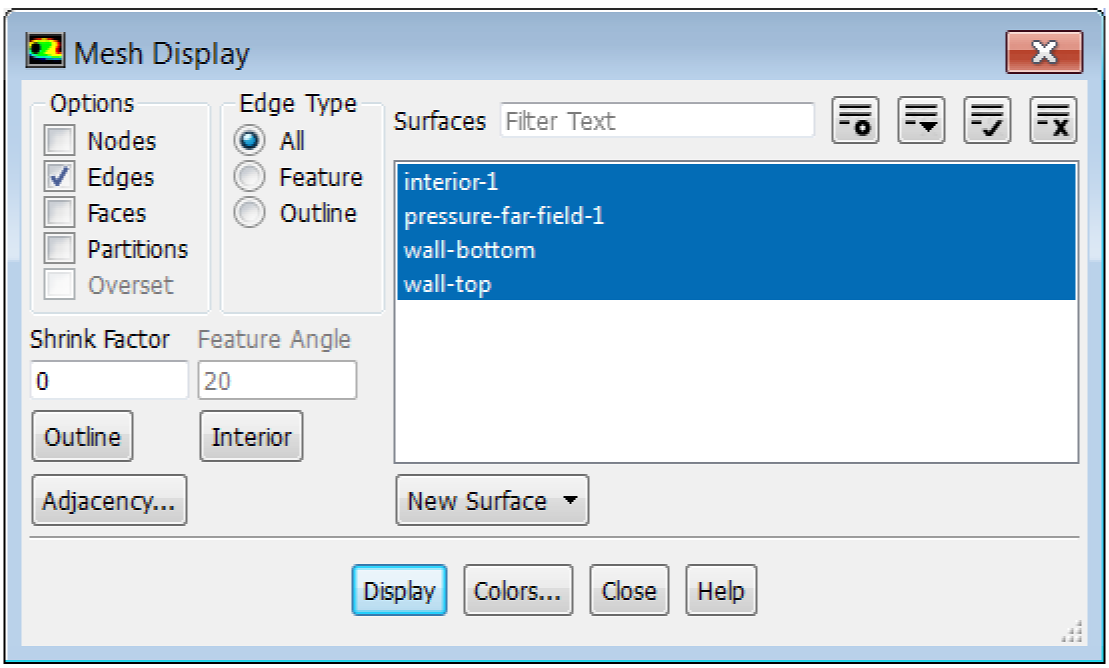
11. Display filled contours of pressure overlaid with the mesh in preparation for creating a surface report definition (Figure 5.4: Pressure Contours After 50 Iterations (p. 249) and Figure 5.5: Magnified View of Pressure Contours Showing Wall-Adjacent Cells (p. 250)).

 **Postprocessing → Graphics → Contours → Edit...**



a. Enable **Filled** in the **Options** group box.

b. Enable **Draw Mesh** to open the **Mesh Display** dialog box.

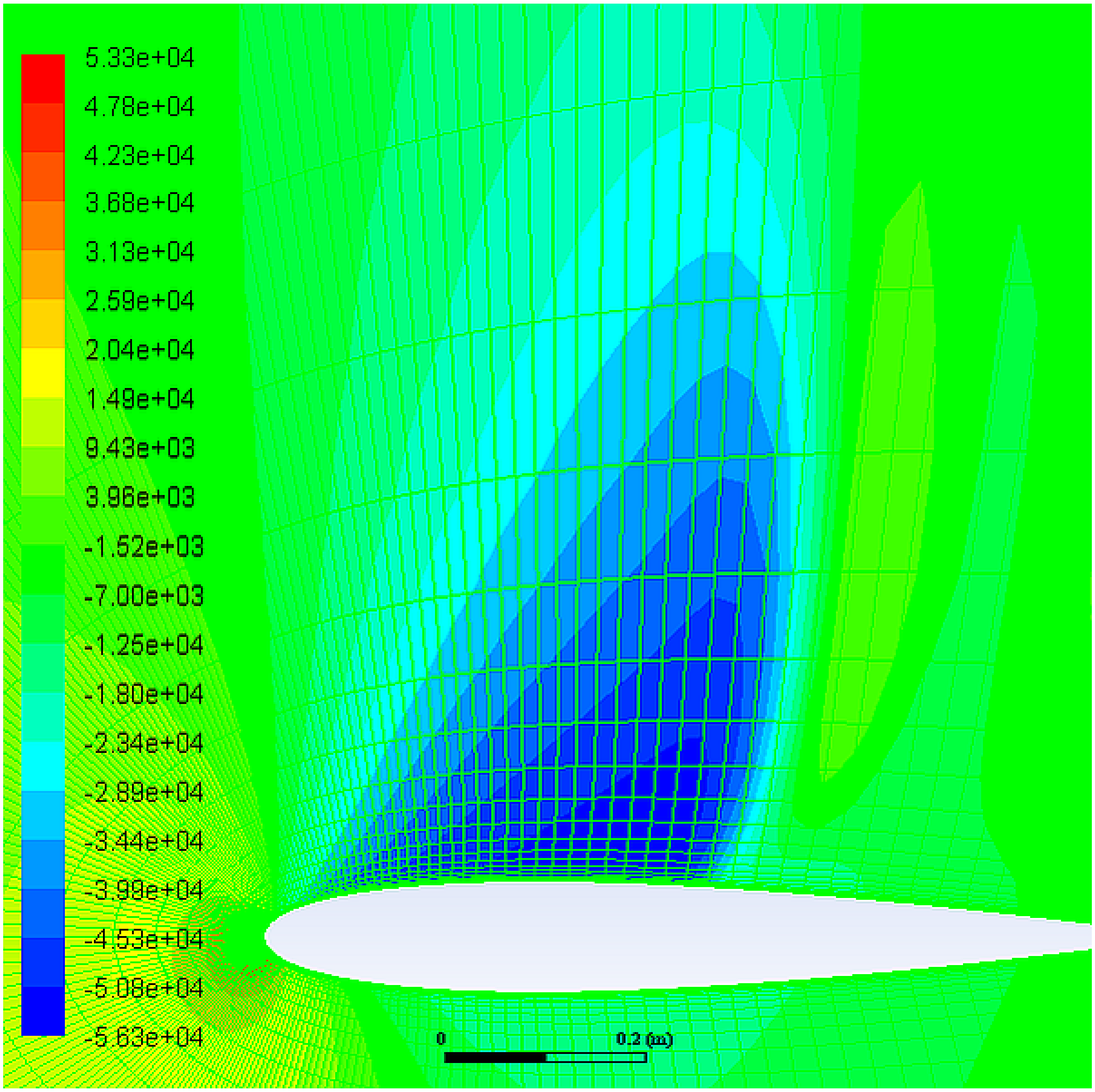


i. Retain the default settings.

ii. Close the **Mesh Display** dialog box.

c. Click **Display** and close the **Contours** dialog box.

Figure 5.4: Pressure Contours After 50 Iterations



The shock is clearly visible on the upper surface of the airfoil, where the pressure jumps to a higher value downstream of the low pressure area.

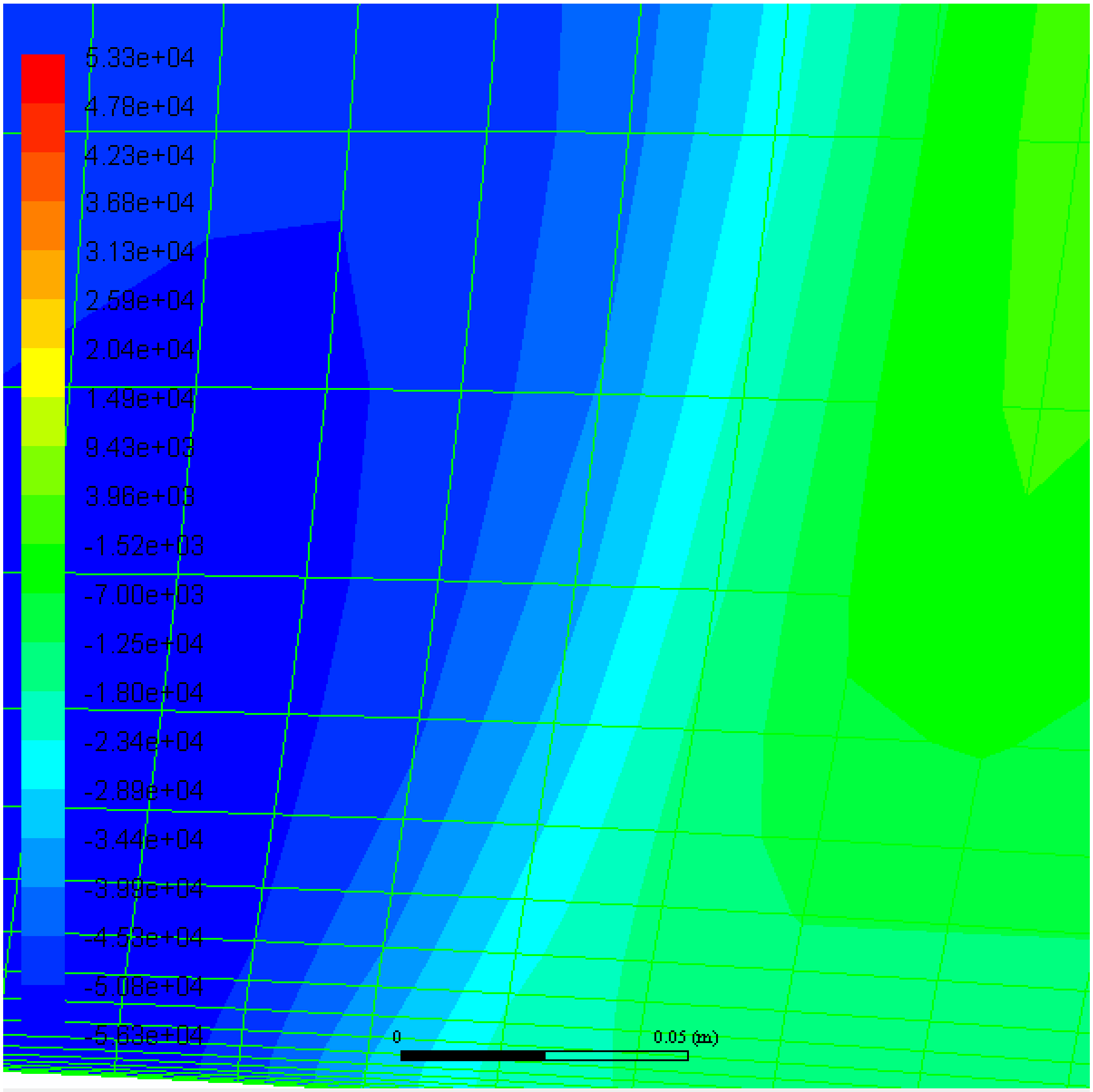
Note

The color indicating a high pressure area near the leading edge of the airfoil is obscured by the overlaid green mesh. To view this contour, simply disable the **Draw Mesh** option in the **Contours** dialog box and click **Display**.

---

d. Zoom in on the shock wave, until individual cells adjacent to the upper surface (**wall-top** boundary) are visible, as shown in Figure 5.5: Magnified View of Pressure Contours Showing Wall-Adjacent Cells (p. 250).

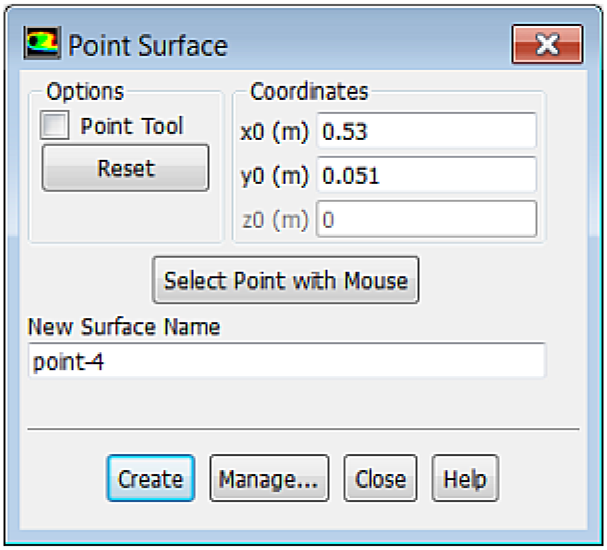
Figure 5.5: Magnified View of Pressure Contours Showing Wall-Adjacent Cells



The magnified region contains cells that are just downstream of the shock and adjacent to the upper surface of the airfoil. In the following step, you will create a point surface inside a wall-adjacent cell, which you will use to create a surface report definition.

12. Create a point surface just downstream of the shock wave.

Postprocessing → **Surface** → **Create** → **Point...**



a. Enter `0.53` m for **x0** and `0.051` m for **y0** in the **Coordinates** group box.

b. Retain the default entry of `point-4` for **New Surface Name**.

c. Click **Create** and close the **Point Surface** dialog box.

Note

You have entered the exact coordinates of the point surface so that your convergence history will match the plots and description in this tutorial. In general, however, you will not know the exact coordinates in advance, so you will need to select the desired location in the graphics window. You do not have to apply the following instructions at this point in the tutorial; they are added here for your information:

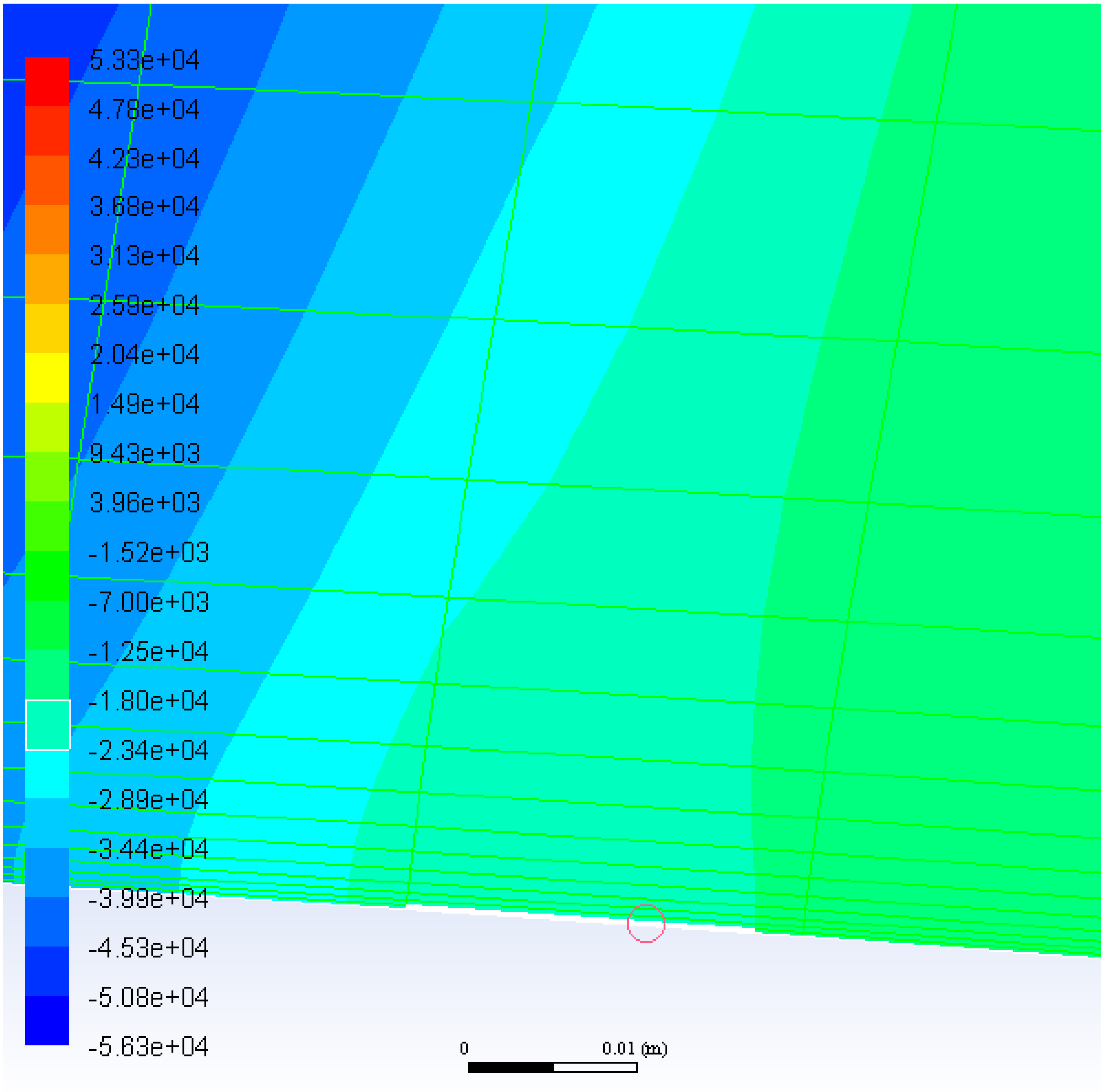
a. In the **Point Surface** dialog box, click the **Select Point with Mouse** button. A **Working** dialog box will open telling you to "Click on a location in the graphics window with the MOUSE-PROBE mouse button."

b. Position the mouse pointer at a point located inside one of the cells adjacent to the upper surface (**wall-top** boundary), downstream of the shock (see Figure 5.6: Pressure Contours after Creating a Point with the Mouse (p. 252)).

c. Click the right mouse button.

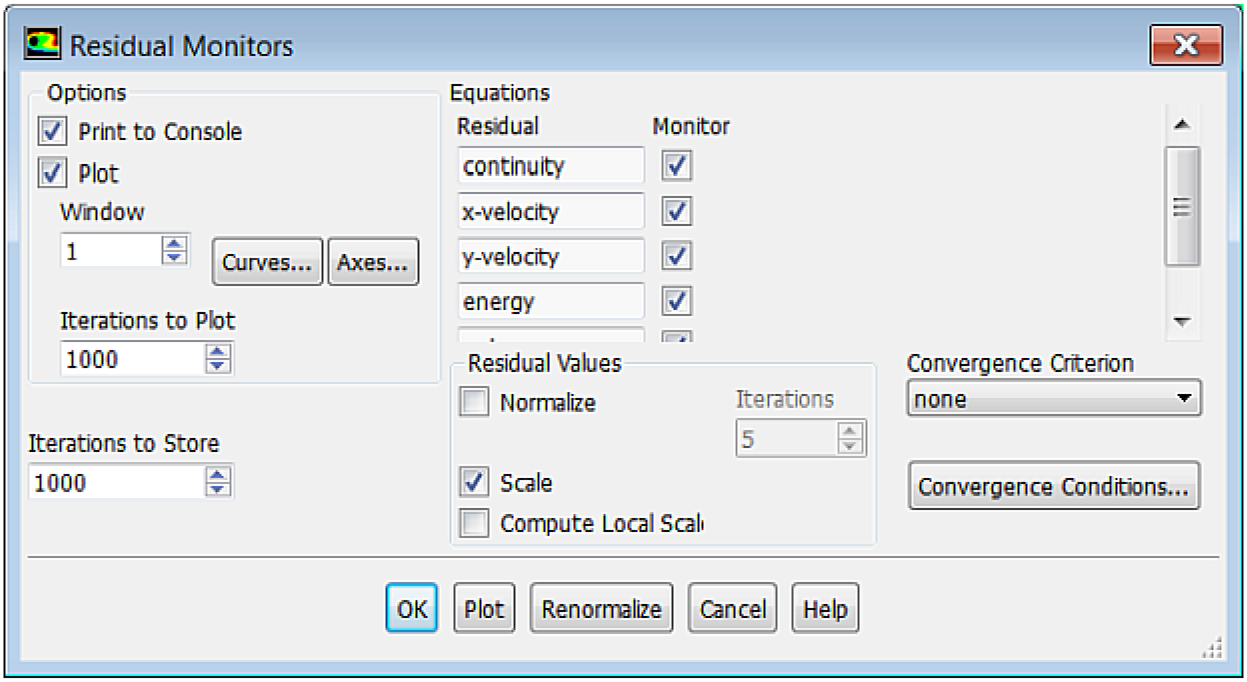
d. Click **Create** to create the point surface and then close the **Point Surface** dialog box.

Figure 5.6: Pressure Contours after Creating a Point with the Mouse



13. Enable residual plotting during the calculation.

Solving → Reports → Residuals...



a. Ensure that **Plot** is enabled in the **Options** group box.

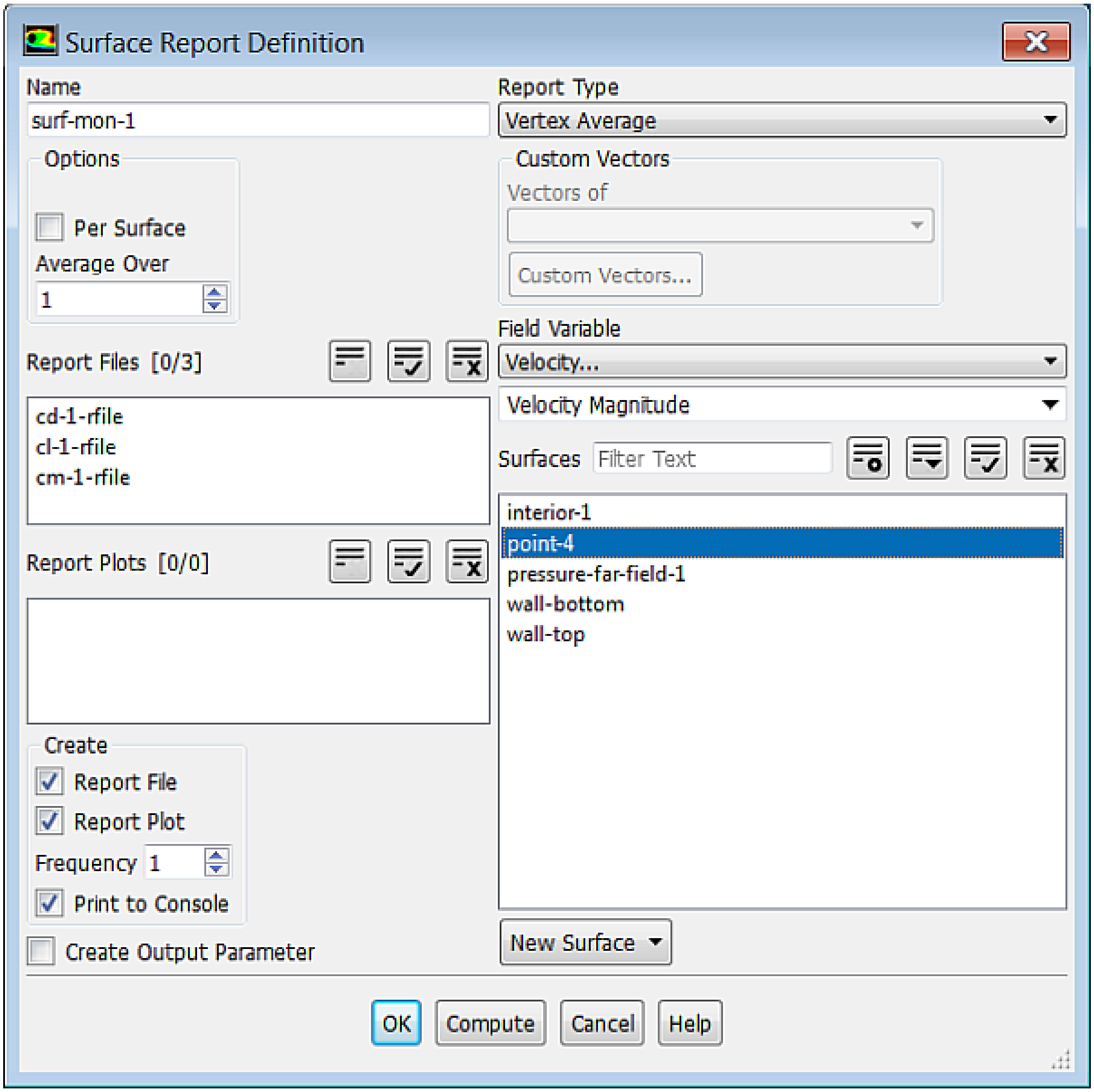
b. Select **none** from the **Convergence Criterion** drop-down list so that automatic convergence checking does not occur.

c. Click **OK** to close the **Residual Monitors** dialog box.

14. Create a surface report definition for tracking the velocity magnitude value at the point created in the previous step.

Since the drag, lift, and moment coefficients are global variables, indicating certain overall conditions, they may converge while local conditions at specific points are still varying from one iteration to the next. To account for this, create a report definition at a point (just downstream of the shock) where there is likely to be significant variation, and monitor the value of the velocity magnitude.

Solving → **Reports** → **Definitions** → **New** → **Surface Report** → **Vertex Average...**



a. Enter **`surf-mon-1`** for **Name**.

b. Select **Velocity...** and **Velocity Magnitude** from the **Field Variable** drop-down list.

c. Select **point-4** in the **Surfaces** selection list.

d. In the **Create** group box, enable **Report File**, **Report Plot** and **Print to Console**.

e. Click **OK** to close the **Surface Report Definition** dialog box.

15. Save the case and data files (`airfoil-1.cas.gz` and `airfoil-1.dat.gz`).

File → Write → Case & Data...

16. Continue the calculation for 200 more iterations.

Solving → Run Calculation → Calculate

The force reports (Figure 5.8: Drag Coefficient Convergence History (p. 256) and Figure 5.9: Lift Coefficient Convergence History (p. 257)) show that the case is converged after approximately 200 iterations.

Figure 5.7: Velocity Magnitude History

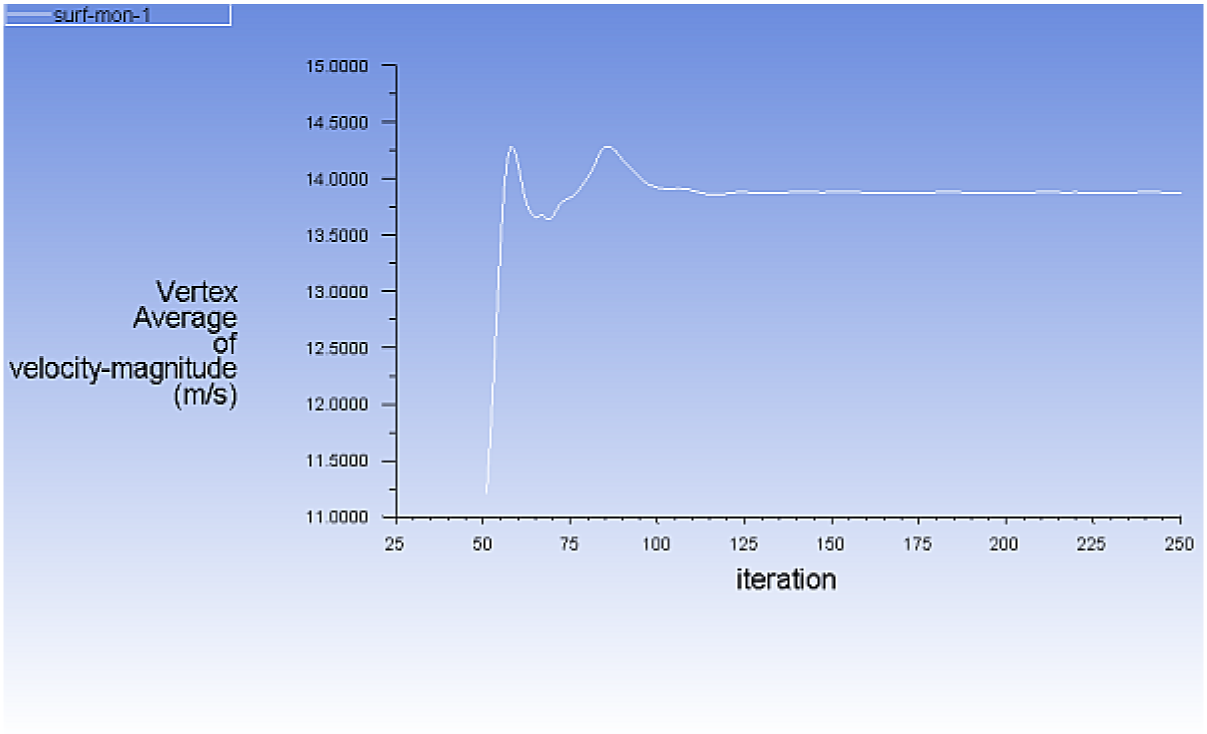


Figure 5.8: Drag Coefficient Convergence History

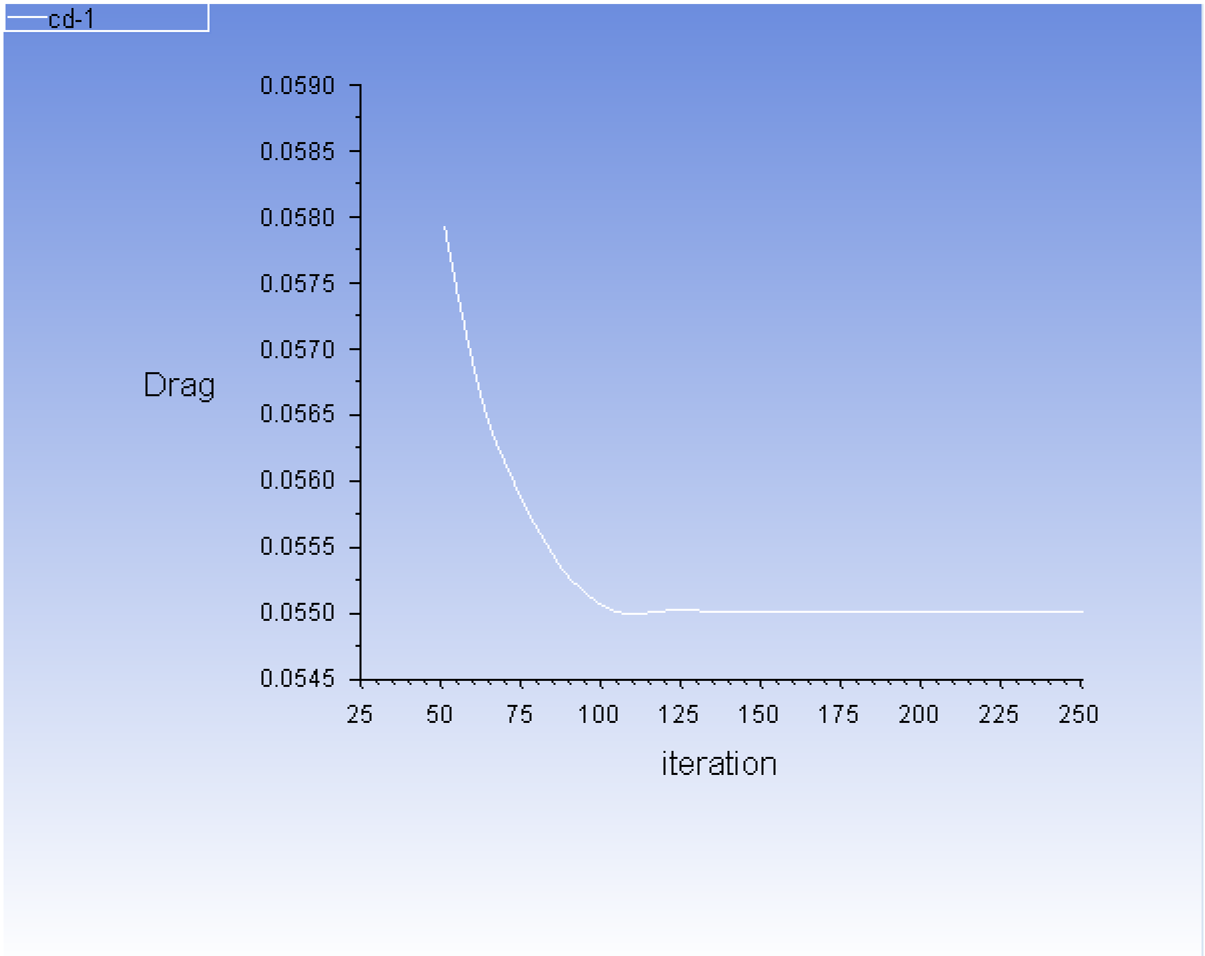


Figure 5.9: Lift Coefficient Convergence History

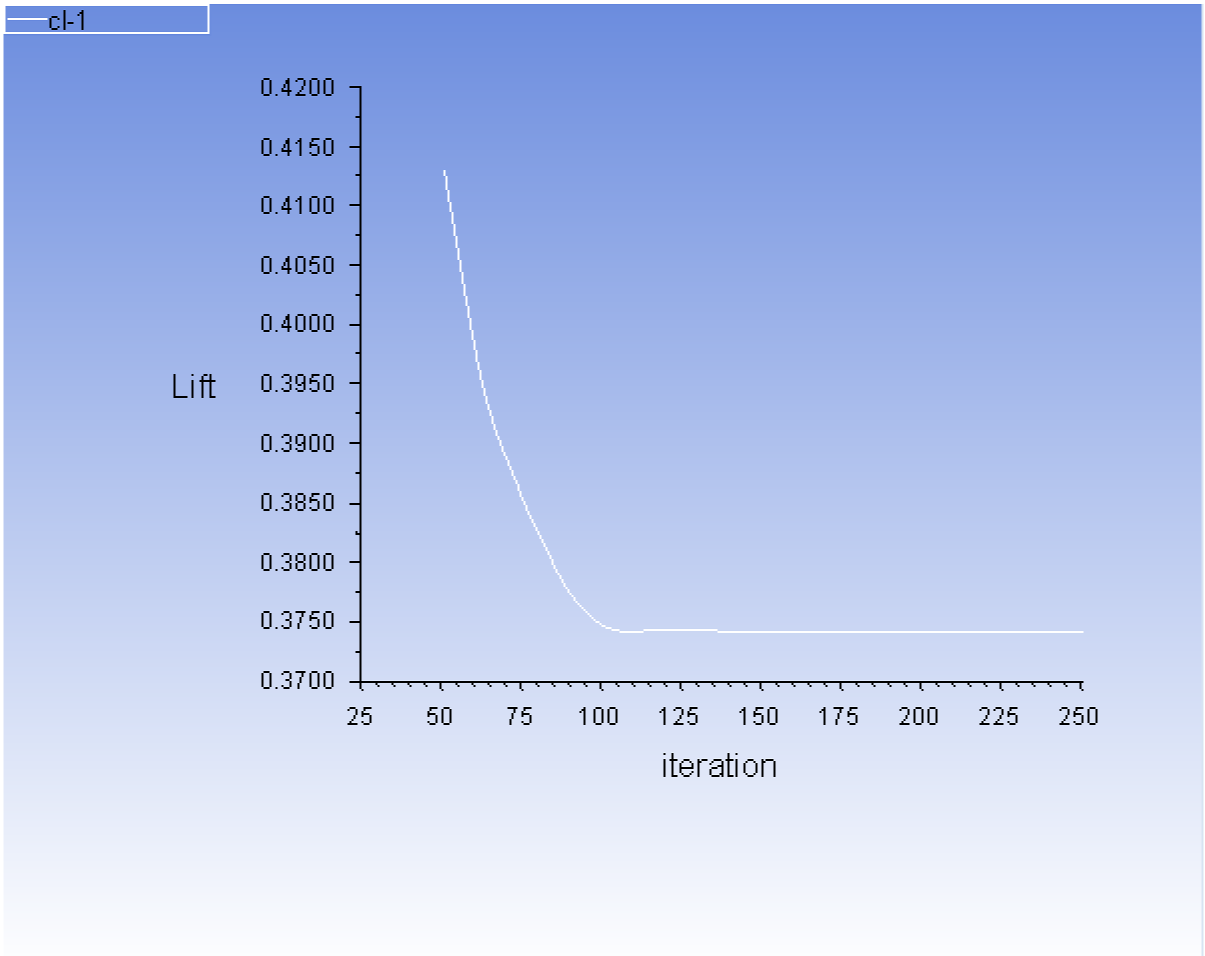
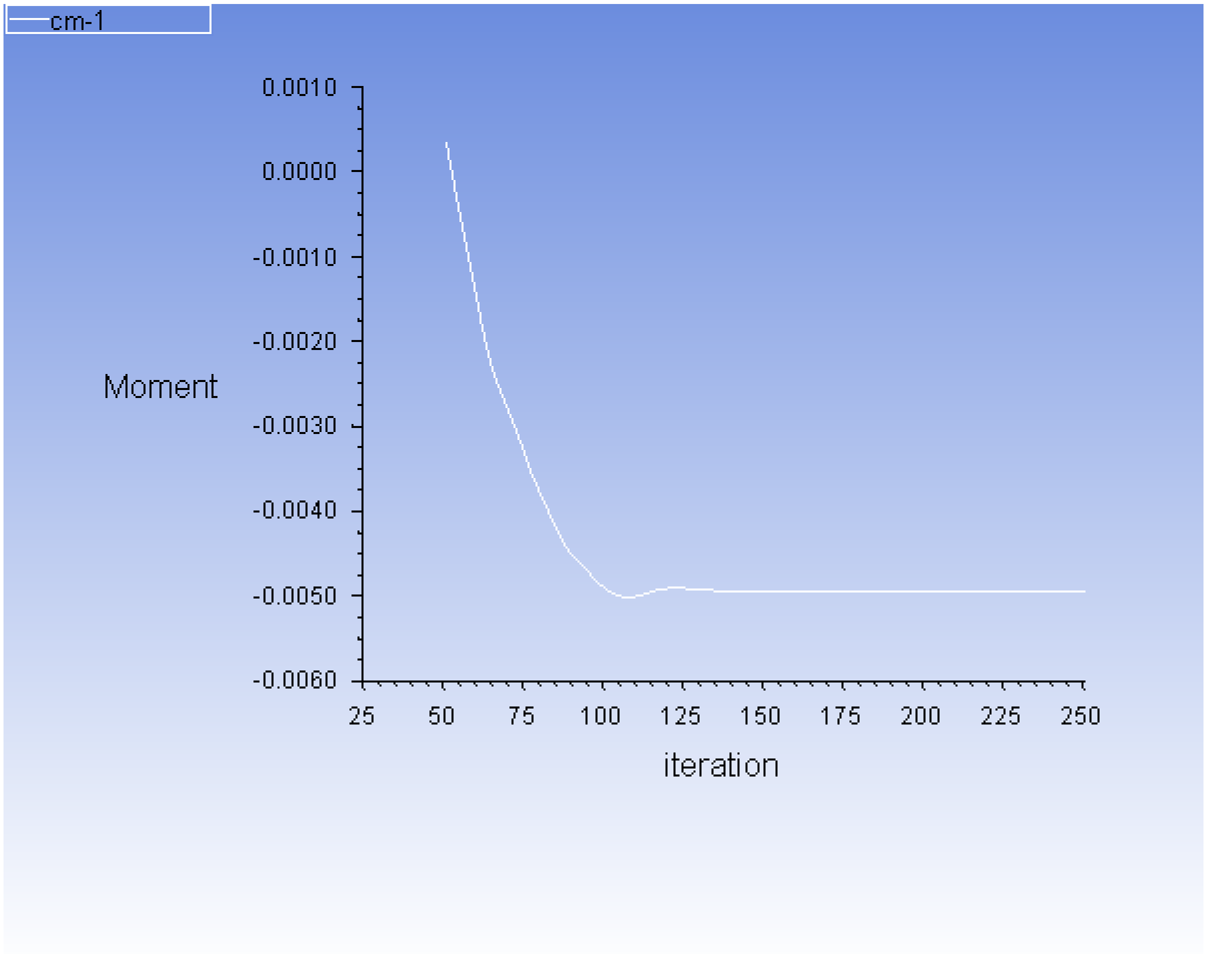


Figure 5.10: Moment Coefficient Convergence History



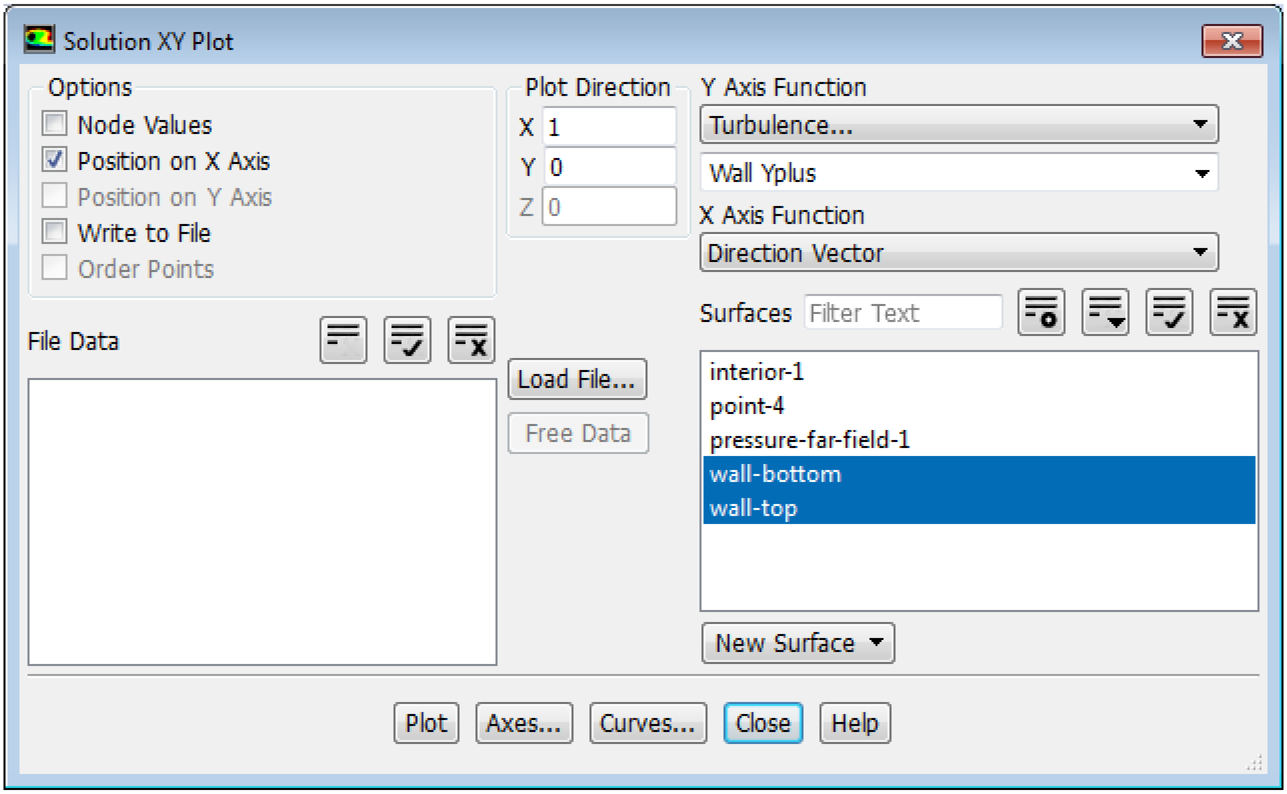
17. Save the case and data files (`airfoil-2.cas.gz` and `airfoil-2.dat.gz`).

File → **Write** → **Case & Data...**

5.4.9. Postprocessing

1. Plot the y^+ distribution on the airfoil (Figure 5.11: XY Plot of y+ Distribution (p. 260)).

Postprocessing → **Plots** → **XY Plot** → **Edit...**



a. Disable **Node Values** in the **Options** group box.

b. Select **Turbulence...** and **Wall Yplus** from the **Y Axis Function** drop-down list.

   Wall Yplus *is available only for cell values.*

c. Select **wall-bottom** and **wall-top** in the **Surfaces** selection list.

d. Click **Plot** and close the **Solution XY Plot** dialog box.

Note

The values of y^+ are dependent on the resolution of the mesh and the Reynolds number of the flow, and are defined only in wall-adjacent cells. The value of y^+ in the wall-adjacent cells dictates how wall shear stress is calculated. When you use the Spalart-Allmaras model, you should check that y^+ of the wall-adjacent cells is either very small (on the order of $y^+=1$), or approximately 30 or greater. Otherwise, you should modify your mesh.

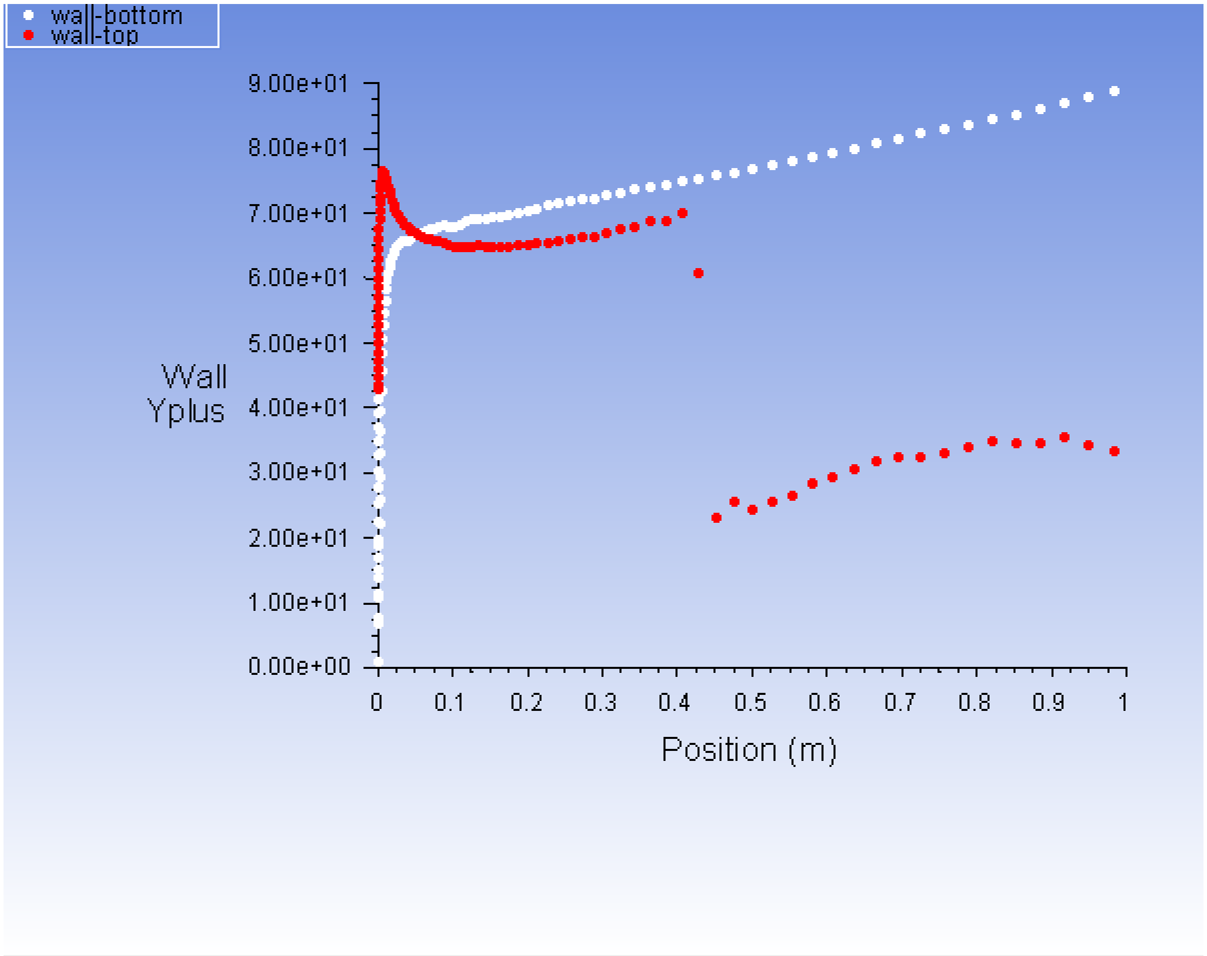
The equation for y^+ is

$$y^+=\frac{y}{\mu}\sqrt{\rho\tau_w} \tag{5.1}$$

where y is the distance from the wall to the cell center, μ is the molecular viscosity, ρ is the density of the air, and τ_w is the wall shear stress.

Figure 5.11: XY Plot of y+ Distribution (p. 260) indicates that, except for a few small regions (notably at the shock and the trailing edge), $y^+>30$ and for much of these regions it does not drop significantly below 30. Therefore, you can conclude that the near-wall mesh resolution is acceptable.

Figure 5.11: XY Plot of y+ Distribution



2. Display filled contours of Mach number (Figure 5.12: Contour Plot of Mach Number (p. 261)).

   Postprocessing → **Graphics** → **Contours** → **Edit...**

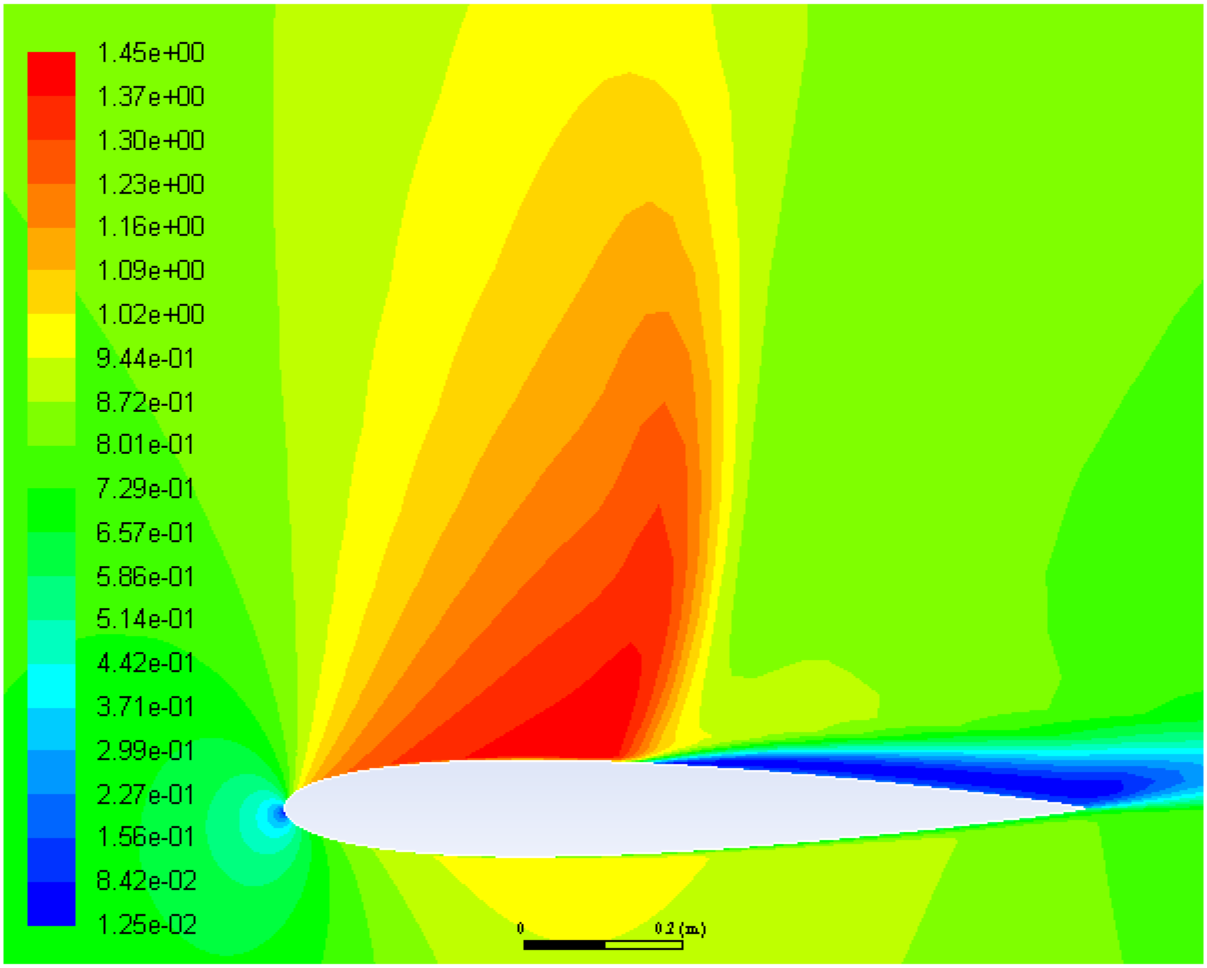
   a. Ensure **Filled** is enabled in the **Options** group box.

   b. Select **Velocity...** and **Mach Number** from the **Contours of** drop-down list.

   c. Click **Display** and close the **Contours** dialog box.

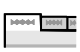
   d. Zoom in on the region around the airfoil, as shown in Figure 5.12: Contour Plot of Mach Number (p. 261).

Figure 5.12: Contour Plot of Mach Number



Note the discontinuity, in this case a shock, on the upper surface of the airfoil in Figure 5.12: Contour Plot of Mach Number (p. 261) at about $x/c \approx 0.45$.

3. Plot the pressure distribution on the airfoil (Figure 5.13: XY Plot of Pressure (p. 262)).

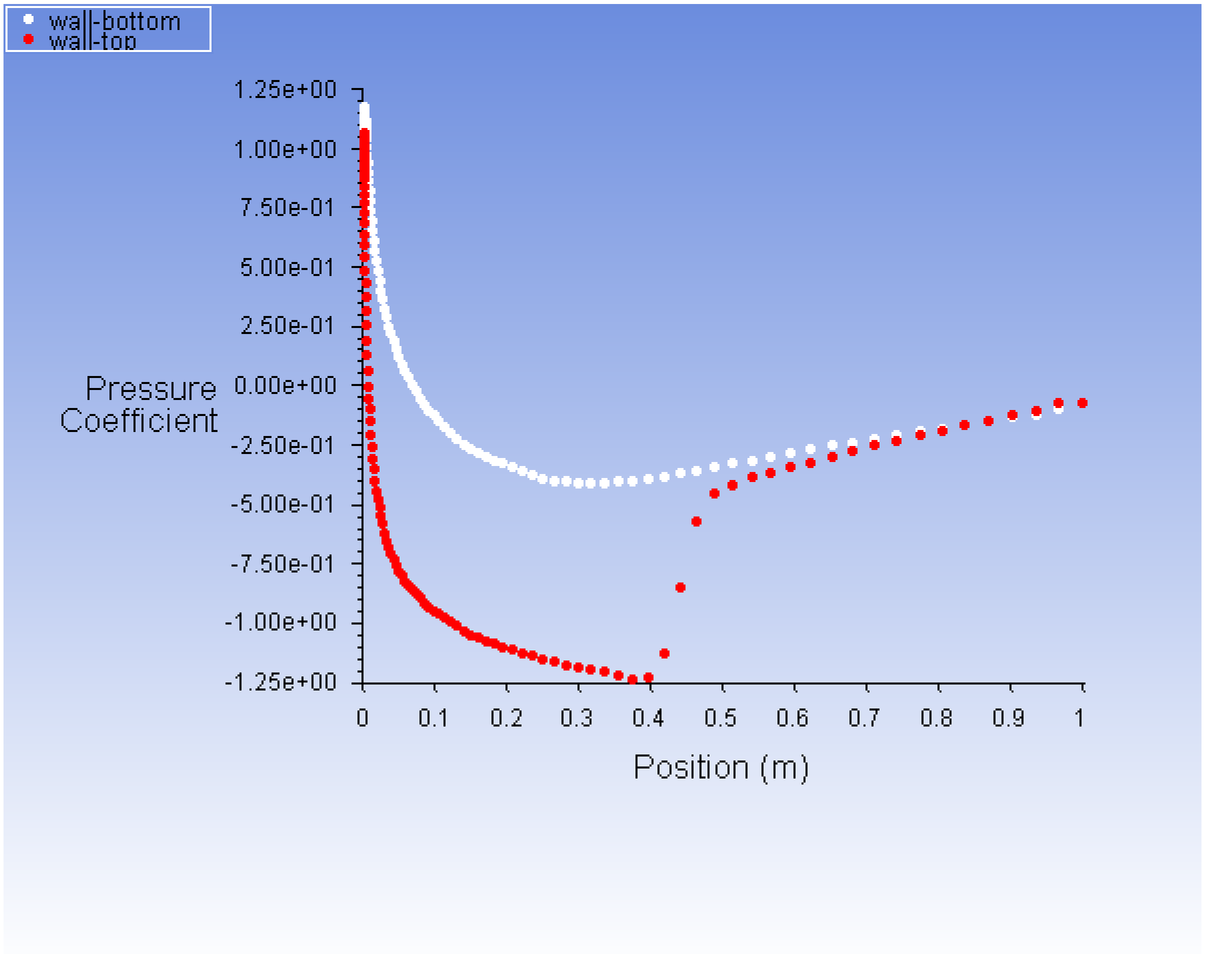
    **Postprocessing** → **Plots** → **XY Plot** → **Edit...**

   a. Enable **Node Values**.

   b. Select **Pressure...** and **Pressure Coefficient** from the **Y Axis Function** drop-down lists.

   c. Click **Plot**.

Figure 5.13: XY Plot of Pressure



Notice the effect of the shock wave on the upper surface in Figure 5.13: XY Plot of Pressure (p. 262).

4. Plot the x component of wall shear stress on the airfoil surface (Figure 5.14: XY Plot of x Wall Shear Stress (p. 263)).

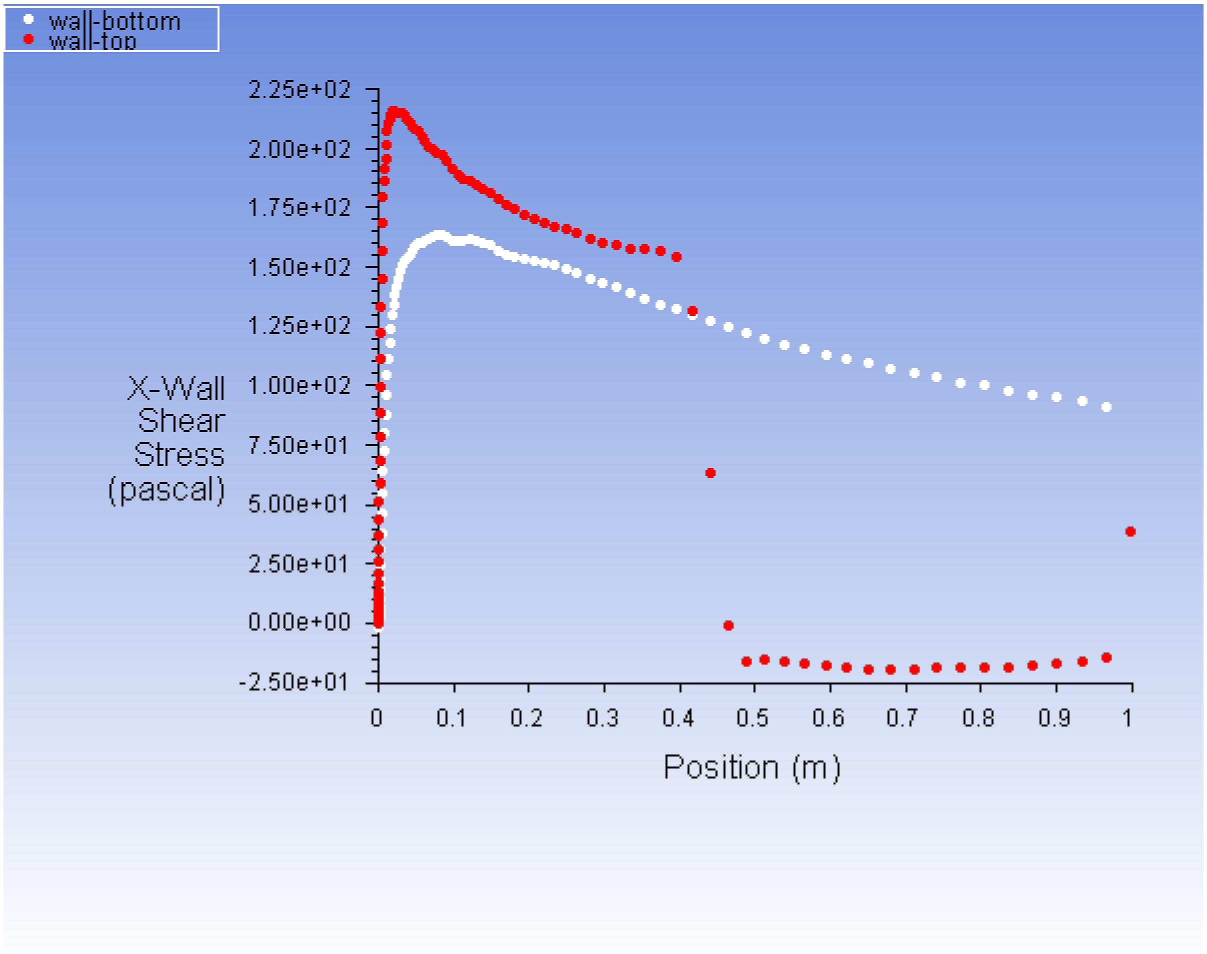
   a. Disable **Node Values**.

   b. Select **Wall Fluxes...** and **X-Wall Shear Stress** from the **Y Axis Function** drop-down lists.

   c. Click **Plot** and close the **Solution XY Plot** dialog box.

   As shown in Figure 5.14: XY Plot of x Wall Shear Stress (p. 263), the large, adverse pressure gradient induced by the shock causes the boundary layer to separate. The point of separation is where the wall shear stress vanishes. Flow reversal is indicated here by negative values of the x component of the wall shear stress.

Figure 5.14: XY Plot of x Wall Shear Stress



5. Display filled contours of the *x* component of velocity (Figure 5.15: Contour Plot of x Component of Velocity (p. 264)).

   Postprocessing → **Graphics** → **Contours** → **Edit...**

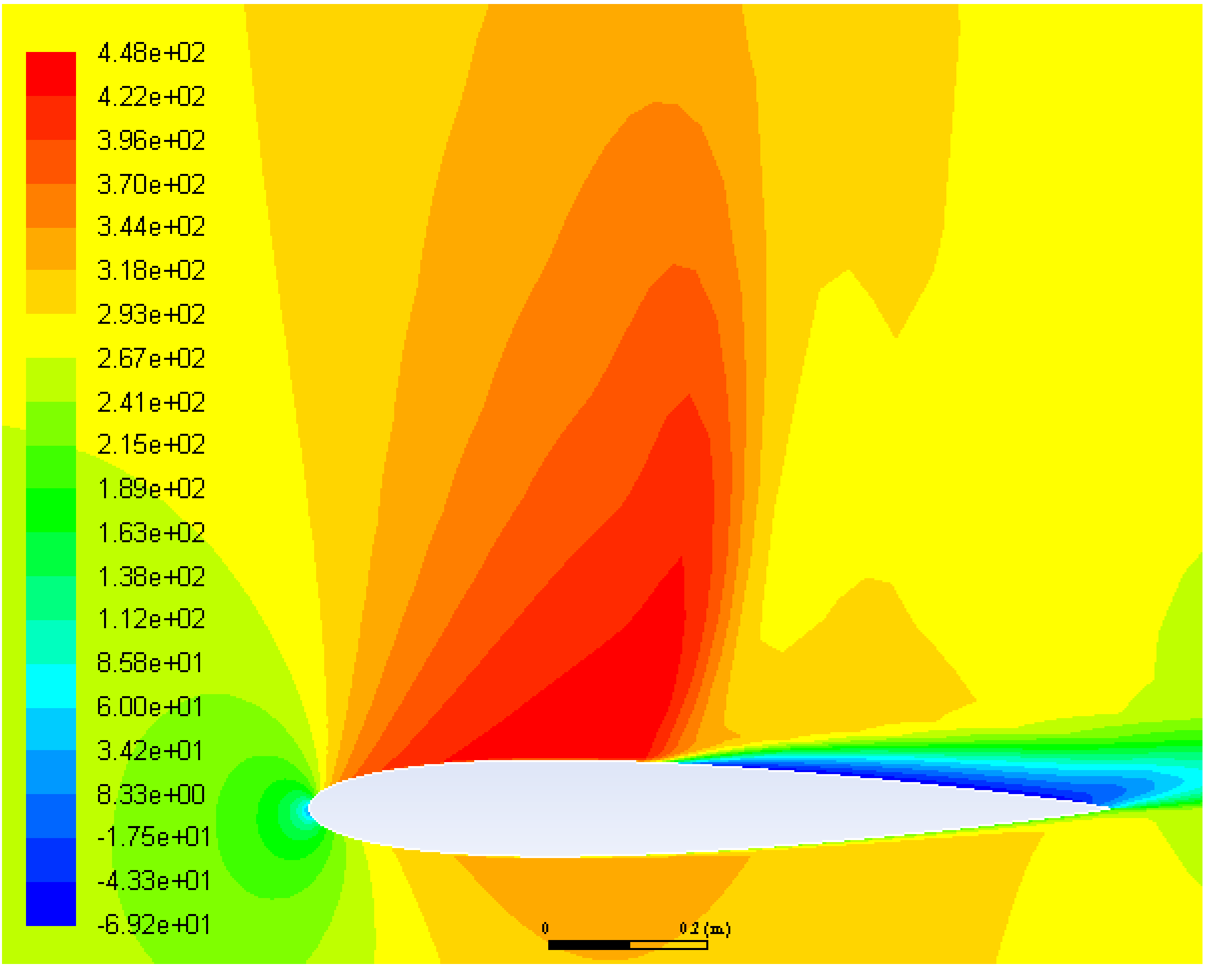
   a. Ensure **Filled** is enabled in the **Options** group box.

   b. Select **Velocity...** and **X Velocity** from the **Contours of** drop-down lists.

   c. Click **Display** and close the **Contours** dialog box.

   Note the flow reversal downstream of the shock in Figure 5.15: Contour Plot of x Component of Velocity (p. 264).

Figure 5.15: Contour Plot of x Component of Velocity



6. Plot velocity vectors (Figure 5.16: Plot of Velocity Vectors Downstream of the Shock (p. 265)).

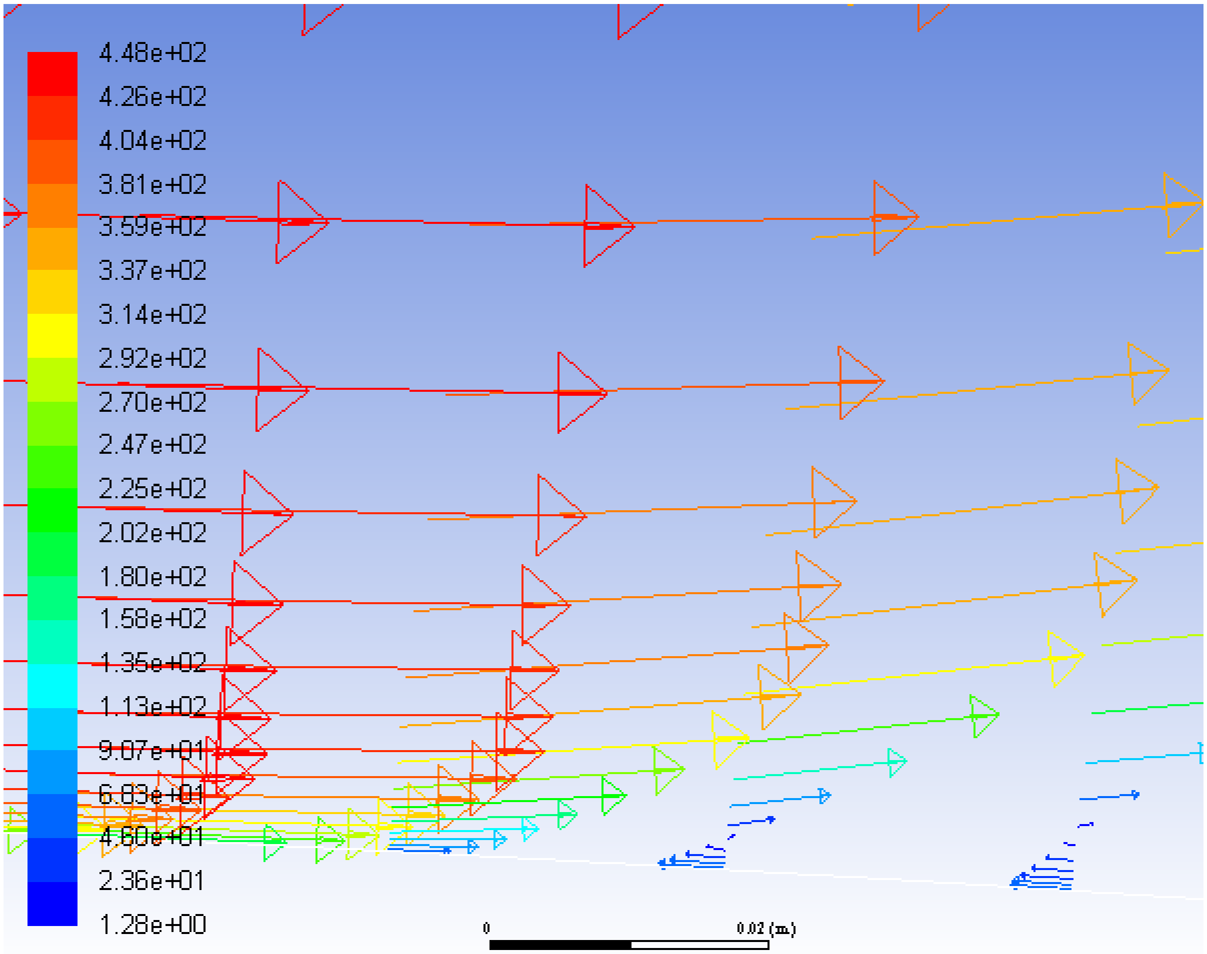
   Postprocessing → **Graphics** → **Vectors** → **Edit...**

   a. Enter 15 for **Scale**.

   b. Click **Display** and close the **Vectors** dialog box.

   c. Zoom in on the flow above the upper surface at a point downstream of the shock, as shown in Figure 5.16: Plot of Velocity Vectors Downstream of the Shock (p. 265).

Figure 5.16: Plot of Velocity Vectors Downstream of the Shock



Flow reversal is clearly visible in Figure 5.16: Plot of Velocity Vectors Downstream of the Shock (p. 265).

5.5. Summary

This tutorial demonstrated how to set up and solve an external aerodynamics problem using the pressure-based coupled solver with pseudo transient under-relaxation and the Spalart-Allmaras turbulence model. It showed how to monitor convergence using force and surface report definitions, and demonstrated the use of several postprocessing tools to examine the flow phenomena associated with a shock wave.

5.6. Further Improvements

This tutorial guides you through the steps to reach an initial solution. You may be able to obtain a more accurate solution by using an appropriate higher-order discretization scheme and by adapting the mesh. Mesh adaption can also ensure that the solution is independent of the mesh. These steps are demonstrated in Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121).

Chapter 6: Modeling Transient Compressible Flow

This tutorial is divided into the following sections:

6.1. Introduction

In this tutorial, ANSYS Fluent's density-based implicit solver is used to predict the time-dependent flow through a two-dimensional nozzle. As an initial condition for the transient problem, a steady-state solution is generated to provide the initial values for the mass flow rate at the nozzle exit.

This tutorial demonstrates how to do the following:

- Calculate a steady-state solution (using the density-based implicit solver) as an initial condition for a transient flow prediction.
- Define a transient boundary condition using a user-defined function (UDF).
- Use dynamic mesh adaption for both steady-state and transient flows.
- Calculate a transient solution using the second-order implicit transient formulation and the density-based implicit solver.
- Create an animation of the transient flow using ANSYS Fluent's transient solution animation feature.

6.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

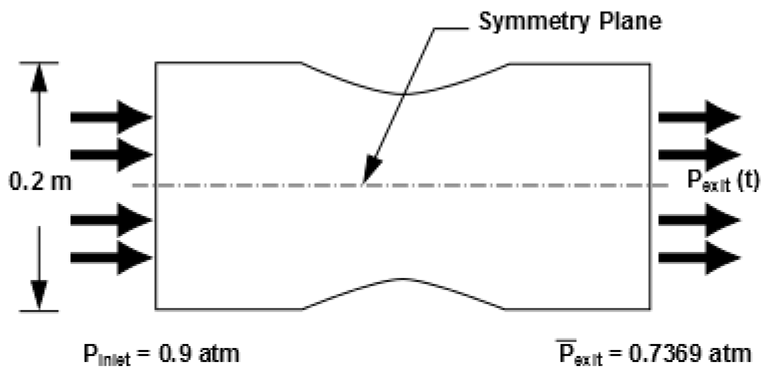
- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

and that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

6.3. Problem Description

The geometry to be considered in this tutorial is shown in Figure 6.1: Problem Specification (p. 268). Flow through a simple nozzle is simulated as a 2D planar model. The nozzle has an inlet height of 0.2 m, and the nozzle contours have a sinusoidal shape that produces a 20% reduction in flow area. Symmetry allows only half of the nozzle to be modeled.

Figure 6.1: Problem Specification



6.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

6.4.1. Preparation

To prepare for running this tutorial:

1. Set up a working folder on the computer you will be using.

2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

   Note

   If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

3. Enter the name of this tutorial into the search bar.

4. Narrow the results by using the filter on the left side of the page.

a. Click **ANSYS Fluent** under **Product**.

b. Click **18.0** under **Version**.

5. Select this tutorial from the list.

6. Click the **unsteady_compressible_R180.zip** link to download the input files.

7. Unzip the `unsteady_compressible_R180.zip` file you downloaded to your working folder.

   The files `nozzle.msh` *and* `pexit.c` *can be found in the* `unsteady_compressible` *folder created after unzipping the file.*

8. Use Fluent Launcher to start the **2D** version of ANSYS Fluent.

   Fluent Launcher displays your **Display Options** preferences from the previous session.

   For more information about the Fluent Launcher, see starting ANSYS Fluent using the Fluent Launcher in the Fluent Getting Started Guide.

9. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.

10. Ensure that **Serial** is selected under **Processing Options**.

11. Disable the **Double Precision** option.

6.4.2. Reading and Checking the Mesh

1. Read the mesh file `nozzle.msh`.

   File → **Read** → **Mesh...**

   The mesh for the half of the geometry is displayed in the graphics window.

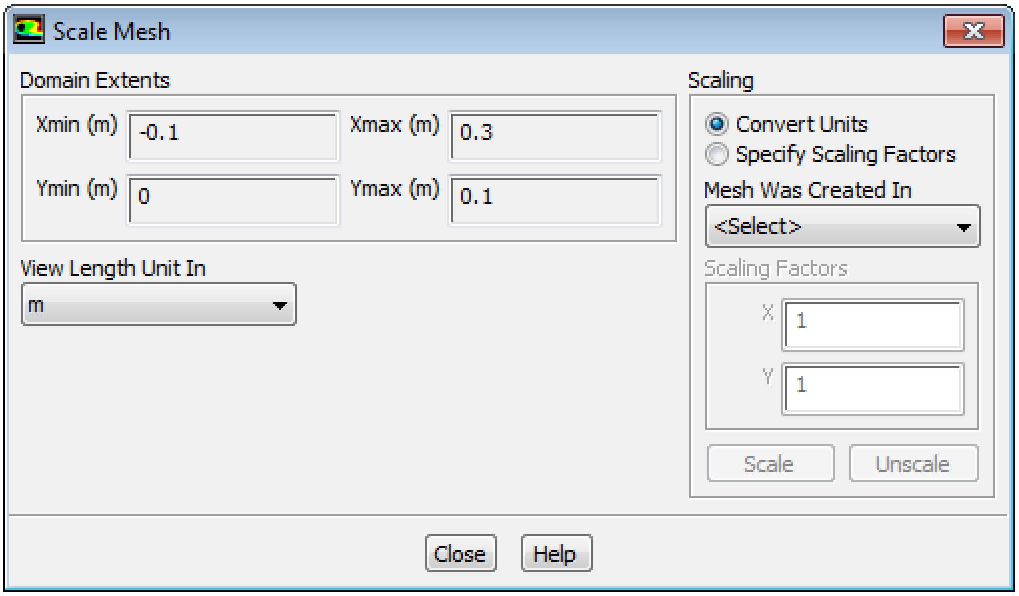
2. Check the mesh.

   Setting Up Domain → **Mesh** → **Check**

   ANSYS Fluent will perform various checks on the mesh and will report the progress in the console. Ensure that the reported minimum volume is a positive number.

3. Verify that the mesh size is correct.

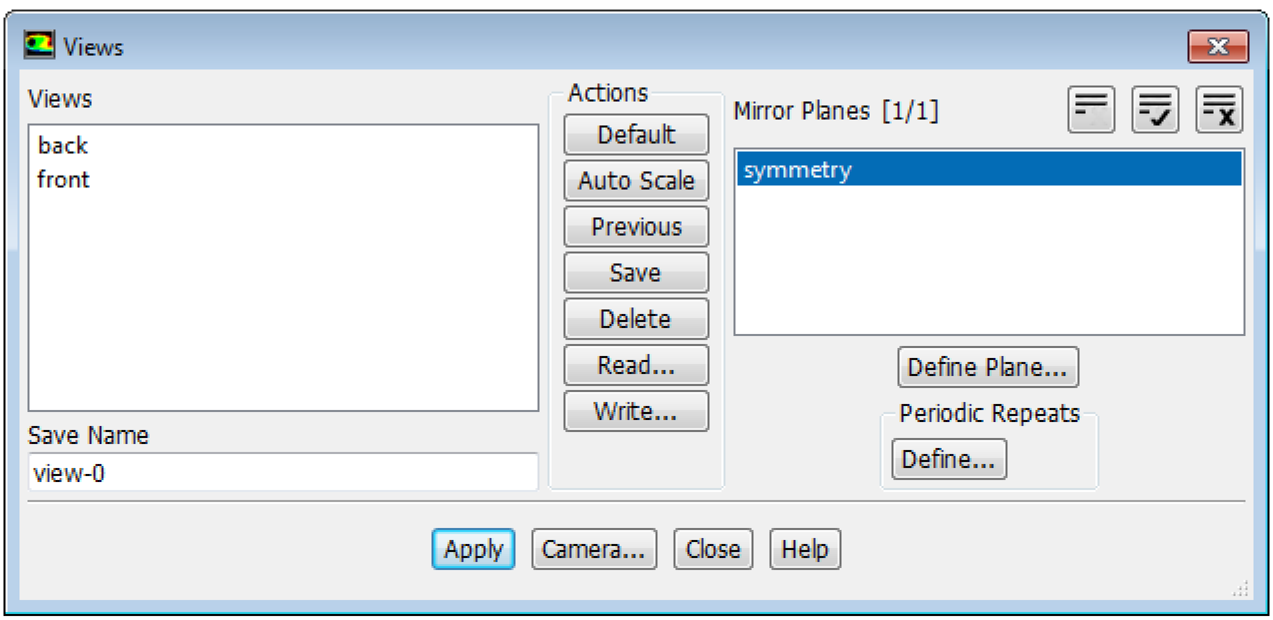
   Setting Up Domain → **Mesh** → **Scale**



Close the **Scale Mesh** dialog box.

4. Mirror the mesh across the centerline (Figure 6.2: 2D Nozzle Mesh Display with Mirroring (p. 271)).

   Viewing → **Display** → **Views...**

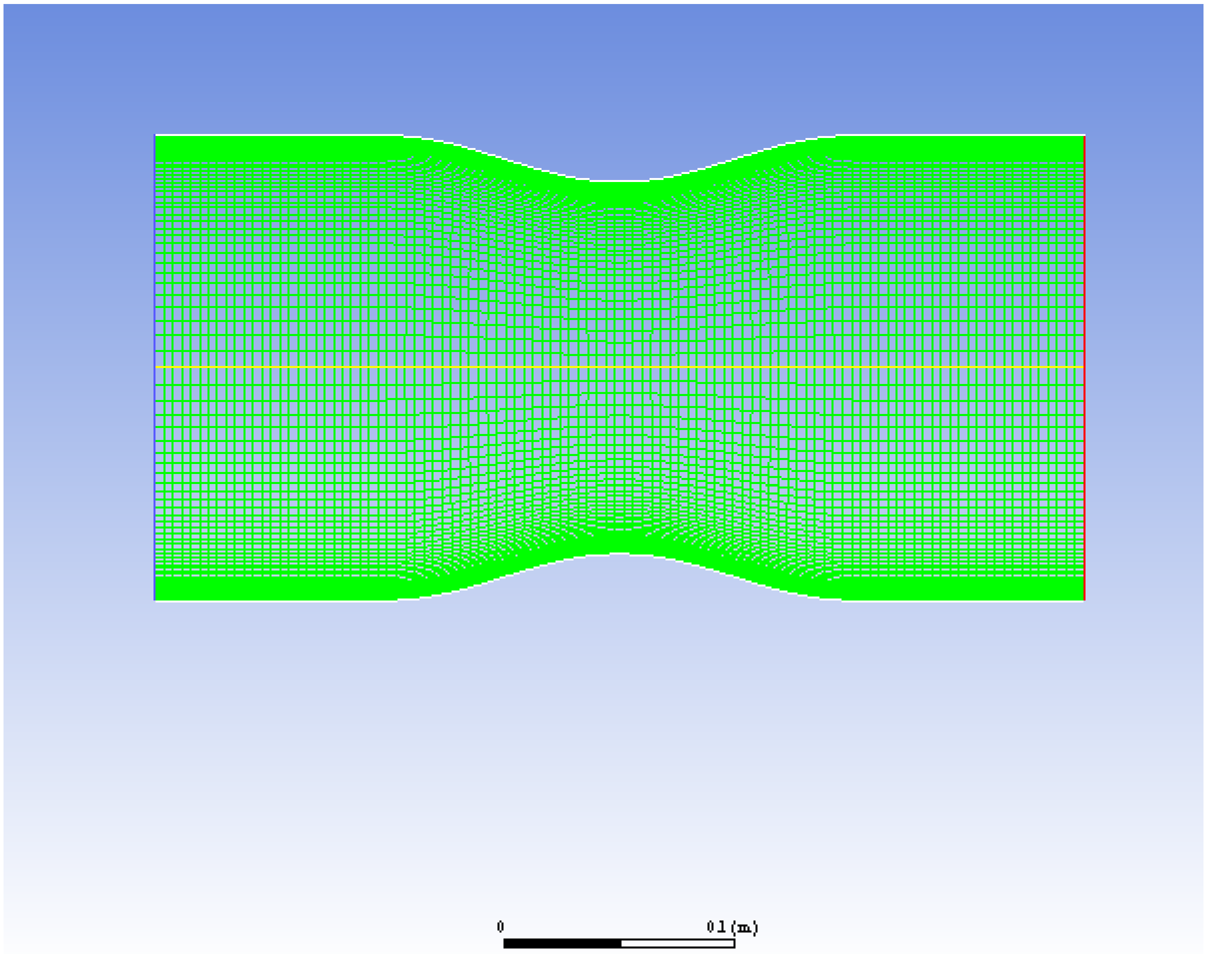


   a. Select **symmetry** in the **Mirror Planes** selection list.

   b. Click **Apply** to refresh the display.

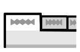
   c. Close the **Views** dialog box.

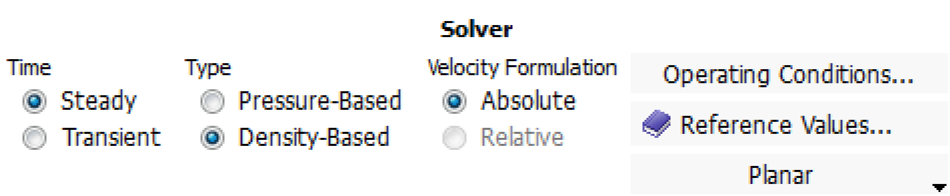
Figure 6.2: 2D Nozzle Mesh Display with Mirroring



6.4.3. Solver and Analysis Type

1. Select the solver settings.

 **Setting Up Physics** → **Solver** → **Density-Based**



a. In the **Solver** group of the **Setting Up Physics** tab, select **Density-Based** from the **Type** list.

The density-based implicit solver is the solver of choice for compressible, transonic flows without significant regions of low-speed flow. In cases with significant low-speed flow regions, the pressure-based solver is preferred. Also, for transient cases with traveling shocks, the density-based explicit solver with explicit time stepping may be the most efficient.

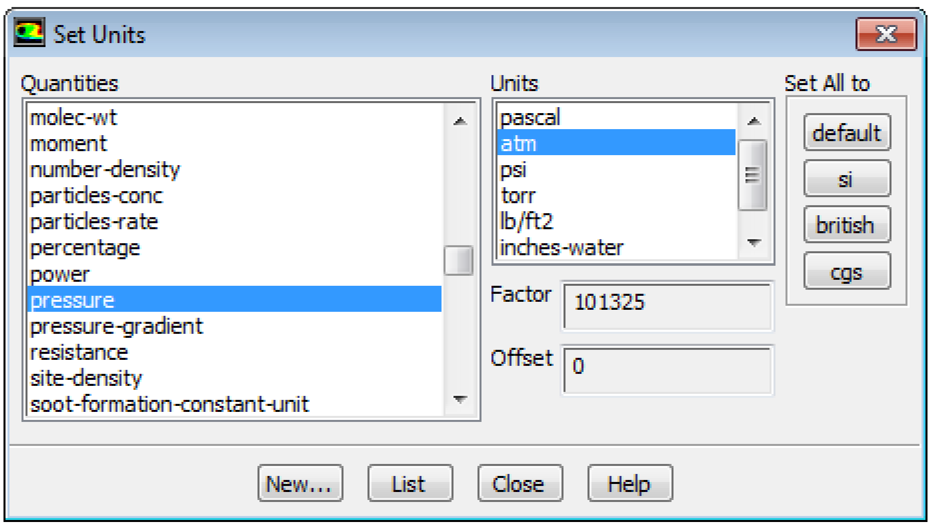
b. Retain the default selection of **Steady** from the **Time** list.

> **Note**
>
> You will solve for the steady flow through the nozzle initially. In later steps, you will use these initial results as a starting point for a transient calculation.

2. For convenience, change the unit of measurement for pressure.

   The pressure for this problem is specified in atm, which is not the default unit in ANSYS Fluent. You must redefine the pressure unit as atm.

   Setting Up Domain → Mesh → Units...



   a. Select **pressure** in the **Quantities** selection list.

      Scroll down the list to find **pressure**.

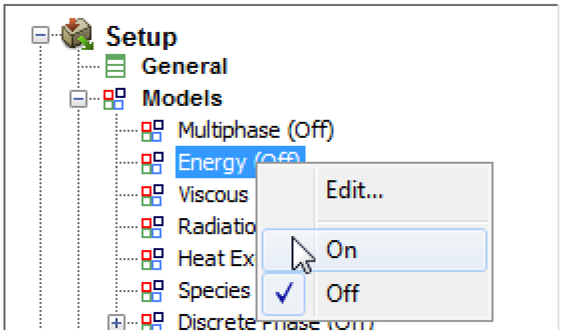
   b. Select **atm** in the **Units** selection list.

   c. Close the **Set Units** dialog box.

6.4.4. Models

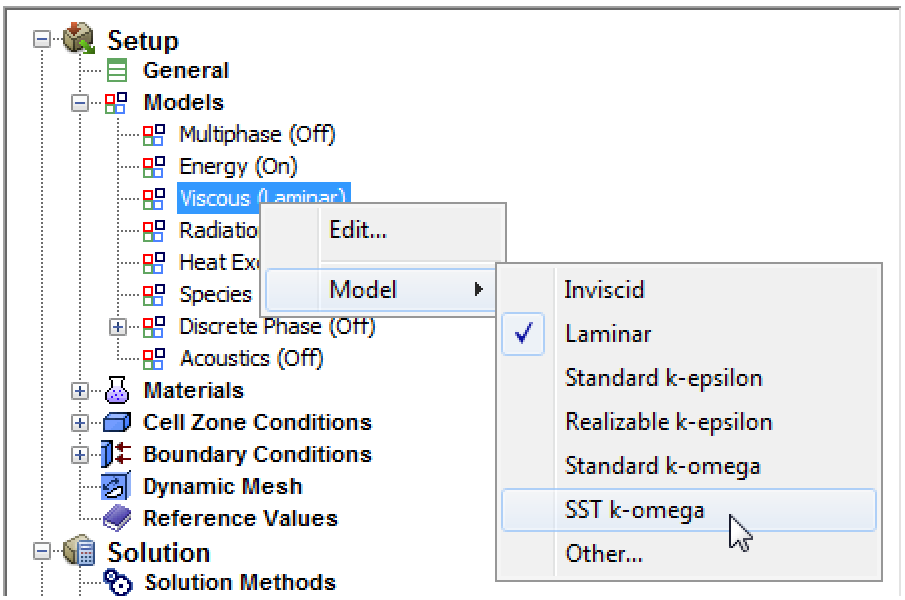
1. Enable the energy equation.

   Setup → Models → Energy **Model → On**



2. Select the k-omega SST turbulence model.

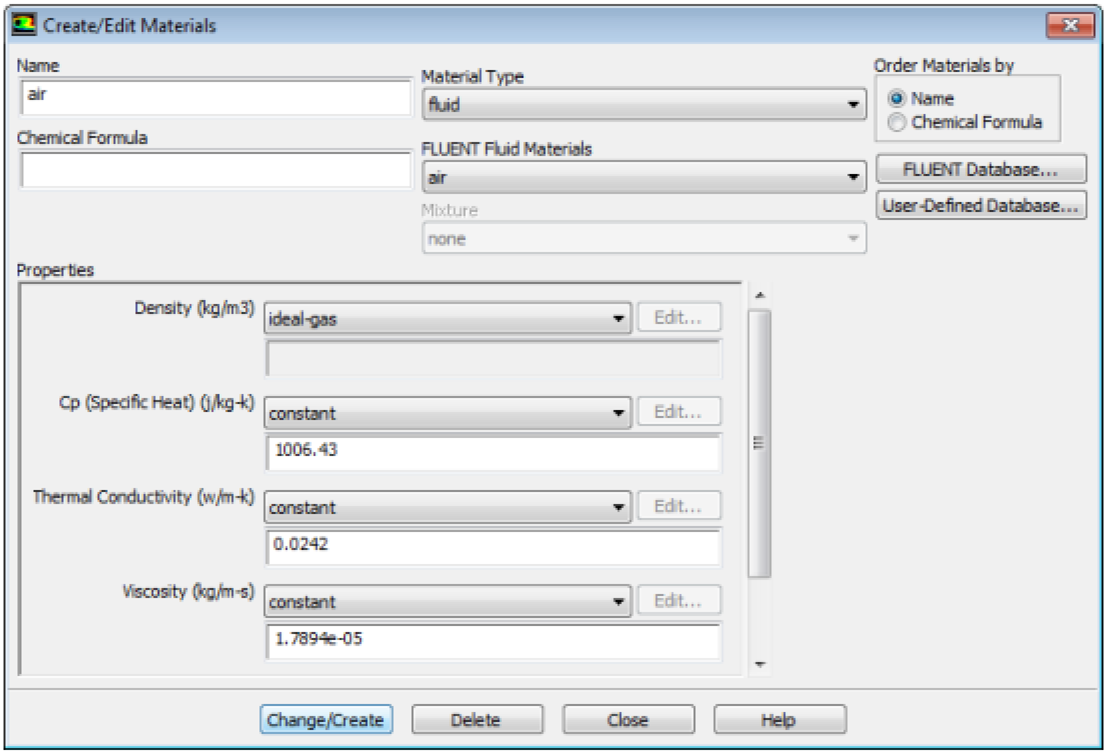
Setup → **Models** → **Viscous** **Model** → **SST k-omega**



6.4.5. Materials

1. Define the settings for air, the default fluid material.

Setup → **Materials** → **air** → **Create/Edit...**



a. Select **ideal-gas** from the **Density** drop-down list in the **Properties** group box, so that the ideal gas law is used to calculate density.

Note

ANSYS Fluent automatically enables the solution of the energy equation when the ideal gas law is used, in case you did not already enable it manually in the **Energy** dialog box.

---

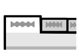
b. Retain the default values for all other properties.

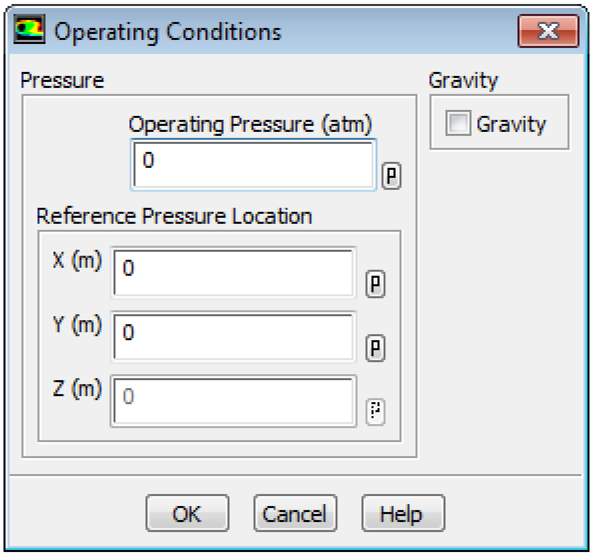
c. Click the **Change/Create** button to save your change.

d. Close the **Create/Edit Materials** dialog box.

6.4.6. Operating Conditions

1. Define the operating pressure.

 **Setting Up Physics → Solver → Operating Conditions...**



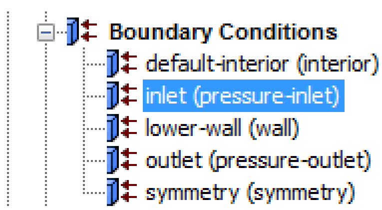
a. Enter 0 atm for **Operating Pressure**.

b. Click **OK** to close the **Operating Conditions** dialog box.

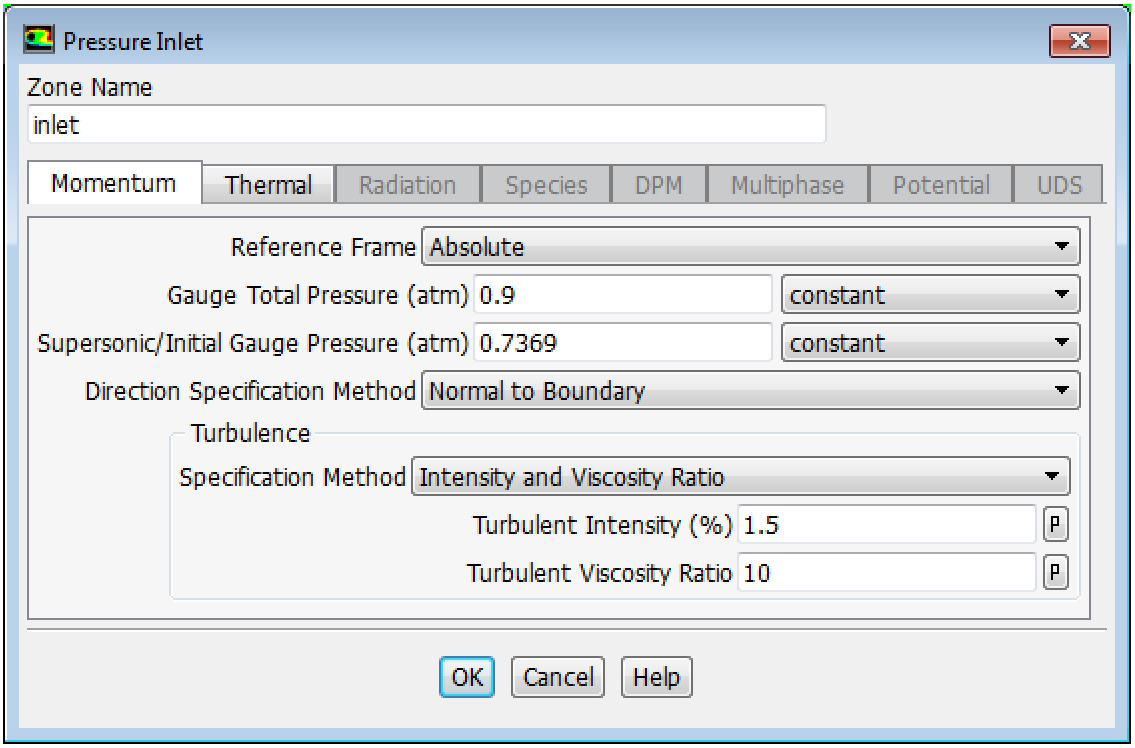
Since you have set the operating pressure to zero, you will specify the boundary condition inputs for pressure in terms of absolute pressures when you define them in the next step. Boundary condition inputs for pressure should always be relative to the value used for operating pressure.

6.4.7. Boundary Conditions

1. Define the boundary conditions for the nozzle inlet (**inlet**).



Setup → **Boundary Conditions** → **inlet (pressure-inlet)** **Edit...**



a. Enter `0.9` atm for **Gauge Total Pressure**.

b. Enter `0.7369` atm for **Supersonic/Initial Gauge Pressure**.

The inlet static pressure estimate is the mean pressure at the nozzle exit. This value will be used during the solution initialization phase to provide a guess for the nozzle velocity.

c. Retain **Intensity and Viscosity Ratio** from the **Specification Method** drop-down list in the **Turbulence** group box.

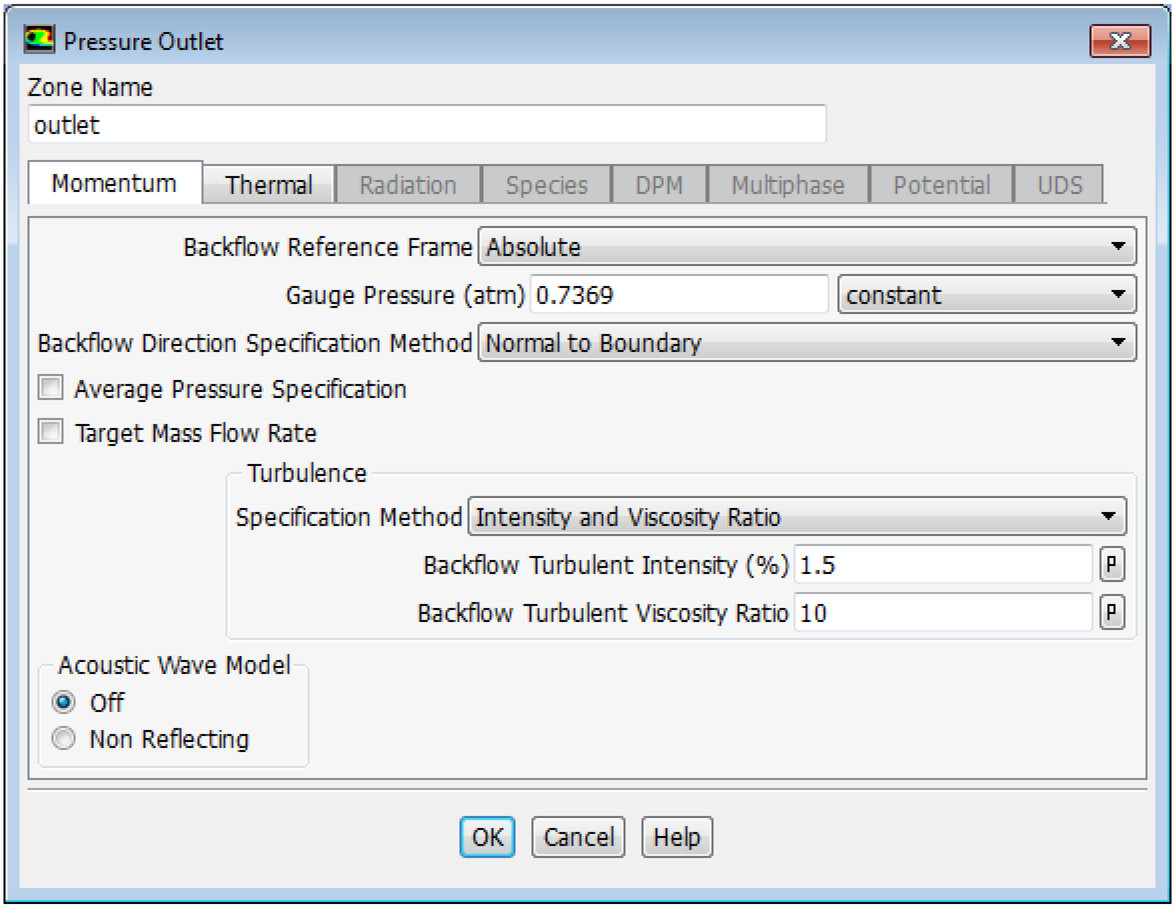
d. Enter `1.5`% for **Turbulent Intensity**.

e. Retain the setting of `10` for **Turbulent Viscosity Ratio**.

f. Click **OK** to close the **Pressure Inlet** dialog box.

2. Define the boundary conditions for the nozzle exit (**outlet**).

Setup → **Boundary Conditions** → **outlet (pressure-outlet)** **Edit...**



a. Enter `0.7369` atm for **Gauge Pressure**.

b. Retain **Intensity and Viscosity Ratio** from the **Specification Method** drop-down list in the **Turbulence** group box.

c. Enter `1.5`% for **Backflow Turbulent Intensity**.

d. Retain the setting of `10` for **Backflow Turbulent Viscosity Ratio**.

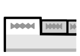
If substantial backflow occurs at the outlet, you may need to adjust the backflow values to levels close to the actual exit conditions.

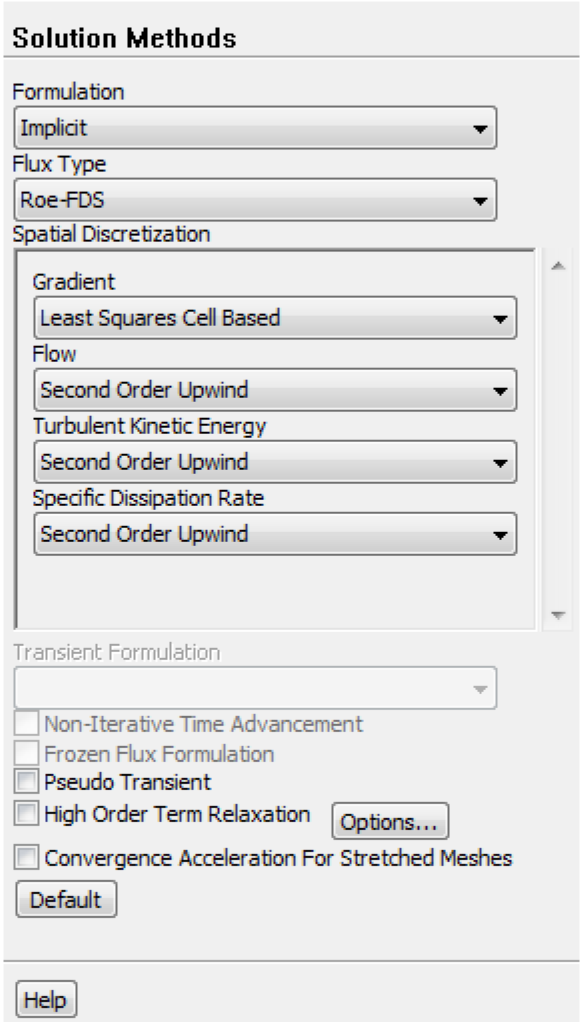
e. Click **OK** to close the **Pressure Outlet** dialog box.

6.4.8. Solution: Steady Flow

In this step, you will generate a steady-state flow solution that will be used as an initial condition for the time-dependent solution.

1. Define the solution parameters.

 **Solving** → **Solution** → **Methods...**

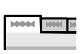


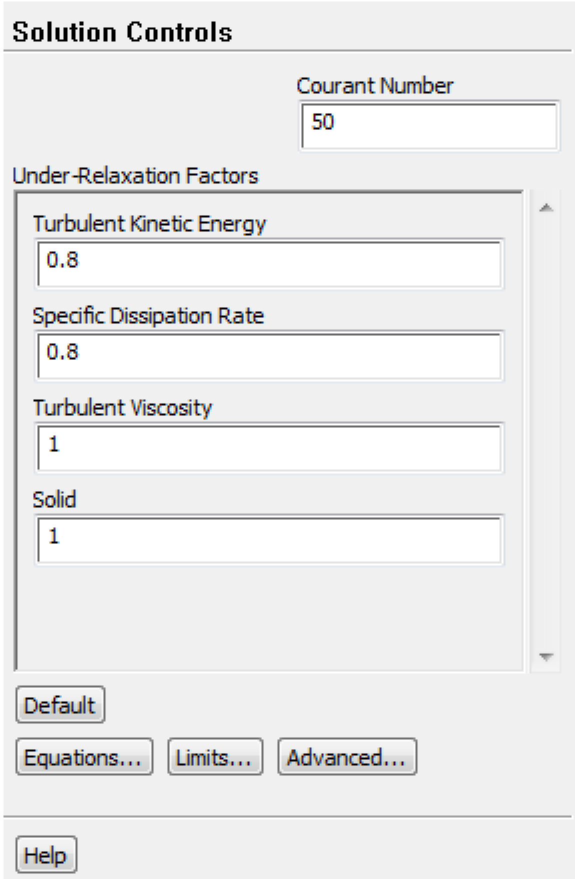
a. Retain the default selection of **Least Squares Cell Based** from the **Gradient** drop-down list in the **Spatial Discretization** group box.

b. Select **Second Order Upwind** from the **Turbulent Kinetic Energy** and **Specific Dissipation Rate** drop-down lists.

   Second-order discretization provides optimum accuracy.

2. Modify the Courant Number.

 **Solving → Controls → Controls...**



a. Enter `50` for the **Courant Number**.

Note

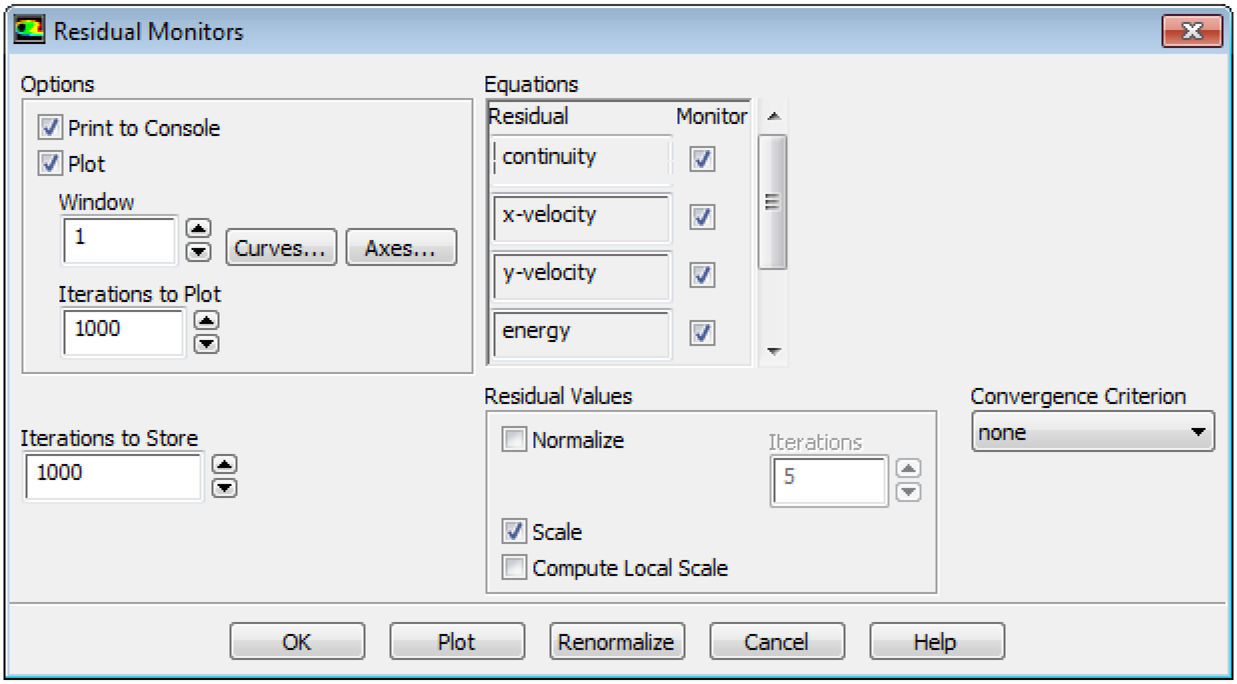
The default Courant number for the density-based implicit formulation is 5. For relatively simple problems, setting the Courant number to 10, 20, 100, or even higher value may be suitable and produce fast and stable convergence. However, if you encounter convergence difficulties at the startup of the simulation of a properly set up problem, then you should consider setting the Courant number to its default value of 5. As the solution progresses, you can start to gradually increase the Courant number until the final convergence is reached.

---

b. Retain the default values for the **Under-Relaxation Factors**.

3. Enable the plotting of residuals.

Solving → **Reports** → **Residuals...**



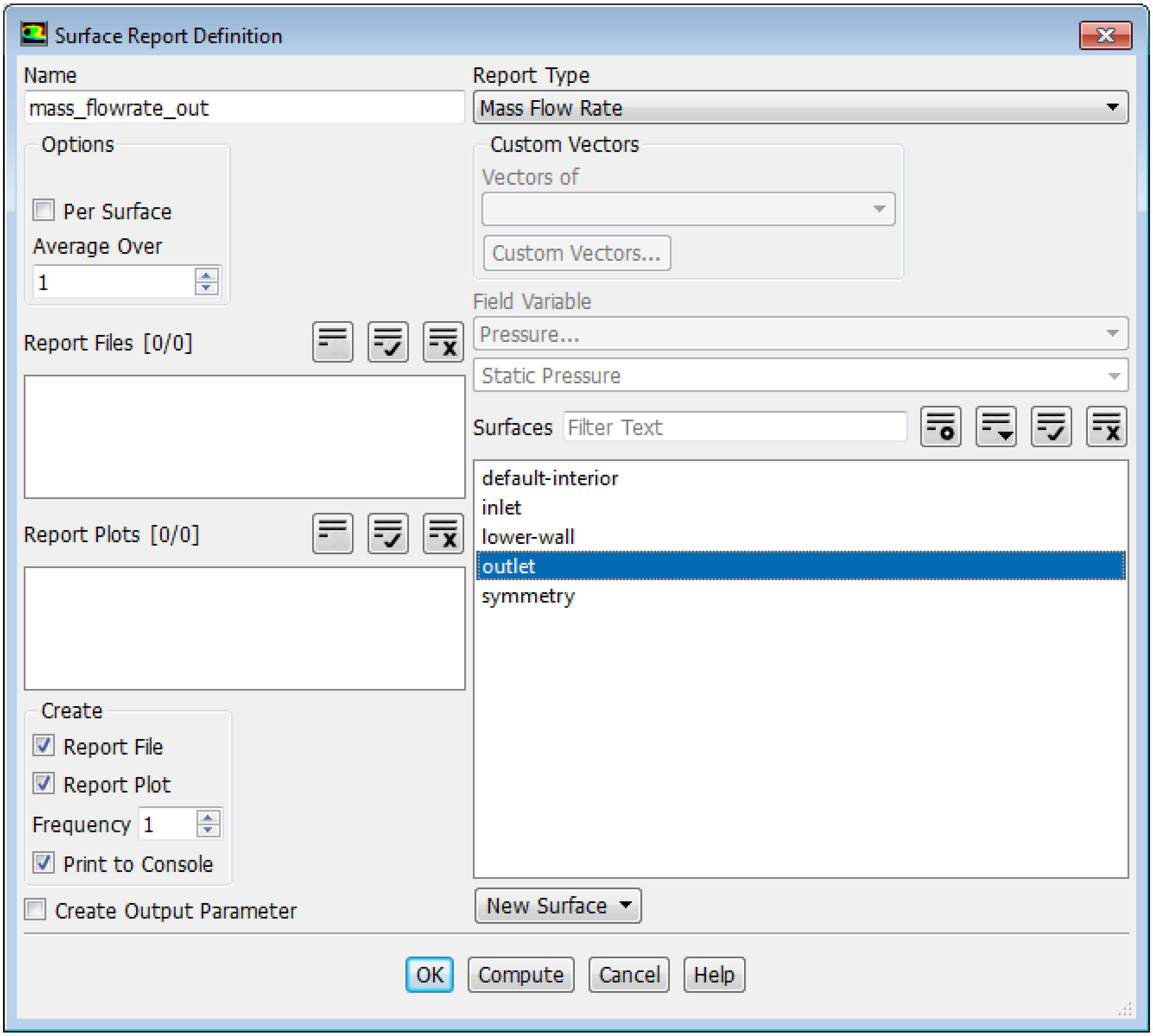
a. Ensure that **Plot** is enabled in the **Options** group box.

b. Select **none** from the **Convergence Criterion** drop-down list.

c. Click **OK** to close the **Residual Monitors** dialog box.

4. Create the surface report definition for mass flow rate at the flow exit.

Solving → **Reports** → **Definitions** → **New** → **Surface Report** → **Mass Flow Rate...**



a. Enter **`mass_flowrate_out`** for **Name**.

b. Select **outlet** in the **Surfaces** selection list.

c. In the **Create** group box, enable **Report File**, **Report Plot** and **Print to Console**.

Note

When **Report File** is enabled in the **Surface Report Definition** dialog box, the mass flow rate history will be written to a file. If you do not enable this option, the history information will be lost when you exit ANSYS Fluent.

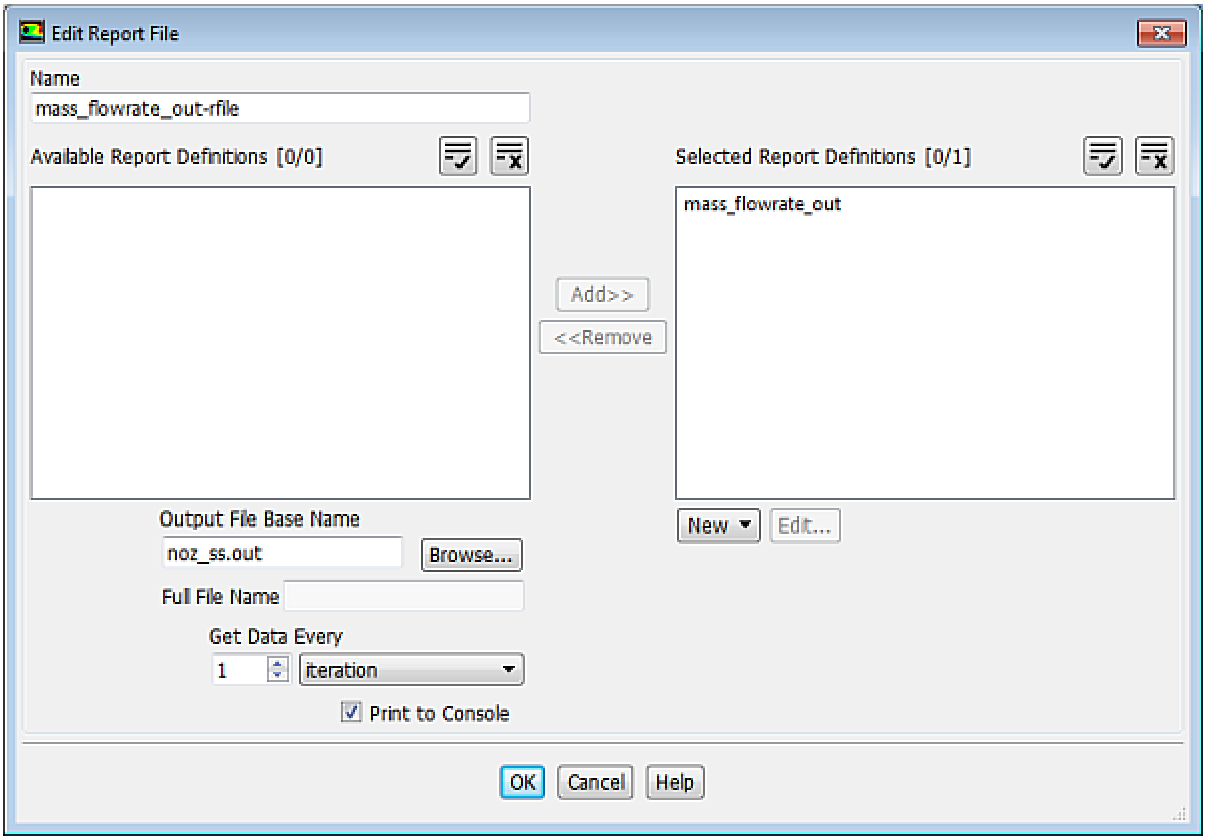
---

d. Click **OK** to close the **Surface Report Definition** dialog box.

mass_flowrate_out-rplot and **mass_flowrate_out-rfile** are automatically generated by Fluent and appear in the tree (under **Solution/Monitors/Report Plots** and **Solution/Monitors/Report Files**, respectively).

e. Modify the output file name.

Solution → **Monitors** → **Report Files** → **mass_flowrate_out-rfile** **Edit...**



i. Enter **`noz_ss.out`** for **Output File Base Name**.

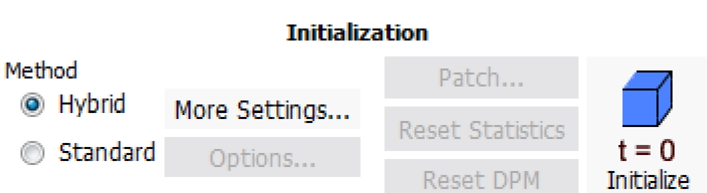
ii. Click **OK** to close the **Edit Report File** dialog box.

5. Save the case file (`noz_ss.cas.gz`).

File → **Write** → **Case...**

6. Initialize the solution.

Solving → **Initialization**



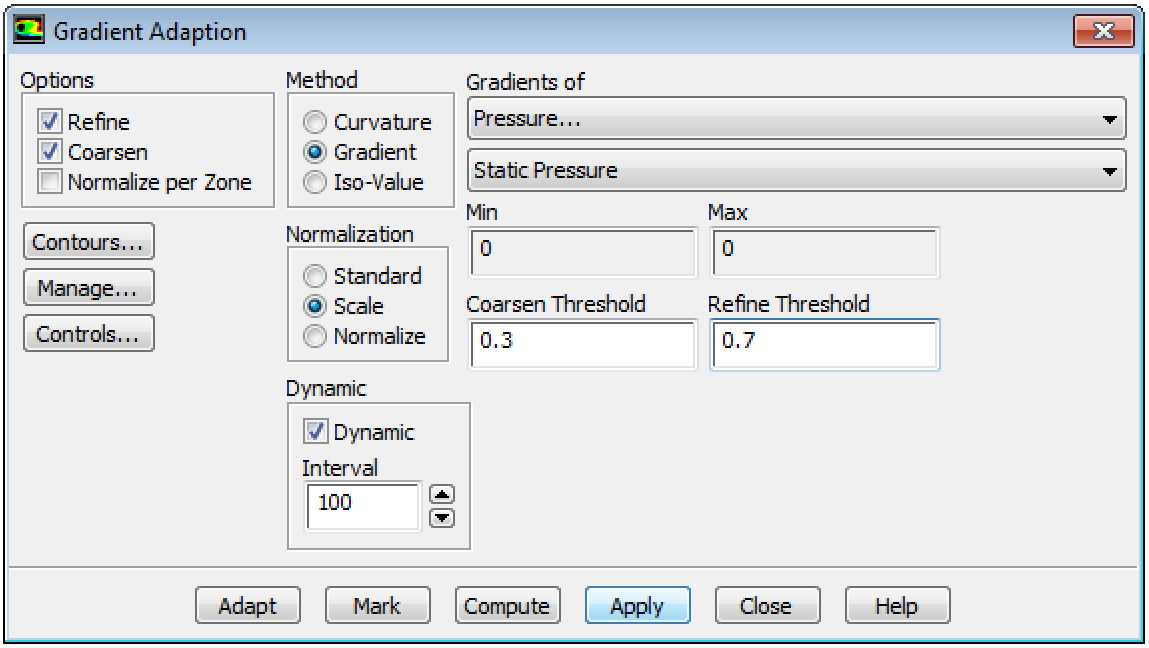
a. Keep the **Method** at the default of **Hybrid**.

b. Click **Initialize**.

7. Set up gradient adaption for dynamic mesh refinement.

You will enable dynamic adaption so that the solver periodically refines the mesh in the vicinity of the shocks as the iterations progress. The shocks are identified by their large pressure gradients.

Setting Up Domain → **Adapt** → **Mark/Adapt Cells** → **Gradient...**



a. Select **Gradient** from the **Method** group box.

The mesh adaption criterion can either be the gradient or the curvature (second gradient). Because strong shocks occur inside the nozzle, the gradient is used as the adaption criterion.

b. Select **Scale** from the **Normalization** group box.

Mesh adaption can be controlled by the raw (or standard) value of the gradient, the scaled value (by its average in the domain), or the normalized value (by its maximum in the domain). For dynamic mesh adaption, it is recommended that you use either the scaled or normalized value because the raw values will probably change strongly during the computation, which would necessitate a readjustment of the coarsen and refine thresholds. In this case, the scaled gradient is used.

c. Enable **Dynamic** in the **Dynamic** group box.

d. Enter `100` for the **Interval**.

For steady-state flows, it is sufficient to only seldomly adapt the mesh—in this case an interval of 100 iterations is chosen. For time-dependent flows, a considerably smaller interval must be used.

e. Retain the default selection of **Pressure...** and **Static Pressure** from the **Gradients of** drop-down lists.

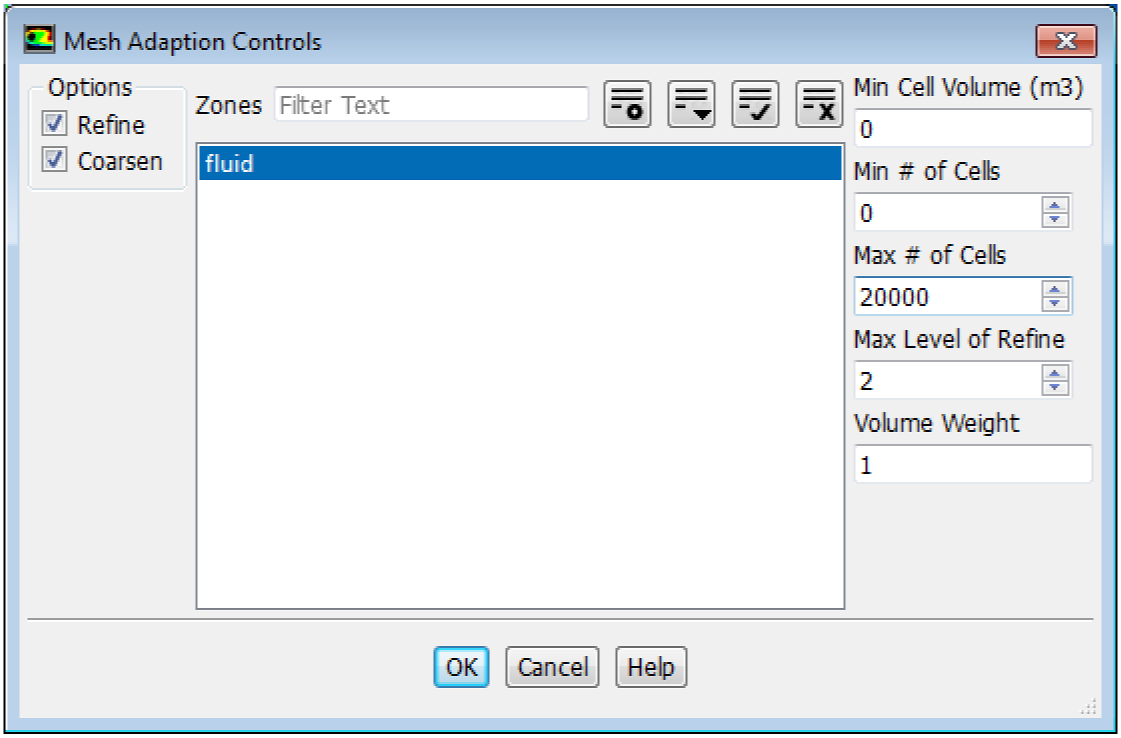
f. Enter `0.3` for **Coarsen Threshold**.

g. Enter `0.7` for **Refine Threshold**.

As the refined regions of the mesh get larger, the coarsen and refine thresholds should get smaller. A coarsen threshold of 0.3 and a refine threshold of 0.7 result in a “medium” to “strong” mesh refinement in combination with the scaled gradient.

h. Click **Apply** to store the information.

i. Click the **Controls...** button to open the **Mesh Adaption Controls** dialog box.



i. Retain the default selection of **fluid** in the **Zones** selection list.

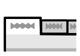
ii. Enter `20000` for **Max # of Cells**.

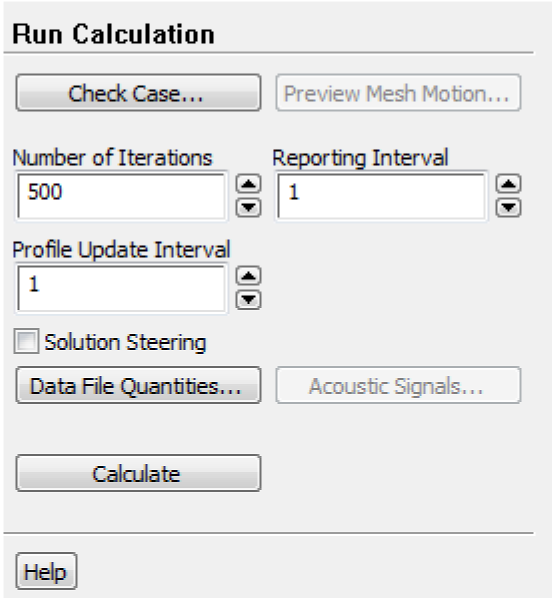
To restrict the mesh adaption, the maximum number of cells can be limited. If this limit is violated during the adaption, the coarsen and refine thresholds are adjusted to respect the maximum number of cells. Additional restrictions can be placed on the minimum cell volume, minimum number of cells, and maximum level of refinement.

iii. Click **OK** to save your settings and close the **Mesh Adaption Controls** dialog box.

j. Click **Close** to close the **Gradient Adaption** dialog box.

8. Start the calculation by requesting `500` iterations.

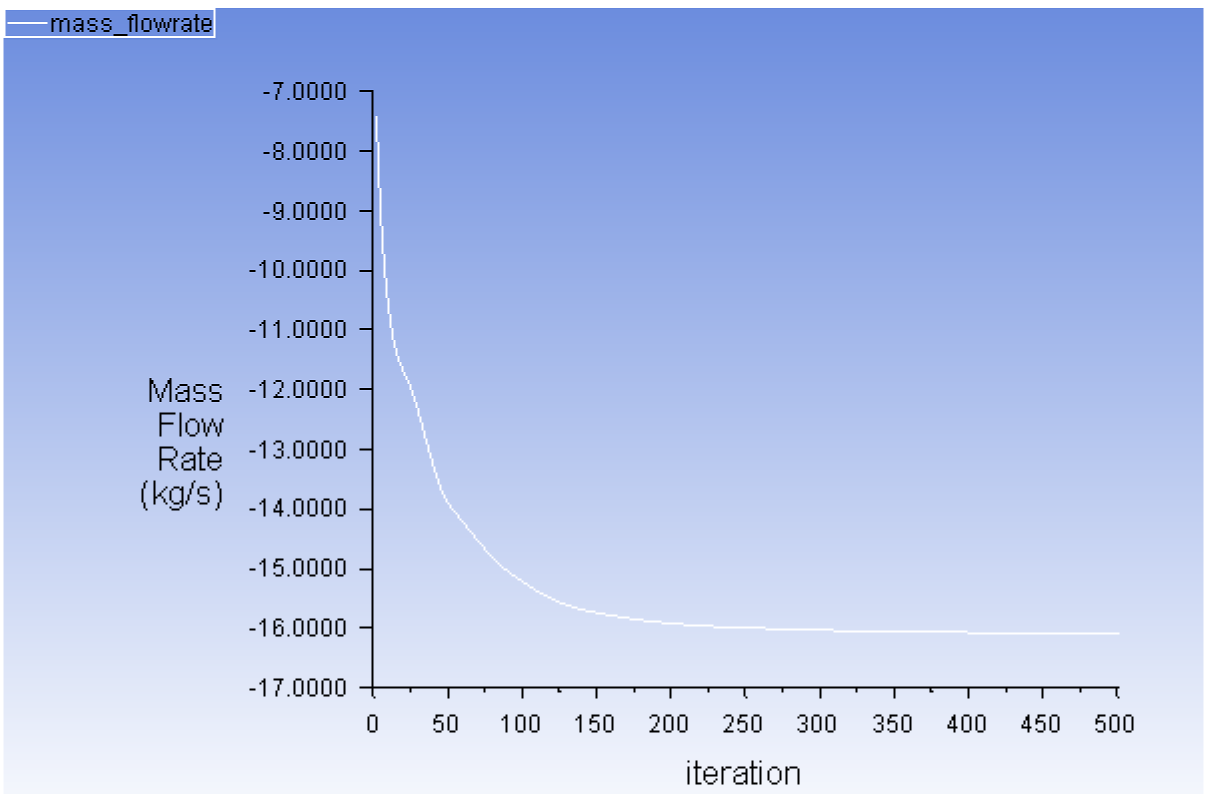
 **Solving → Run Calculation → Advanced...**



a. Enter **500** for **Number of Iterations**.

b. Click **Calculate** to start the steady flow simulation.

Figure 6.3: Mass Flow Rate History



9. Save the case and data files (`noz_ss.cas.gz` and `noz_ss.dat.gz`).

File → **Write** → **Case & Data...**

Note

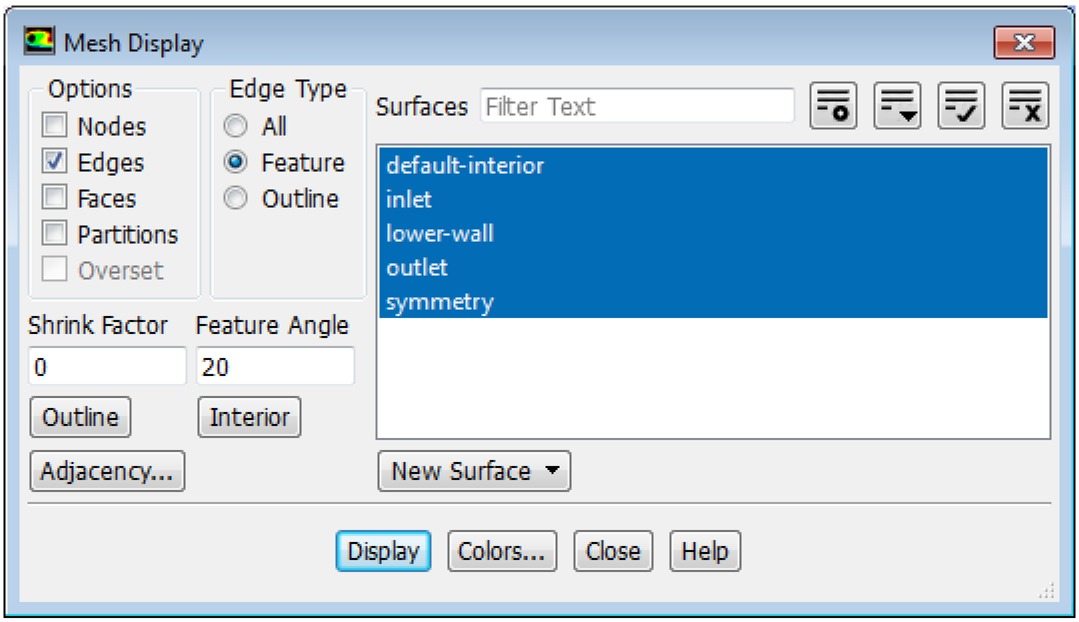
When you write the case and data files at the same time, it does not matter whether you specify the file name with a `.cas` or `.dat` extension, as both will be saved.

---

10. Click **OK** in the **Question** dialog box to overwrite the existing file.

11. Review a mesh that resulted from the dynamic adaption performed during the computation.

Results → **Graphics** → **Mesh** **Edit...**



a. Ensure that only the **Edges** option is enabled in the **Options** group box.

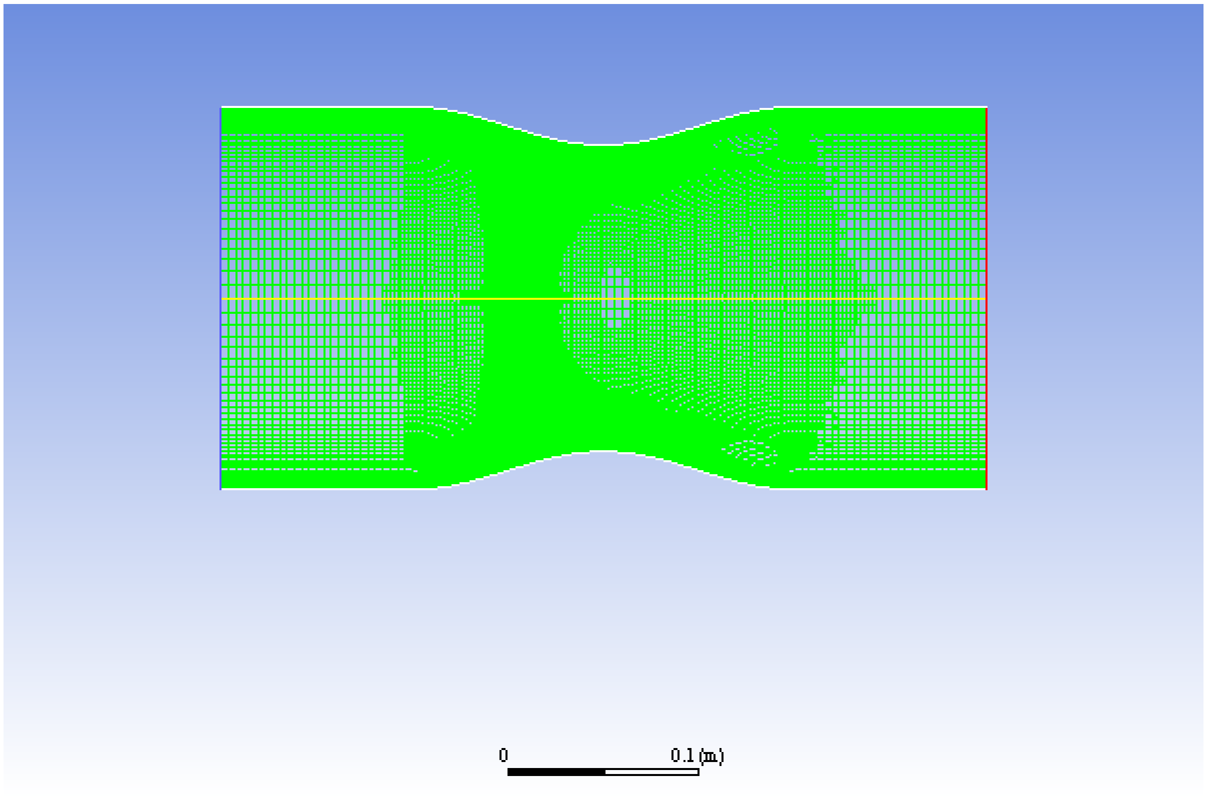
b. Select **Feature** from the **Edge Type** list.

c. Ensure that all of the items are selected from the **Surfaces** selection list.

d. Click **Display** and close the **Mesh Display** dialog box.

The mesh after adaption is displayed in the graphics window (Figure 6.4: 2D Nozzle Mesh after Adaption (p. 287))

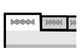
Figure 6.4: 2D Nozzle Mesh after Adaption

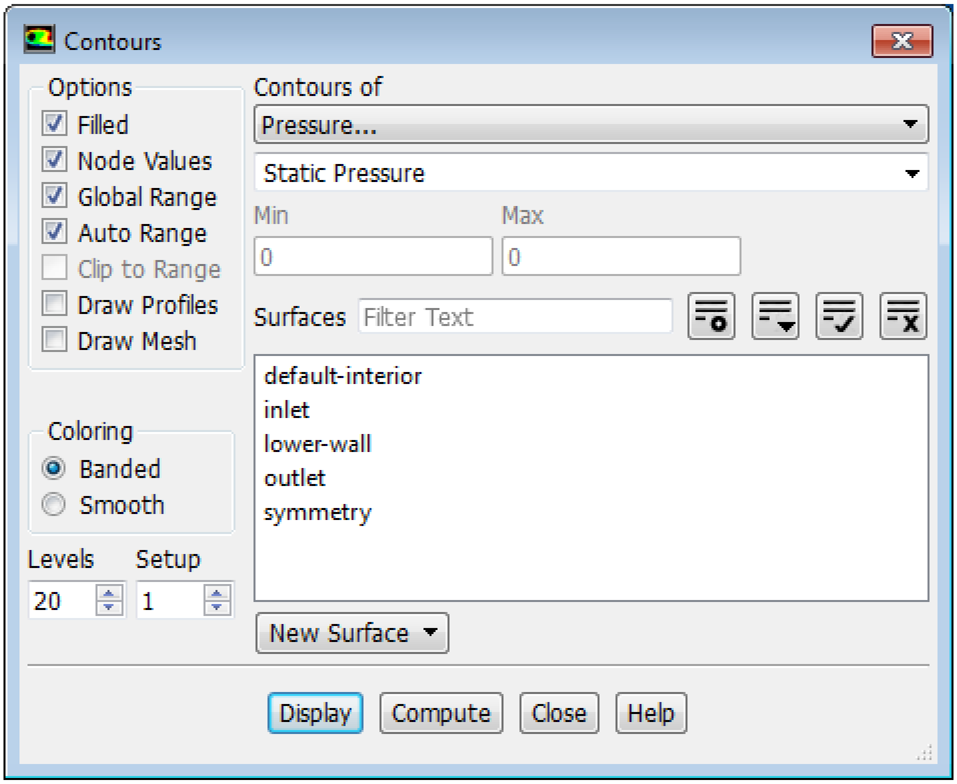


e. Zoom in using the middle mouse button to view aspects of your mesh.

   Notice that the cells in the regions of high pressure gradients have been refined.

12. Display the steady flow contours of static pressure (Figure 6.5: Contours of Static Pressure (Steady Flow) (p. 288)).

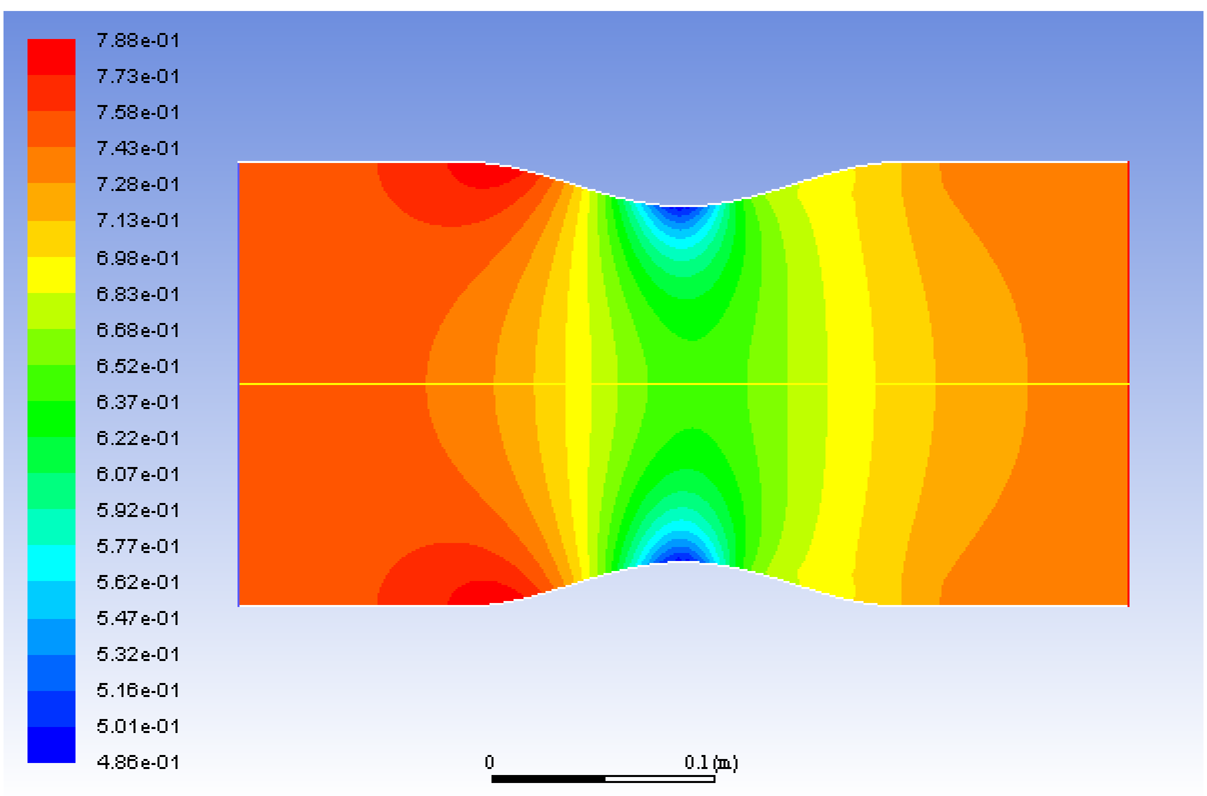
 **Postprocessing → Graphics → Contours → Edit...**



a. Enable **Filled** in the **Options** group box.

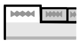
b. Click **Display** and close the **Contours** dialog box.

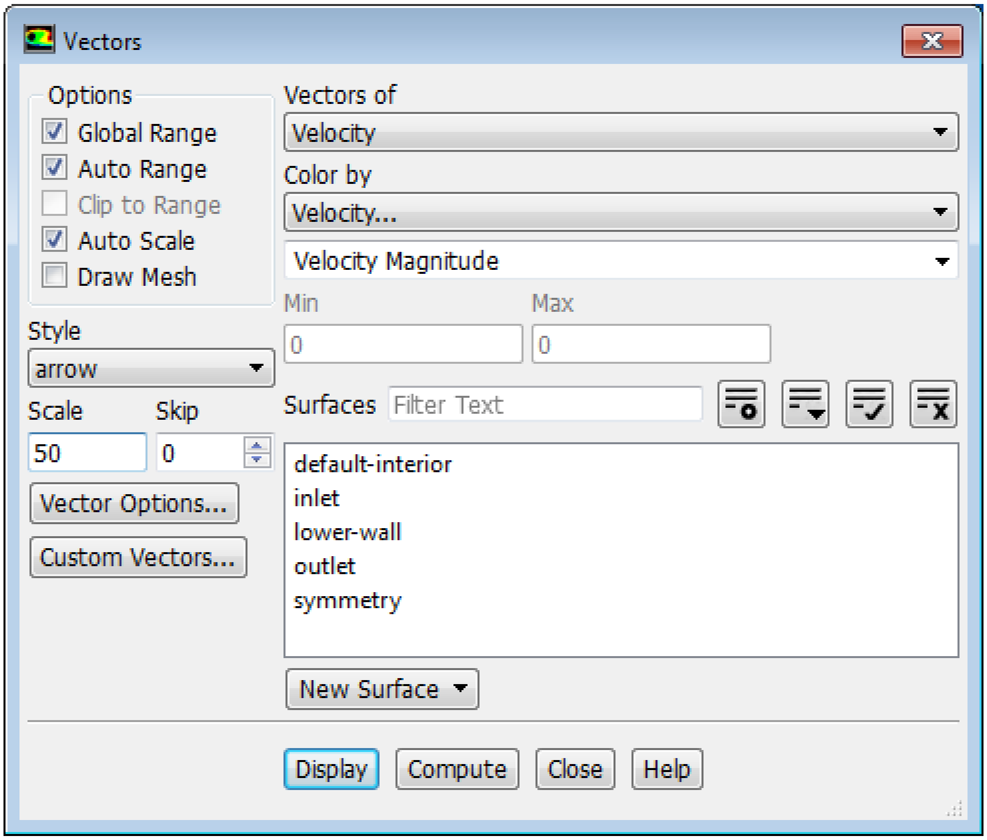
Figure 6.5: Contours of Static Pressure (Steady Flow)



The steady flow prediction in [Figure 6.5: Contours of Static Pressure (Steady Flow) (p. 288)] shows the expected pressure distribution, with low pressure near the nozzle throat.

13. Display the steady-flow velocity vectors ([Figure 6.6: Velocity Vectors Showing Recirculation (Steady Flow) (p. 290)]).

 **Postprocessing** → **Graphics** → **Vectors** → **Edit...**



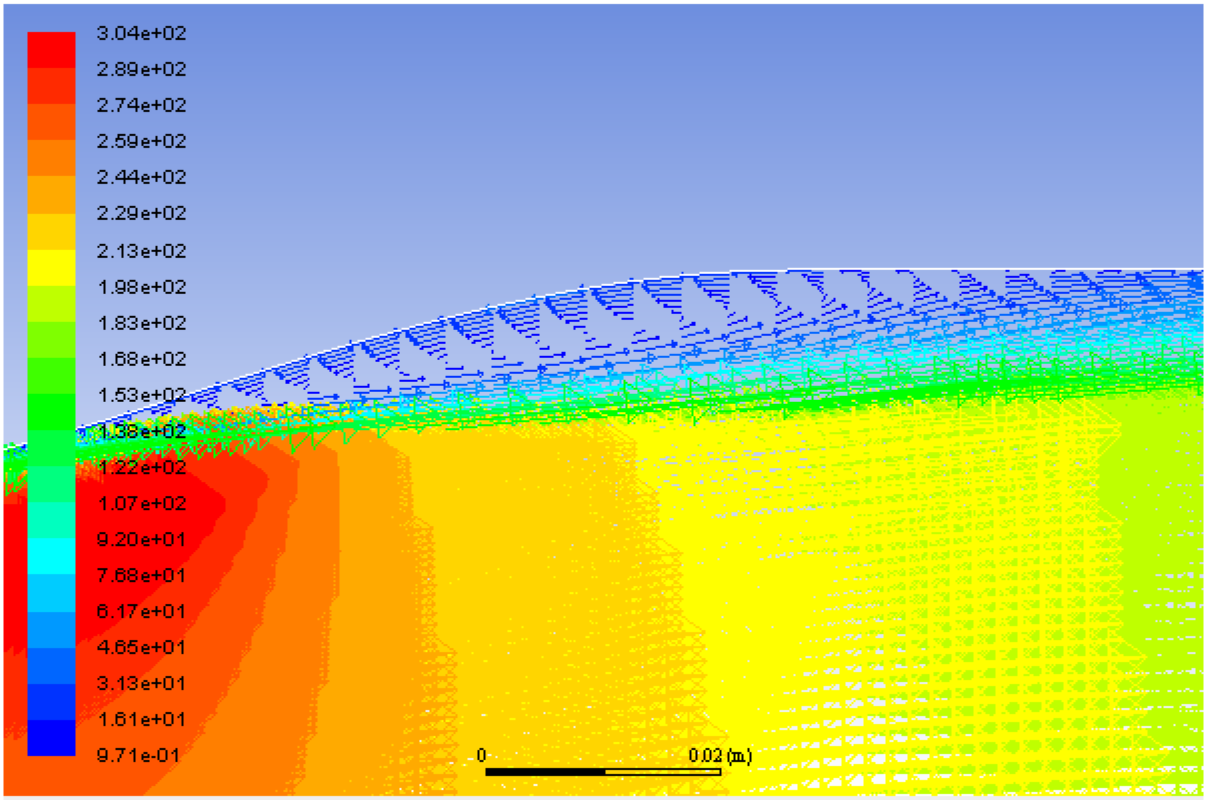
a. Enter `50` under **Scale**.

b. Click **Display** and close the **Vectors** dialog box.

The steady flow prediction shows the expected form, with a peak velocity of approximately 300 m/s through the nozzle.

You can zoom in on the wall in the expansion region of the nozzle to view the recirculation of the flow as shown in [Figure 6.6: Velocity Vectors Showing Recirculation (Steady Flow) (p. 290)] .

Figure 6.6: Velocity Vectors Showing Recirculation (Steady Flow)

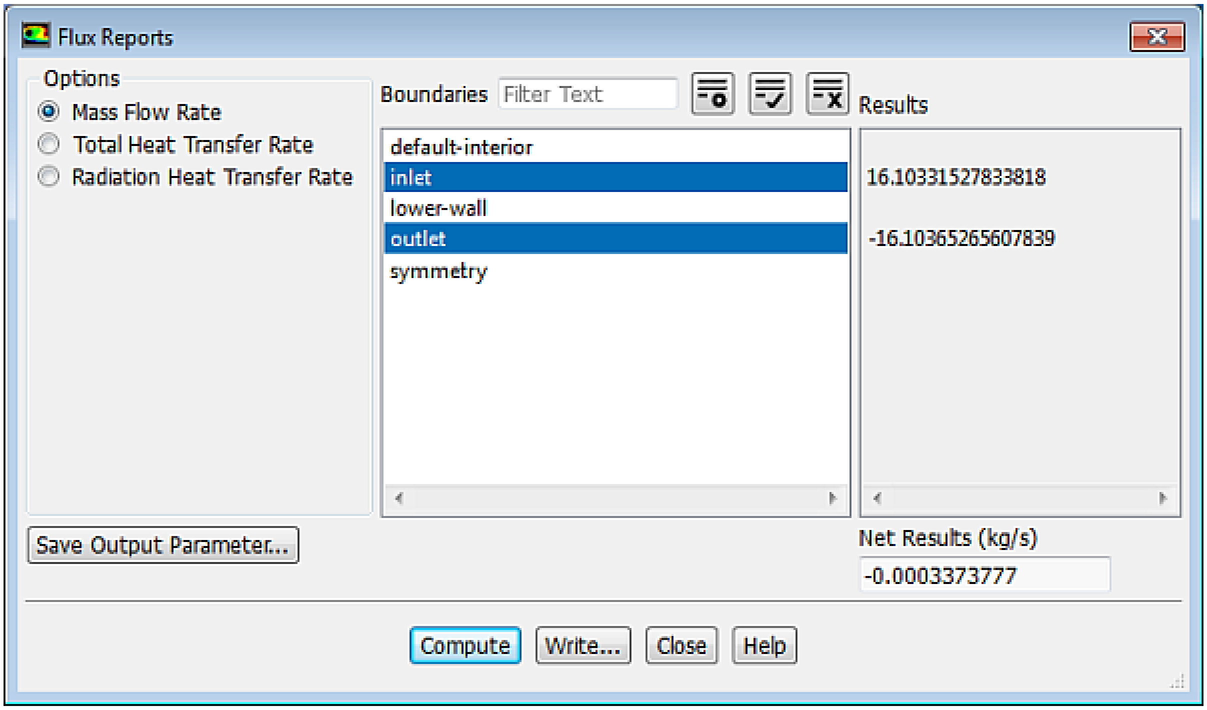


14. Check the mass flux balance.

Important

Although the mass flow rate history indicates that the solution is converged, you should also check the mass flux throughout the domain to ensure that mass is being conserved.

Postprocessing → Reports → Fluxes...



a. Retain the default selection of **Mass Flow Rate**.

b. Select **inlet** and **outlet** in the **Boundaries** selection list.

c. Click **Compute** and examine the values displayed in the dialog box.

Important

The net mass imbalance should be a small fraction (for example, 0.1%) of the total flux through the system. The imbalance is displayed in the lower right field under **Net Results**. If a significant imbalance occurs, you should decrease your residual tolerances by at least an order of magnitude and continue iterating.

---

d. Close the **Flux Reports** dialog box.

6.4.9. Enabling Time Dependence and Setting Transient Conditions

In this step you will define a transient flow by specifying a transient pressure condition for the nozzle.

1. Enable a time-dependent flow calculation.

   Setting Up Physics → **Solver** → **Transient**

2. Read the user-defined function (`pexit.c`), in preparation for defining the transient condition for the nozzle exit.

   The pressure at the outlet is defined as a wave-shaped profile, and is described by the following equation:

$$p_{exit}(t) = 0.12\sin(\omega t) + \overline{p}_{exit} \tag{6.1}$$

   where

ω = circular frequency of transient pressure (rad/s)

$\overline{p}_{exit}$ = mean exit pressure (atm)

In this case, ω=2200 *rad/s, and* $\overline{p}_{exit}$= 0.7369 *atm.*

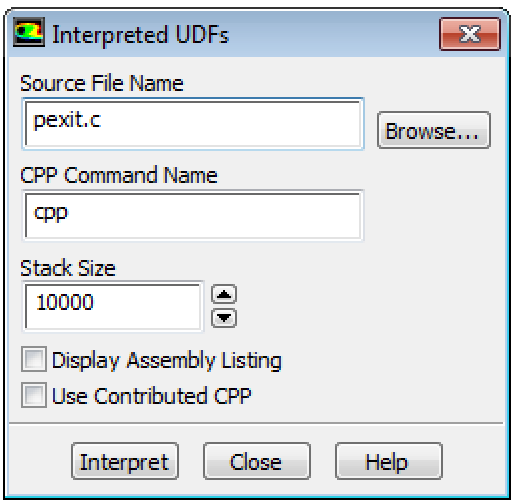
A user-defined function (`pexit.c` *) has been written to define the equation (Equation 6.1 (p. 291)) required for the pressure profile.*

Note

To input the value of Equation 6.1 (p. 291) in the correct units, the function `pexit.c` has to be written in SI units.

More details about user-defined functions can be found in the Fluent Customization Manual.

User Defined → **User Defined** → **Functions** → **Interpreted...**



a. Enter `pexit.c` for **Source File Name**.

   If the UDF source file is not in your working directory, then you must enter the entire directory path for **Source File Name** *instead of just entering the file name.*

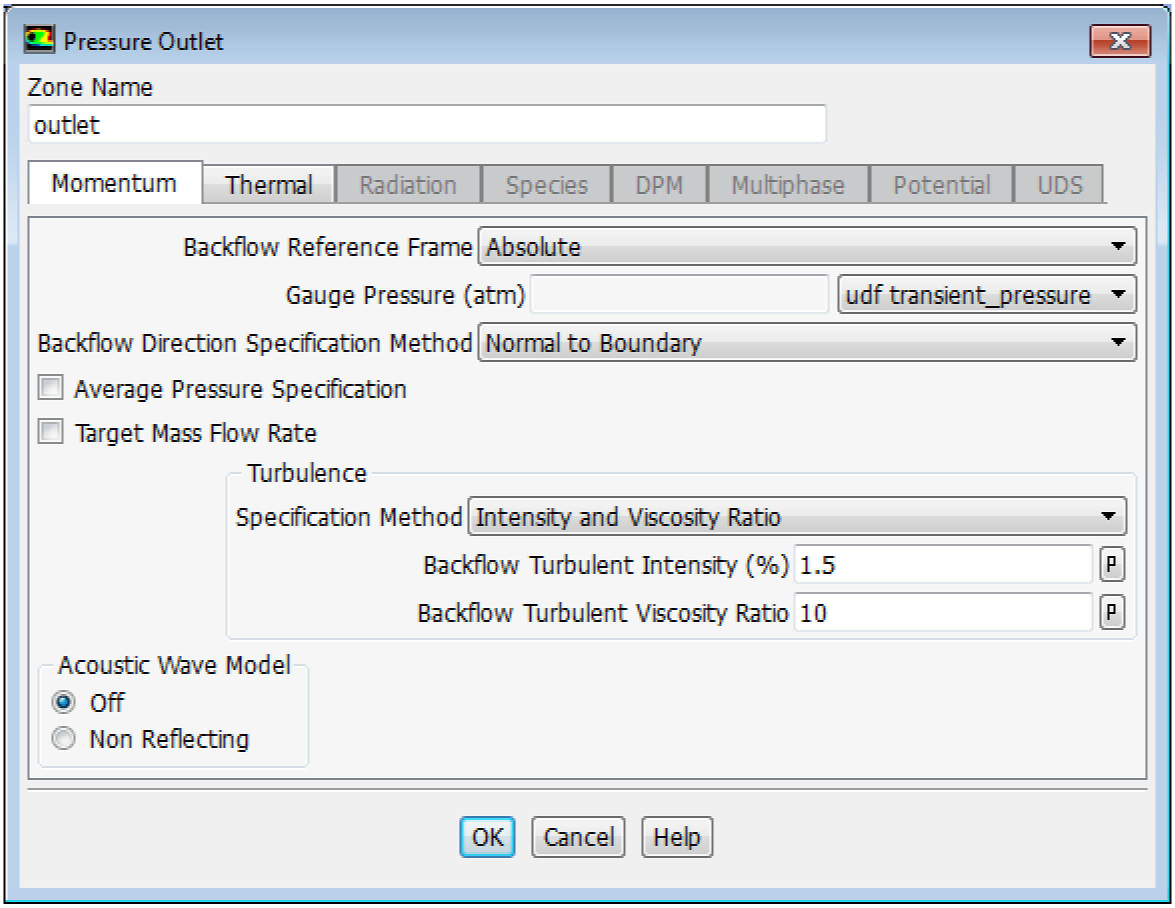
b. Click **Interpret**.

   The user-defined function has already been defined, but it must be compiled within ANSYS Fluent before it can be used in the solver.

c. Close the **Interpreted UDFs** dialog box.

3. Define the transient boundary conditions at the nozzle exit (**outlet**).

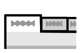
Setup → **Boundary Conditions** → **outlet** **Edit...**



a. Select **udf transient_pressure** (the user-defined function) from the **Gauge Pressure** drop-down list.

b. Click **OK** to close the **Pressure Outlet** dialog box.

4. Update the gradient adaption parameters for the transient case.

 **Setting Up Domain → Adapt → Mark/Adapt Cells → Gradient...**

a. Enter `10` for **Interval** in the **Dynamic** group box.

For the transient case, the mesh adaption will be done every 10 time steps.

b. Enter `0.3` for **Coarsen Threshold**.

c. Enter `0.7` for **Refine Threshold**.

The refine and coarsen thresholds have been changed during the steady-state computation to meet the limit of 20000 cells. Therefore, you must reset these parameters to their original values.

d. Click **Apply** to store the values.

e. Click **Controls...** to open the **Mesh Adaption Controls** dialog box.

i. Enter `8000` for **Min # of Cells**.

ii. Enter `30000` for **Max # of Cells**.

You must increase the maximum number of cells to try to avoid readjustment of the coarsen and refine thresholds. Additionally, you must limit the minimum number of cells to 8000, because you should not have a coarse mesh during the computation (the current mesh has approximately 20000 cells).

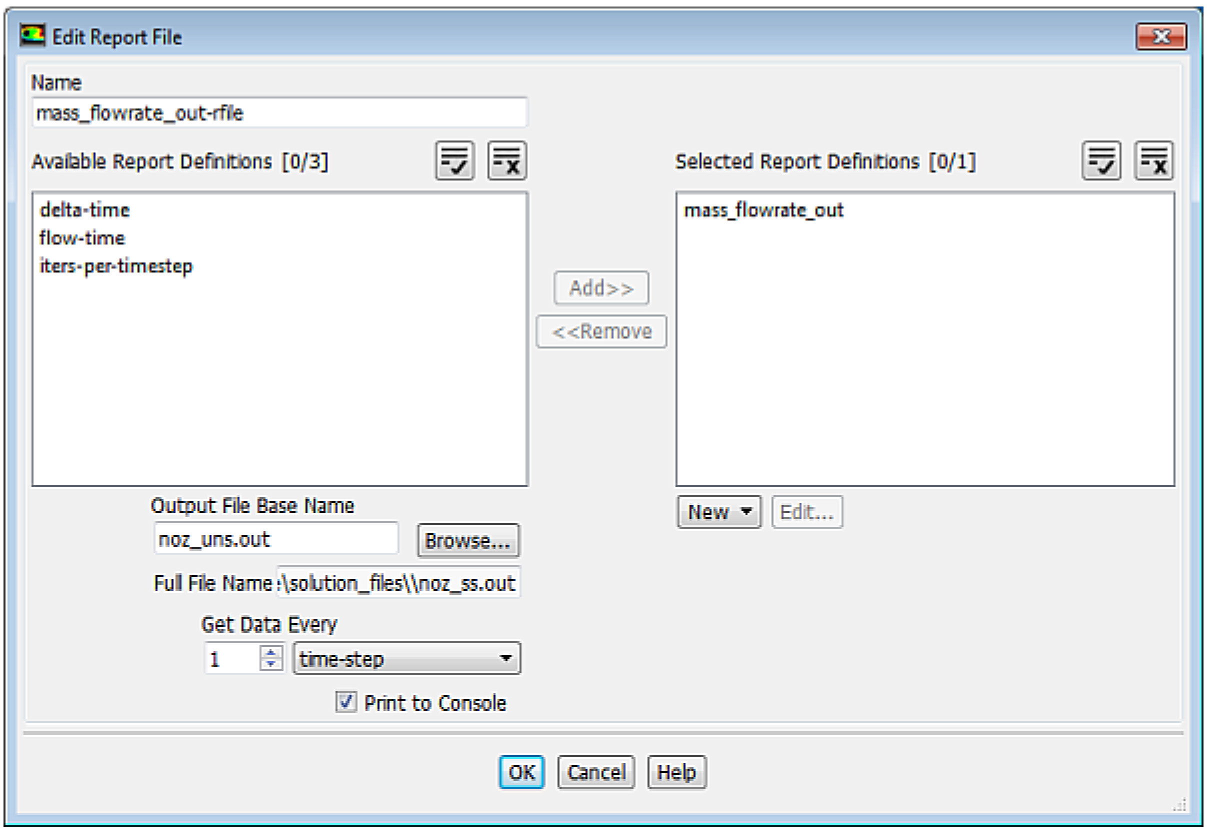
iii. Click **OK** to close the **Mesh Adaption Controls** dialog box.

f. Close the **Gradient Adaption** dialog box.

6.4.10. Specifying Solution Parameters for Transient Flow and Solving

1. Modify the **mass_flowrate_out-rfile** report file definition.

Solution → **Monitors** → **Report Files** → **mass_flowrate_out-rfile** **Edit...**



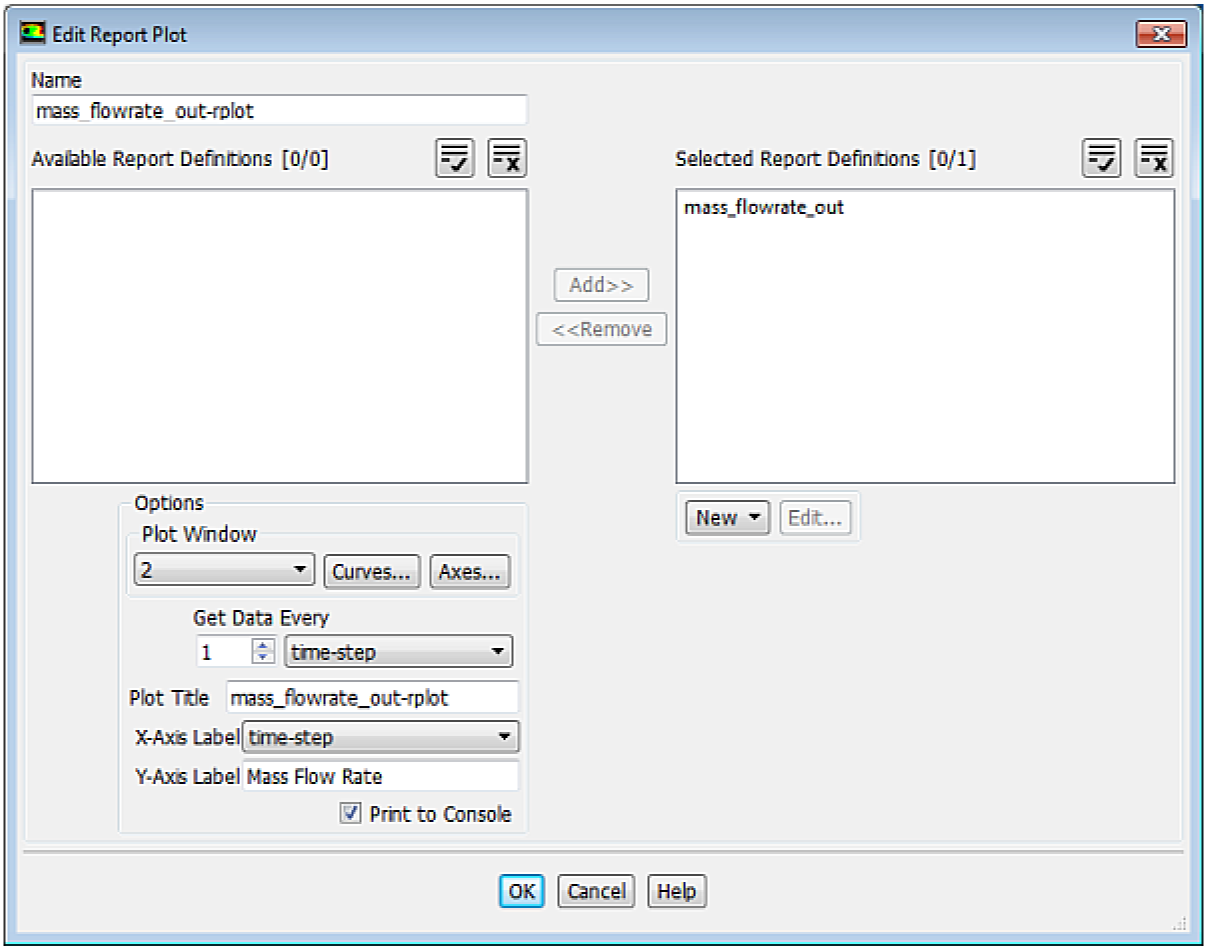
a. Enter `noz_uns.out` for **Output File Base Name**.

b. Select **time-step** from the **Get Data Every** drop-down list.

c. Click **OK** to close the **Edit Report File** dialog box.

2. Modify the **mass_flowrate_out-rplot** report plot definition.

Solution → **Monitors** → **Report Plots** → **mass_flowrate_out-rplot** **Edit...**



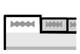
a. For **Get Data Every**, retain the value of **1** and select **time-step** from the drop-down list.

   Because each time step requires 10 iterations, a smoother plot will be generated by plotting at every time step.

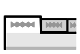
b. Select **time-step** from the **X-Axis Label** drop-down list.

c. Click **OK** to close the **Edit Report File** dialog box.

3. Save the transient solution case file (`noz_uns.cas.gz`).

    **File → Write → Case...**

4. Modify the plotting of residuals.

    **Solving → Reports → Residuals...**

   a. Ensure that **Plot** is enabled in the **Options** group box.

   b. Ensure **none** is selected from the **Convergence Criterion** drop-down list.

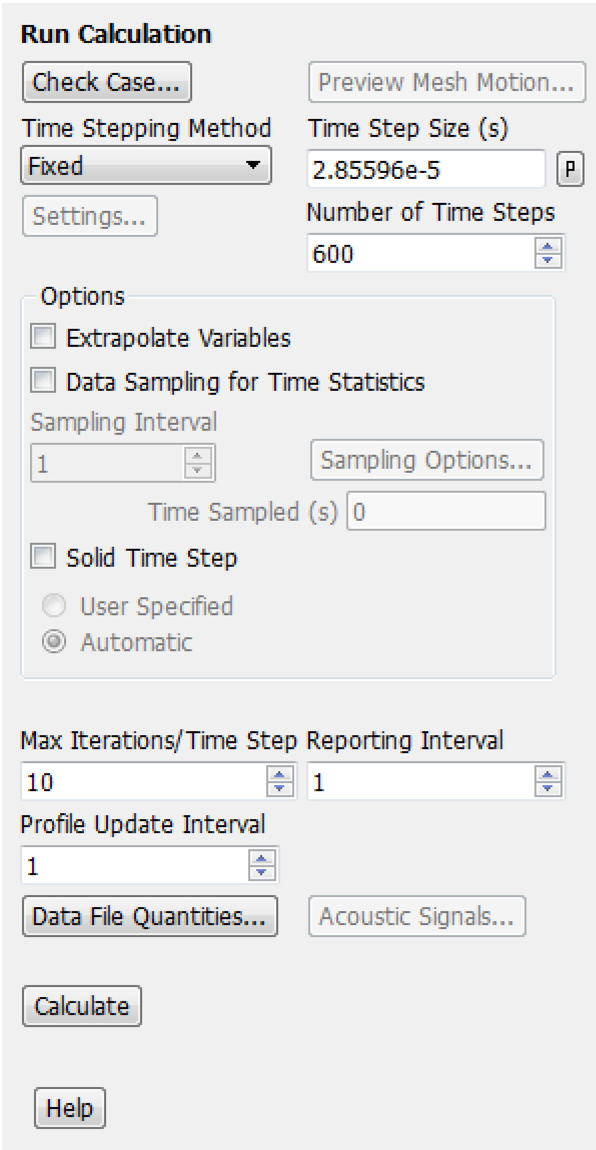
   c. Set the **Iterations to Plot** to `100`.

   d. Click **OK** to close the **Residual Monitors** dialog box.

5. Define the time step parameters.

The selection of the time step is critical for accurate time-dependent flow predictions. Using a time step of 2.85596 x 10^{-5} seconds, 100 time steps are required for one pressure cycle. The pressure cycle begins and ends with the initial pressure at the nozzle exit.

Solving → **Run Calculation** → **Advanced**



a. Enter `2.85596e-5` s for **Time Step Size**.

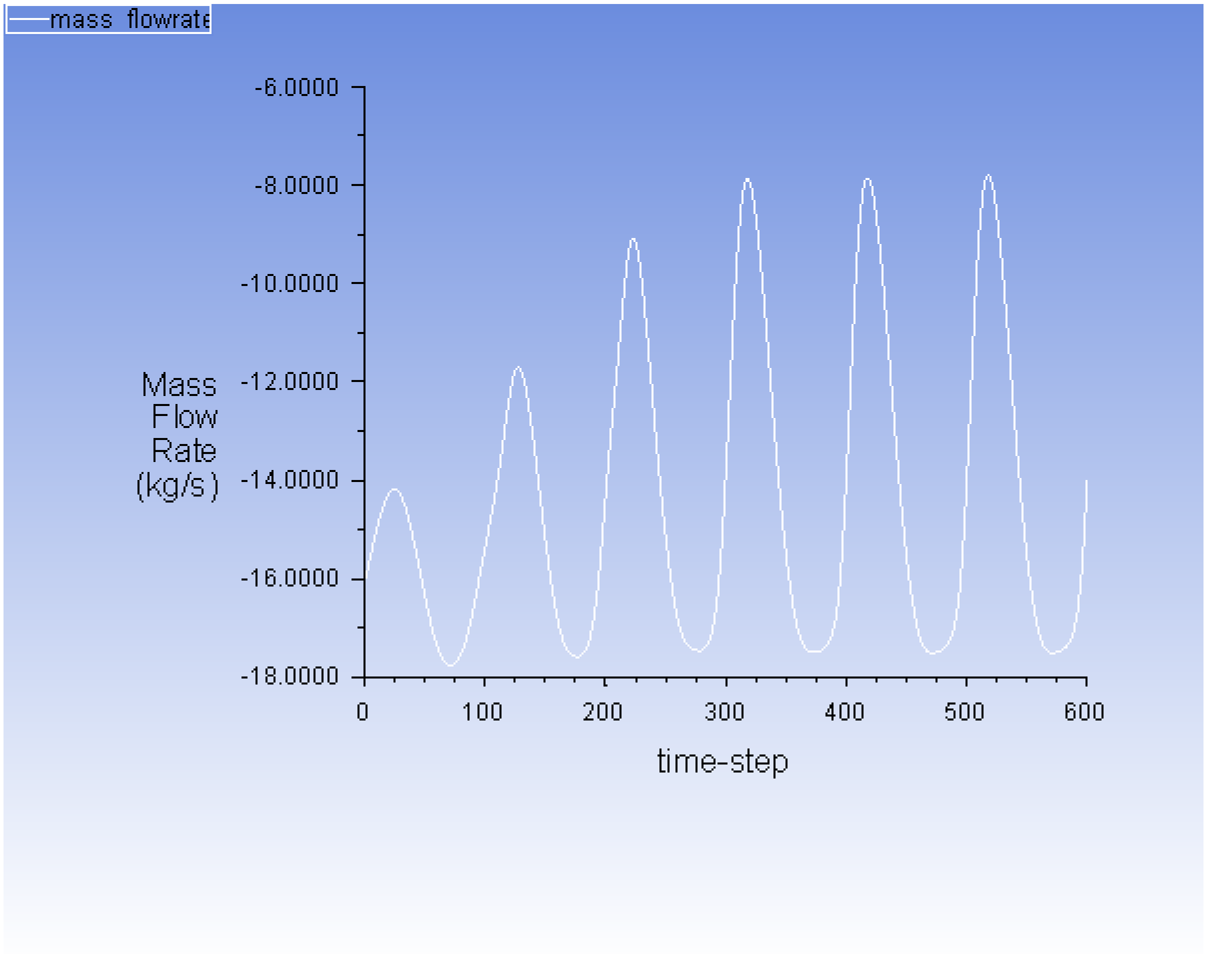
b. Enter `600` for **Number of Time Steps**.

c. Enter `10` for **Max Iterations/Time Step**.

d. Click **Calculate** to start the transient simulation.

By requesting 600 time steps, you are asking ANSYS Fluent to compute six pressure cycles. The mass flow rate history is shown in *Figure 6.7: Mass Flow Rate History (Transient Flow) (p. 297)*.

Figure 6.7: Mass Flow Rate History (Transient Flow)



6. Optionally, you can review the effect of dynamic mesh adaption performed during transient flow computation as you did in steady-state flow case.

7. Save the transient case and data files (`noz_uns.cas.gz` and `noz_uns.dat.gz`).

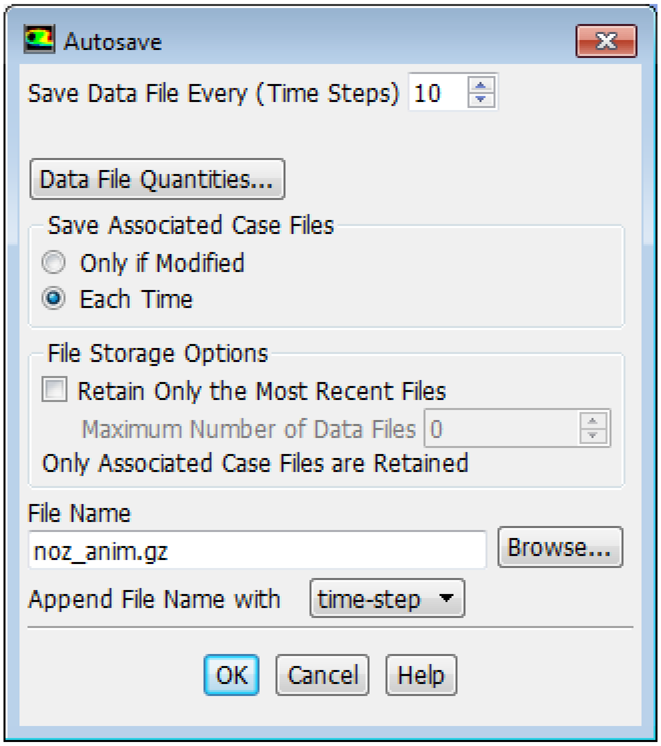
   File → **Write** → **Case & Data...**

6.4.11. Saving and Postprocessing Time-Dependent Data Sets

At this point, the solution has reached a time-periodic state. To study how the flow changes within a single pressure cycle, you will now continue the solution for 100 more time steps. You will use ANSYS Fluent's solution animation feature to save contour plots of pressure and Mach number at each time step, and the autosave feature to save case and data files every 10 time steps. After the calculation is complete, you will use the solution animation playback feature to view the animated pressure and Mach number plots over time.

1. Request the saving of case and data files every 10 time steps.

   Solving → **Activities** → **Autosave...**



a. Enter `10` for **Save Data File Every**.

b. Select **Each Time** for **Save Associated Case Files**.

c. Retain the default selection of **time-step** from the **Append File Name with** drop-down list.

d. Enter `noz_anim.gz` for **File Name**.

When ANSYS Fluent saves a file, it will append the time step value to the file name prefix (`noz_anim` *). The standard extensions (* `.cas` *and* `.dat` *) will also be appended. By adding the optional extension* `.gz` *to the end of the file name, you instruct ANSYS Fluent to save the case and data files in compressed format. This will yield file names of the form* `noz_anim-1-00640.cas.gz` *and* `noz_anim-1-00640.dat.gz` *, where* `00640` *is the time step number.*

e. Click **OK** to close the **Autosave** dialog box.

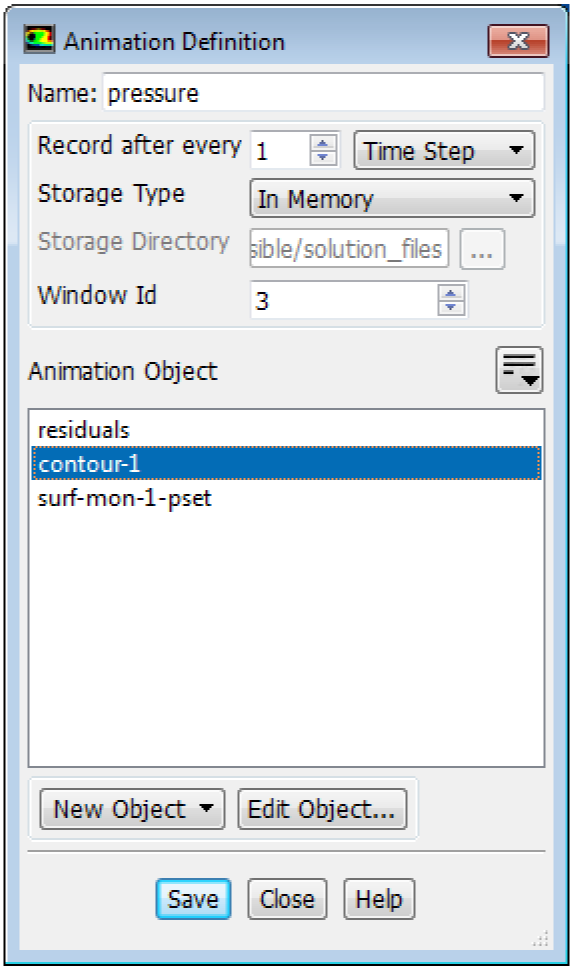
Tip

If you have constraints on disk space, you can restrict the number of files saved by ANSYS Fluent by enabling the **Retain Only the Most Recent Files** option and setting the **Maximum Number of Data Files** to a nonzero number.

---

2. Create an animation definition for the nozzle pressure contour plot.

Solving → **Activities** → **Create** → **Solution Animations...**



a. Enter `pressure` for the **Name**.

b. Select **Time Step** for **Record after every**.

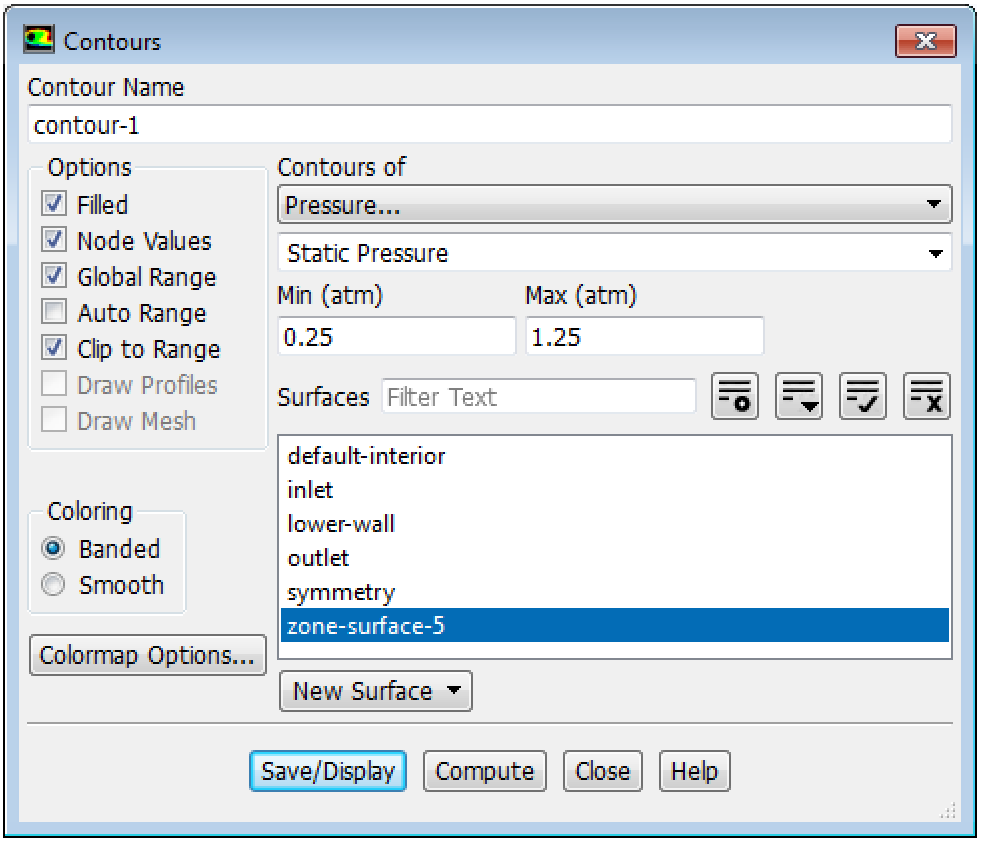
The default value of `1` *in the integer number entry box instructs ANSYS Fluent to update the animation sequence at every time step.*

c. Select **In Memory** from the **Storage Type** drop-down.

The **In Memory** *option is acceptable for a small 2D case such as this. For larger 2D or 3D cases, saving animation files with either the* **Metafile** *or* **PPM Image** *option is preferable, to avoid using too much of your machine's memory.*

d. Enter **3** for the **Window Id**.

e. Click **New Object** and select **Contours...** from the drop-down list to open the associated dialog box.



i. Select **Pressure...** and **Static Pressure** from the **Contours of** drop-down lists.

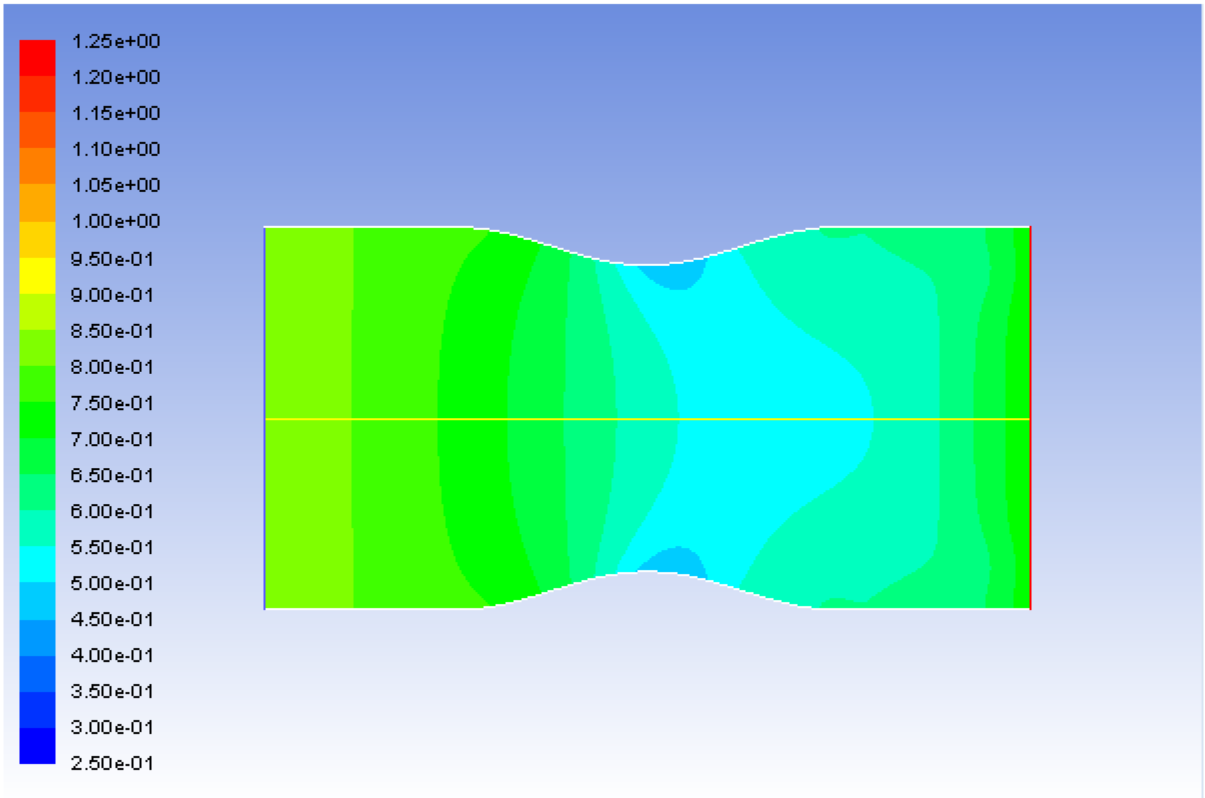
ii. Ensure that **Filled** is enabled in the **Options** group box.

iii. Disable **Auto Range**.

iv. Enter `0.25` atm for **Min** and `1.25` atm for **Max**.

v. Click **Save/Display** and close the **Contours** dialog box.

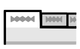
vi. **Figure 6.8: Pressure Contours at t=0.017136 s**

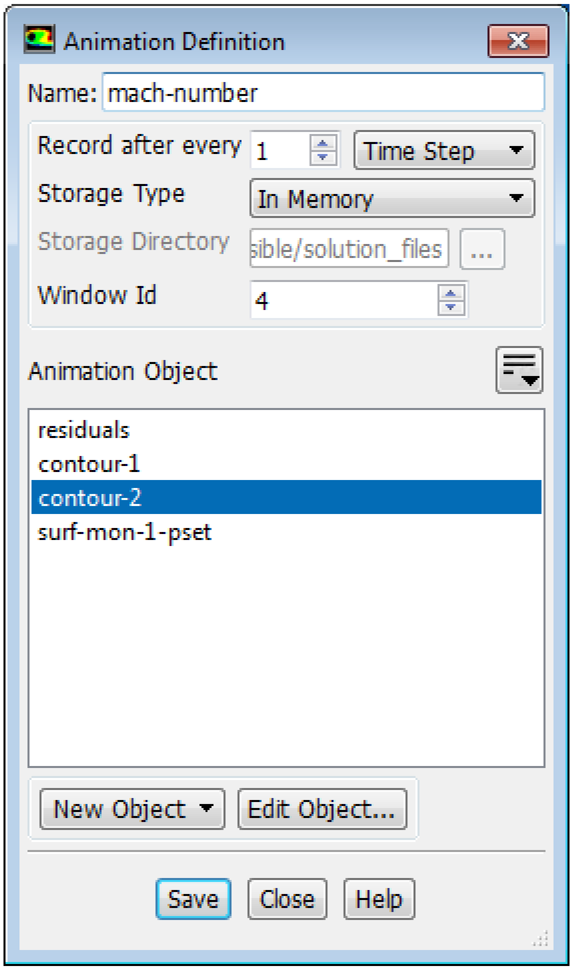


f. Ensure **contour-1** is selected in the **Animation Object** group box.

g. Click **Save** and close the **Animation Definition** dialog box.

3. Create an animation definition for the Mach number contour plot.

 **Solving** → **Activities** → **Create** → **Solution Animations...**



a. Enter **mach-number** for the **Name**.

b. Select **Time Step** for **Record after every**.

   The default value of `1` *in the integer number entry box instructs ANSYS Fluent to update the animation sequence at every time step.*

c. Ensure that **In Memory** is selected from the **Storage Type** drop-down.

d. Enter **4** for the **Window Id**.

e. Click **New Object** and select **Contours...** from the drop-down list to open the associated dialog box.

   i. Select **Velocity...** and **Mach Number** from the **Contours of** drop-down lists.

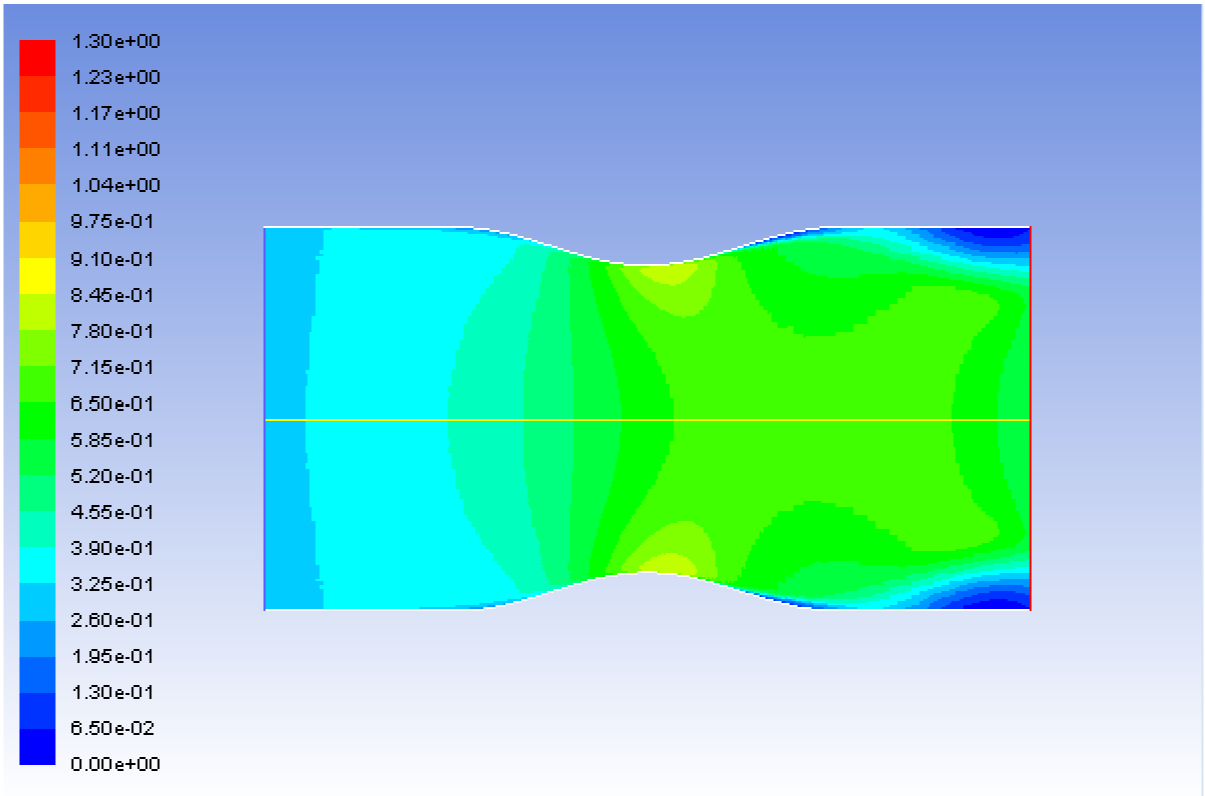
   ii. Ensure that **Filled** is enabled in the **Options** group box.

   iii. Disable **Auto Range**.

   iv. Enter `0.00` for **Min** and `1.30` for **Max**.

   v. Click **Save/Display** and close the **Contours** dialog box.

vi. **Figure 6.9: Mach Number Contours at t=0.017136 s**



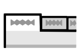
f. Ensure **contour-2** is selected in the **Animation Object** group box.

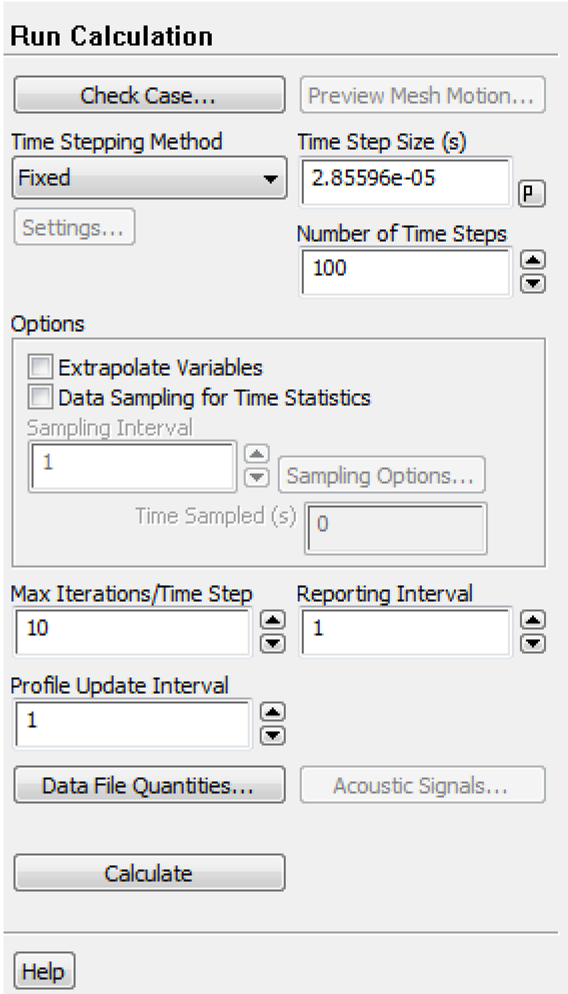
g. Click **Save** and close the **Animation Definition** dialog box.

4. Continue the calculation by requesting 100 time steps.

By requesting 100 time steps, you will march the solution through an additional 0.0028 seconds, or roughly one pressure cycle.

With the autosave and animation features active (as defined previously), the case and data files will be saved approximately every 0.00028 seconds of the solution time; animation files will be saved every 0.000028 seconds of the solution time.

 **Solving → Run Calculation → Advanced...**

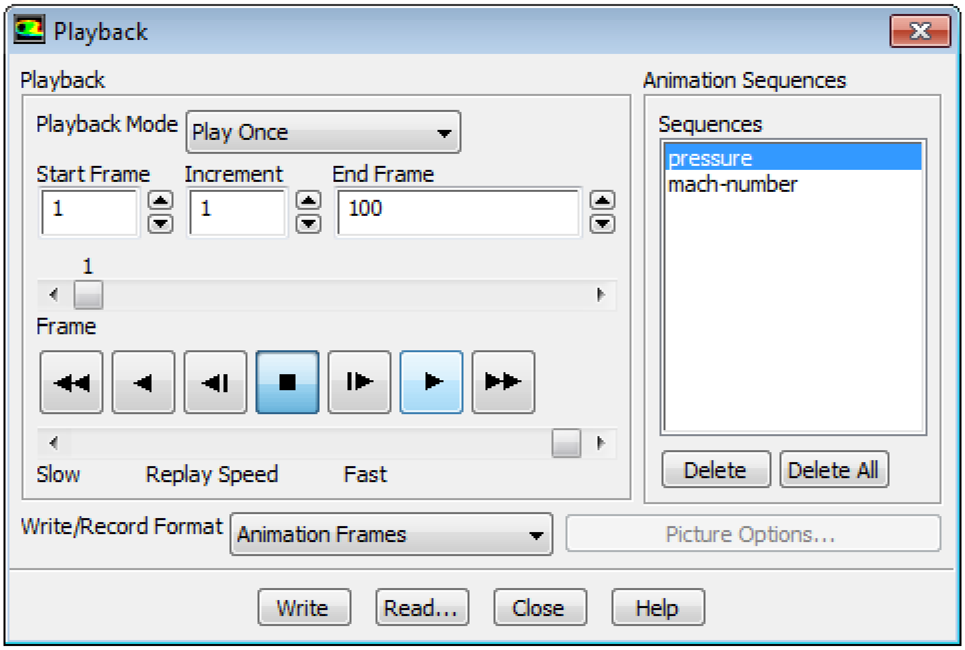


Enter `100` for **Number of Time Steps** and click **Calculate**.

When the calculation finishes, you will have ten pairs of case and data files and there will be 100 pairs of contour plots stored in memory. In the next few steps, you will play back the animation sequences and examine the results at several time steps after reading in pairs of newly saved case and data files.

5. Play the animation of the pressure contours.

Results → **Animations** → **Solution Animation Playback** **Edit...**



a. Retain the default selection of **pressure** in the **Sequences** selection list.

   Ensure that tab window 4 is open in the graphics window.

b. Click the play button (the second from the right in the group of buttons in the **Playback** group box).

c. Close the **Playback** dialog box.

Examples of pressure contours at $t=0.017993$ *s (the 630th time step) and* $t=0.019135$ *s (the 670th time step) are shown in* *Figure 6.10: Pressure Contours at t=0.017993 s(p. 306)* *and* *Figure 6.11: Pressure Contours at t=0.019135 s (p. 306)*.

Figure 6.10: Pressure Contours at t=0.017993 s

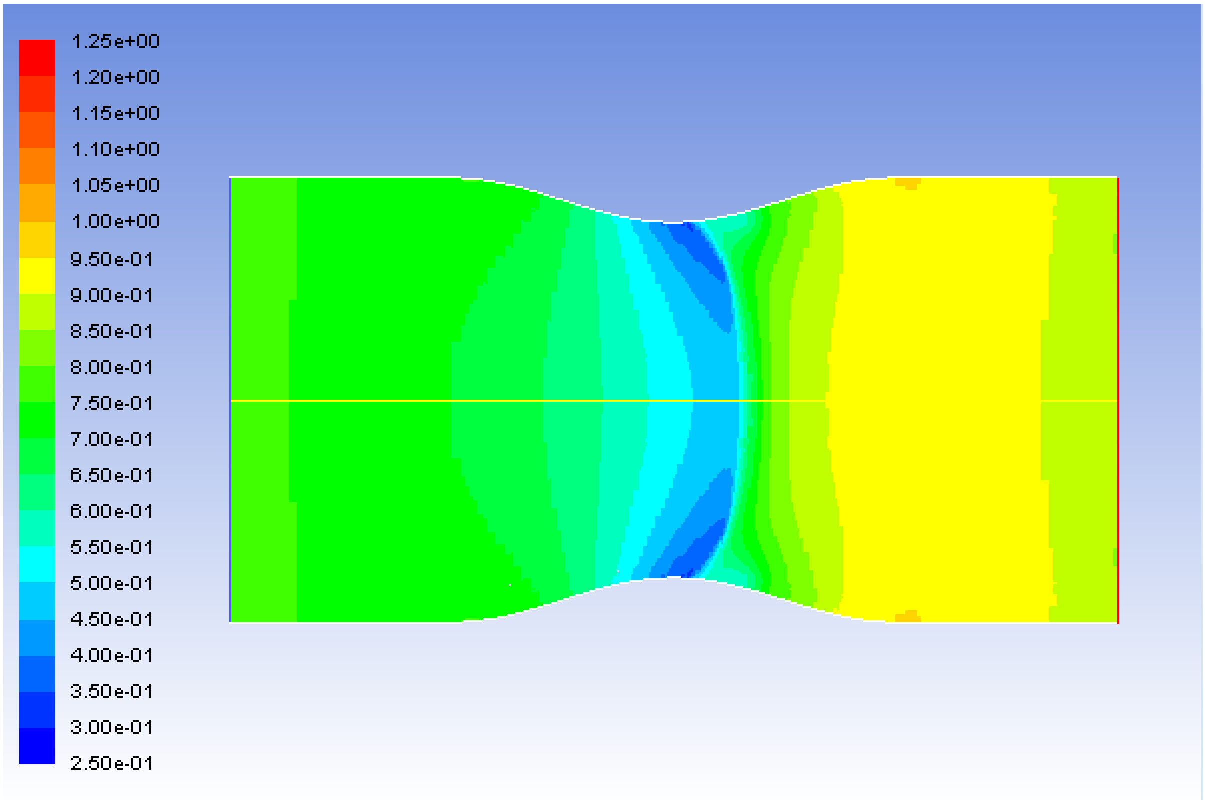
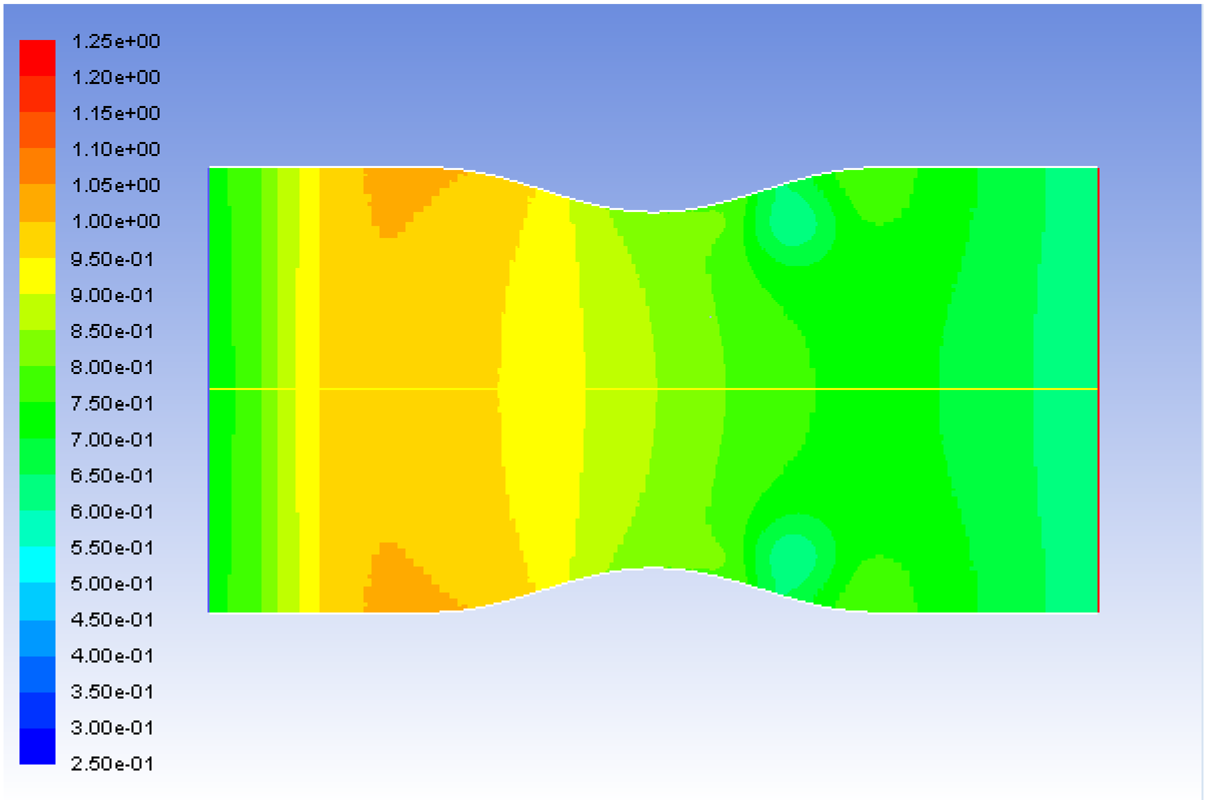


Figure 6.11: Pressure Contours at t=0.019135 s



6. In a similar manner to steps 4 and 5, select the appropriate active window and animation sequence name for the Mach number contours.

   Examples of Mach number contours at t=0.017993 s and t=0.019135 s are shown in *Figure 6.12: Mach Number Contours at t=0.017993 s (p. 307)* *and* *Figure 6.13: Mach Number Contours at t=0.019135 s(p. 308)*.

Figure 6.12: Mach Number Contours at t=0.017993 s

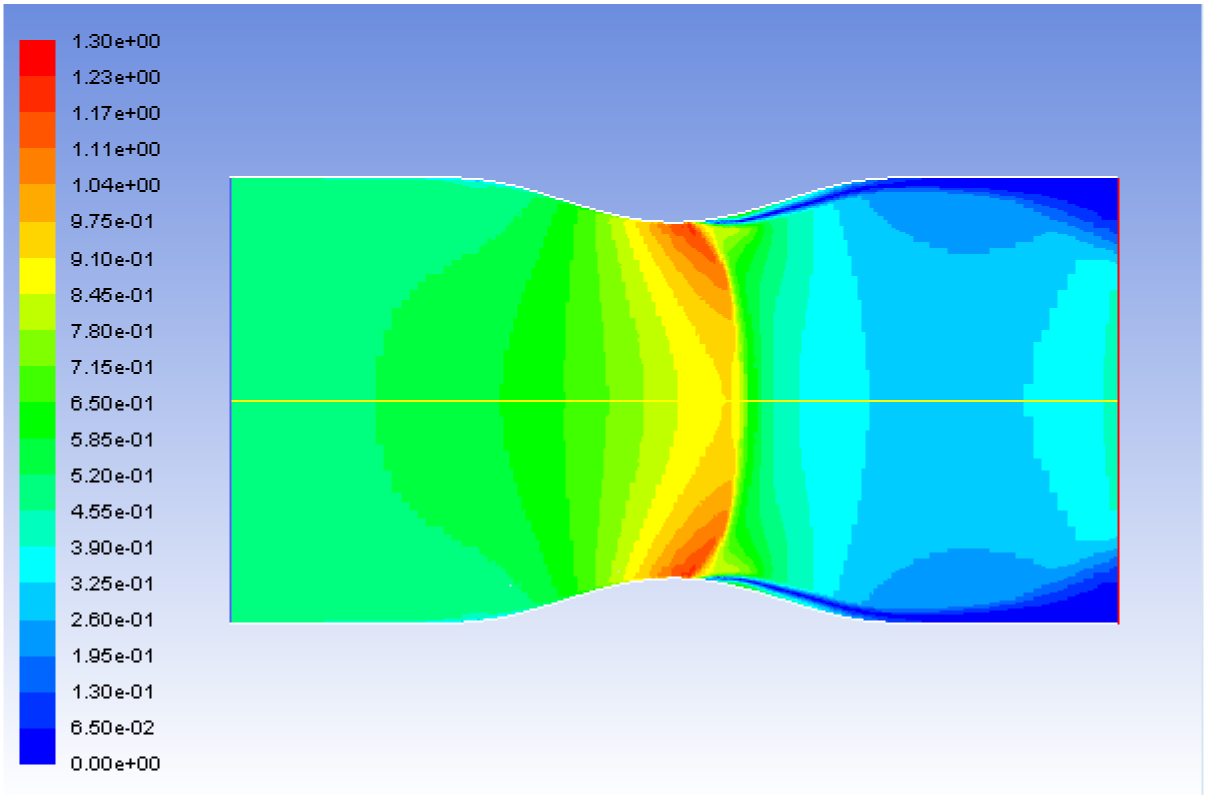
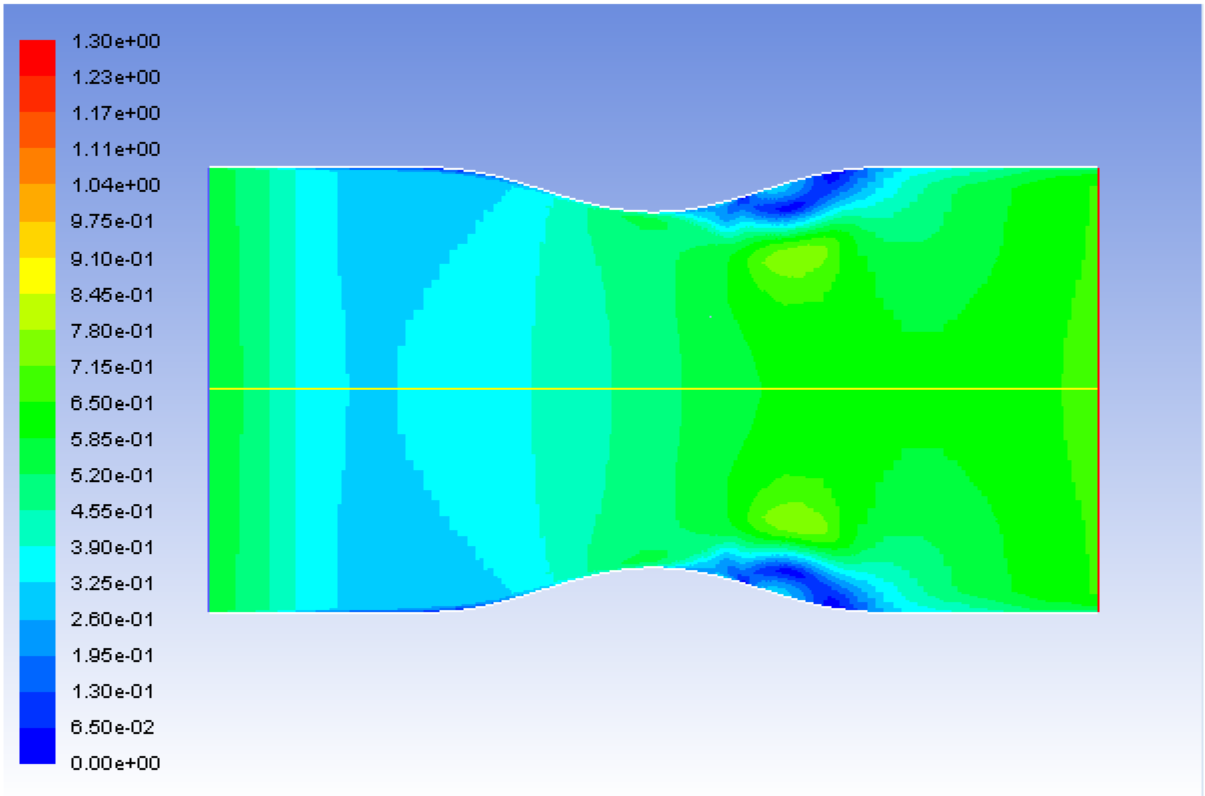


Figure 6.13: Mach Number Contours at t=0.019135 s



Tip

ANSYS Fluent gives you the option of exporting an animation as an MPEG file or as a series of files in any of the hardcopy formats available in the **Save Picture** dialog box (including TIFF and PostScript).

To save an MPEG file, select **MPEG** from the **Write/Record Format** drop-down list in the **Playback** dialog box and then click the **Write** button. The MPEG file will be saved in your working folder. You can view the MPEG movie using an MPEG player (for example, Windows Media Player or another MPEG movie player).

To save a series of TIFF, PostScript, or other hardcopy files, select **Picture Frames** in the **Write/Record Format** drop-down list in the **Playback** dialog box. Click the **Picture Options...** button to open the **Save Picture** dialog box and set the appropriate parameters for saving the hardcopy files. Click **Apply** in the **Save Picture** dialog box to save your modified settings. Click **Save...** to select a directory in which to save the files. In the **Playback** dialog box, click the **Write** button. ANSYS Fluent will replay the animation, saving each frame to a separate file in your working folder.

If you want to view the solution animation in a later ANSYS Fluent session, you can select **Animation Frames** as the **Write/Record Format** and click **Write**.

Warning

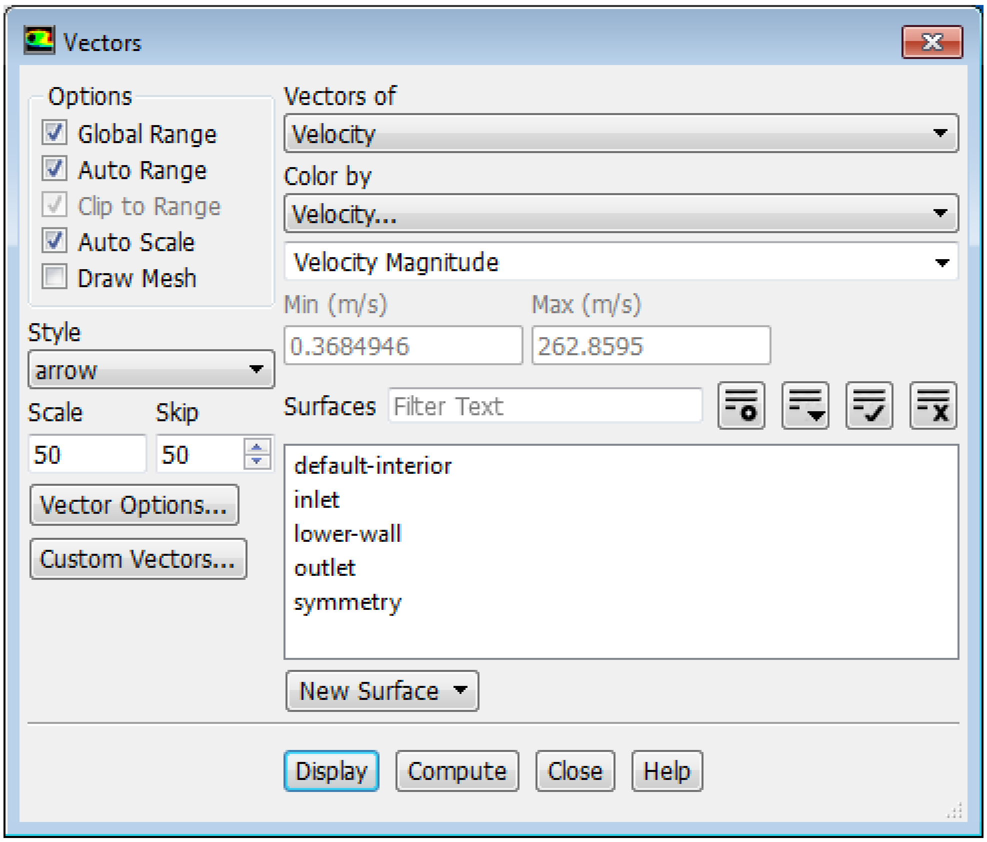
Because the solution animation was stored in memory, it will be lost if you exit ANSYS Fluent without saving it in one of the formats described previously. Note

> that only the animation-frame format can be read back into the **Playback** dialog box for display in a later ANSYS Fluent session.

7. Read the case and data files for the 660th time step (**noz_anim–1–00660.cas.gz** and **noz_anim–1–00660.dat.gz**) into ANSYS Fluent.

8. Plot vectors at t=0.018849 s (Figure 6.14: Velocity Vectors at t=0.018849 s (p. 310)).

   Postprocessing → **Graphics** → **Vectors** → **Edit...**



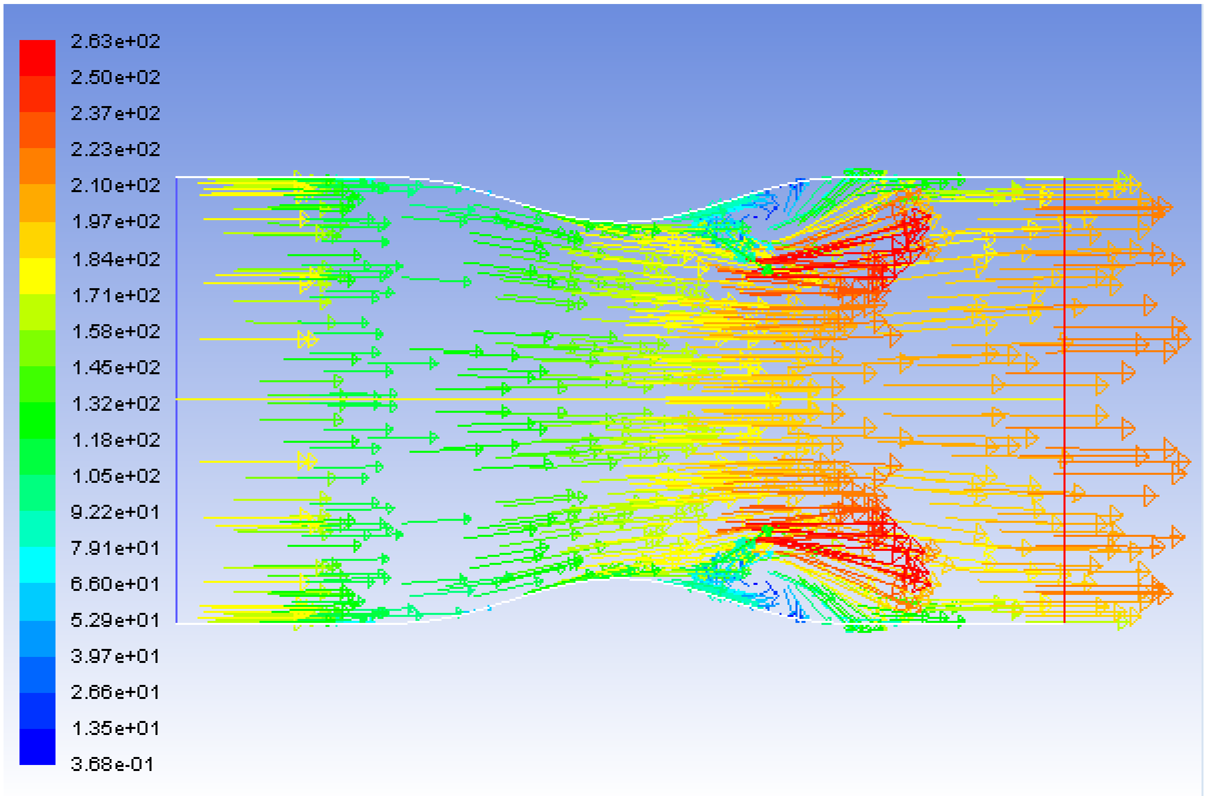
   a. Ensure **Auto Scale** is enabled under **Options**.

   b. Enter `50` under **Scale**.

   c. Enter `50` under **Skip**.

   d. Click **Display** and close the **Vectors** dialog box.

Figure 6.14: Velocity Vectors at t=0.018849 s



The transient flow prediction in Figure 6.14: Velocity Vectors at t=0.018849 s (p. 310) shows the expected form, with peak velocity of approximately 260 m/s through the nozzle at $t = 0.018849$ seconds.

9. In a similar manner to steps 7 and 8, read the case and data files saved for other time steps of interest and display the vectors.

6.5. Summary

In this tutorial, you modeled the transient flow of air through a nozzle. In doing so, you learned how to:

- generate a steady-state solution as an initial condition for the transient case.
- set solution parameters for implicit time-stepping and apply a user-defined transient pressure profile at the outlet.
- use mesh adaption to refine the mesh in areas with high pressure gradients to better capture the shocks.
- automatically save solution information as the transient calculation proceeds.
- create and view solution animations of the transient flow.

6.6. Further Improvements

This tutorial guides you through the steps to generate a second-order solution. You may be able to increase the accuracy of the solution even further by using an appropriate higher-order discretization scheme and by adapting the mesh further. Mesh adaption can also ensure that the solution is independent of the mesh. These steps are demonstrated in Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121).

Chapter 7: Modeling Radiation and Natural Convection

This tutorial is divided into the following sections:

7.1. Introduction

In this tutorial, combined radiation and natural convection are solved in a three-dimensional square box on a mesh consisting of hexahedral elements.

This tutorial demonstrates how to do the following:

- Use the surface-to-surface (S2S) radiation model in ANSYS Fluent.
- Set the boundary conditions for a heat transfer problem involving natural convection and radiation.
- Calculate a solution using the pressure-based solver.
- Display velocity vectors and contours of wall temperature, surface cluster ID, and radiation heat flux.

7.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

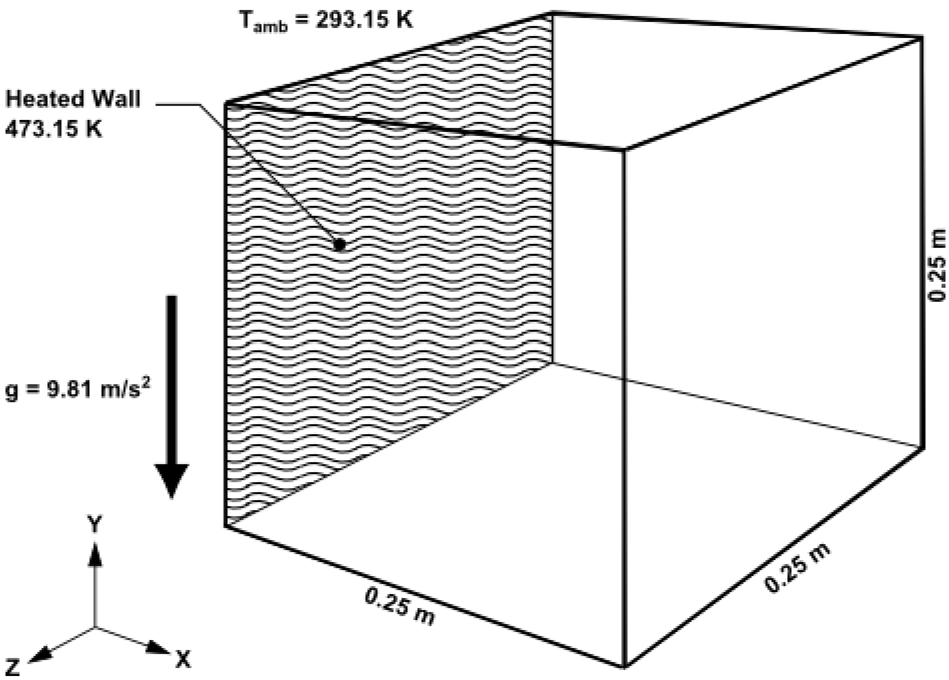
and that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

7.3. Problem Description

The problem to be considered is shown schematically in Figure 7.1: Schematic of the Problem (p. 314). A three-dimensional box $\left(0.25\text{m} \times 0.25\text{m} \times 0.25\text{m}\right)$ has a hot wall of aluminum at 473.15 K. All other walls are made of an insulation material and are subject to radiative and convective heat transfer to the surroundings, which are at 293.15 K. Gravity acts downwards. The medium contained in the box is assumed not to emit, absorb, or scatter radiation. All walls are gray. The objective is to compute the flow and temperature patterns in the box, as well as the wall heat flux, using the surface-to-surface (S2S) model available in ANSYS Fluent.

The working fluid has a Prandtl number of approximately 0.71, and the Rayleigh number based on L (0.25) is 1×10^{8}. This means the flow is most likely laminar. The Planck number $k/\left(4\sigma LT_0^3\right)$ is 0.006, and measures the relative importance of conduction to radiation.

Figure 7.1: Schematic of the Problem



7.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

7.4.1. Preparation

To prepare for running this tutorial:

1. Set up a working folder on the computer you will be using.

2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

> **Note**
>
> If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

3. Enter the name of this tutorial into the search bar.

4. Narrow the results by using the filter on the left side of the page.

   a. Click **ANSYS Fluent** under **Product**.

   b. Click **18.0** under **Version**.

5. Select this tutorial from the list.

6. Click the **radiation_natural_convection_R180.zip** link to download the input files.

7. Unzip `radiation_natural_convection_R180.zip` to your working folder.

   The mesh file `rad.msh.gz` can be found in the `radiation_natural_convection` folder created after unzipping the file.

8. Use Fluent Launcher to start the **3D**, single precision (disable **Double Precision**) version of ANSYS Fluent.

   Fluent Launcher displays your **Display Options** preferences from the previous session.

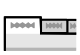
   For more information about Fluent Launcher, see starting ANSYS Fluent using the Fluent Launcher in the Fluent Getting Started Guide.

9. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.

10. Ensure that **Serial** is selected under **Processing Options**.

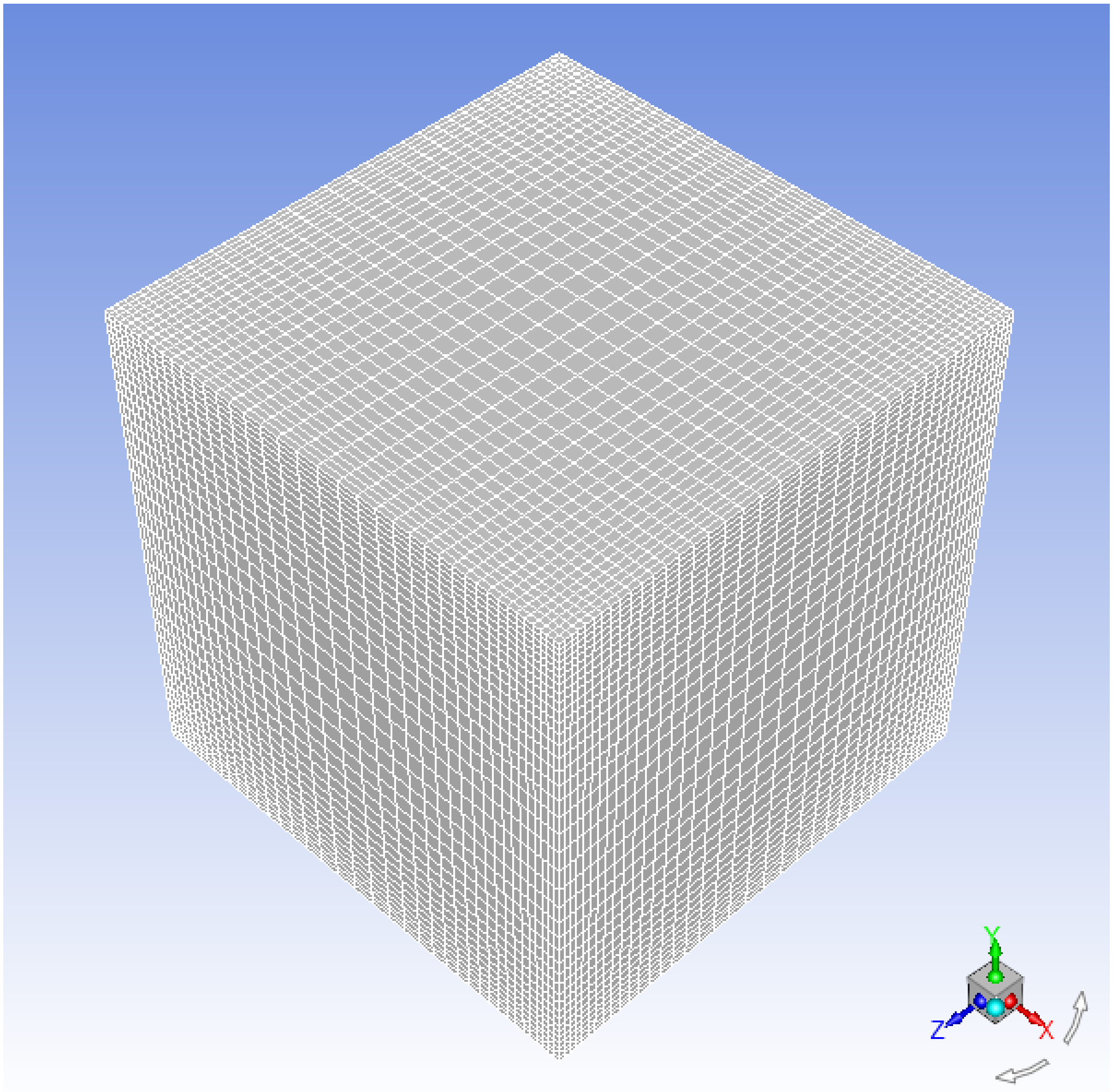
7.4.2. Reading and Checking the Mesh

1. Read the mesh file `rad.msh.gz`.

    **File → Read → Mesh...**

   As the mesh is read, messages will appear in the console reporting the progress of the reading and the mesh statistics. The mesh size will be reported as 64,000 cells. Once reading is complete, the mesh will be displayed in the graphics window.

Figure 7.2: Graphics Display of Mesh



2. Check the mesh.

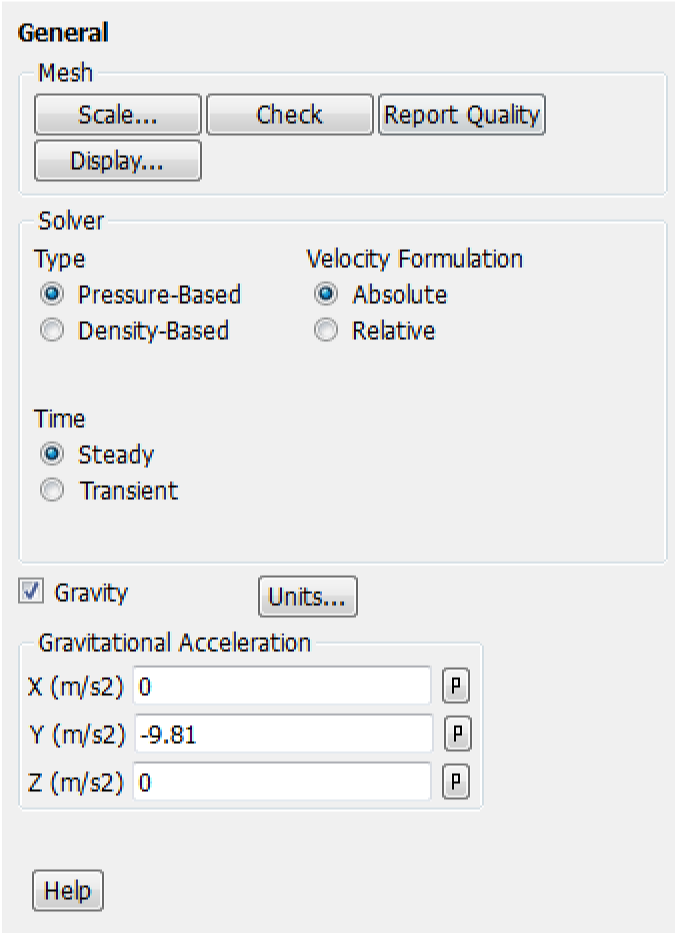
   Setting Up Domain → Mesh → Check

   ANSYS Fluent will perform various checks on the mesh and report the progress in the console. Make sure that the reported minimum volume is a positive number.

7.4.3. Solver and Analysis Type

1. Confirm the solver settings and enable gravity.

   Setup → General



a. Retain the default settings of pressure-based steady-state solver in the **Solver** group box.

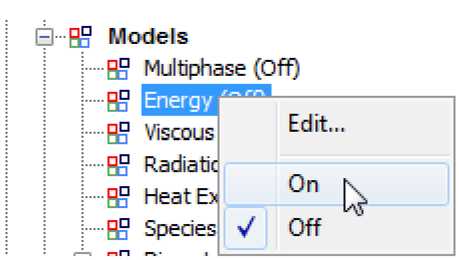
b. Enable the **Gravity** option.

c. Enter `-9.81` m/s^2 for **Y** in the **Gravitational Acceleration** group box.

7.4.4. Models

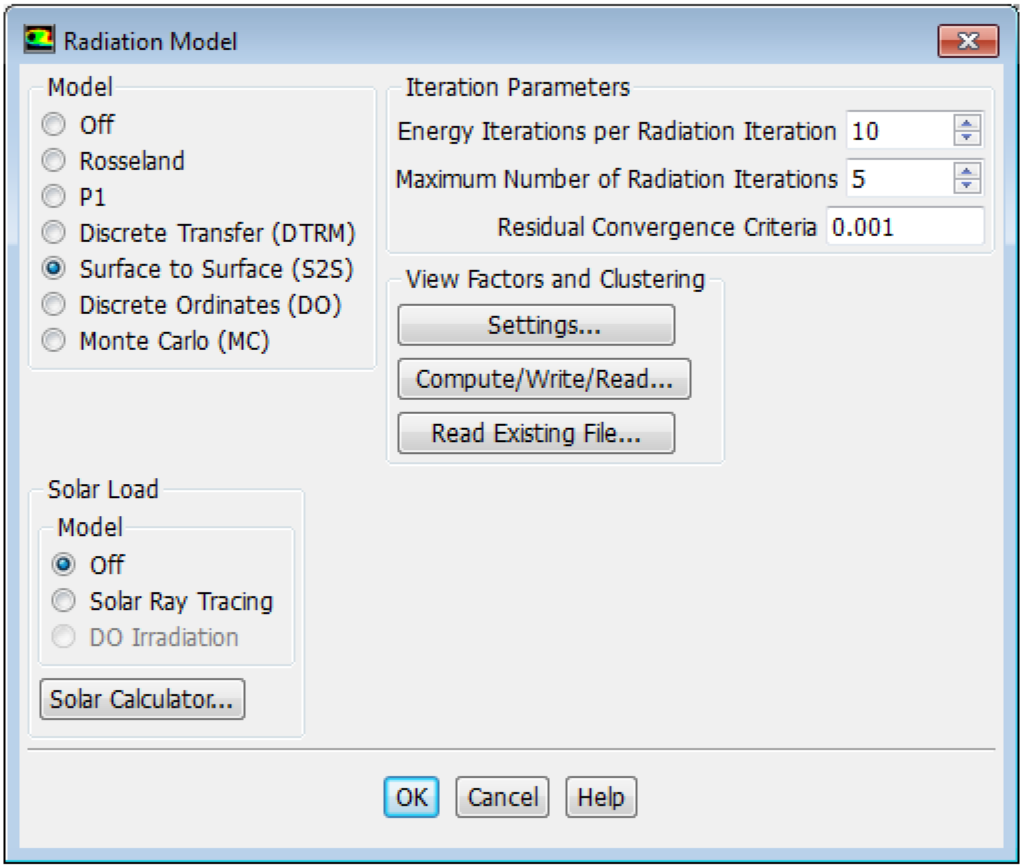
1. Enable the energy equation.

Setup → **Models** → **Energy** **On**



2. Set up the **Surface to Surface (S2S)** radiation model.

Setup → **Models** → **Radiation** **Model** → **Surface to Surface (S2S)**



The surface-to-surface (S2S) radiation model can be used to account for the radiation exchange in an enclosure of gray-diffuse surfaces. The energy exchange between two surfaces depends in part on their size, separation distance, and orientation. These parameters are accounted for by a geometric function called a "view factor".

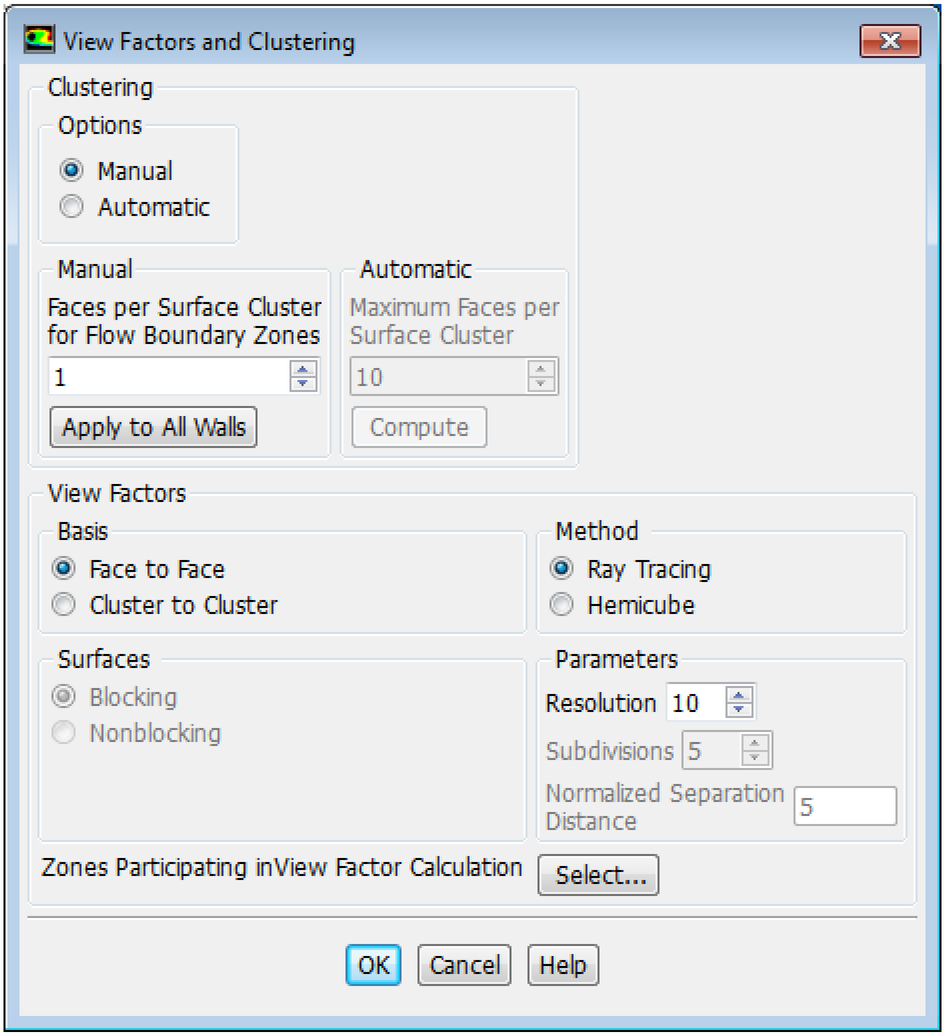
The S2S model assumes that all surfaces are gray and diffuse. Thus according to the gray-body model, if a certain amount of radiation is incident on a surface, then a fraction is reflected, a fraction is absorbed, and a fraction is transmitted. The main assumption of the S2S model is that any absorption, emission, or scattering of radiation by the medium can be ignored. Therefore only "surface-to-surface" radiation is considered for analysis.

For most applications the surfaces in question are opaque to thermal radiation (in the infrared spectrum), so the surfaces can be considered opaque. For gray, diffuse, and opaque surfaces it is valid to assume that the emissivity is equal to the absorptivity and that reflectivity is equal to 1 minus the emissivity.

When the S2S model is used, you also have the option to define a "partial enclosure". This option allows you to disable the view factor calculation for walls with negligible emission/absorption or walls that have uniform temperature. The main advantage of this option is to speed up the view factor calculation and the radiosity calculation.

a. Click the **Settings...** button to open the **View Factors and Clustering** dialog box.

   You will define the view factor and cluster parameters.



i. Retain the value of 1 for **Faces per Surface Cluster for Flow Boundary Zones** in the **Manual** group box.

ii. Click **Apply to All Walls**.

The S2S radiation model is computationally very expensive when there are a large number of radiating surfaces. The number of radiating surfaces is reduced by clustering surfaces into surface "clusters". The surface clusters are made by starting from a face and adding its neighbors and their neighbors until a specified number of faces per surface cluster is collected.

For a small problem, the default value of 1 for ***Faces per Surface Cluster for Flow Boundary Zones*** *is acceptable. For a large problem you can increase this number to reduce the memory requirement for the view factor file that is saved in a later step. This may also lead to some reduction in the computational expense. However, this is at the cost of some accuracy. This tutorial illustrates the influence of clusters.*

iii. Ensure **Ray Tracing** is selected from the **Method** list in the **View Factors** group box.

iv. Click **OK** to close the **View Factors and Clustering** dialog box.

b. Click the **Compute/Write/Read...** button in the **View Factors and Clustering** group box to open the **Select File** dialog box and to compute the view factors.

The file created in this step will store the cluster and view factor parameters.

i. Enter **rad_1.s2s.gz** as the file name for **S2S File**.

ii. Click **OK** in the **Select File** dialog box.

Note

The size of the view factor file can be very large if not compressed. It is highly recommended to compress the view factor file by providing `.gz` or `.Z` extension after the name (that is, `rad_1.gz` or `rad_1.Z`). For small files, you can provide the .s2s extension after the name.

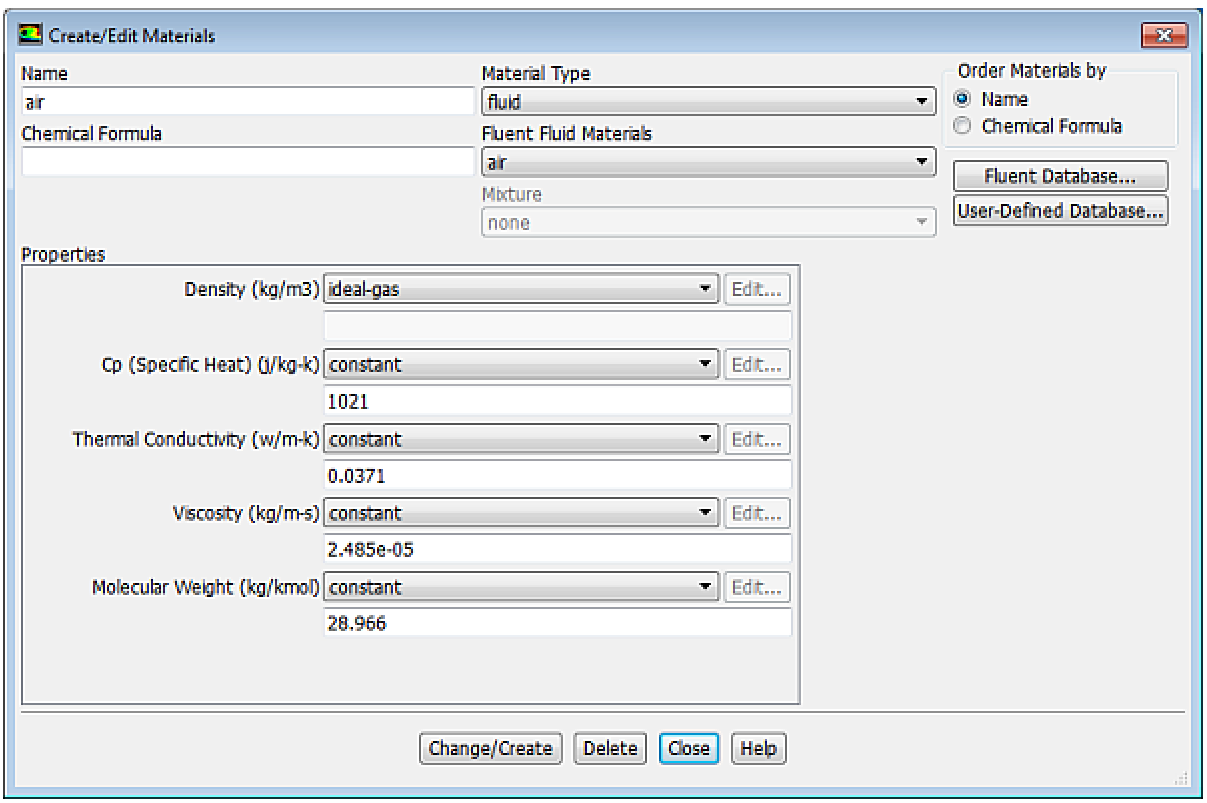
ANSYS Fluent will print an informational message describing the progress of the view factor calculation in the console.

c. Click **OK** to close the **Radiation Model** dialog box.

7.4.5. Defining the Materials

1. Set the properties for air.

Setup → **Materials** → **Fluids** → **air** **Edit...**



a. Select **ideal-gas** from the **Density** drop-down list.

b. Enter **1021** J/kg-K for **Cp (Specific Heat)**.

c. Enter **0.0371** W/m-K for **Thermal Conductivity**.

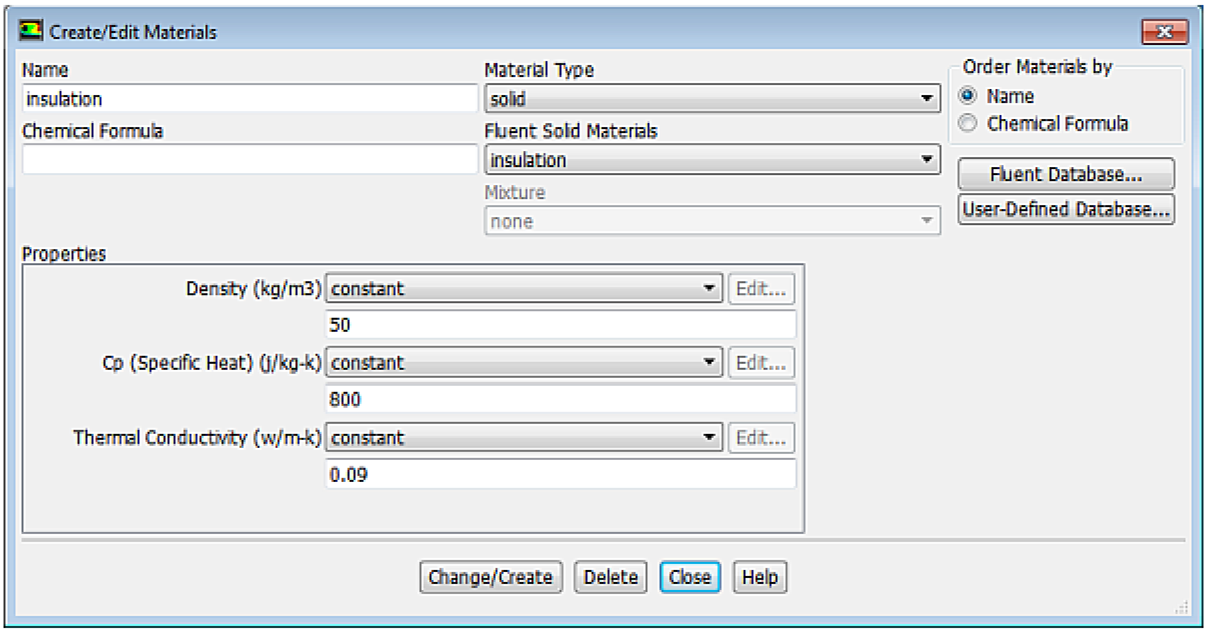
d. Enter **2.485e-05** kg/m-s for **Viscosity**.

e. Retain the default value of **28.966** kg/kmol for **Molecular Weight**.

f. Click **Change/Create** and close the **Create/Edit Materials** dialog box.

2. Define the new material, **insulation**.

Setup → **Materials** → **Solid** → **aluminum** **Edit...**



a. Enter **insulation** for **Name**.

b. Delete the entry in the **Chemical Formula** field.

c. Enter **50** kg/m^3 for **Density**.

d. Enter **800** J/kg-K for **Cp (Specific Heat)**.

e. Enter **0.09** W/m-K for **Thermal Conductivity**.

f. Click **Change/Create**.

g. Click **No** when the **Question** dialog box appears, asking if you want to overwrite aluminum.

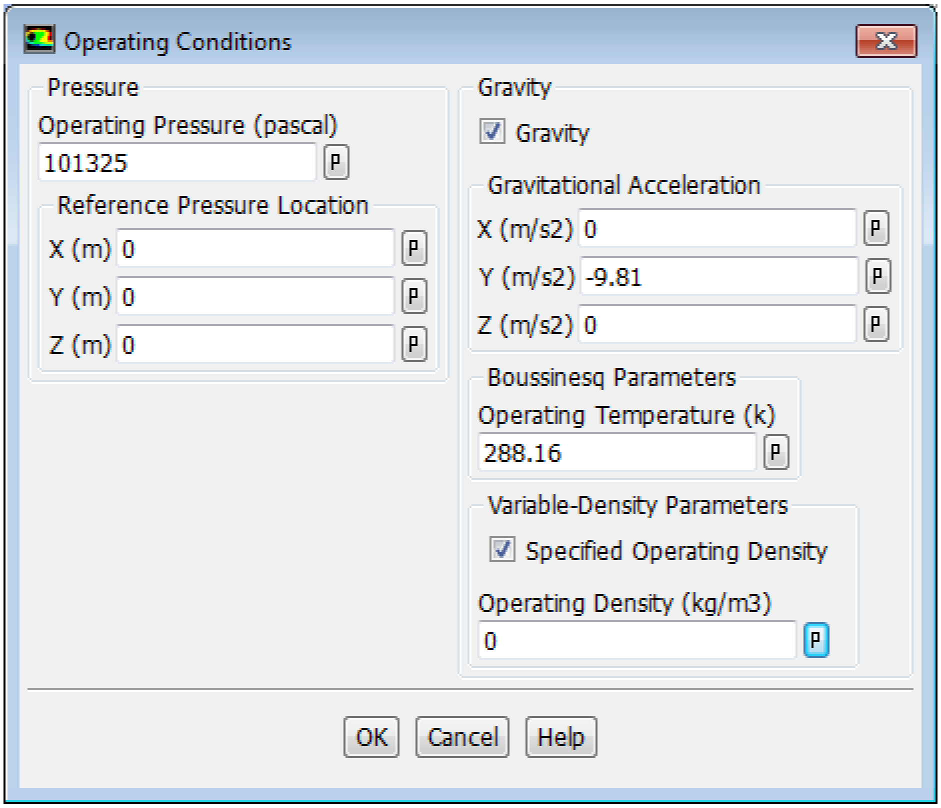
*The **Create/Edit Materials** dialog box will be updated to show the new material, **insulation**, in the **Fluent Solid Materials** drop-down list.*

h. Close the **Create/Edit Materials** dialog box.

7.4.6. Operating Conditions

Specify operating density.

Setting Up Physics → **Solver** → **Operating Conditions...**



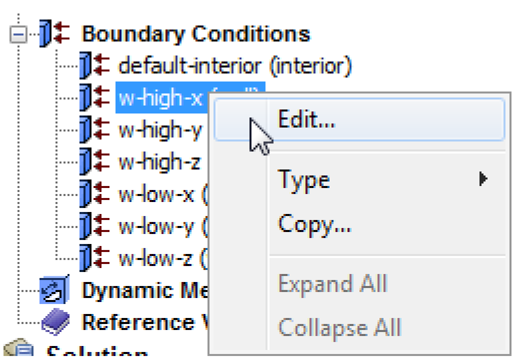
1. In the **Operating Conditions** dialog box, select the **Specified Operating Density** check box.

2. Enter **0** for **Operating Density** and click **OK** to close the **Operating Conditions** dialog box.

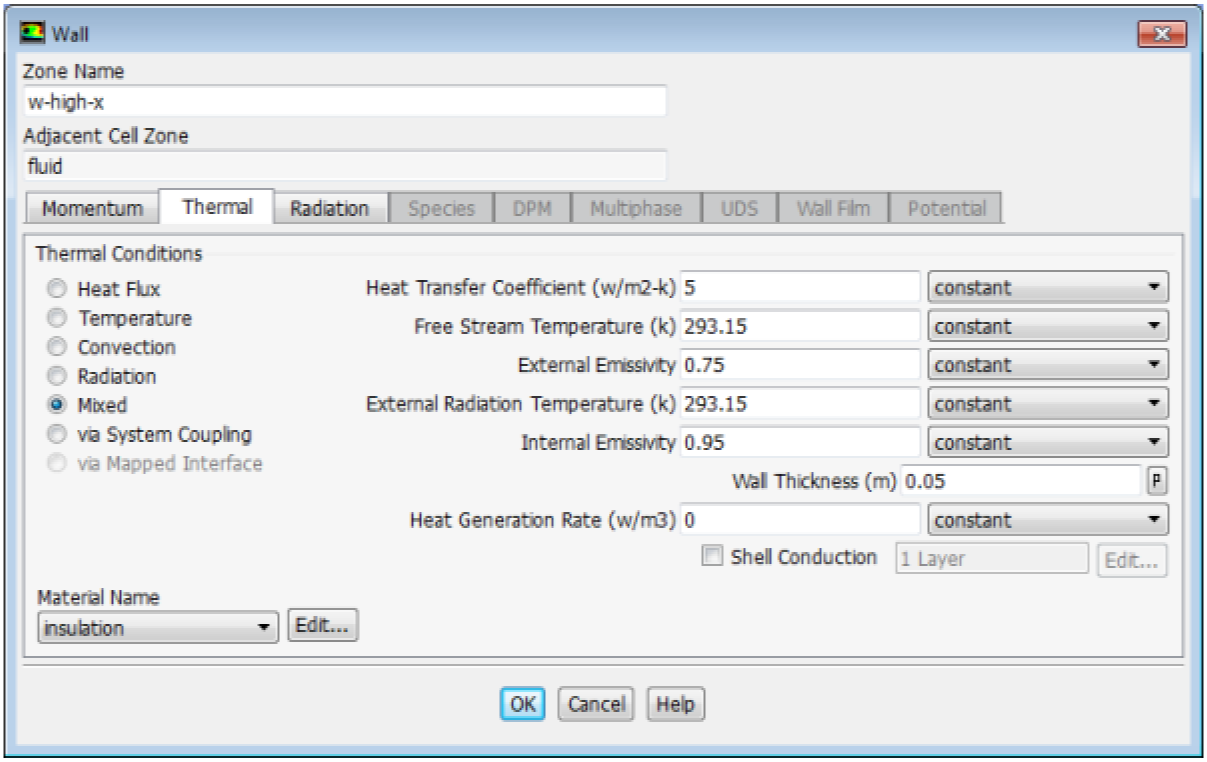
7.4.7. Boundary Conditions

1. Set the boundary conditions for the front wall (**w-high-x**).

   Setup → **Boundary Conditions** → **w-high-x** **Edit...**



   The **Wall** dialog box opens.



a. Click the **Thermal** tab and select **Mixed** from the **Thermal Conditions** list.

b. Select **insulation** from the **Material Name** drop-down list.

c. Enter **5** W/m^2-K for **Heat Transfer Coefficient**.

d. Enter **`293.15`** K for **Free Stream Temperature**.

e. Enter **`0.75`** for **External Emissivity**.

f. Enter **`293.15`** K for **External Radiation Temperature**.

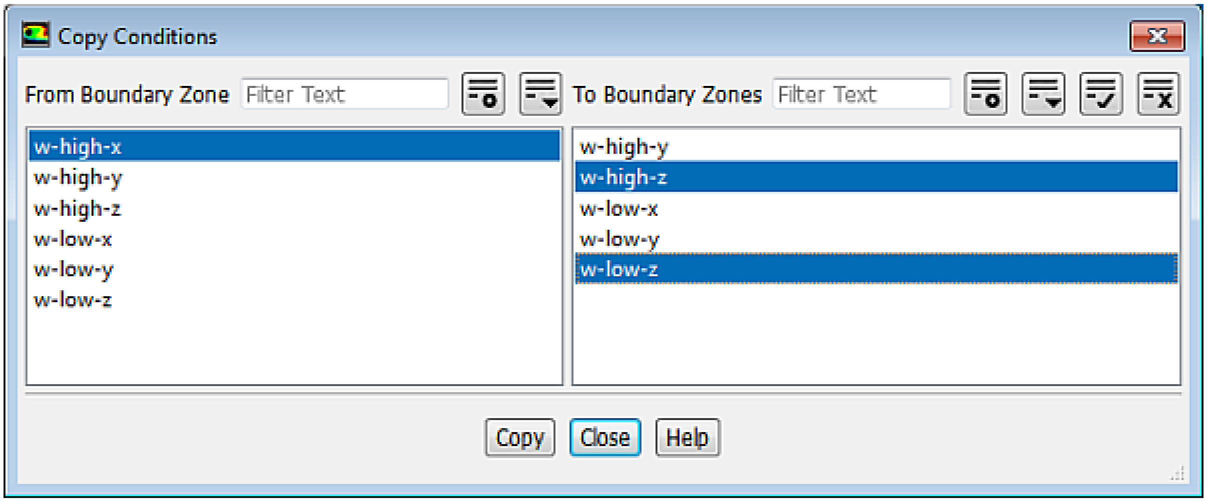
g. Enter **`0.95`** for **Internal Emissivity**.

h. Enter **`0.05`** m for **Wall Thickness**.

i. Click **OK** to close the **Wall** dialog box.

2. Copy boundary conditions to define the side walls **w-high-z** and **w-low-z**.

Setup → **Boundary Conditions** → **w-high-x** **Copy...**



a. Ensure **w-high-x** is selected in the **From Boundary Zone** selection list.

b. Select **w-high-z** and **w-low-z** from the **To Boundary Zones** selection list.

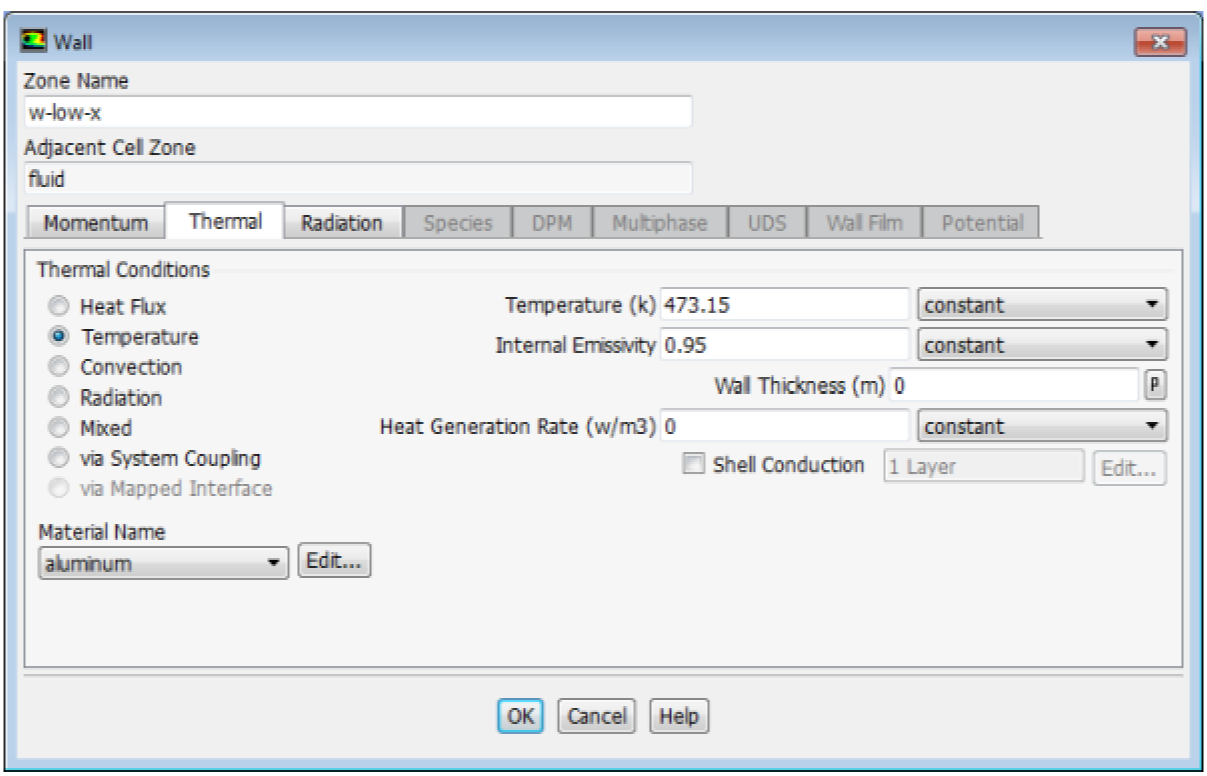
c. Click **Copy**.

d. Click **OK** when the **Question** dialog box opens asking whether you want to copy the boundary conditions of w-high-x to all the selected zones.

e. Close the **Copy Conditions** dialog box.

3. Set the boundary conditions for the heated wall (**w-low-x**).

Setup → **Boundary Conditions** → **w-low-x** **Edit...**



a. Click the **Thermal** tab and select **Temperature** from the **Thermal Conditions** list.

b. Retain the default selection of **aluminum** from the **Material Name** drop-down list.

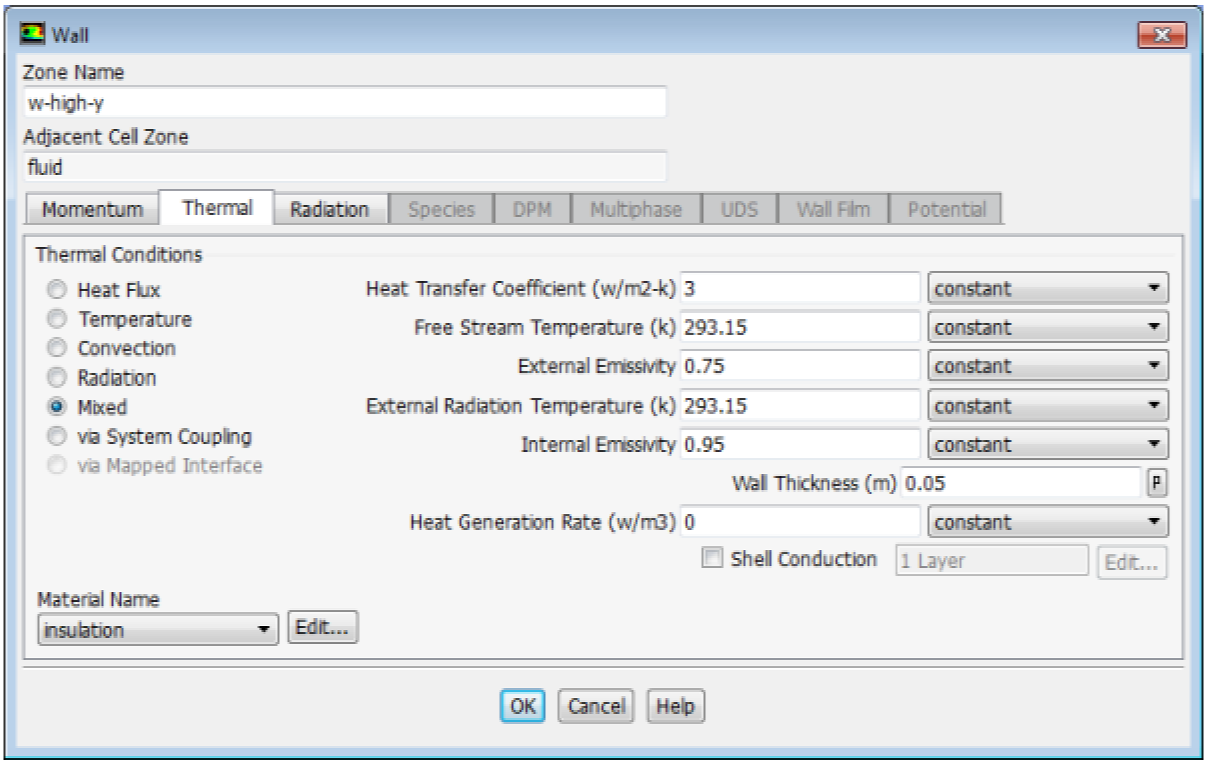
c. Enter **473.15** K for **Temperature**.

d. Enter **0.95** for **Internal Emissivity**.

e. Click **OK** to close the **Wall** dialog box.

4. Set the boundary conditions for the top wall (**w-high-y**).

Setup → **Boundary Conditions** → **w-high-y** **Edit...**



a. Click the **Thermal** tab and select **Mixed** from the **Thermal Conditions** list.

b. Select **insulation** from the **Material Name** drop-down list.

c. Enter **3** W/m^2-K for **Heat Transfer Coefficient**.

d. Enter **293.15** K for **Free Stream Temperature**.

e. Enter **0.75** for **External Emissivity**.

f. Enter **293.15** K for **External Radiation Temperature**.

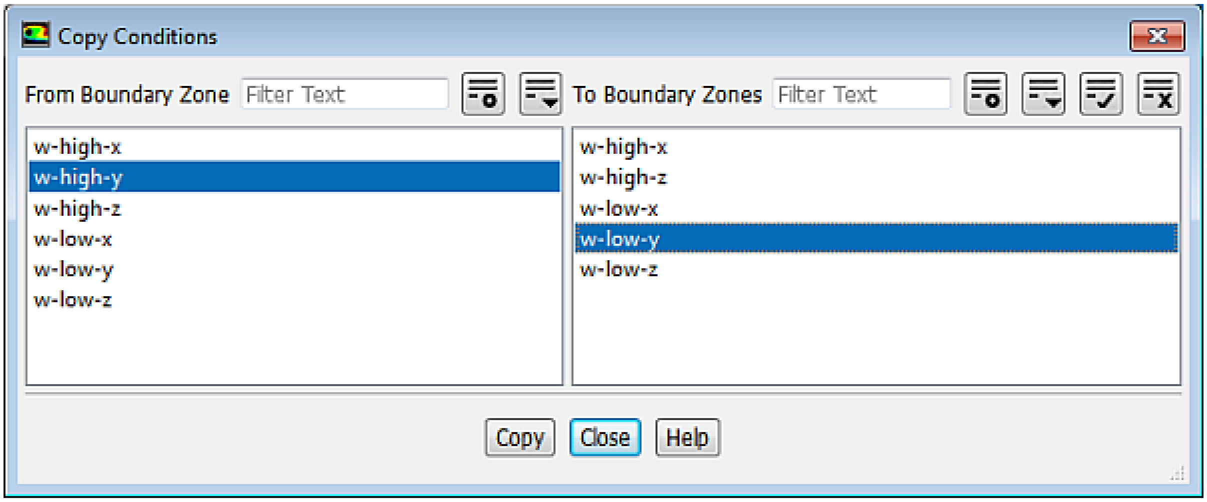
g. Enter **0.95** for **Internal Emissivity**.

h. Enter **0.05** m for **Wall Thickness**.

i. Click **OK** to close the **Wall** dialog box.

5. Copy boundary conditions to define the bottom wall (**w-low-y**) as previously done in this tutorial.

   Setup → **Boundary Conditions** → **w-high-y** **Copy...**



   a. Ensure **w-high-y** is selected in the **From Boundary Zone** selection list.

   b. Select **w-low-y** from the **To Boundary Zones** selection list.

   c. Click **Copy**.

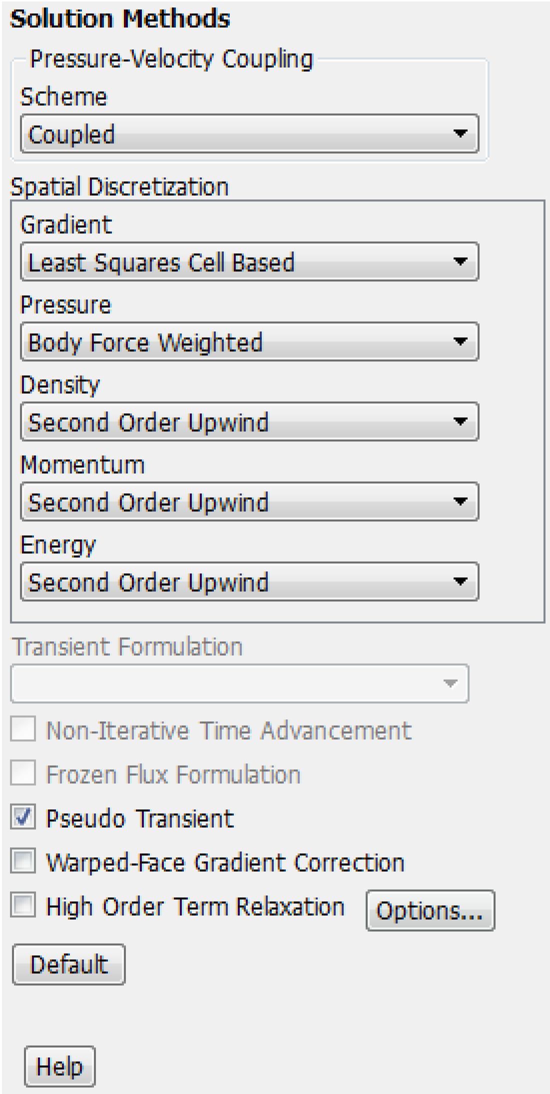
   d. Click **OK** when the **Question** dialog box opens asking whether you want to copy the boundary conditions of w-high-y to all the selected zones.

   e. Close the **Copy Conditions** dialog box.

7.4.8. Obtaining the Solution

1. Set the solution parameters.

   Solving → **Solution** → **Methods...**



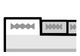
a. Select **Coupled** from the **Scheme** drop-down list in the **Pressure-Velocity Coupling** group box.

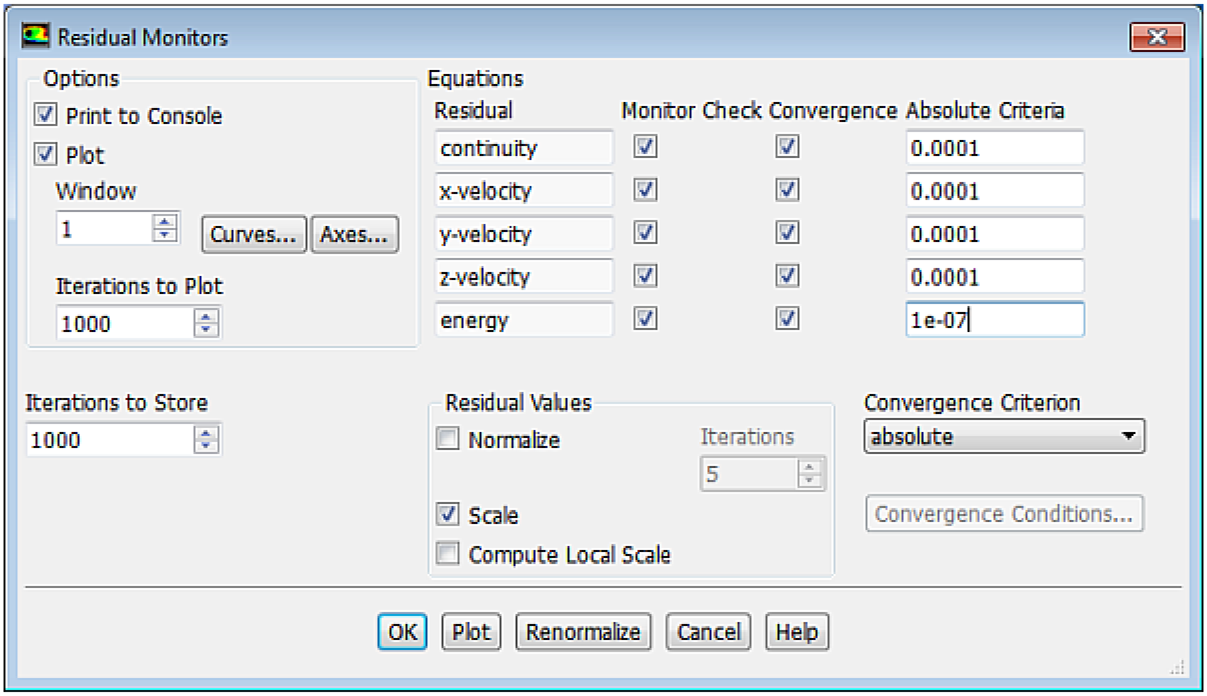
b. Select **Body Force Weighted** from the **Pressure** drop-down list in the **Spatial Discretization** group box.

c. Retain the default selection of **Second Order Upwind** from the **Momentum** and **Energy** drop-down lists.

d. Enable the **Pseudo Transient** option.

2. Set the convergence criteria for you simulation.

 **Solving** → **Reports** → **Residuals...**



a. Ensure that **Plot** is enabled in the **Options** group box.

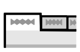
b. Enter `0.0001` for the **Absolute Criteria** of **continuity**, **x-**, **y-**, and **z-velocity**.

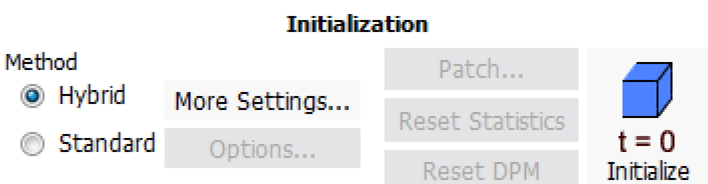
c. Enter `1e-7` for the **Absolute Criteria** of **energy**.

Decreasing the criteria for these residuals will improve the accuracy of the solution.

d. Click **OK**

3. Initialize the solution.

 **Solving** → **Initialization**



a. Retain the default selection of **Hybrid Initialization** from the **Initialization Methods** list.

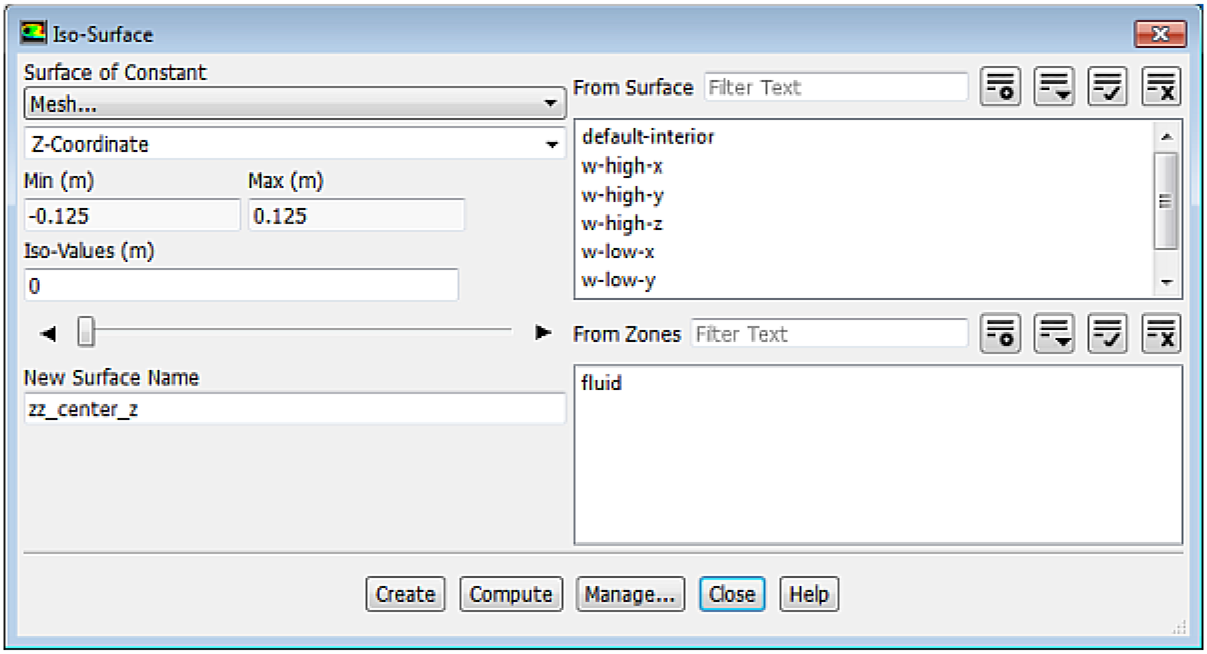
b. Click **Initialize**.

4. Create a surface report definition to aid in judging convergence.

It is good practice to use reports of physical solution quantities together with residual monitors when determining whether a solution is converged. In this step you will create a surface report definition for the average temperature on the z=0 plane.

a. Create the new surface, **zz_center_z**.

Setting Up Domain → Surface → Create → Iso-Surface...



i. Select **Mesh...** and **Z-Coordinate** from the **Surface of Constant** drop-down lists.

ii. Click **Compute** and retain the default value of `0` for **Iso-Values**.

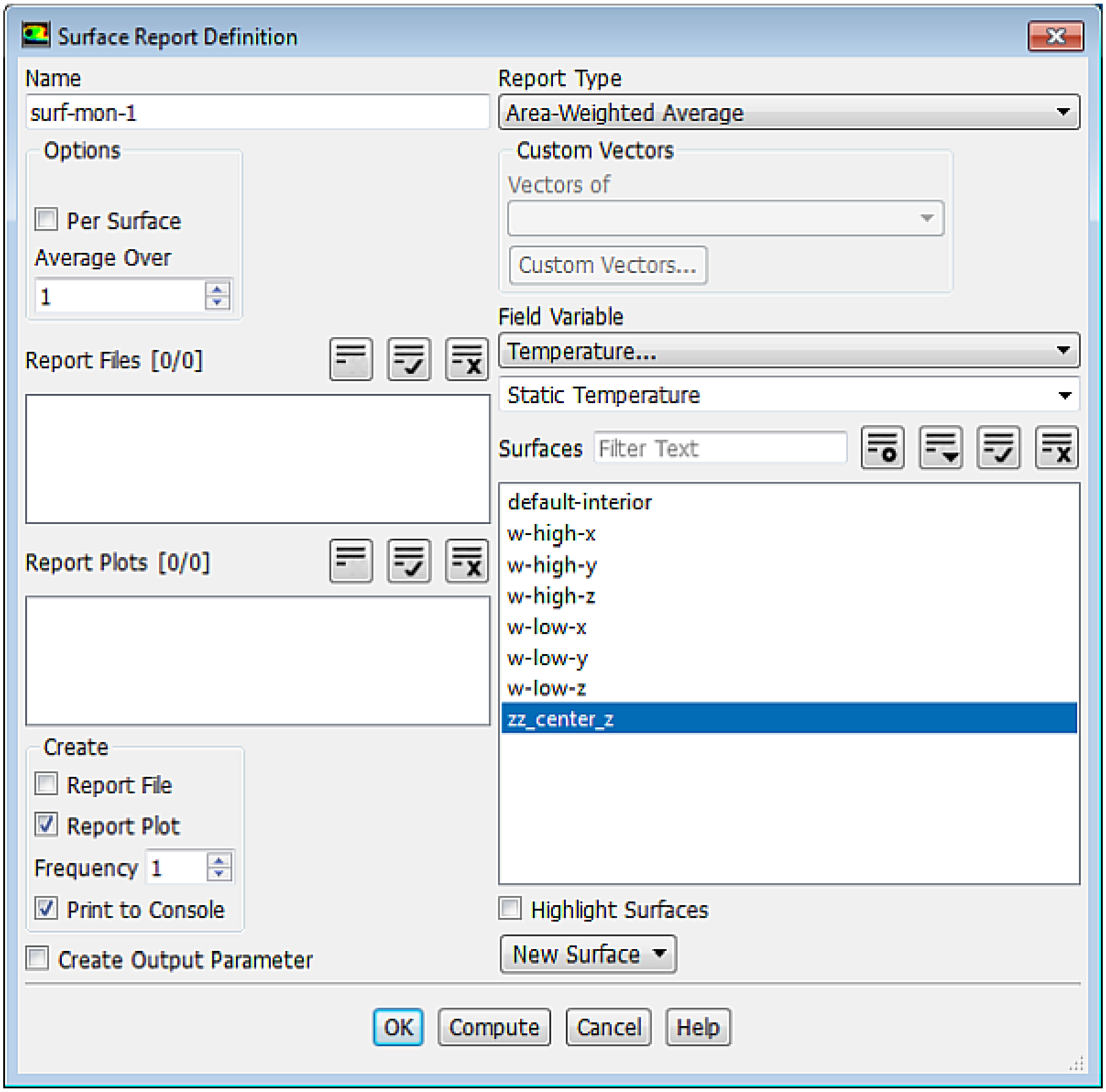
iii. Enter `zz_center_z` for **New Surface Name**.

> **Note**
>
> If you want to delete or otherwise manipulate any surfaces, click **Manage...** to open the **Surfaces** dialog box.

iv. Click **Create** and close the **Iso-Surface** dialog box.

b. Create the surface report definition.

Solving → Reports → Definitions → New → Surface Report → Area-Weighted Average...



i. Enter `surf-mon-1` for the **Name** of the surface report definition.

ii. In the **Create** group box, enable **Report Plot** and **Print to Console**.

Note

Unlike residual values, data from other reports is not saved as part of the solution set when the ANSYS Fluent data file is saved. If you want to access the surface report data in future ANSYS Fluent sessions, you can enable the **Report File** option. The report file will be saved in your working directory.

---

iii. Select **Temperature...** and **Static Temperature** from the **Field Variable** drop-down lists.

iv. Select **zz_center_z** from the **Surfaces** selection list.

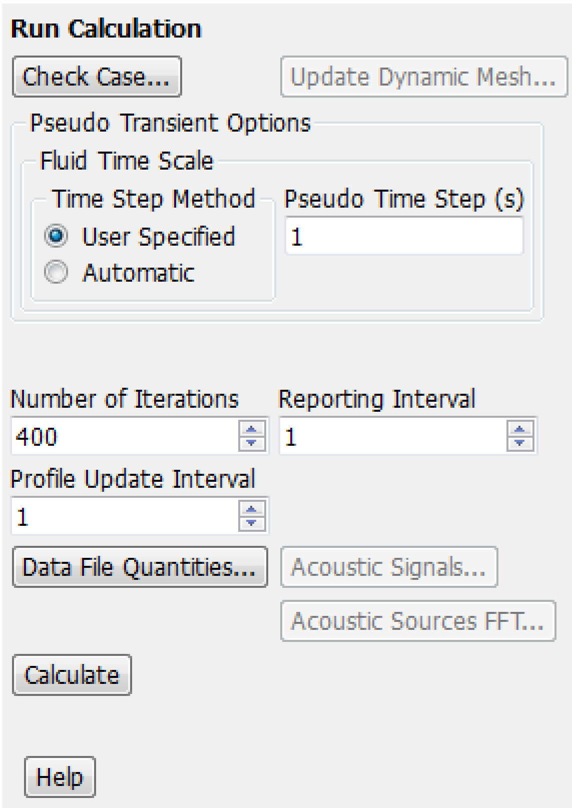
v. Click **OK** to save the surface report definition settings and close the **Surface Report Definition** dialog box.

5. Save the case file (`rad_a_1.cas.gz`).

   File → **Write** → **Case...**

6. Start the calculation by requesting 400 iterations.

   Solving → **Run Calculation**→**Advanced...**



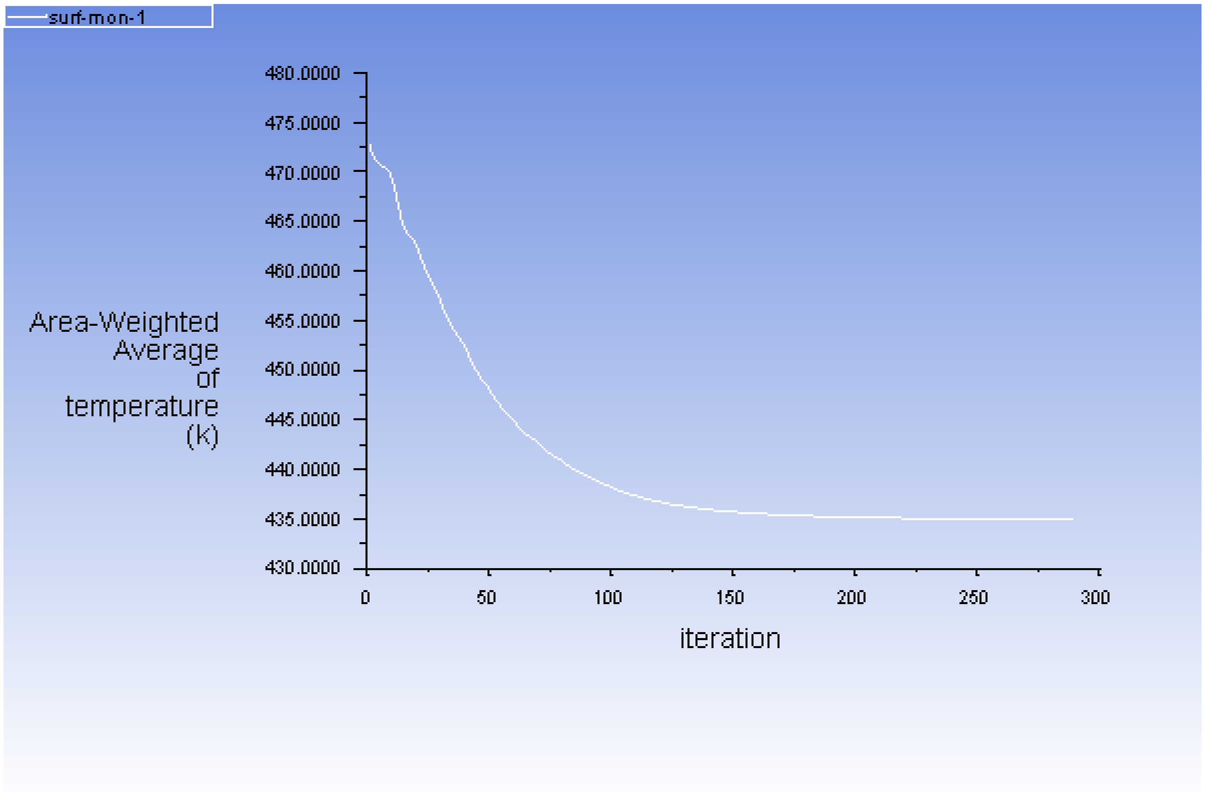
   a. Select **User Specified** from the **Time Step Method** list.

   b. Retain the default value of `1` for **Pseudo Time Step**.

   c. Enter **`400`** for **Number of Iterations**.

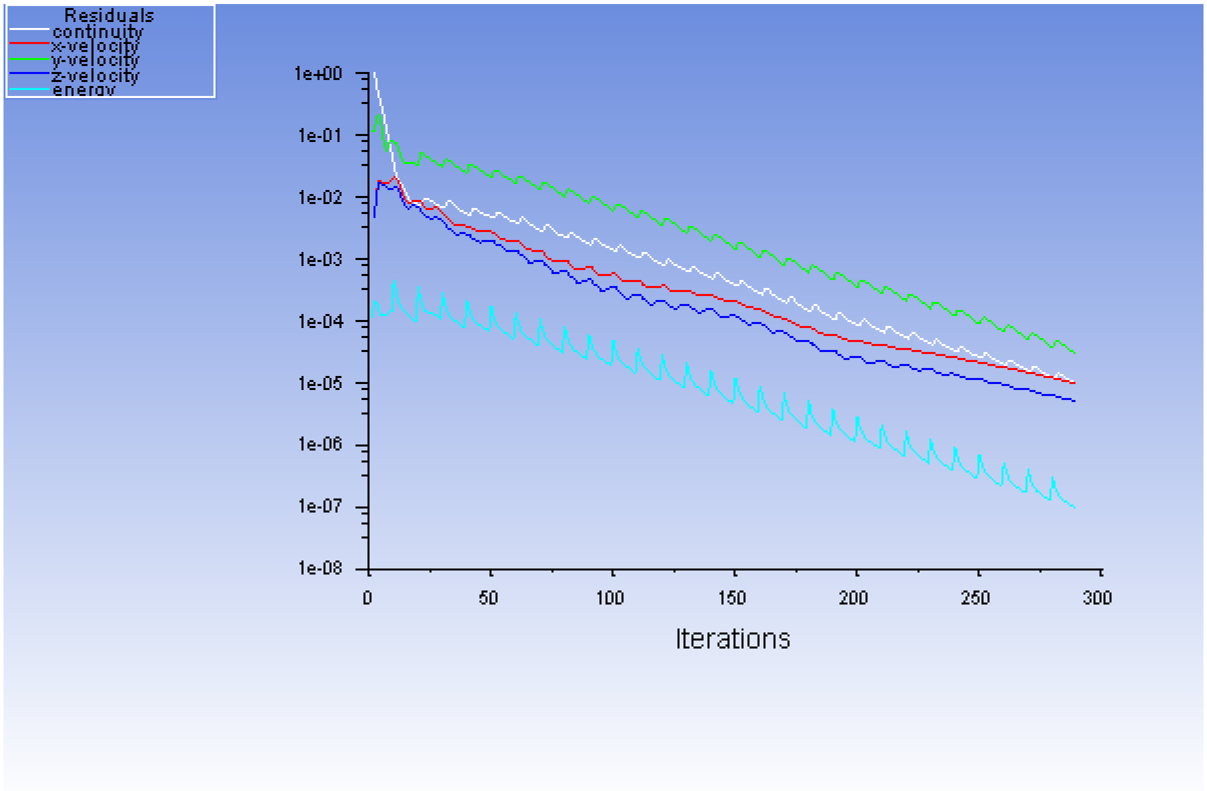
   d. Click **Calculate**.

Figure 7.3: Temperature Surface Report



*The surface report history shows that the average temperature on **zz_center_z** has stabilized, thus confirming that the solution has indeed reached convergence. You can view the behavior of the residuals (Figure 7.4: Scaled Residuals (p. 333)) in the **Scaled Residuals** graphics window tab.*

Figure 7.4: Scaled Residuals



7. Save the case and data files (`rad_a_1.cas.gz`, `rad_a_1.dat.gz`).

   File → **Write** → **Case & Data...**

7.4.9. Postprocessing

1. Enable the postprocessing view by clicking in the objects toolbar and selecting **Post Processing**.

2. Disable lighting.

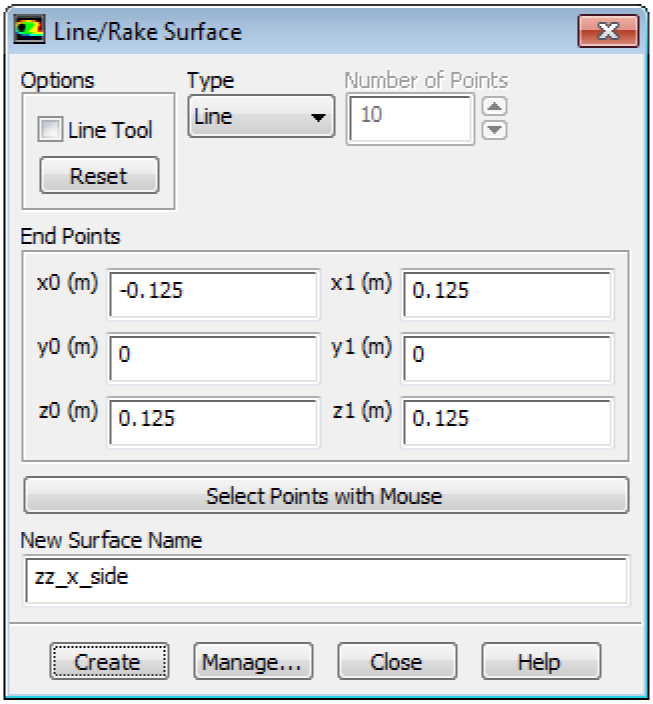
   Results → **Graphics** **Lights...**

   a. Disable **Light On** and click **Apply**.

   b. Close the **Lights** dialog box.

3. Create a new surface, **zz_x_side**, which will be used later to plot wall temperature.

   Postprocessing → **Surface** → **Create** → **Line/Rake...**



a. Enter (**`-0.125`**, **`0`**, **`0.125`**) for (**x0**, **y0**, **z0**), respectively.

b. Enter (**`0.125`**, **`0`**, **`0.125`**) for (**x1**, **y1**, **z1**), respectively.

c. Enter **`zz_x_side`** for **New Surface Name**.

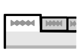
Note

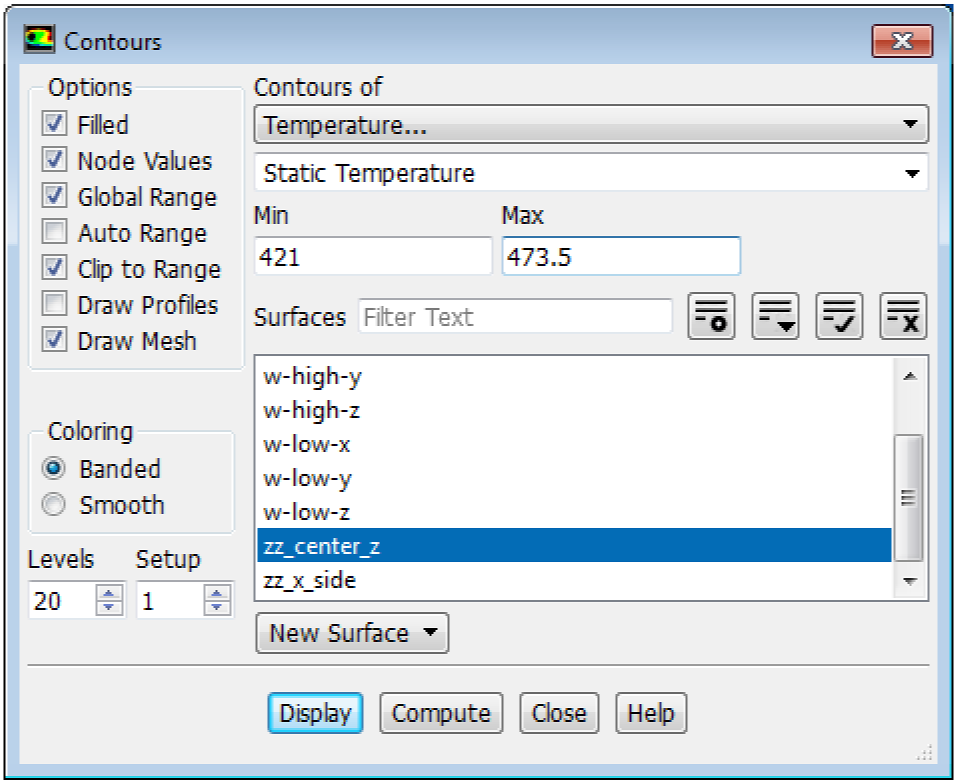
If you want to delete or otherwise manipulate any surfaces, click **Manage...** to open the **Surfaces** dialog box.

---

d. Click **Create** and close the **Line/Rake Surface** dialog box.

4. Display contours of static temperature.

 **Postprocessing** → **Graphics** → **Contours** → **Edit...**



a. Enable the **Filled** option in the **Options** group box.

b. Select **Temperature...** and **Static Temperature** from the **Contours of** drop-down lists.

c. Select **zz_center_z** from the **Surfaces** selection list.

d. Enable the **Draw Mesh** option in the **Options** group box to open the **Mesh Display** dialog box.

   i. Ensure **Outline** is selected in the **Edge Type** group box.

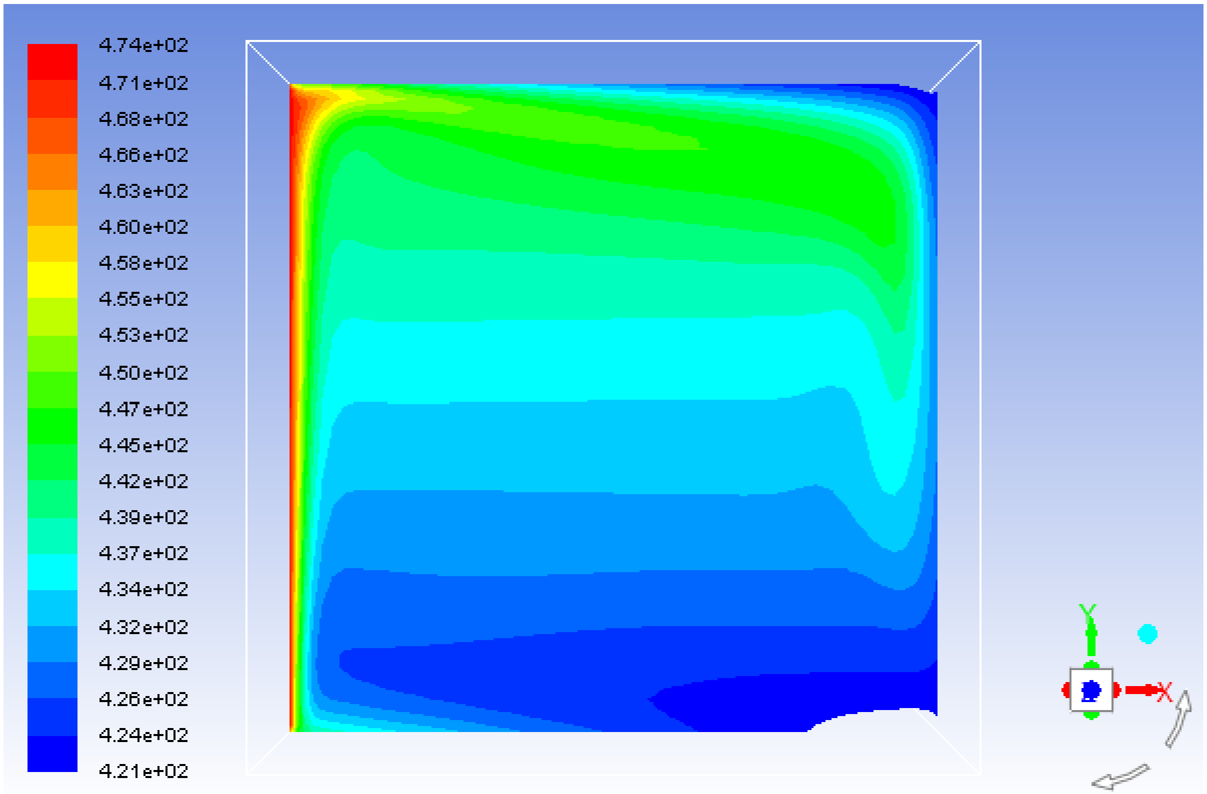
   ii. Close the **Mesh Display** dialog box.

e. Disable the **Auto Range** option.

f. Enter **421** K for **Min** and **473.15** K for **Max**.

g. Click **Display**, rotate the view as shown in Figure 7.5: Contours of Static Temperature (p. 336), and close the **Contours** dialog box.

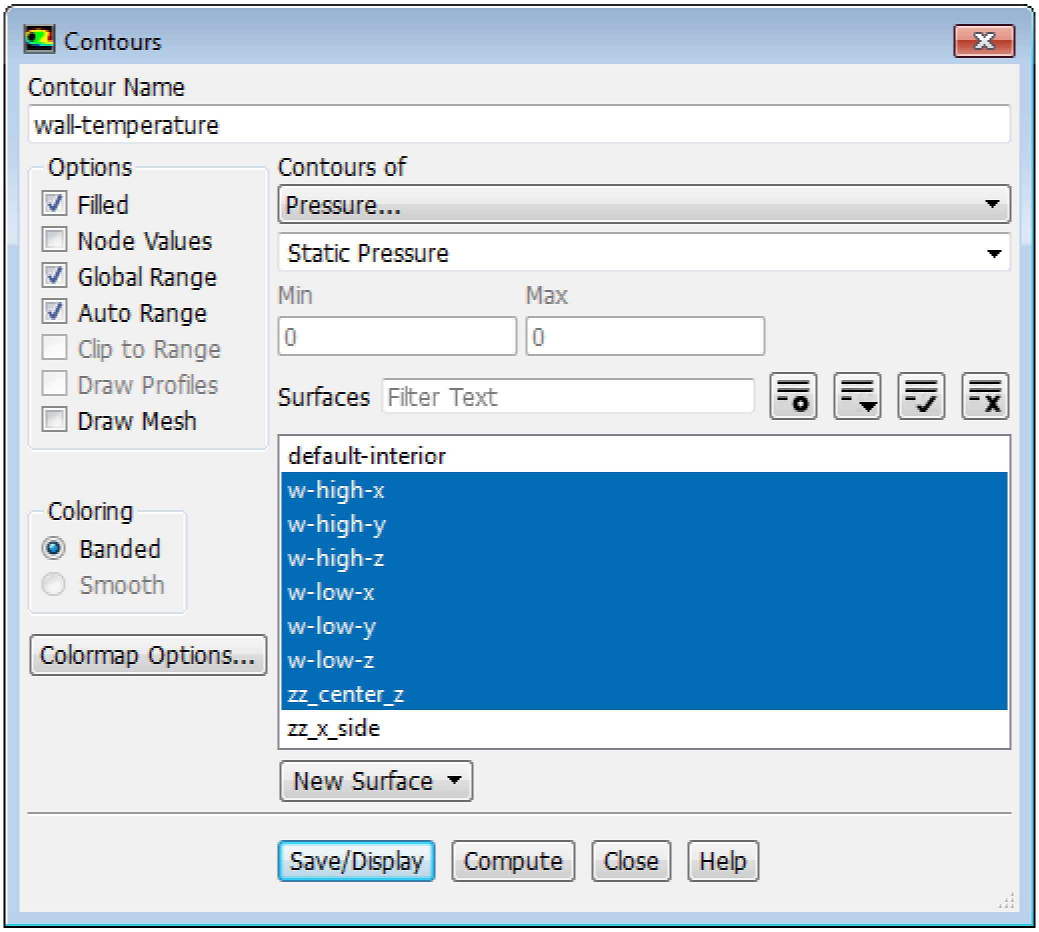
Figure 7.5: Contours of Static Temperature



A regular check for most buoyant cases is to look for evidence of stratification in the temperature field. This is observed as nearly horizontal bands of similar temperature. These may be broken or disturbed by buoyant plumes. For this case you can expect reasonable stratification with some disturbance at the vertical walls where the air is driven around. Inspection of the temperature contours in *Figure 7.5: Contours of Static Temperature (p. 336)* *reveals that the solution appears as expected.*

5. Create and display a contour definition for wall temperature (surfaces in contact with the fluid).

   Postprocessing → **Graphics** → **Contours** → **New...**



a. Enter **`wall-temperature`** for **Contour Name**.

b. Ensure that the **Filled** option is enabled in the **Options** group box.

c. Disable the **Node Values** option.

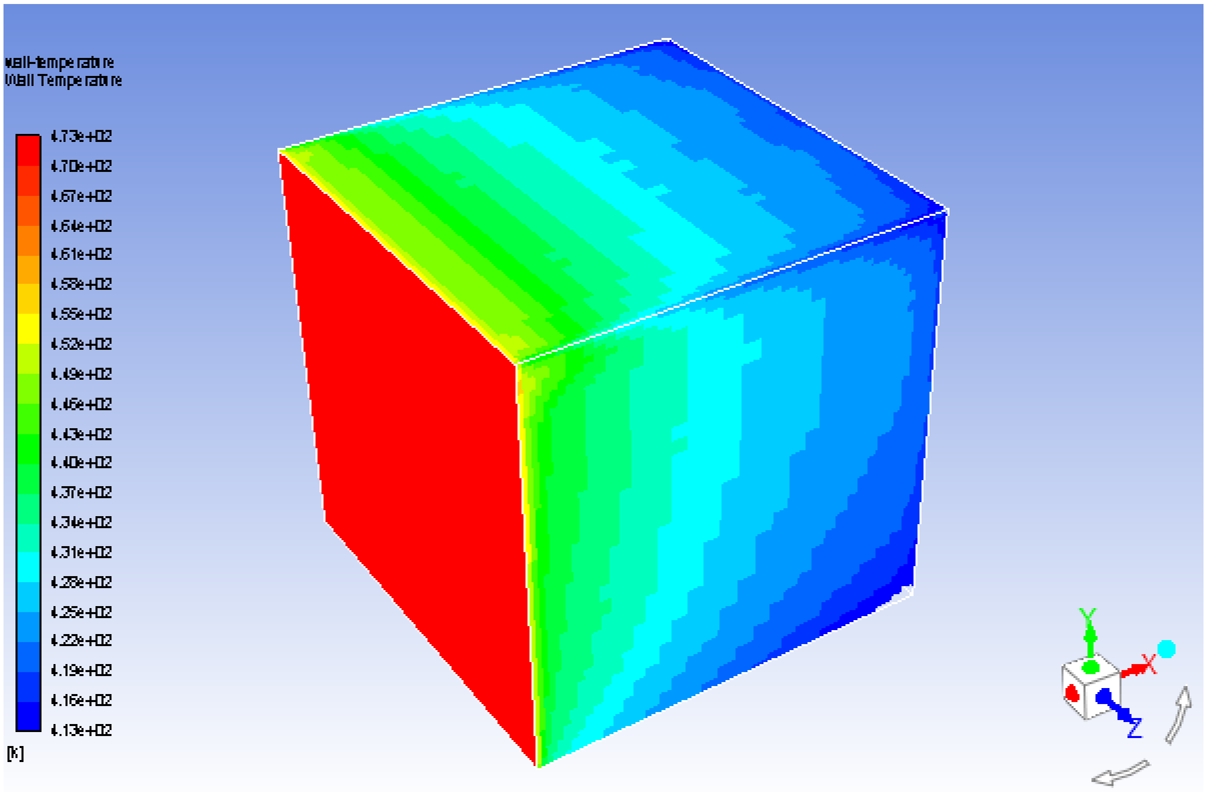
d. Select **Temperature...** and **Wall Temperature** from the **Contours of** drop-down lists.

e. Select all surfaces except **default-interior** and **zz_x_side** in the **Surfaces** selection list.

f. Enter **`413`** K for **Min** and **`473.15`** K for **Max**.

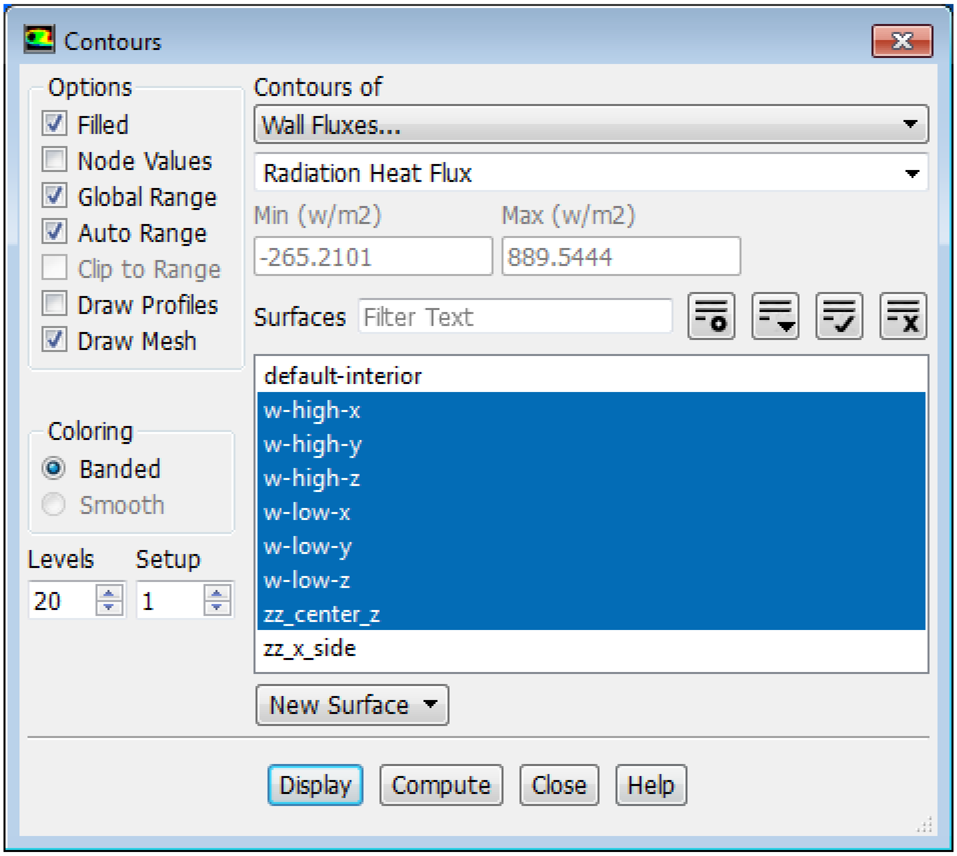
g. Click **Save/Display**, and rotate the view as shown in Figure 7.6: Contours of Wall Temperature (p. 338).

Figure 7.6: Contours of Wall Temperature



6. Display contours of radiation heat flux.

 **Postprocessing** → **Graphics** → **Contours** → **Edit...**



a. Ensure that the **Filled** option is enabled in the **Options** group box.

b. Enable **Auto Range**.

c. Select **Wall Fluxes...** and **Radiation Heat Flux** from the **Contours of** drop-down list.

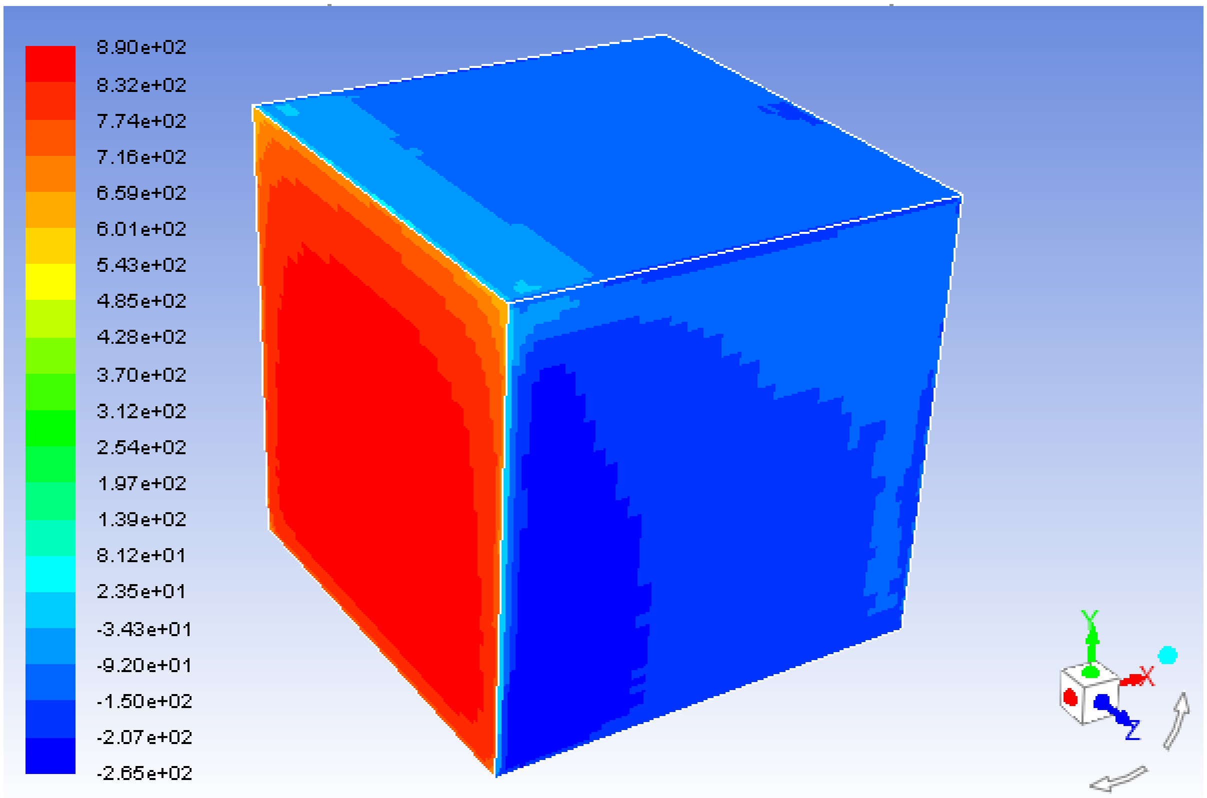
d. Make sure that all surfaces except **default-interior** and **zz_x_side** are selected in the **Surfaces** selection list.

e. Click **Display**.

f. Close the **Contours** dialog box.

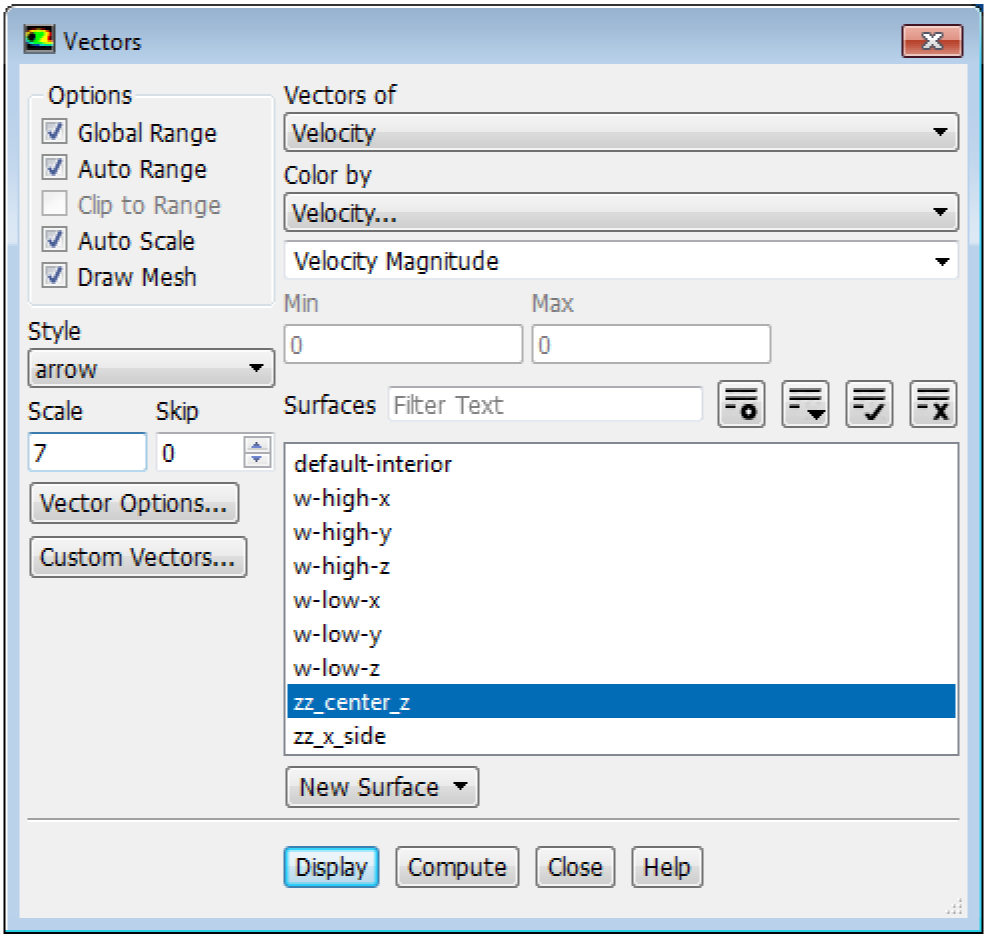
*Figure 7.7: Contours of Radiation Heat Flux (p. 340) shows the radiating wall (**w-low-x**) with positive heat flux and all other walls with negative heat flux.*

Figure 7.7: Contours of Radiation Heat Flux



7. Display vectors of velocity magnitude.

   Postprocessing → **Graphics** → **Vectors** → **Edit...**



a. Retain the default selection of **Velocity** from the **Vectors of** drop-down list.

b. Retain the default selection of **Velocity...** and **Velocity Magnitude** from the **Color by** drop-down lists.

c. Select **zz_center_z** from the **Surfaces** selection list.

d. Enable the **Draw Mesh** option in the **Options** group box to open the **Mesh Display** dialog box.

   i. Ensure that **Outline** is selected from the **Edge Type** list.

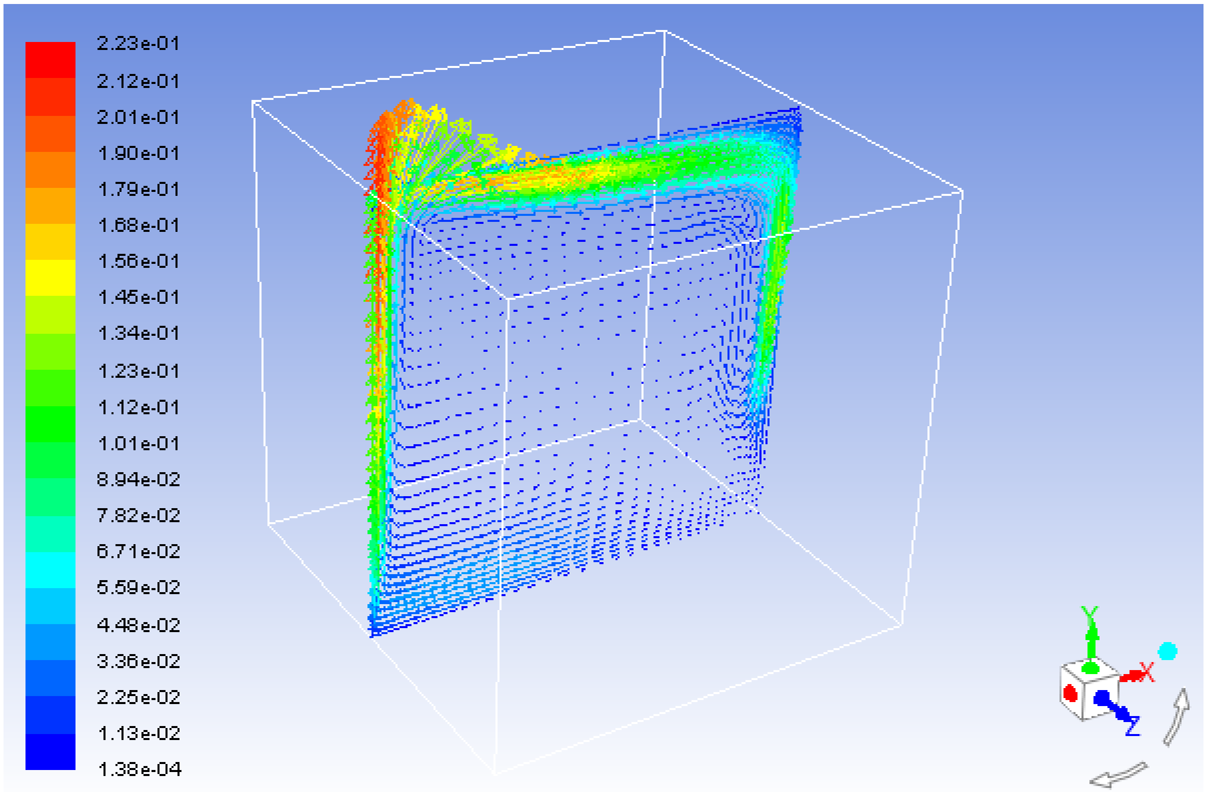
   ii. Click **Display** and close the **Mesh Display** dialog box.

e. Enter 7 for **Scale**.

f. Click **Display** and rotate the view as shown in Figure 7.8: Vectors of Velocity Magnitude (p. 342).

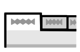
g. Close the **Vectors** dialog box.

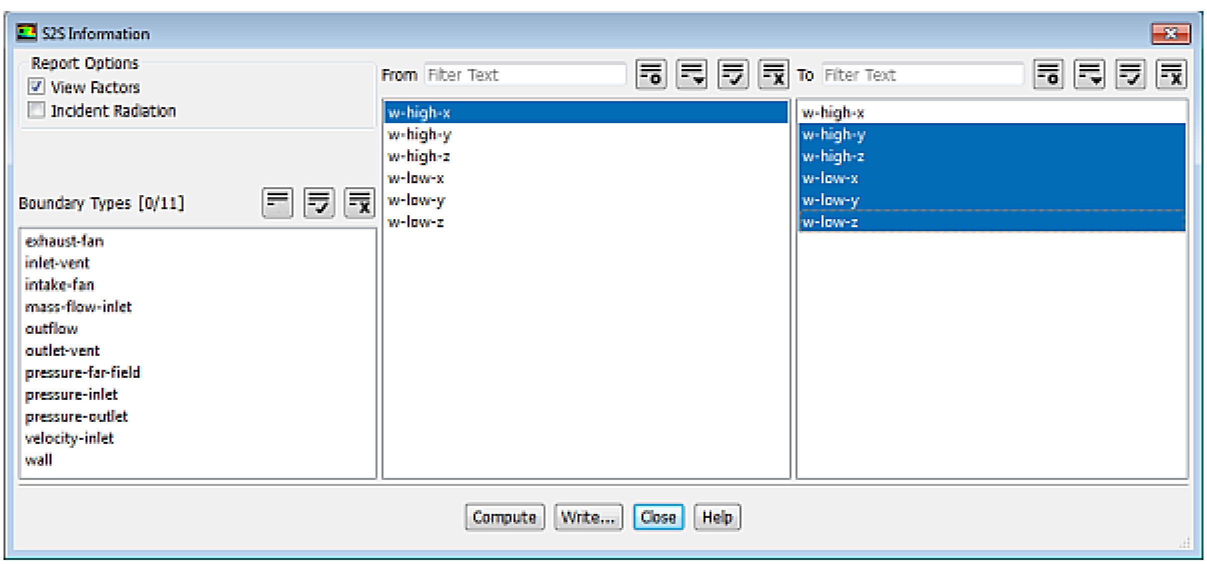
Figure 7.8: Vectors of Velocity Magnitude



8. Compute view factors and radiation emitted from the front wall (**w-high-x**) to all other walls.

   In the **Postprocessing** tab, click **S2S Information...** (**Model Specific** group box).

    **Postprocessing** → **Model Specific** → **S2S Information...**

   a. Ensure that the **View Factors** option is enabled in the **Report Options** group box.

   b. Enable the **Incident Radiation** option.

c. Select **w-high-x** from the **From** selection list.

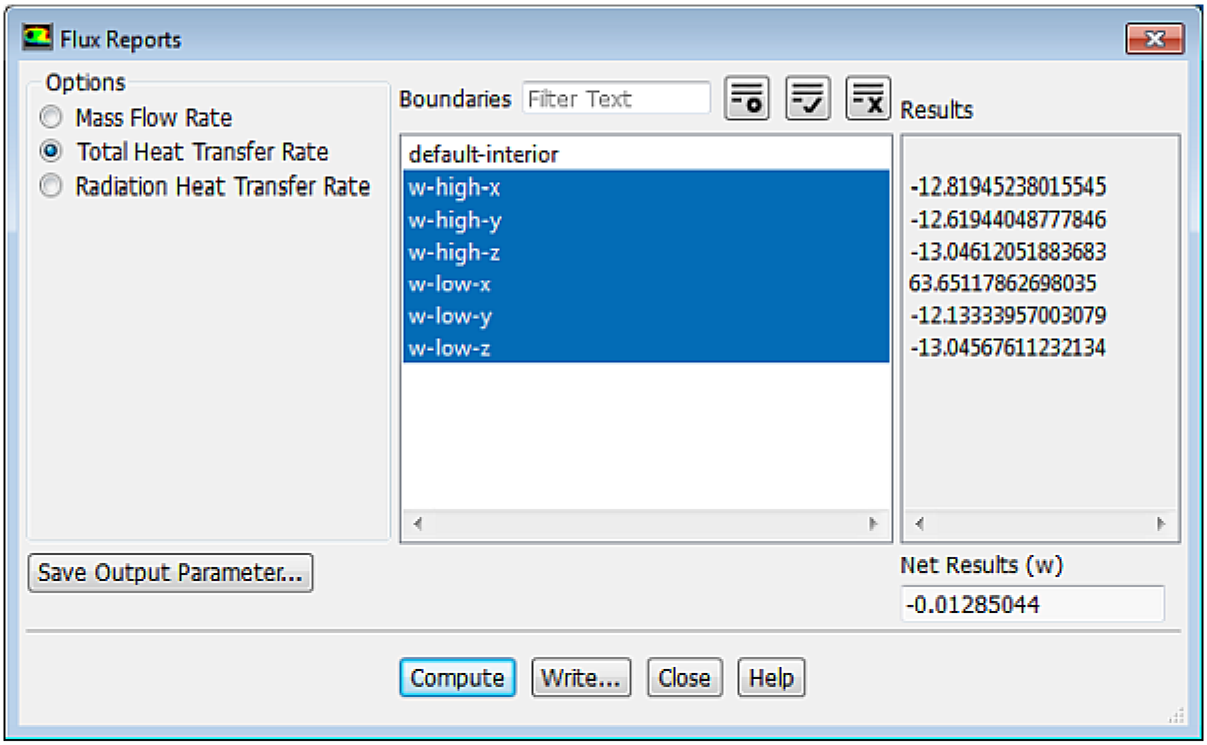
d. Select all zones except **w-high-x** from the **To** selection list.

e. Click **Compute** and close the **S2S Information** dialog box.

The computed values of the view factors and incident radiation are displayed in the console. A view factor of approximately 0.2 for each wall is a good value for the square box.

9. Compute the total heat transfer rate.

Postprocessing → **Reports** → **Fluxes...**



a. Select **Total Heat Transfer Rate** from the **Options** list.

b. Select all boundary zones except **default-interior** from the **Boundaries** selection list.

c. Click **Compute**.

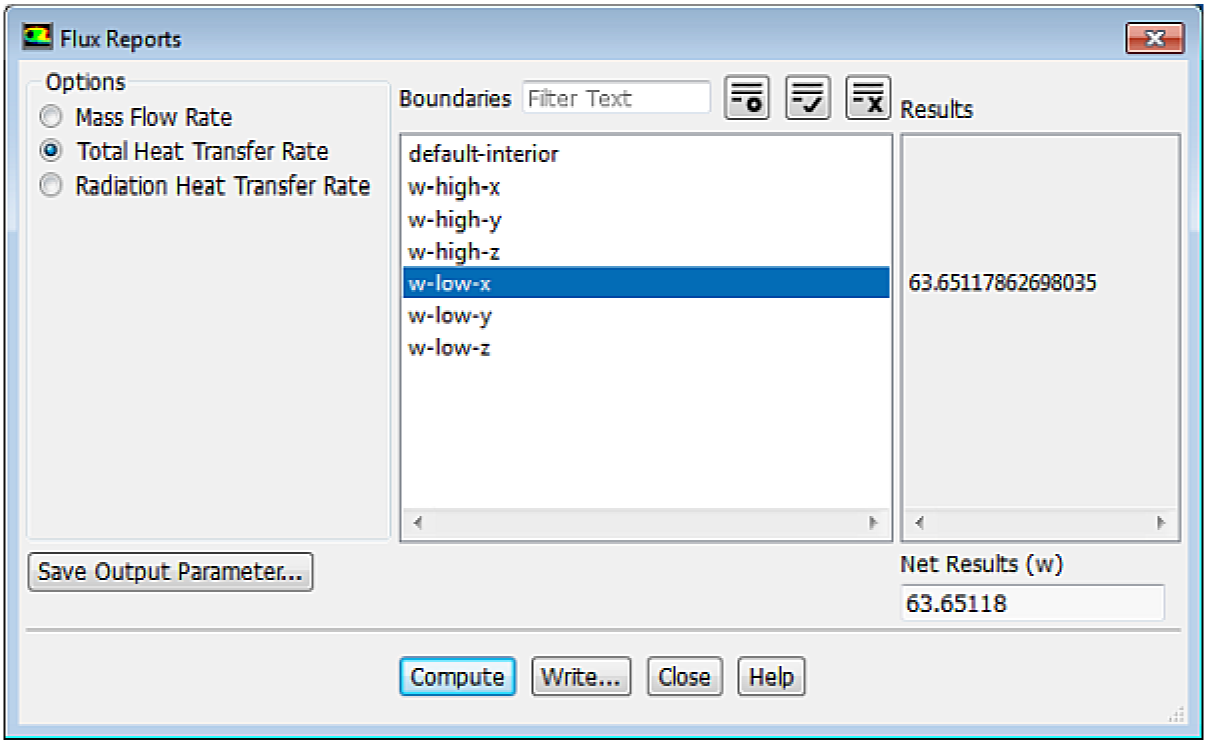
Note

The energy imbalance is approximately 0.004%.

---

10. Compute the total heat transfer rate for **w-low-x**.

Postprocessing → **Reports** → **Fluxes...**



a. Retain the selection of **Total Heat Transfer Rate** from the **Options** list.

b. Deselect all boundary zones and select **w-low-x** from the **Boundaries** selection list.

c. Click **Compute**.

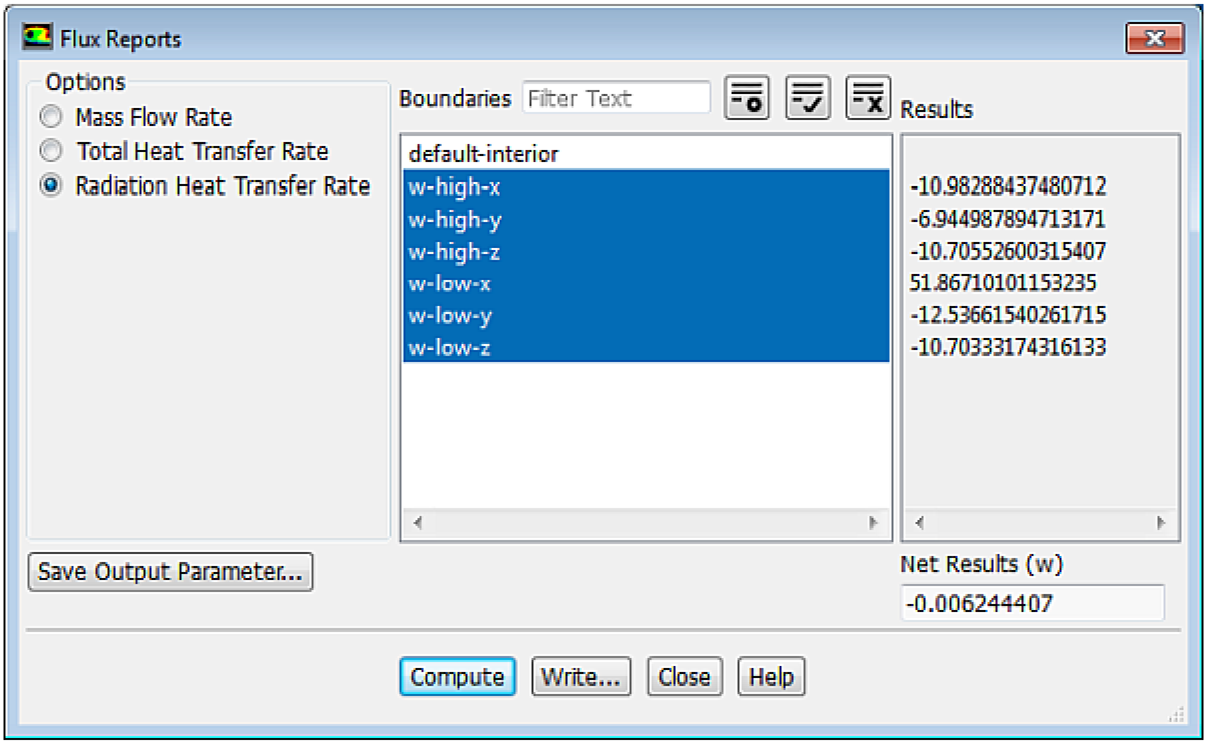
Note

The net heat load is approximately 63 W.

---

11. Compute the radiation heat transfer rate.

Postprocessing → **Reports** → **Fluxes...**



a. Select **Radiation Heat Transfer Rate** from the **Options** list.

b. Select all boundary zones except **default-interior** from the **Boundaries** selection list.

c. Click **Compute**.

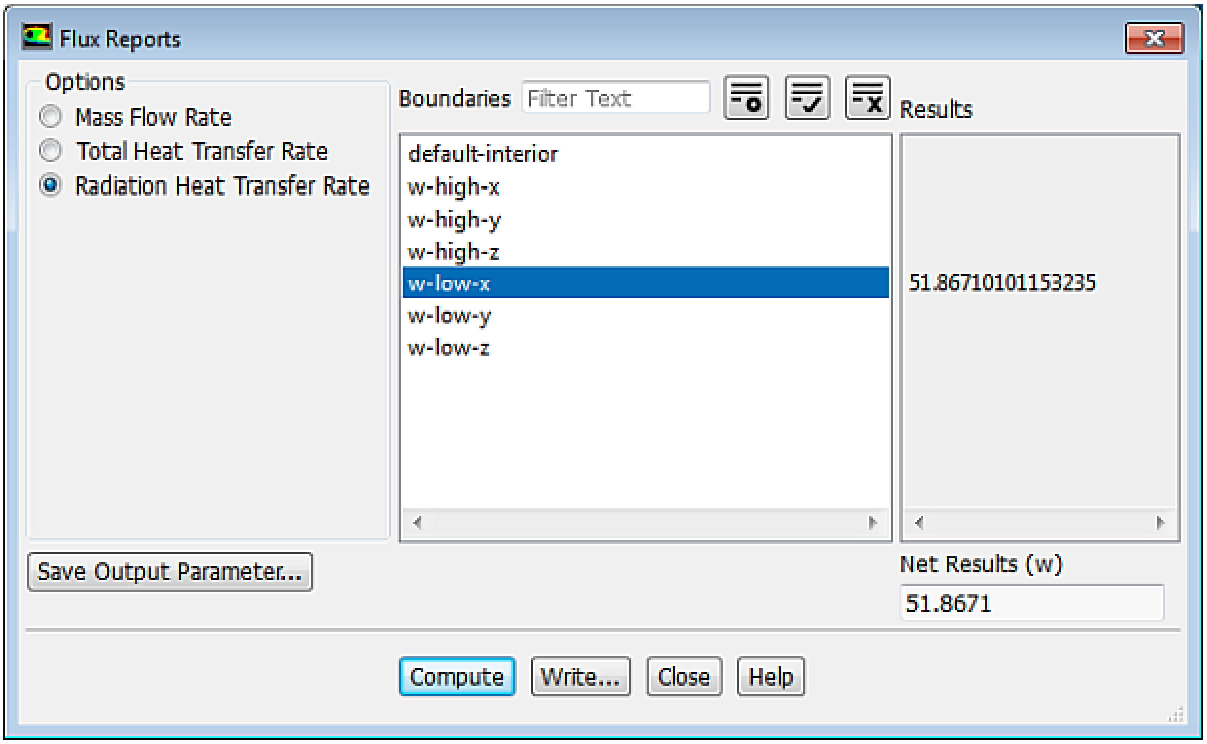
Note

The heat imbalance is approximately -0.007 W.

---

12. Compute the radiation heat transfer rate for **w-low-x**.

Postprocessing → **Reports** → **Fluxes...**



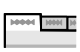
a. Retain the selection of **Radiation Heat Transfer Rate** from the **Options** list.

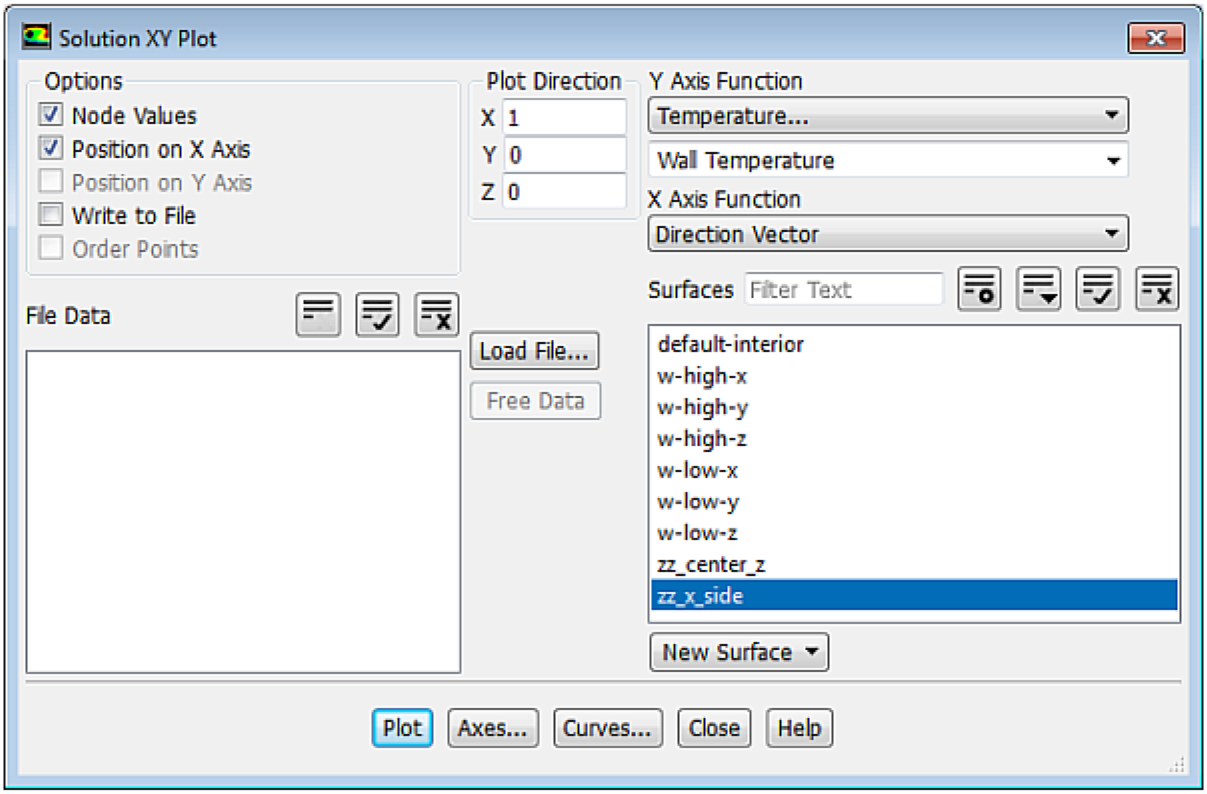
b. Deselect all boundary zones and select **w-low-x** from the **Boundaries** selection list.

c. Click **Compute** and close the **Flux Reports** dialog box.

The net heat load is approximately 51 W. After comparing the total heat transfer rate and radiation heat transfer rate, it can be concluded that radiation is the dominant mode of heat transfer.

13. Display the temperature profile for the side wall.

 **Postprocessing** → **Plots** → **XY Plot** → **Edit...**



a. Select **Temperature...** and **Wall Temperature** from the **Y Axis Function** drop-down lists.

b. Retain the default selection of **Direction Vector** from the **X Axis Function** drop-down list.

c. Select **zz_x_side** from the **Surfaces** selection list.

d. Click **Plot** (Figure 7.9: Temperature Profile Along the Outer Surface of the Box (p. 348)).

e. Enable the **Write to File** option and click the **Write...** button to open the **Select File** dialog box.

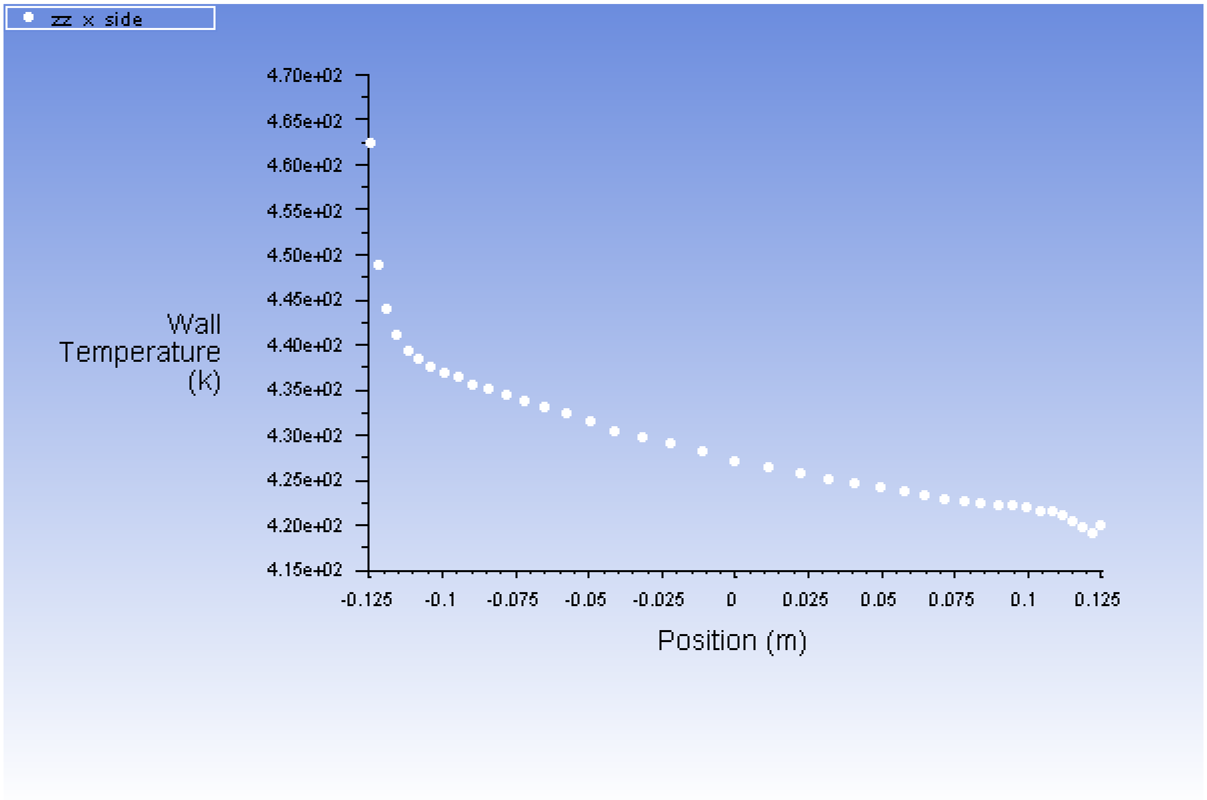
   i. Enter `tp_1.xy` for **XY File**.

   ii. Click **OK** in the **Select File** dialog box.

f. Disable the **Write to File** option.

g. Close the **Solution XY Plot** dialog box.

Figure 7.9: Temperature Profile Along the Outer Surface of the Box



14. Save the case and data files (`rad_b_1.cas.gz` and `rad_b_1.dat.gz`).

File → **Write** → **Case & Data...**

7.4.10. Comparing the Contour Plots after Varying Radiating Surfaces

1. Increase the number of faces per cluster to 10.

Setup → **Models** → **Radiation** **Edit...**

a. Click the **Settings...** button to open the **View Factors and Clustering** dialog box.

i. Enter **`10`** for **Faces per Surface Cluster for Flow Boundary Zones** in the **Manual** group box.

ii. Click **Apply to All Walls**.

iii. Click **OK** to close the **View Factors and Clustering** dialog box.

b. Click the **Compute/Write/Read...** button to open the **Select File** dialog box and to compute the view factors.

Specify a name for the S2S file that will store the cluster and view factor parameters.

i. Enter **`rad_10.s2s.gz`** for **S2S File**.

ii. Click **OK** in the **Select File** dialog box.

c. Click **OK** to close the **Radiation Model** dialog box.

2. Right-click **Solution Initialization** and select **Initialize**.

   Solution → **Solution Initialization** → **Initialize**

3. Start the calculation by requesting 400 iterations.

   Solution → **Run Calculation** → **Calculate**

   The solution will converge in approximately 309 iterations.

4. Save the case and data files (`rad_10.cas.gz` and `rad_10.dat.gz`).

   File → **Write** → **Case & Data...**

5. In a similar manner described in the steps 13.a – 13.g of Postprocessing (p. 333), display the temperature profile for the side wall and write it to a file named `tp_10.xy`.

6. Repeat the procedure, outlined in steps 1 – 5 of this section, for 100, 400, 800, and 1600 faces per surface cluster and save the respective S2S files (for example, `rad_100.s2s.gz`), case and data files (for example, `rad_100.cas.gz`), and temperature profile files (for example, `tp_100.xy`).

7. Display contours of wall temperature for all six cases using the **wall-temperature** definition you created earlier.

   Results → **Graphics** → **Contours** → **wall-temperature** → **Display**

Figure 7.10: Contours of Wall Temperature: 1 Face per Surface Cluster

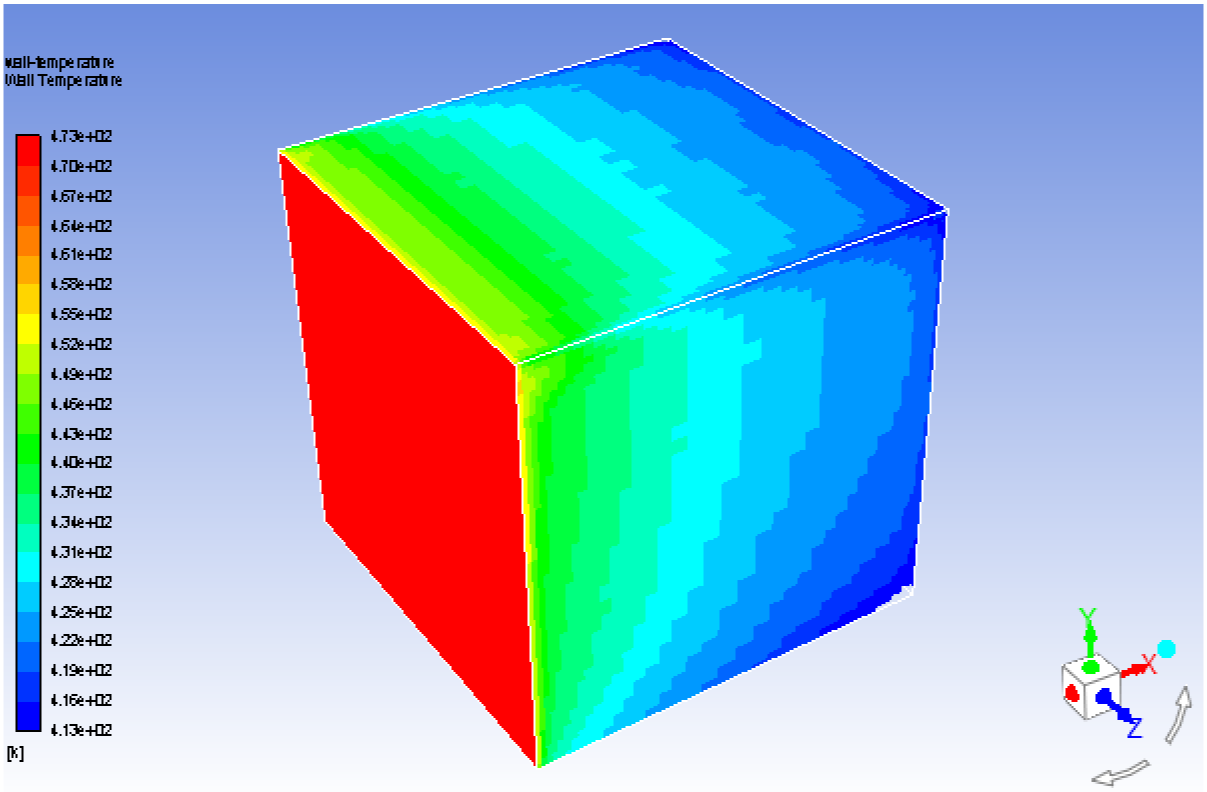


Figure 7.11: Contours of Wall Temperature: 10 Faces per Surface Cluster

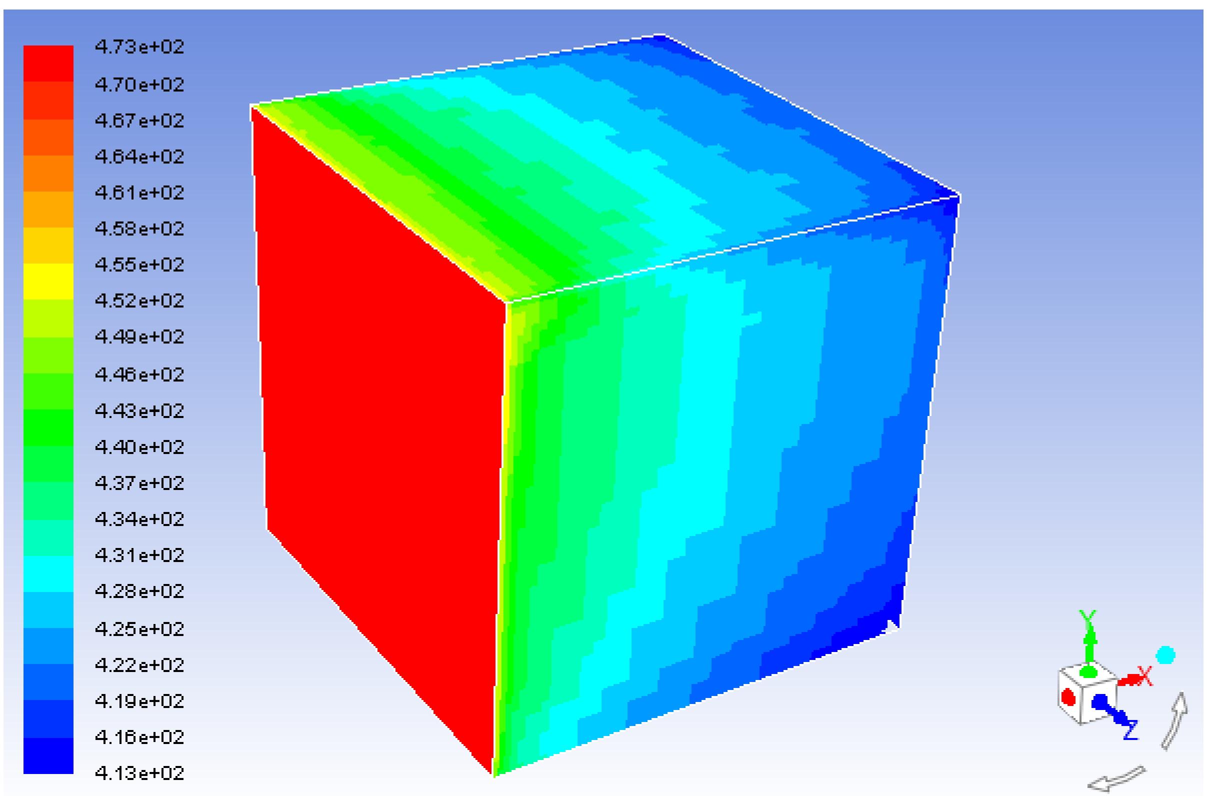


Figure 7.12: Contours of Wall Temperature: 100 Faces per Surface Cluster

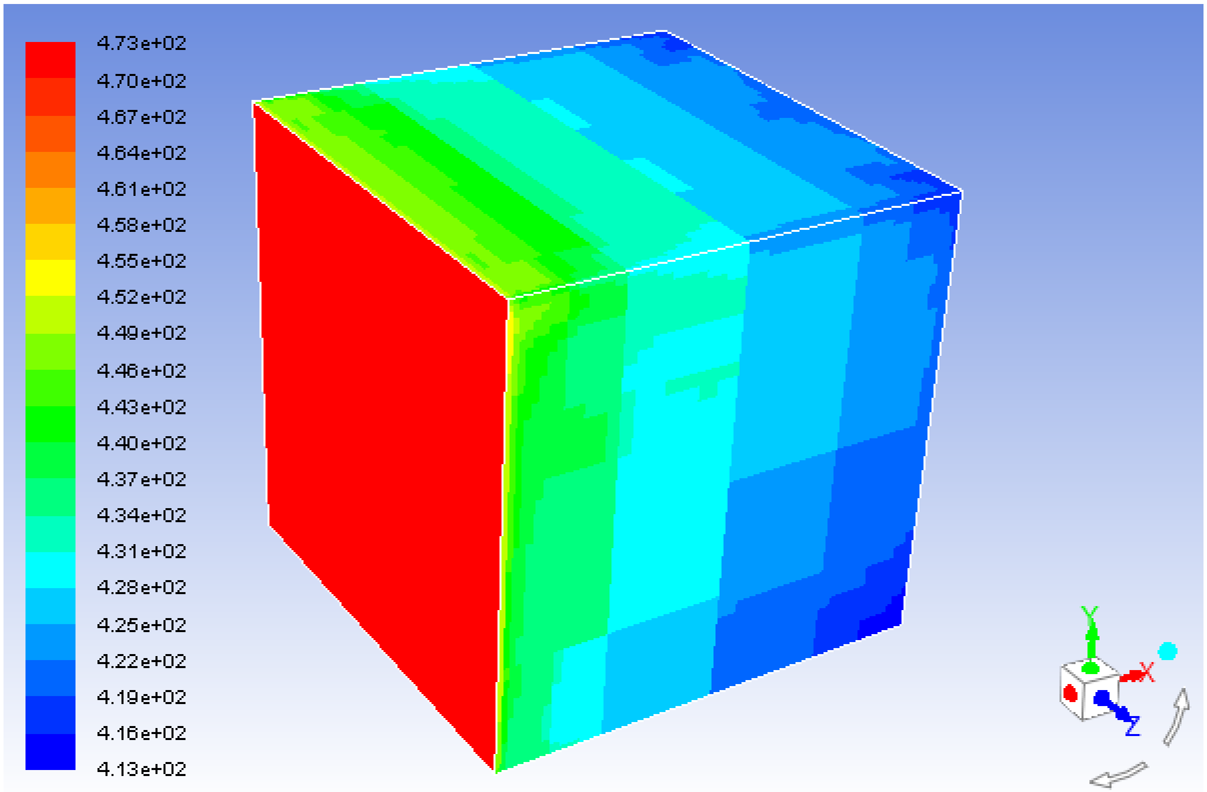


Figure 7.13: Contours of Wall Temperature: 400 Faces per Surface Cluster

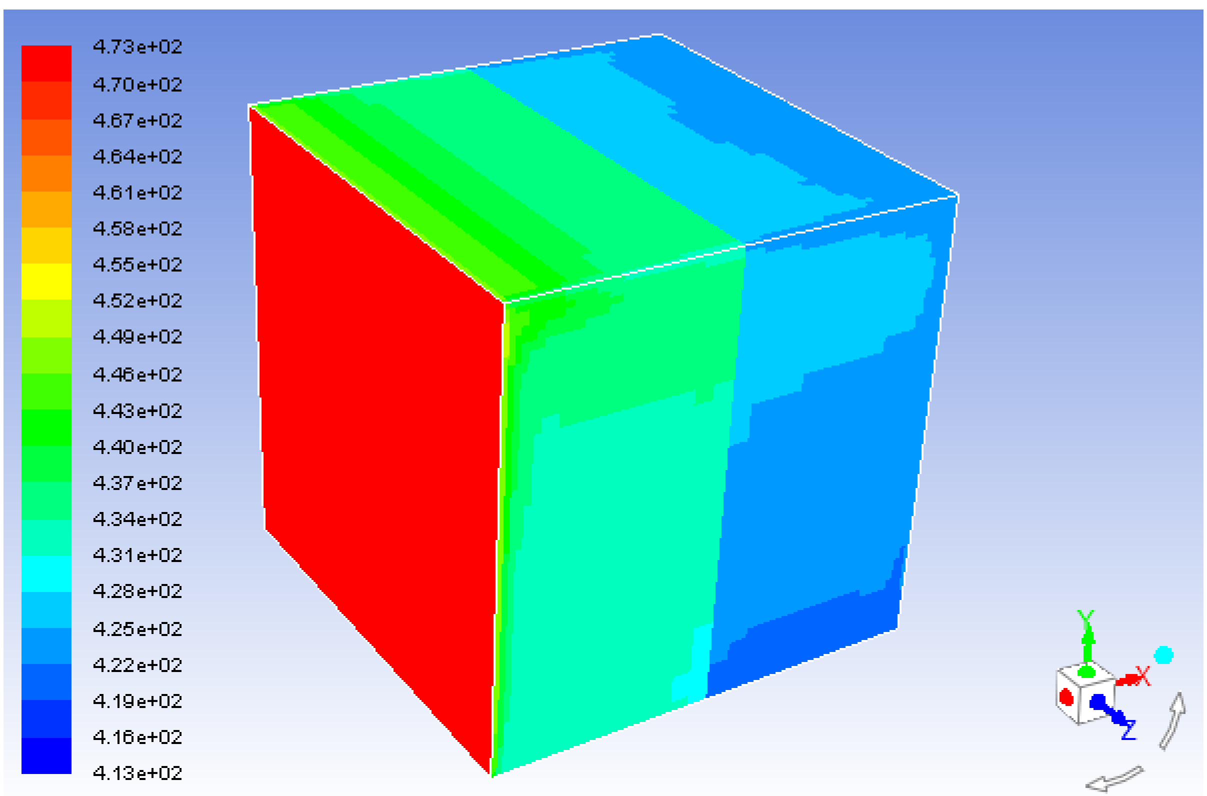


Figure 7.14: Contours of Wall Temperature: 800 Faces per Surface Cluster

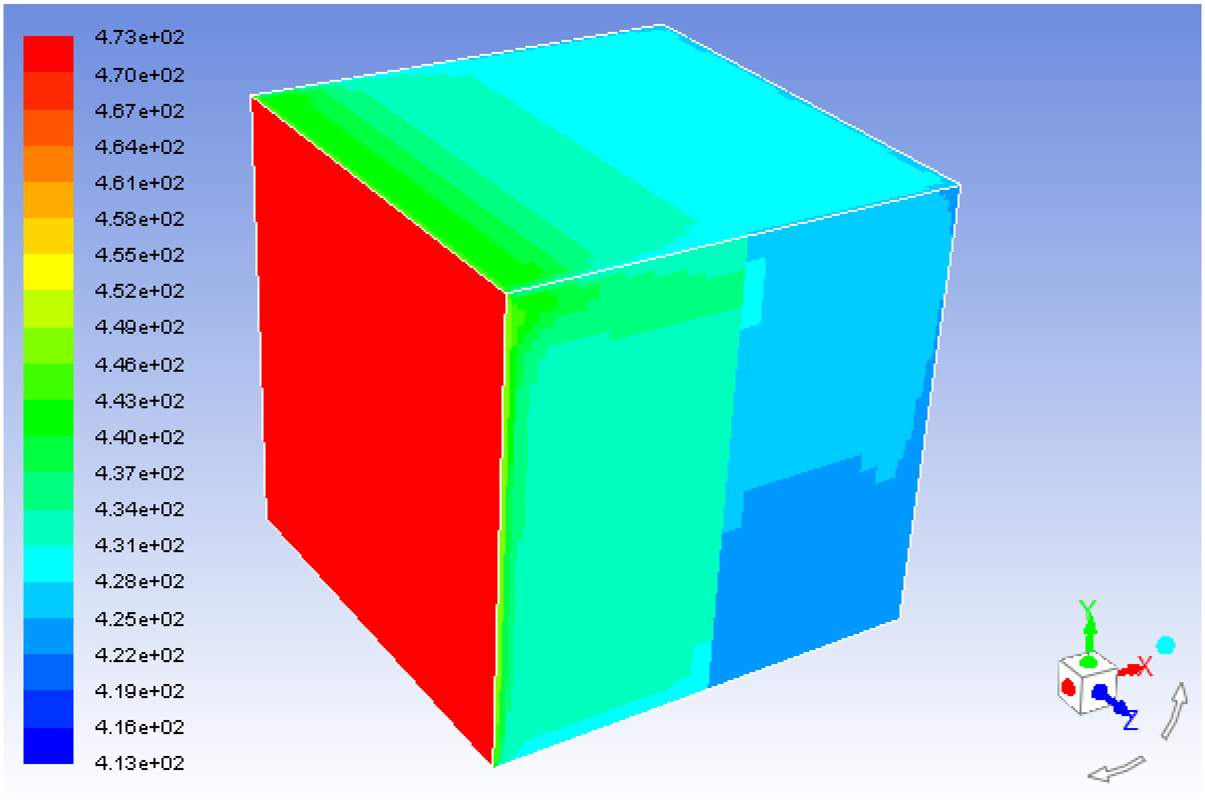
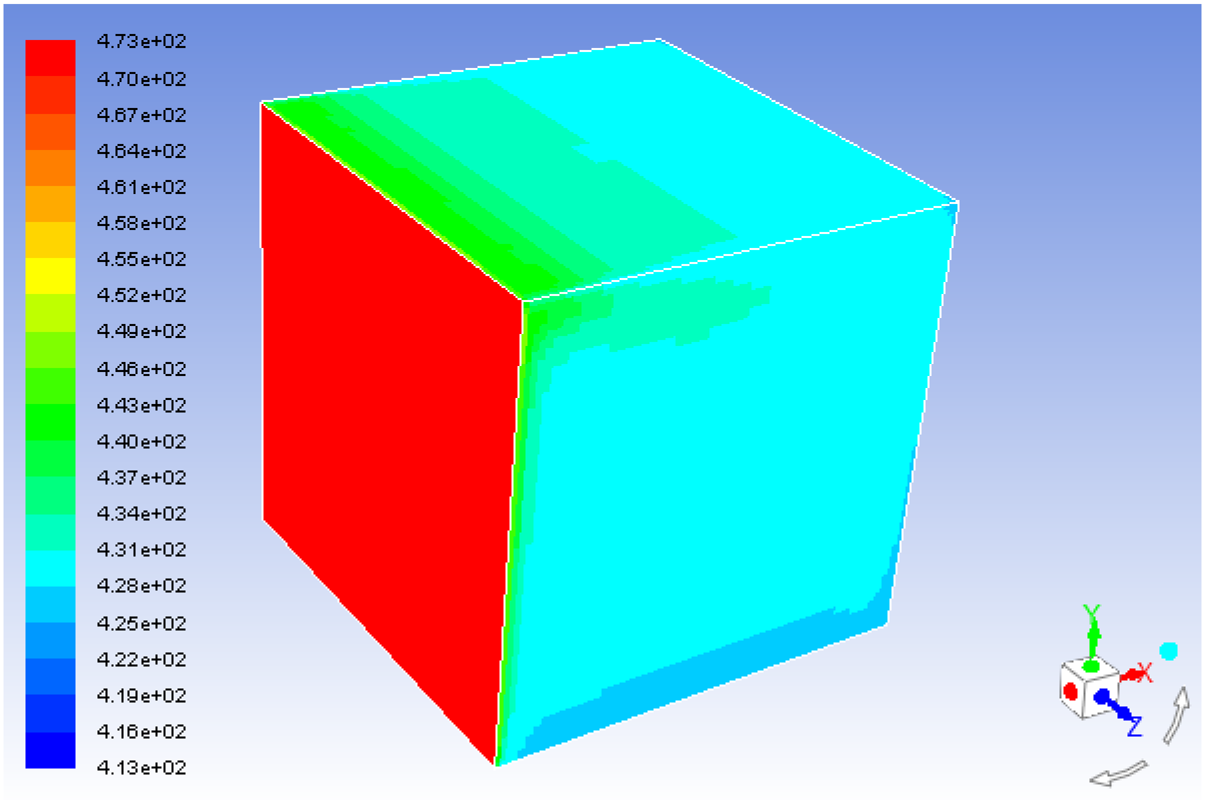
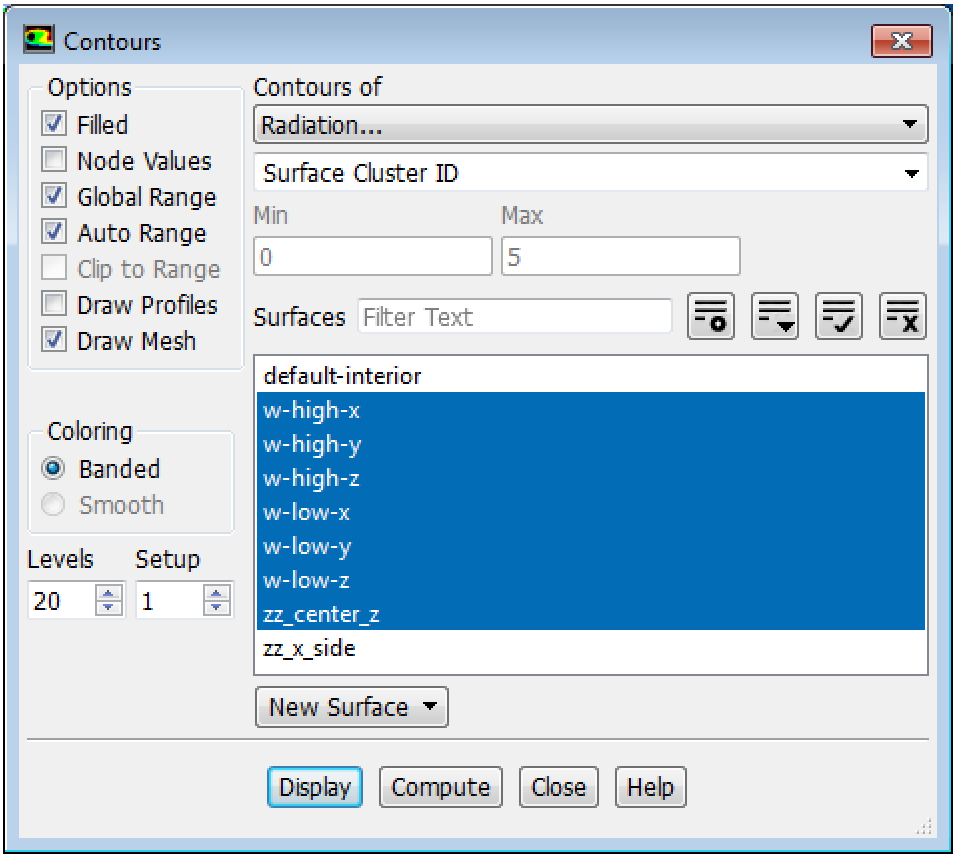


Figure 7.15: Contours of Wall Temperature: 1600 Faces per Surface Cluster



8. Display contours of surface cluster ID for 1600 faces per surface cluster (Figure 7.16: Contours of **Surface Cluster ID**—1600 Faces per Surface Cluster (FPSC) (p. 354)).

Postprocessing → **Graphics** → **Contours** → **Edit...**



a. Ensure that the **Filled** option is enabled in the **Options** group box.

b. Ensure that the **Node Values** option is disabled.

c. Enable the **Auto Range** option.

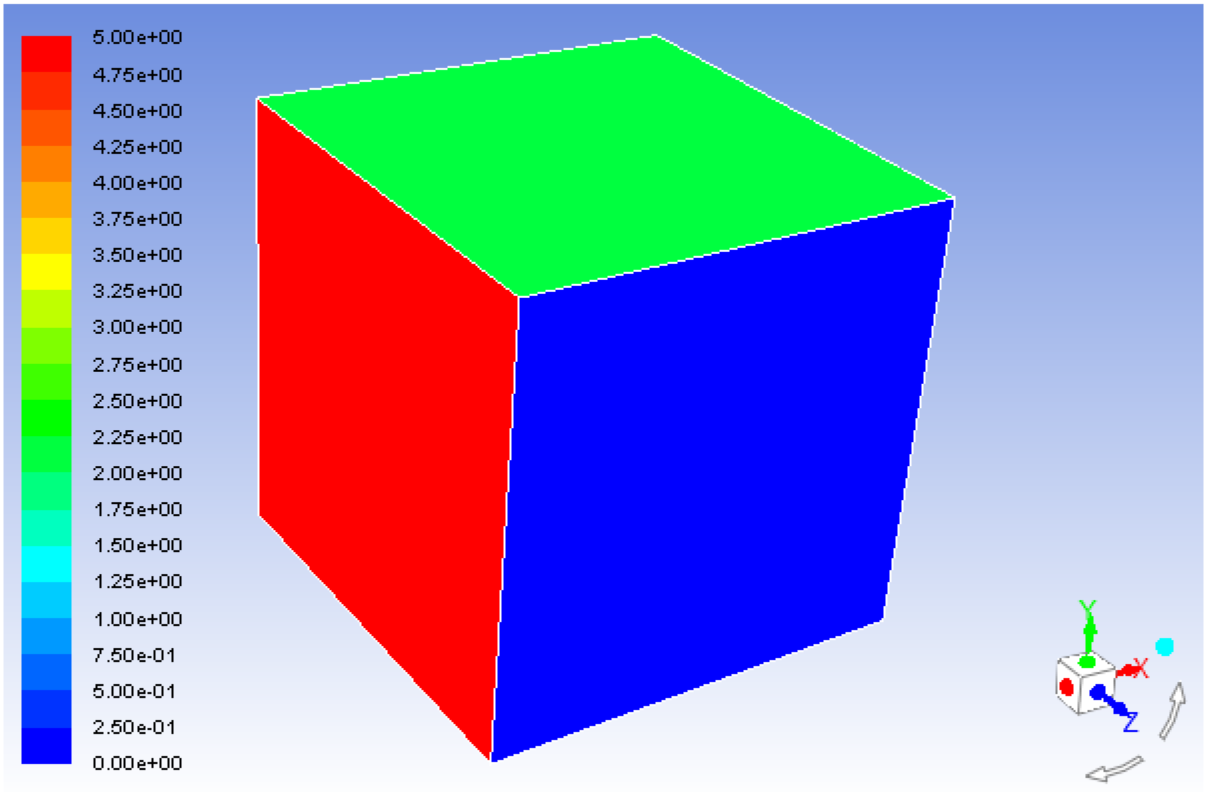
d. Select **Radiation...** and **Surface Cluster ID** from the **Contours of** drop-down lists.

e. Ensure that all surfaces except **default-interior** and **zz_x_side** are selected in the **Surfaces** selection list.

f. Click **Display** and rotate the view as shown in Figure 7.16: Contours of **Surface Cluster ID**—1600 Faces per Surface Cluster (FPSC) (p. 354).

g. Close the **Contours** dialog box.

Figure 7.16: Contours ofSurface Cluster ID—1600 Faces per Surface Cluster (FPSC)



9. Read `rad_400.cas.gz` and `rad_400.dat.gz` and, in a similar manner to the previous step, display contours of surface cluster ID (Figure 7.17: Contours of **Surface Cluster ID**—400 FPSC (p. 355)).

Figure 7.17: Contours of Surface Cluster ID—400 FPSC

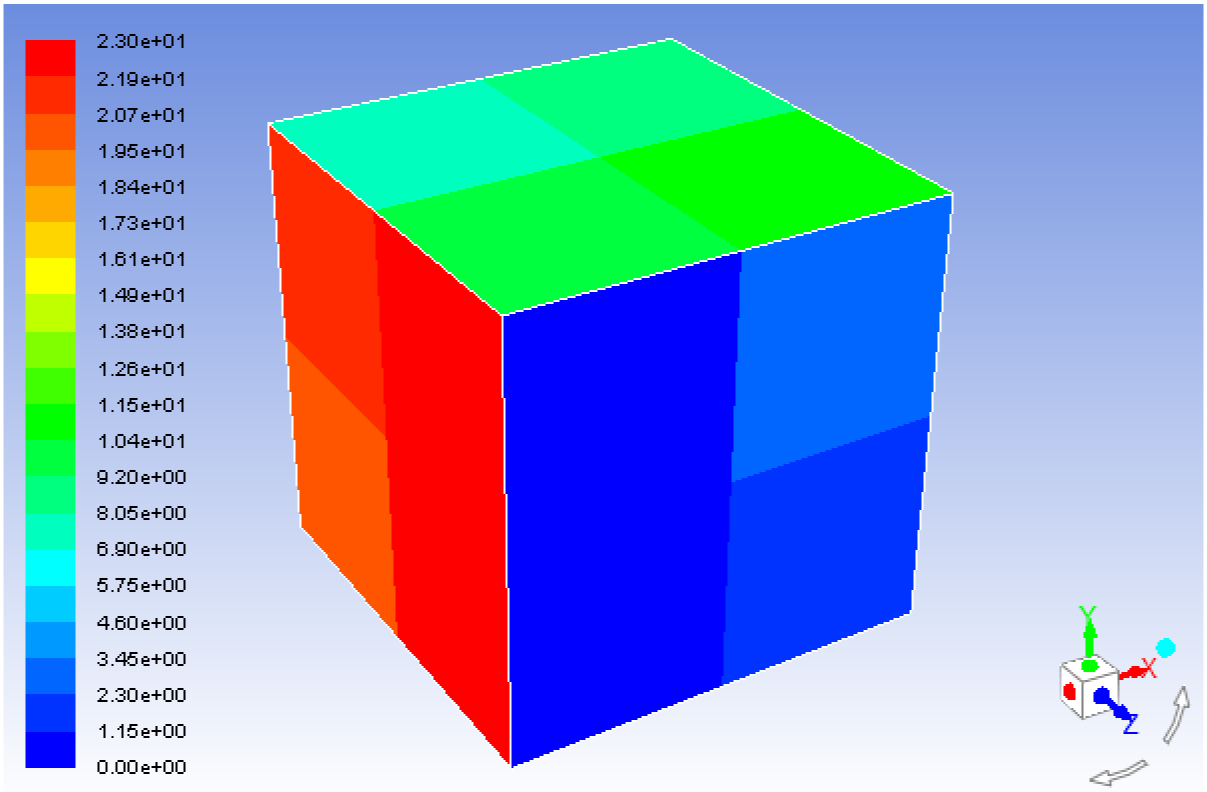


*Figure 7.17: Contours of **Surface Cluster ID**—400 FPSC (p. 355) shows contours of **Surface Cluster ID** for 400 FPSC. This case shows better clustering compared to all of the other cases.*

10. Create a plot that compares the temperature profile plots for 1, 10, 100, 400, 800, and 1600 FPSC.

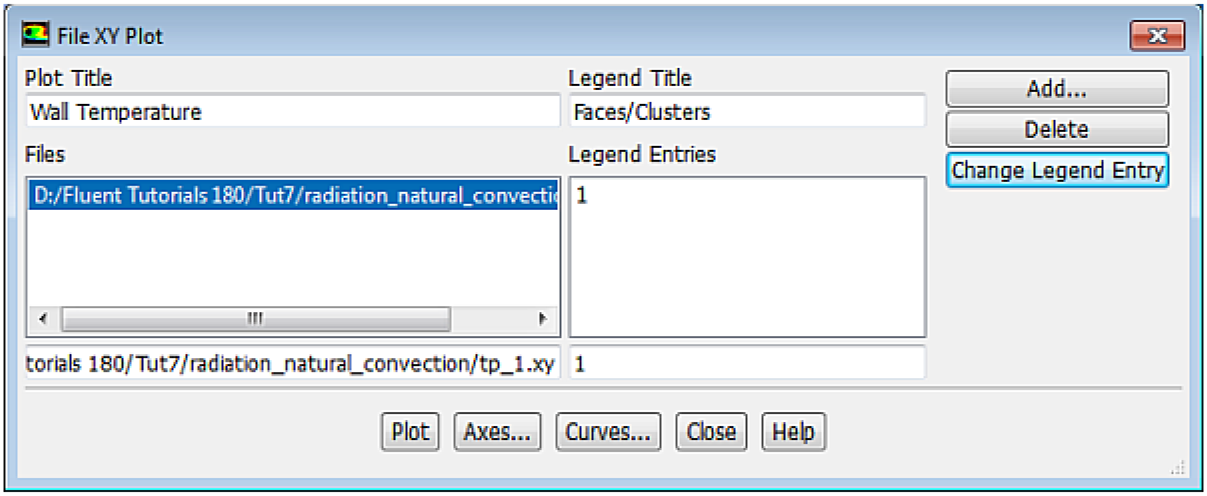
    Postprocessing → **Plots** → **File...**

    a. Click the **Add...** button to open the **Select File** dialog box.

        i. Select the file `tp_1.xy` that you created in step 11 of Postprocessing (p. 333).

        ii. Click **OK** to close the **Select File** dialog box.

    b. Change the legend entry for the data series.



i. Enter **`Faces/Cluster`** in the **Legend Title** text box.

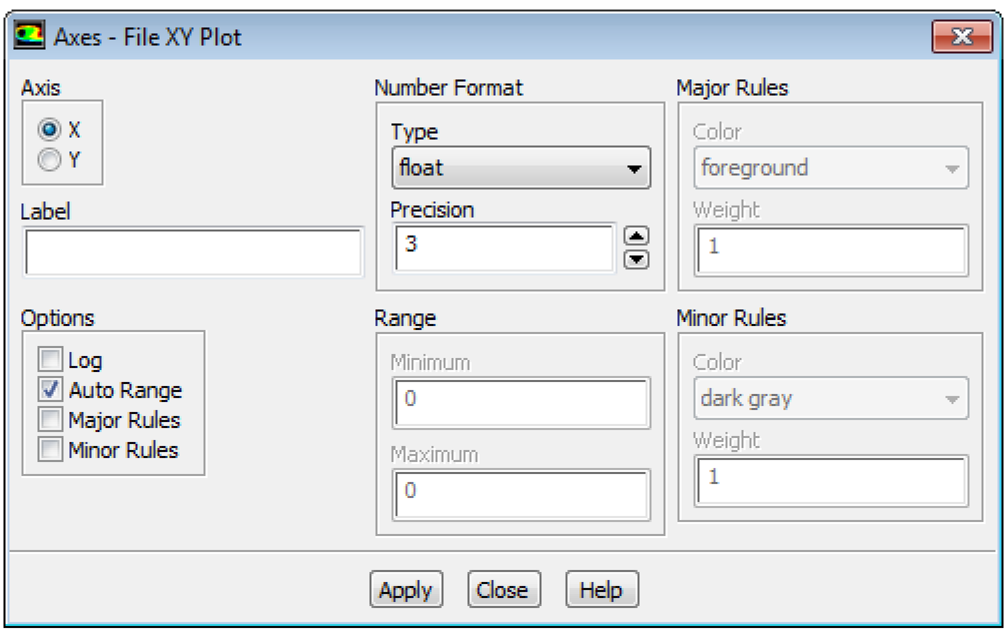
ii. Enter **1** in the text box to the left of the **Change Legend Entry** button.

iii. Click **Change Legend Entry**.

ANSYS Fluent will update the Legend Entry text for the file `tp_1.xy`.

c. Load the files `tp_10.xy`, `tp_100.xy`, `tp_400.xy`, `tp_800.xy`, and `tp_1600.xy` and change their legend entries accordingly, in a manner similar to the previous two steps (a and b).

d. Click the **Axes...** button to open the **Axes** dialog box.



i. Ensure **X** is selected from the **Axis** list.

ii. Enter **3** for **Precision** in the **Number Format** group box and click **Apply**.

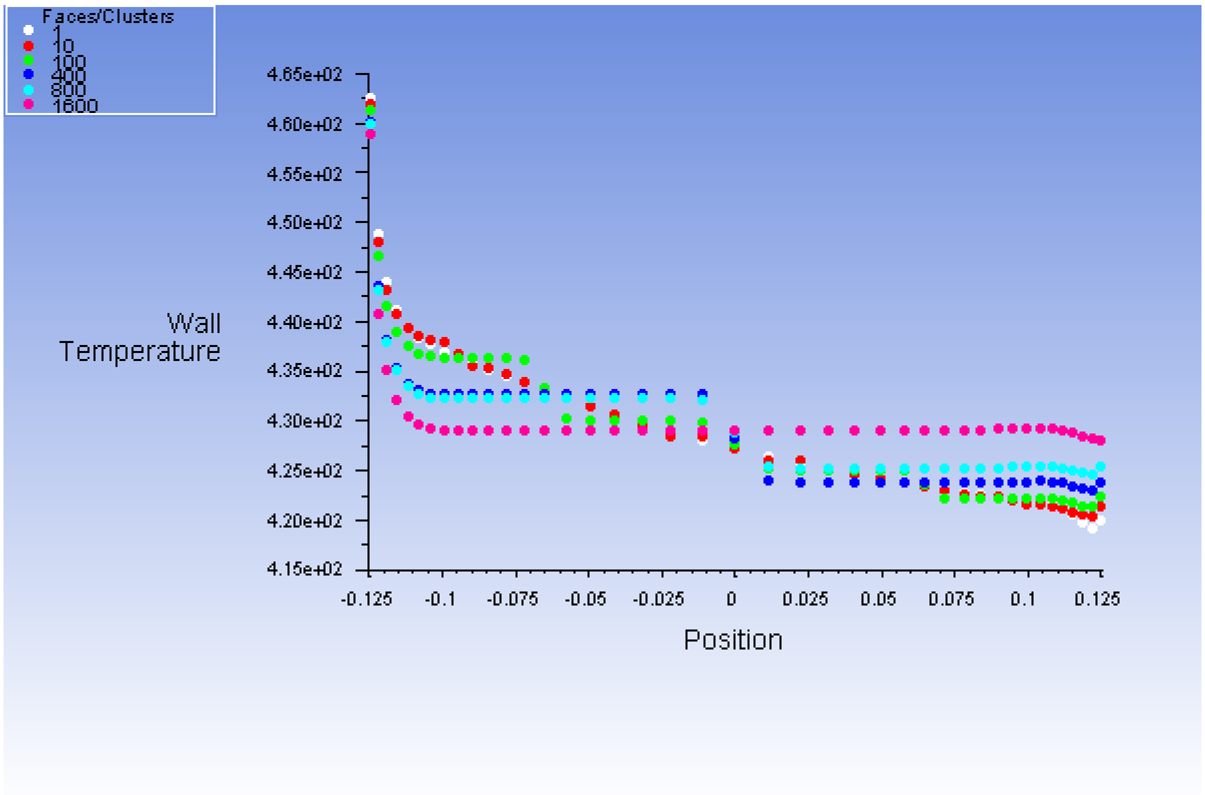
iii. Select **Y** from the **Axis** list.

iv. Enter **2** for **Precision** and click **Apply**.

v. Close the **Axes** dialog box.

e. Click **Plot** (Figure 7.18: A Comparison of Temperature Profiles along the Outer Surface of the Box (p. 357)) and close the **File XY Plot** dialog box.

Figure 7.18: A Comparison of Temperature Profiles along the Outer Surface of the Box



7.4.11. S2S Definition, Solution, and Postprocessing with Partial Enclosure

As mentioned previously, when the S2S model is used, you also have the option to define a "partial enclosure"; that is, you can disable the view factor calculation for walls with negligible emission/absorption, or walls that have uniform temperature. Even though the view factor will not be computed for these walls, they will still emit radiation at a fixed temperature called the "partial enclosure temperature". The main advantage of this is to speed up the view factor and the radiosity calculation.

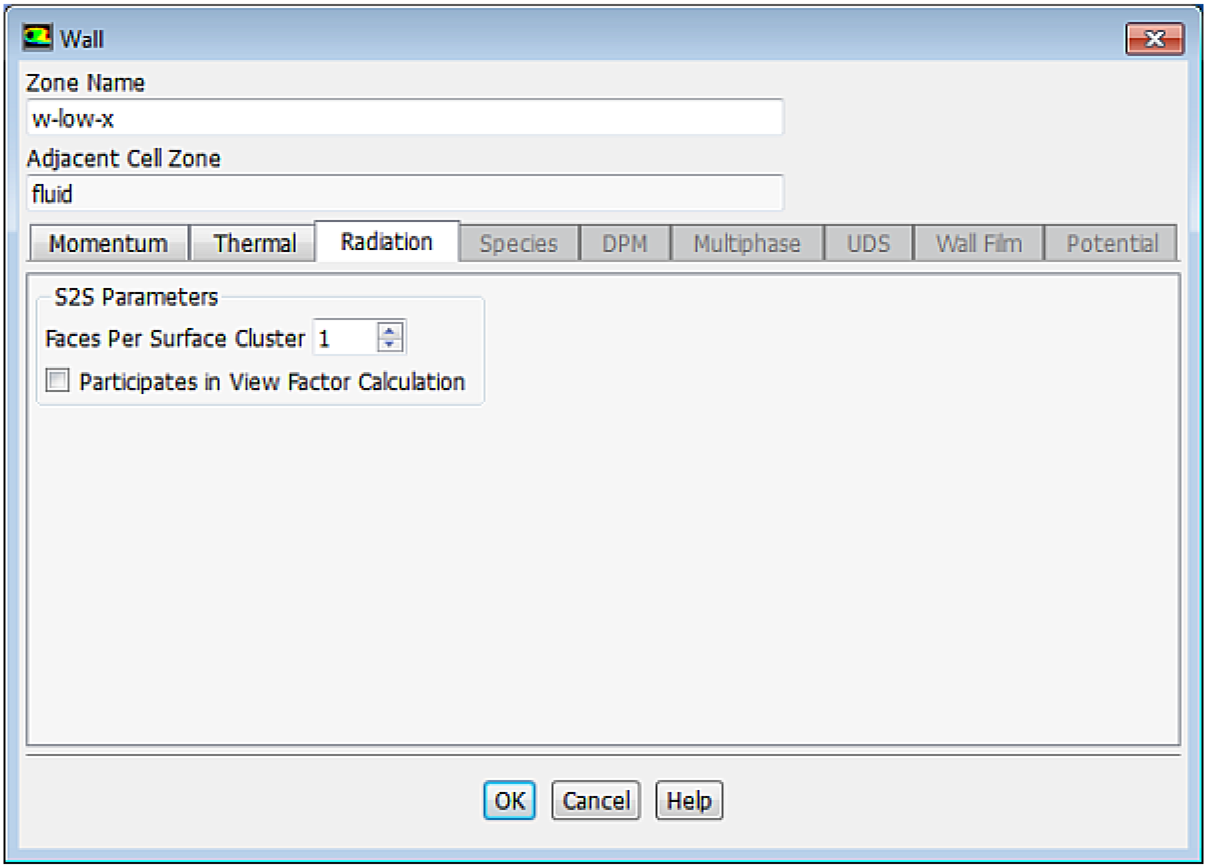
*In the steps that follow, you will specify the radiating wall (**w-low-x**) as a boundary zone that is not participating in the S2S radiation model. Consequently, you will specify the partial enclosure temperature for the wall. Note that the partial enclosure option may not yield accurate results in cases that have multiple wall boundaries that are not participating in S2S radiation and that each have different temperatures. This is because a single partial enclosure temperature is applied to all of the non-participating walls.*

1. Read the case file saved previously for the S2S model (`rad_b_1.cas.gz`).

   File → Read → Case...

2. Set the partial enclosure parameters for the S2S model.

   Setup → Boundary Conditions → w-low-x → Edit...



a. Click the **Radiation** tab.

b. Disable the **Participates in View Factor Calculation** option in the **S2S Parameters** group box.

c. Click **OK** to close the **Wall** dialog box.

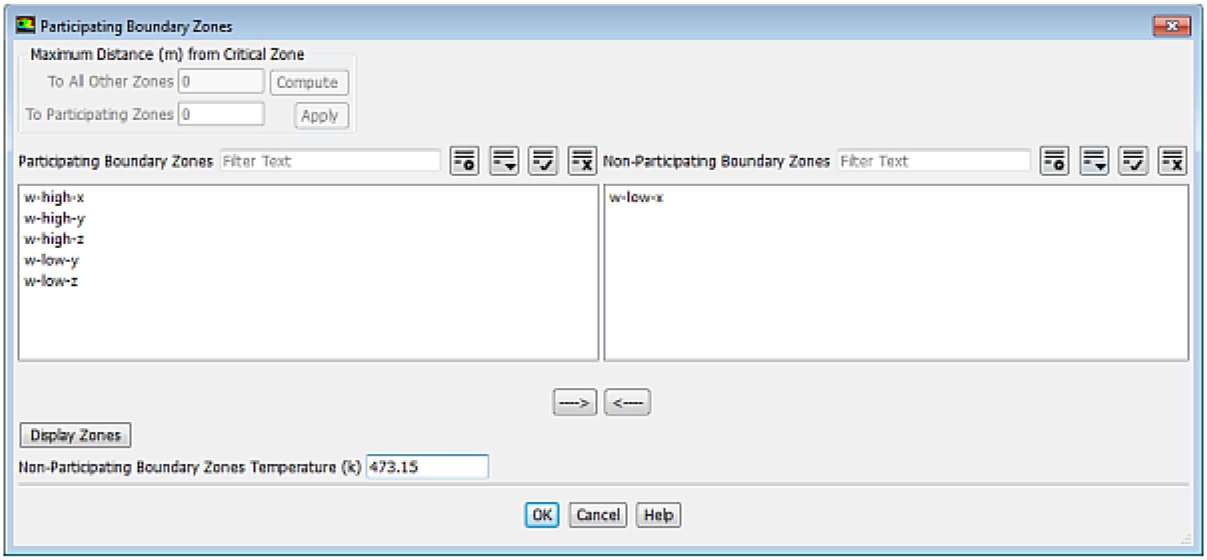
*Click **OK** to close the dialog box informing you that you must recompute viewfactors.*

3. Compute the view factors for the S2S model.

Setup → **Models** → **Radiation** **Edit...**

a. Click the **Settings...** button to open the **View Factors and Clustering** dialog box.

b. Click the **Select...** button to open the **Participating Boundary Zones** dialog box.



i. Enter **`473.15`** K for **Non-Participating Boundary Zones Temperature**.

ii. Click **OK** to close the **Participating Boundary Zones** dialog box.

*Click **OK** to close the dialog box informing you that you must recompute viewfactors.*

c. Click **OK** to close the **View Factors and Clustering** dialog box.

d. Click the **Compute/Write/Read...** button to open the **Select File** dialog box and to compute the view factors.

The view factor file will store the view factors for the radiating surfaces only. This may help you control the size of the view factor file as well as the memory required to store view factors in ANSYS Fluent. Furthermore, the time required to compute the view factors will be reduced, as only the view factors for radiating surfaces will be calculated.

i. Enter **`rad_partial.s2s.gz`** for **S2S File**.

ii. Click **OK** in the **Select File** dialog box.

e. Click **OK** to close the **Radiation Model** dialog box.

4. Right-click **Solution Initialization** and select **Initialize**.

Solution → Solution Initialization Initialize

5. Start the calculation by requesting 400 iterations.

Solution → Run Calculation Calculate

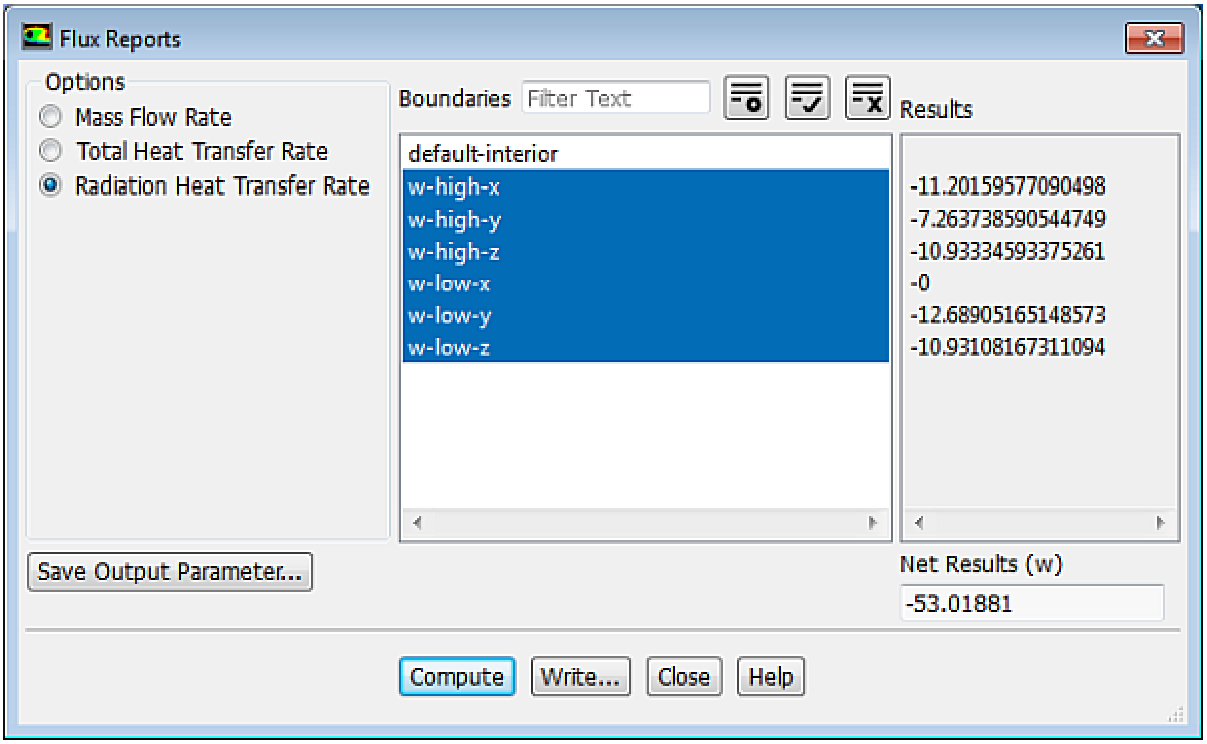
The solution will converge in approximately 300 iterations.

6. Save the case and data files (`rad_partial.cas.gz` and `rad_partial.dat.gz`).

File → Write → Case & Data...

7. Compute the radiation heat transfer rate.

   Postprocessing → **Reports** → **Fluxes...**



   a. Ensure that **Radiation Heat Transfer Rate** is selected from the **Options** list.

   b. Select all boundary zones except **default-interior** from the **Boundaries** selection list.

   c. Click **Compute** and close the **Flux Reports** dialog box.

   *The **Flux Reports** dialog box does not report any heat transfer rate for the radiating wall (**w-low-x**), because you specified that it not participate in the view factor calculation. The remaining walls report similar rates to those obtained in step 9 of Postprocessing (p. 333), indicating that in this case the use of a partial enclosure saved computation time without significantly affecting the results.*

8. Compare the temperature profile for the side wall to the profile saved in `tp_1.xy`.

   Postprocessing → **Plots** → **XY Plot** → **Edit...**

   a. Display the temperature profile for the side wall, **zz_x_side**, and write it to a file named `tp_partial.xy`, in a manner similar to the instructions shown in step 13 of Postprocessing (p. 333).

   b. Click **Load File...** to open the **Select File** dialog box.

      i. Select **tp_1.xy**.

      ii. Click **OK** to close the **Select File** dialog box.

   c. Click **Plot**.

   d. Close the **Solution XY Plot** dialog box.

Figure 7.19: Wall Temperature Profile Comparison

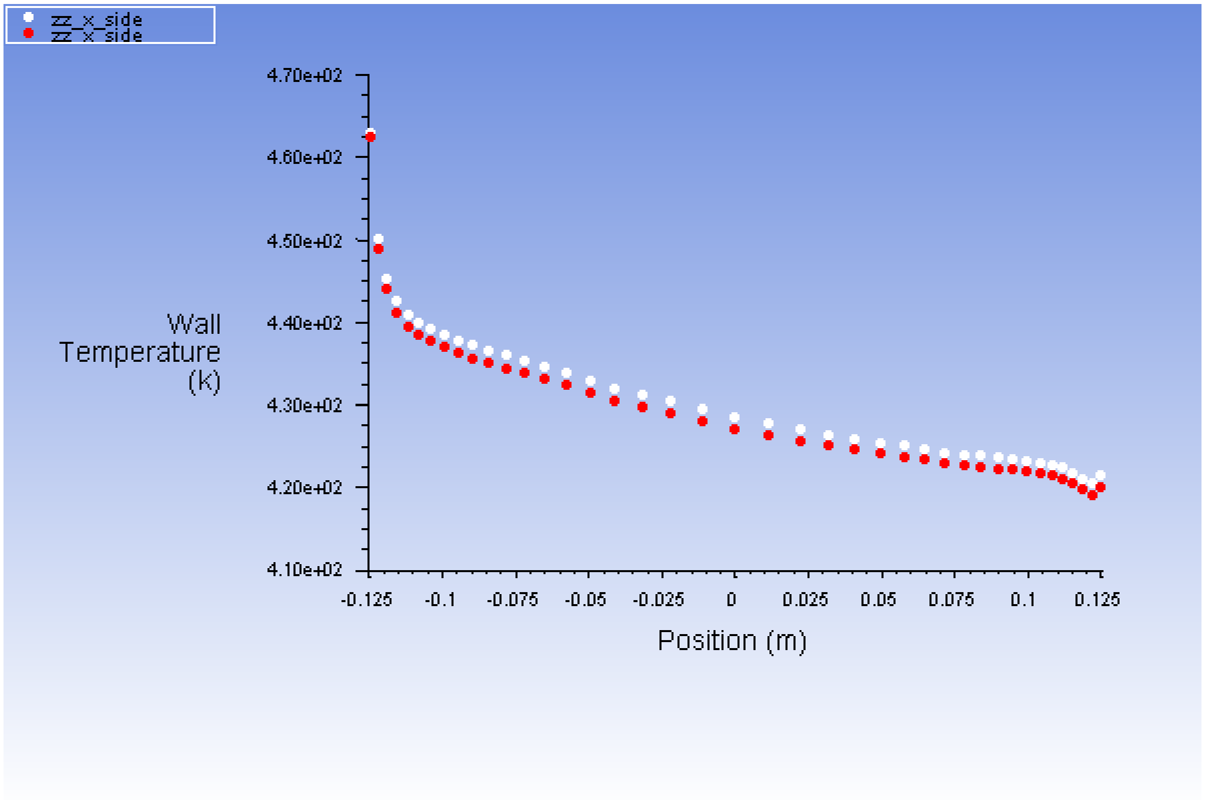


Figure 7.19: Wall Temperature Profile Comparison (p. 361) further confirms that the use of a partial enclosure did not significantly affect the results.

7.5. Summary

In this tutorial you studied combined natural convection and radiation in a three-dimensional square box and compared how varying the settings of the surface-to-surface (S2S) radiation model affected the results. The S2S radiation model is appropriate for modeling the enclosure radiative transfer without participating media, whereas the methods for participating radiation may not always be efficient.

For more information about the surface-to-surface (S2S) radiation model, see the Fluent User's Guide.

7.6. Further Improvements

This tutorial guides you through the steps to reach an initial solution. You may be able to obtain a more accurate solution by using an appropriate higher-order discretization scheme and by adapting the mesh. Mesh adaption can also ensure that the solution is independent of the mesh. These steps are demonstrated in Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121).

Chapter 8: Modeling Flow Through Porous Media

This tutorial is divided into the following sections:

8.1. Introduction

Many industrial applications such as filters, catalyst beds, and packing, involve modeling the flow through porous media. This tutorial illustrates how to set up and solve a problem involving gas flow through porous media.

The industrial problem solved here involves gas flow through a catalytic converter. Catalytic converters are commonly used to purify emissions from gasoline and diesel engines by converting environmentally hazardous exhaust emissions to acceptable substances. Examples of such emissions include carbon monoxide (CO), nitrogen oxides (NOx), and unburned hydrocarbon fuels. These exhaust gas emissions are forced through a substrate, which is a ceramic structure coated with a metal catalyst such as platinum or palladium.

The nature of the exhaust gas flow is a very important factor in determining the performance of the catalytic converter. Of particular importance is the pressure gradient and velocity distribution through the substrate. Hence, CFD analysis is useful for designing efficient catalytic converters. By modeling the exhaust gas flow, the pressure drop and the uniformity of flow through the substrate can be determined. In this tutorial, ANSYS Fluent is used to model the flow of nitrogen gas through a catalytic converter geometry, so that the flow field structure may be analyzed.

This tutorial demonstrates how to do the following:

- Set up a porous zone for the substrate with appropriate resistances.
- Calculate a solution for gas flow through the catalytic converter using the pressure-based solver.
- Plot pressure and velocity distribution on specified planes of the geometry.
- Determine the pressure drop through the substrate and the degree of non-uniformity of flow through cross sections of the geometry using X-Y plots and numerical reports.

8.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)

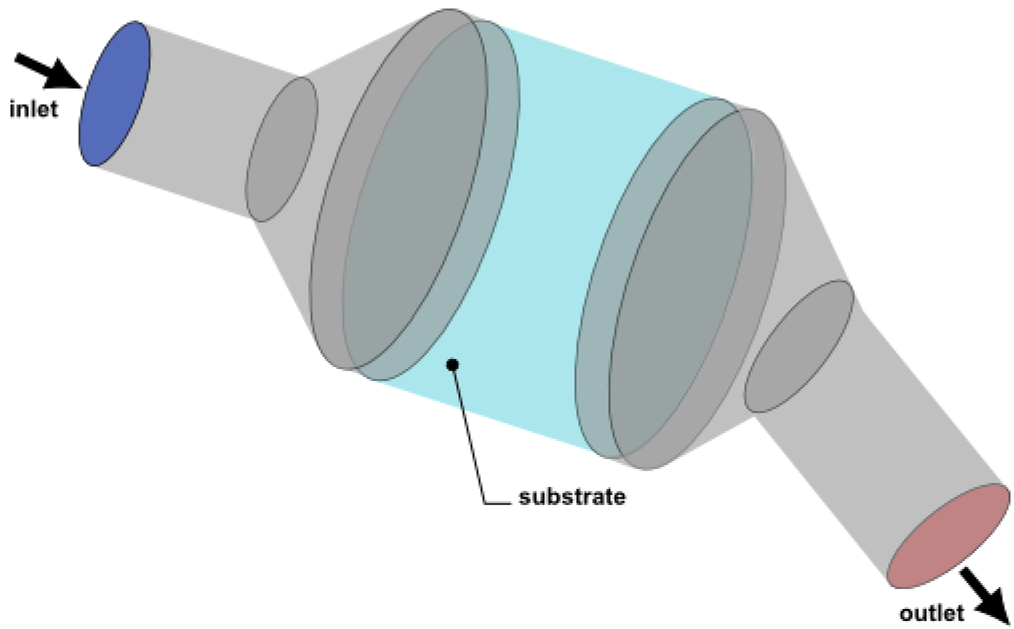
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

and that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

8.3. Problem Description

The catalytic converter modeled here is shown in Figure 8.1: Catalytic Converter Geometry for Flow Modeling (p. 364). The nitrogen flows through the inlet with a uniform velocity of 22.6 m/s, passes through a ceramic monolith substrate with square-shaped channels, and then exits through the outlet.

Figure 8.1: Catalytic Converter Geometry for Flow Modeling



While the flow in the inlet and outlet sections is turbulent, the flow through the substrate is laminar and is characterized by inertial and viscous loss coefficients along the inlet axis. The substrate is impermeable in other directions. This characteristic is modeled using loss coefficients that are three orders of magnitude higher than in the main flow direction.

8.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

8.4.1. Preparation

To prepare for running this tutorial:

1. Set up a working folder on the computer you will be using.

2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

   Note

   If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

3. Enter the name of this tutorial into the search bar.

4. Narrow the results by using the filter on the left side of the page.

   a. Click **ANSYS Fluent** under **Product**.

   b. Click **18.0** under **Version**.

5. Select this tutorial from the list.

6. Click the **porous_R180.zip** link to download the input files.

7. Unzip `porous_R180.zip` to your working folder.

   The mesh file `catalytic_converter.msh` *can be found in the* `porous` *directory created after unzipping the file.*

8. Use Fluent Launcher to start the **3D** version of ANSYS Fluent, with the **Double Precision** and **Display Mesh After Reading** options enabled.

   For more information about starting ANSYS Fluent using the Fluent Launcher, see the Fluent Getting Started Guide, available in the help viewer or on the ANSYS Customer Portal (http://support.ansys.com/documentation).

8.4.2. Mesh

1. Read the mesh file (`catalytic_converter.msh`).

   File → Read → Mesh...

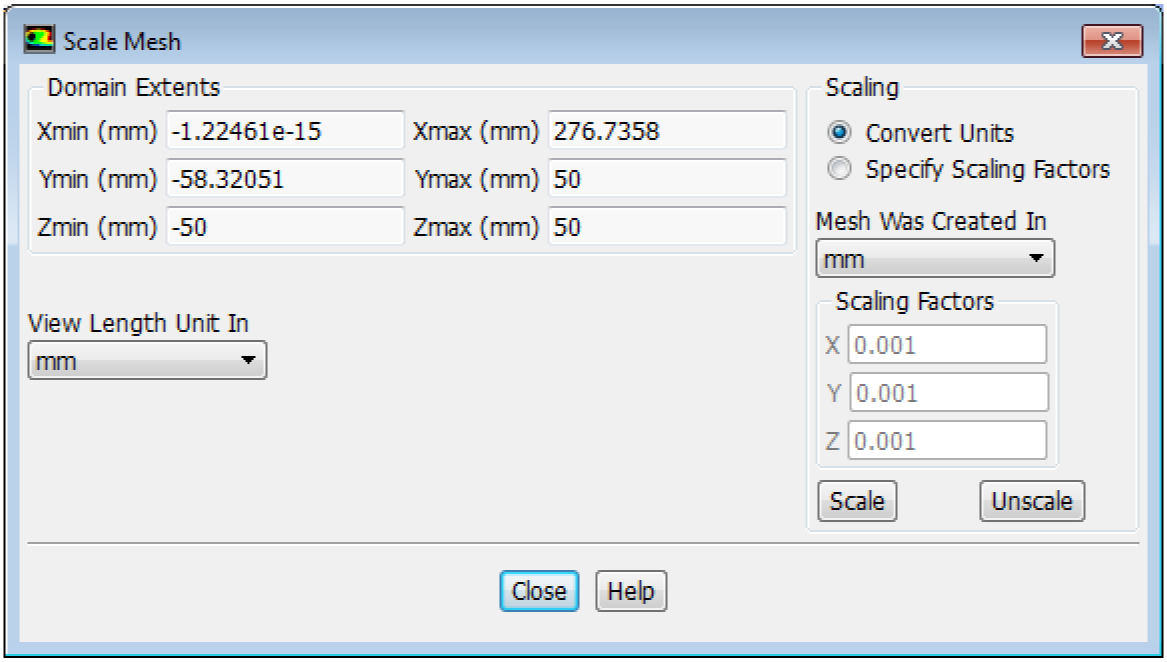
2. Check the mesh.

   Setting Up Domain → Mesh → Check

   ANSYS Fluent will perform various checks on the mesh and report the progress in the console. Make sure that the reported minimum volume is a positive number.

3. Scale the mesh.

Setting Up Domain → Mesh → Scale...



a. Select **mm** from the **Mesh Was Created In** drop-down list.

b. Click **Scale**.

c. Select **mm** from the **View Length Unit In** drop-down list.

All dimensions will now be shown in millimeters.

d. Close the **Scale Mesh** dialog box.

4. Check the mesh.

Setting Up Domain → Mesh → Check

Note

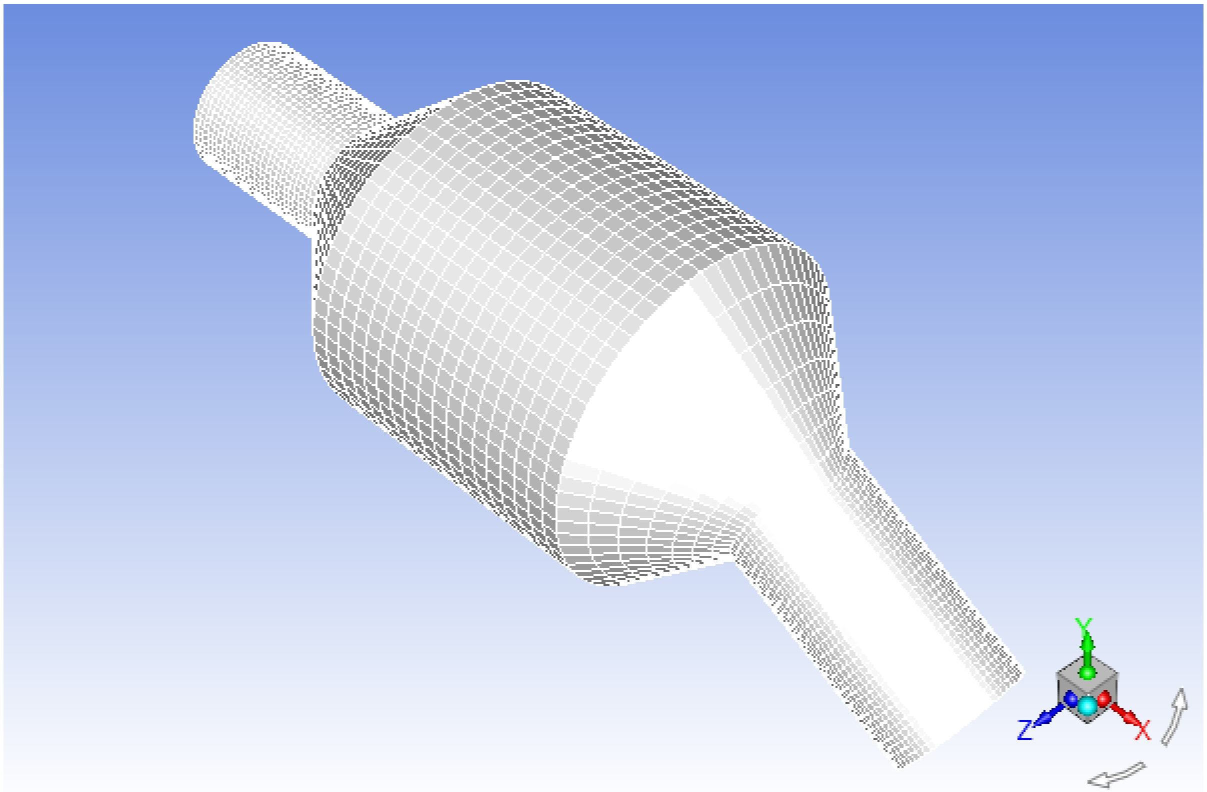
It is a good idea to check the mesh after you manipulate it (that is, scale, convert to polyhedra, merge, separate, fuse, add zones, or smooth and swap.) This will ensure that the quality of the mesh has not been compromised.

---

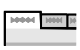
5. Examine the mesh.

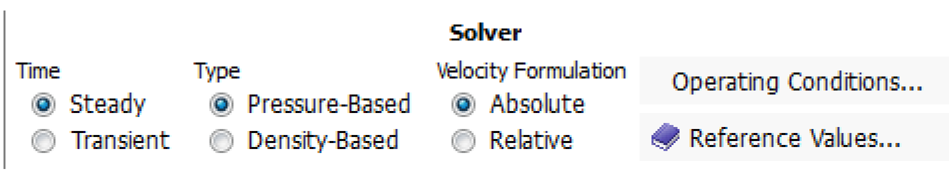
Rotate the view and zoom in to get the display shown in *Figure 8.2: Mesh for the Catalytic Converter Geometry (p. 367)**. The hex mesh on the geometry contains a total of 34,580 cells.*

Figure 8.2: Mesh for the Catalytic Converter Geometry



8.4.3. General Settings

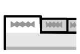
 **Setting Up Physics → Solver**

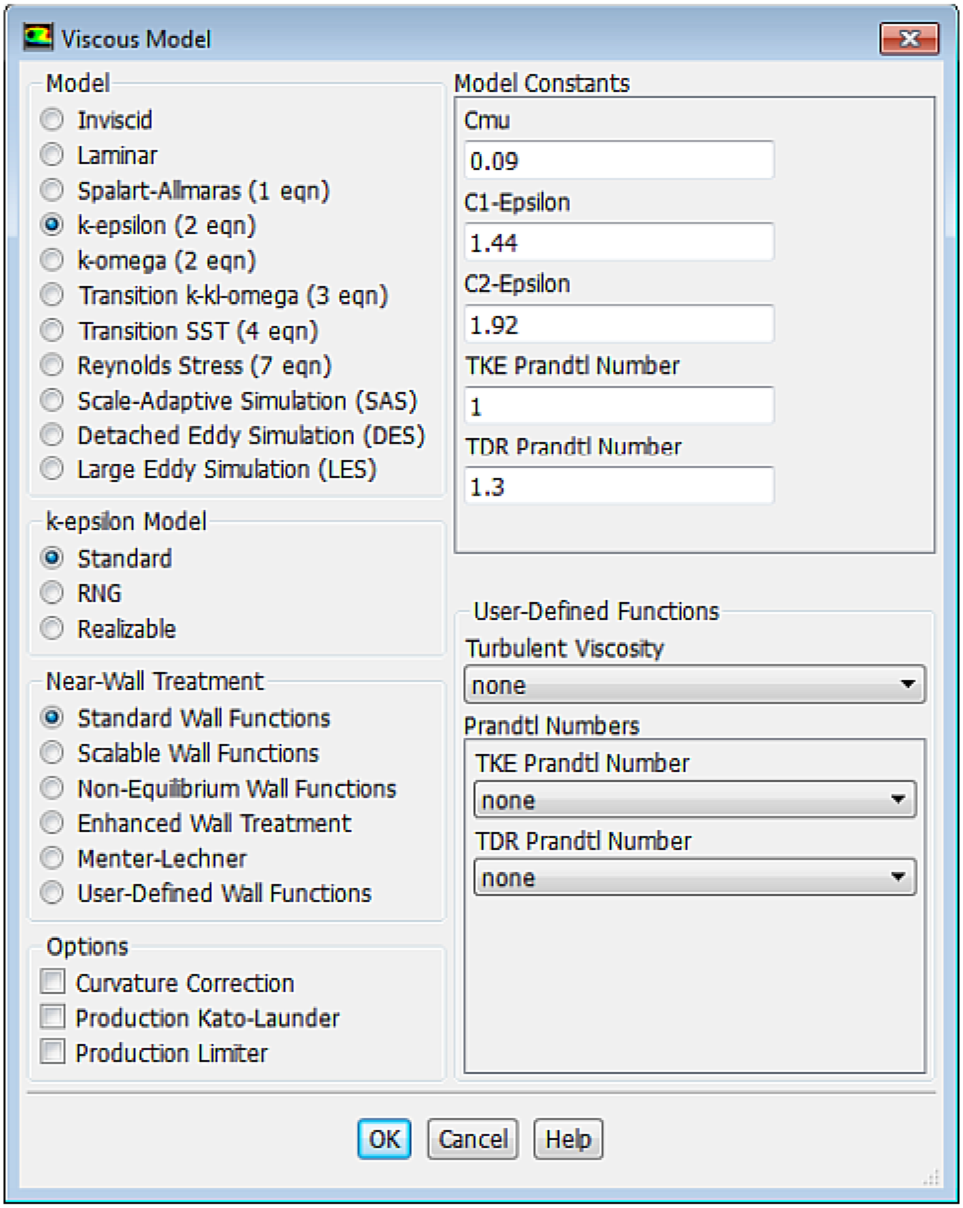


1. Retain the default solver settings.

8.4.4. Models

1. Select the standard k- ε turbulence model.

    **Setting Up Physics → Models → Viscous...**



a. Select **k-epsilon (2eqn)** in the **Model** list.

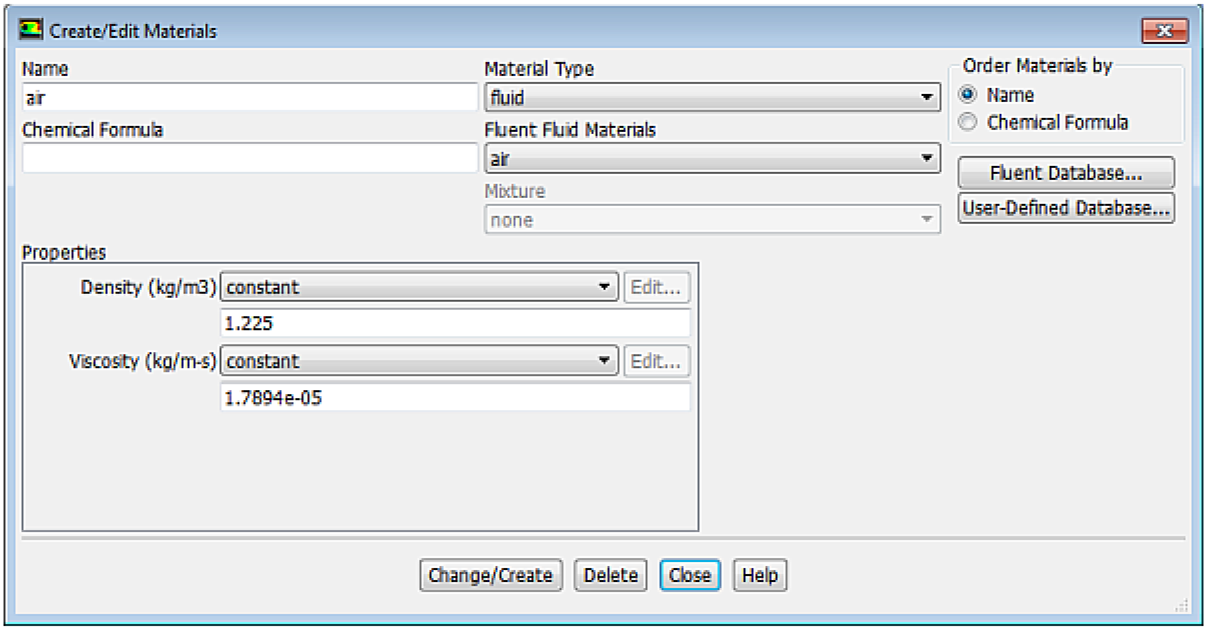
   *The original **Viscous Model** dialog box will now expand.*

b. Retain the default settings for **k-epsilon Model** and **Near-Wall Treatment** and click **OK** to close the **Viscous Model** dialog box.

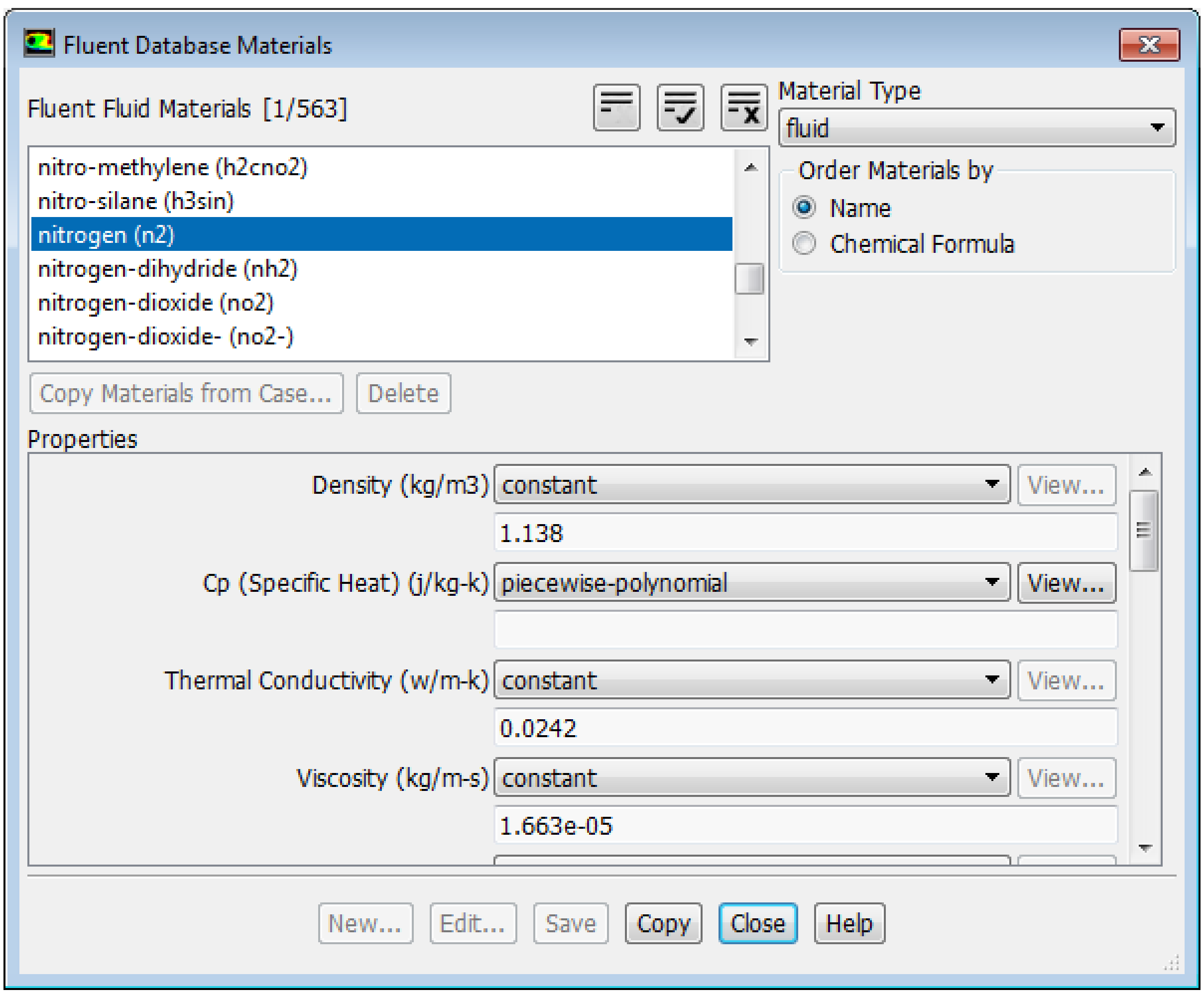
8.4.5. Materials

1. Add nitrogen to the list of fluid materials by copying it from the **Fluent Database** of materials.

   Setting Up Physics → Materials → Create/Edit...



a. Click the **Fluent Database...** button to open the **Fluent Database Materials** dialog box.



i. Select **nitrogen (n2)** in the **Fluent Fluid Materials** selection list.

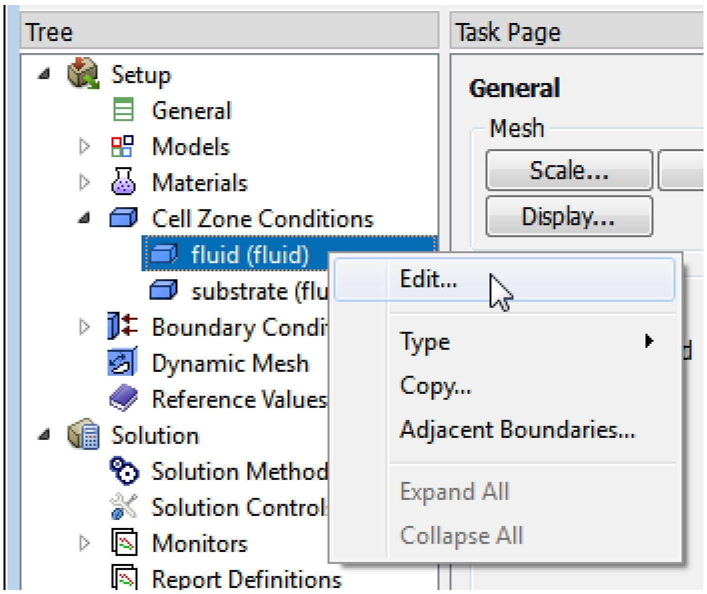
ii. Click **Copy** to copy the information for nitrogen to your list of fluid materials.

iii. Close the **Fluent Database Materials** dialog box.

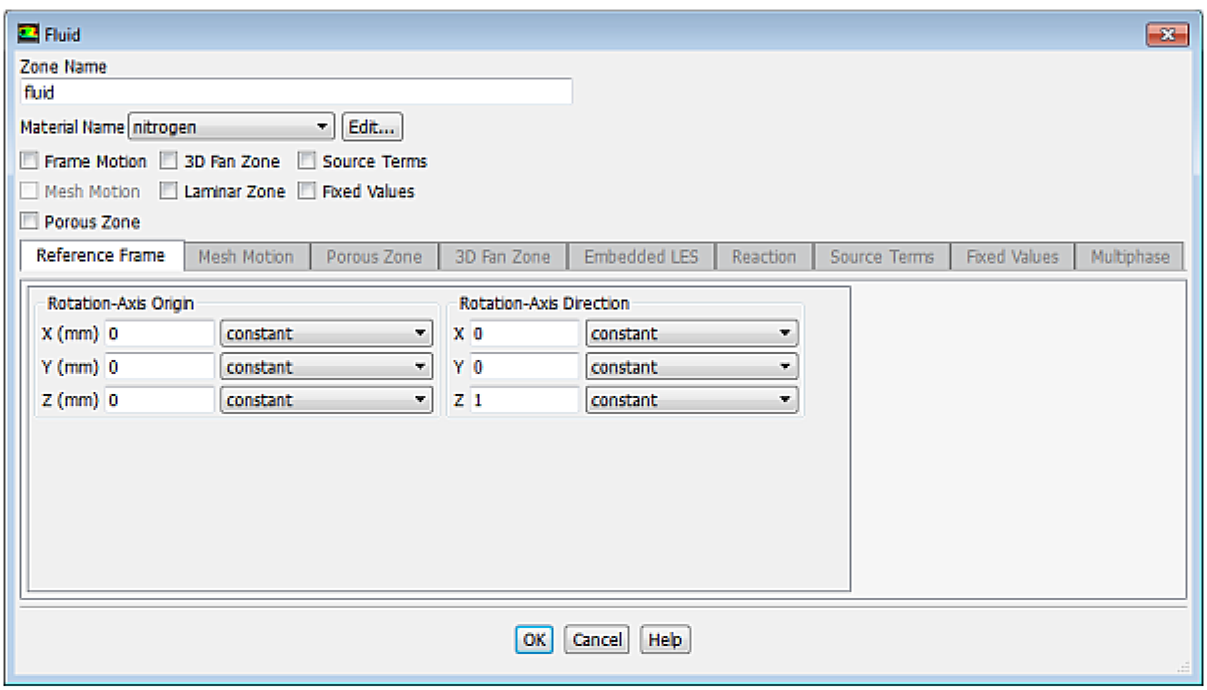
b. Click **Change/Create** and close the **Create/Edit Materials** dialog box.

8.4.6. Cell Zone Conditions

Setup → **Cell Zone Conditions** → **fluid** **Edit...**



1. Set the cell zone conditions for the fluid (**fluid**).

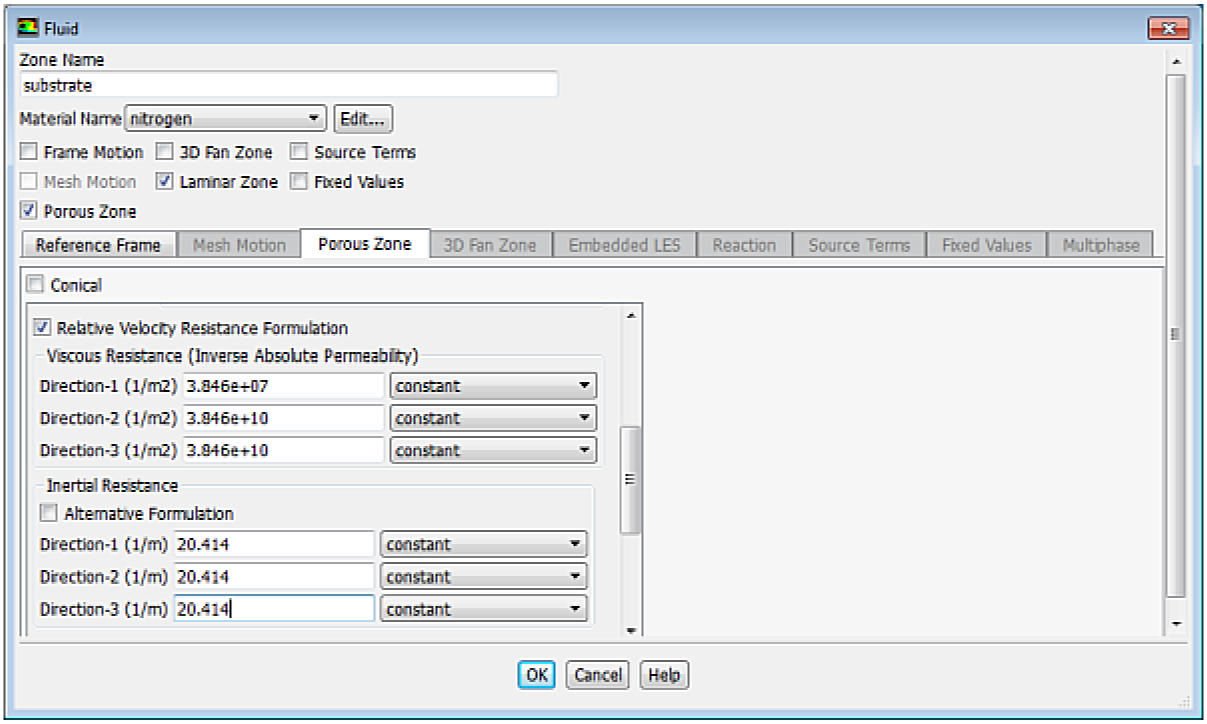


a. Select **nitrogen** from the **Material Name** drop-down list.

b. Click **OK** to close the **Fluid** dialog box.

2. Set the cell zone conditions for the substrate (**substrate**).

Setup → Cell Zone Conditions → substrate Edit...



a. Select **nitrogen** from the **Material Name** drop-down list.

b. Enable **Porous Zone** to activate the porous zone model.

c. Enable **Laminar Zone** to solve the flow in the porous zone without turbulence.

d. Click the **Porous Zone** tab.

   i. Make sure that the principal direction vectors are set as shown in Table 8.1: Values for the Principle Direction Vectors (p. 371).

   ANSYS Fluent automatically calculates the third (Z direction) vector based on your inputs for the first two vectors. The direction vectors determine which axis the viscous and inertial resistance coefficients act upon.

   Table 8.1: Values for the Principle Direction Vectors

Axis	Direction-1 Vector	Direction-2 Vector
X	1	0
Y	0	1
Z	0	0

   ii. For the viscous and inertial resistance directions, enter the values in Table 8.2: Values for the Viscous and Inertial Resistance (p. 372) **Viscous Resistance** and **Inertial Resistance**.

Direction-2 *and* ***Direction-3*** *are set to arbitrary large numbers. These values are several orders of magnitude greater than that of the Direction-1 flow and will make any radial flow insignificant.*

Scroll down to access the fields that are not initially visible.

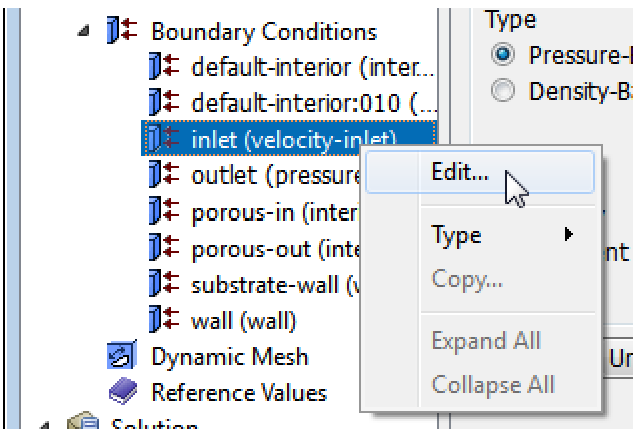
Table 8.2: Values for the Viscous and Inertial Resistance

Direction	Viscous Resistance (1/m2)	Inertial Resistance (1/m)
Direction-1	3.846e+07	20.414
Direction-2	3.846e+10	20414
Direction-3	3.846e+10	20414

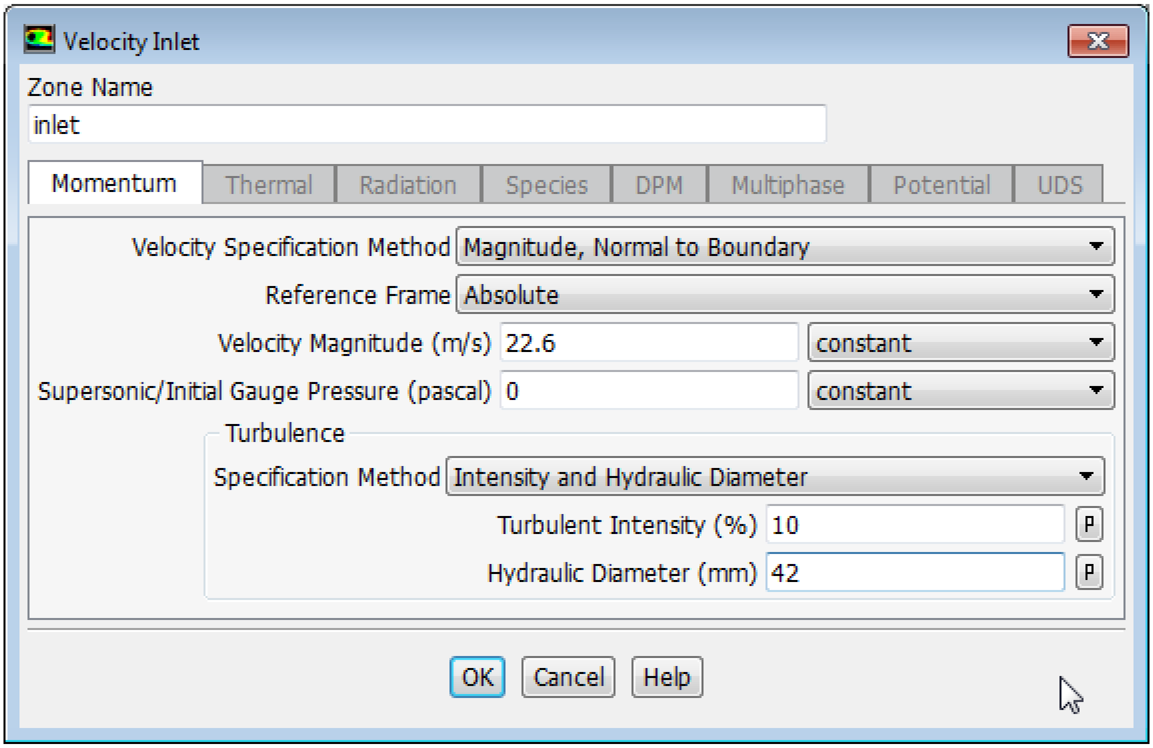
e. Click **OK** to close the **Fluid** dialog box.

8.4.7. Boundary Conditions

Setup → **Boundary Conditions** → **inlet** **Edit...**



1. Set the velocity and turbulence boundary conditions at the inlet (**inlet**).



a. Enter 22.6 m/s for **Velocity Magnitude**.

b. In the **Turbulence** group box, select **Intensity and Hydraulic Diameter** from the **Specification Method** drop-down list.

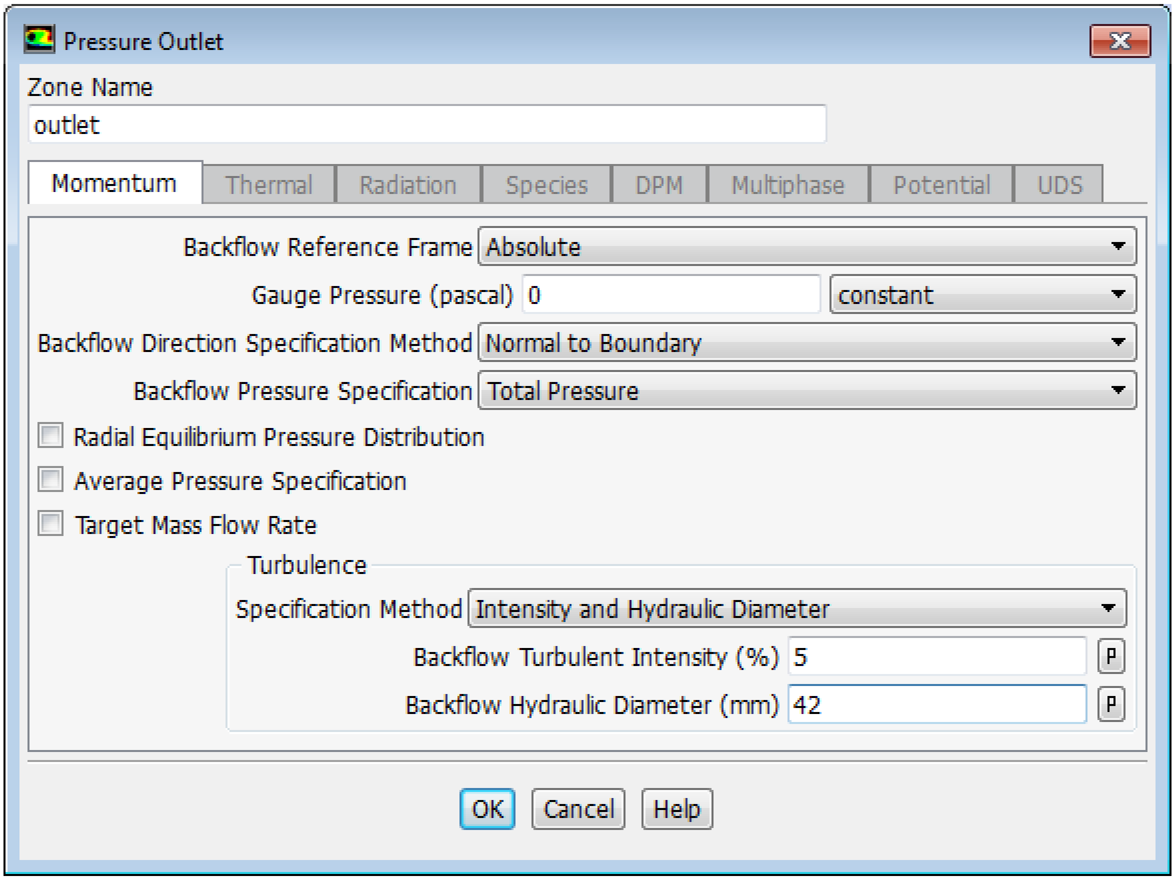
c. Enter **10**% for the **Turbulent Intensity**.

d. Enter 42 mm for the **Hydraulic Diameter**.

e. Click **OK** to close the **Velocity Inlet** dialog box.

2. Set the boundary conditions at the outlet (**outlet**).

Setup → **Boundary Conditions** → **outlet** → **Edit...**



a. Retain the default setting of **0** for **Gauge Pressure**.

b. In the **Turbulence** group box, select **Intensity and Hydraulic Diameter** from the **Specification Method** drop-down list.

c. Retain the default value of 5% for the **Backflow Turbulent Intensity**.

d. Enter 42 mm for the **Backflow Hydraulic Diameter**.

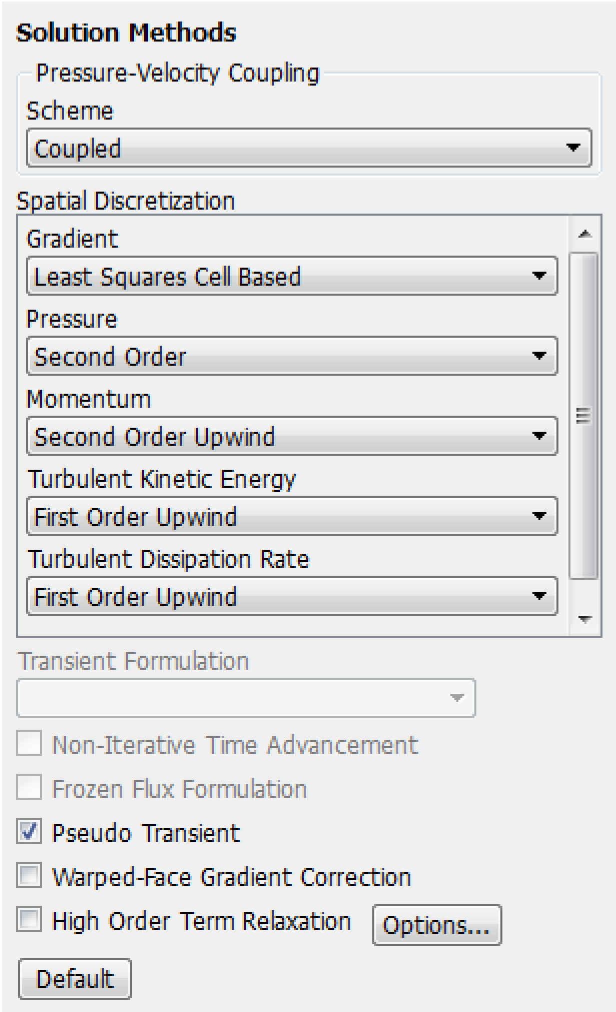
e. Click **OK** to close the **Pressure Outlet** dialog box.

3. Retain the default boundary conditions for the walls (**substrate-wall** and **wall**).

8.4.8. Solution

1. Set the solution parameters.

Solving → **Solution** → **Methods...**



a. Select **Coupled** from the **Scheme** drop-down list.

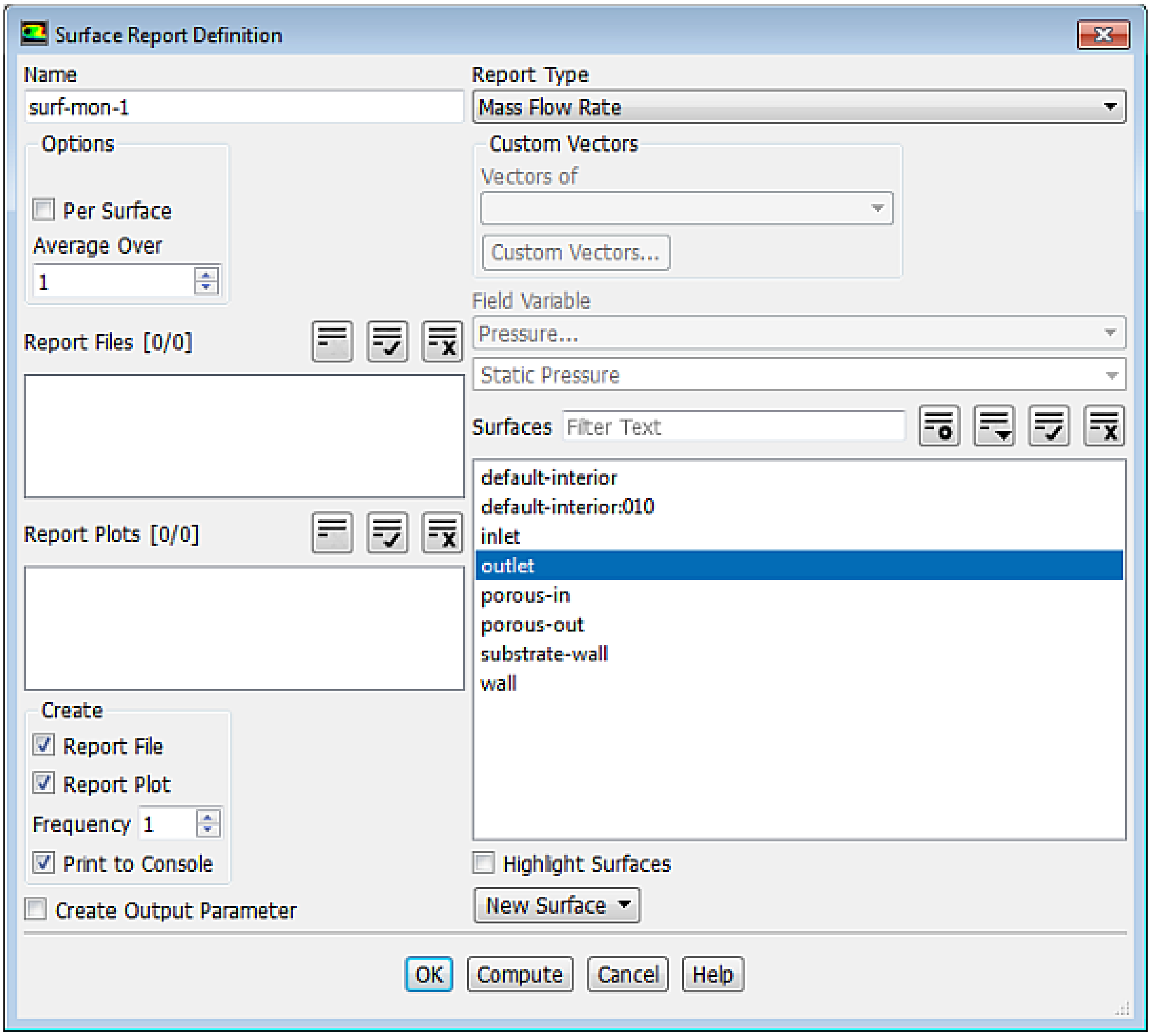
b. Retain the default selection of **Least Squares Cell Based** from the **Gradient** drop-down list in the **Spatial Discretization** group box.

c. Retain the default selection of **Second Order Upwind** from the **Momentum** drop-down list.

d. Enable **Pseudo Transient**.

2. Enable the plotting of the mass flow rate at the outlet.

Solving → **Reports** → **Definitions** → **New** → **Surface Report** → **Mass Flow Rate**



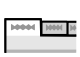
a. Enter **`surf-mon-1`** for the **Name** of the surface report definition.

b. In the **Create** group box, enable **Report File**, **Report Plot** and **Print to Console**.

c. Select **outlet** in the **Surfaces** selection list.

d. Click **OK** to save the surface report definition settings and close the **Surface Report Definition** dialog box.

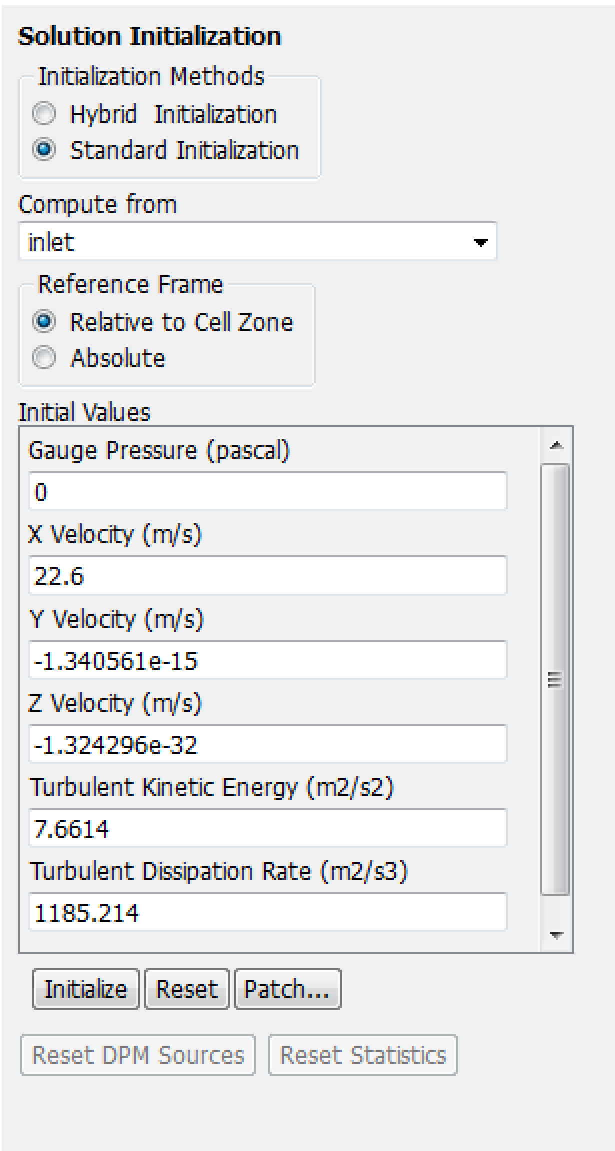
3. Initialize the solution from the inlet.

 **Solving** → **Initialization**

a. Select **Standard** under **Method**.

> **Warning**
>
> **Standard** is the recommended initialization method for porous media simulations. The default **Hybrid** method does not account for the porous media properties, and depending on boundary conditions, may produce an unrealistic initial velocity field. For porous media simulations, the **Hybrid** method should only be used when the **Maintain Constant Velocity Magnitude** option is enabled in the **Hybrid Initialization** dialog box.

b. Click **Options...** to open the **Solution Initialization** task page, which provides access to further settings.



i. Select **inlet** from the **Compute from** drop-down list in the **Solution Initialization** task page.

ii. Retain the default settings for standard initialization method.

iii. Click **Initialize**.

4. Save the case file (`catalytic_converter.cas.gz`).

   File → Write → Case...

5. Run the calculation by requesting 100 iterations.

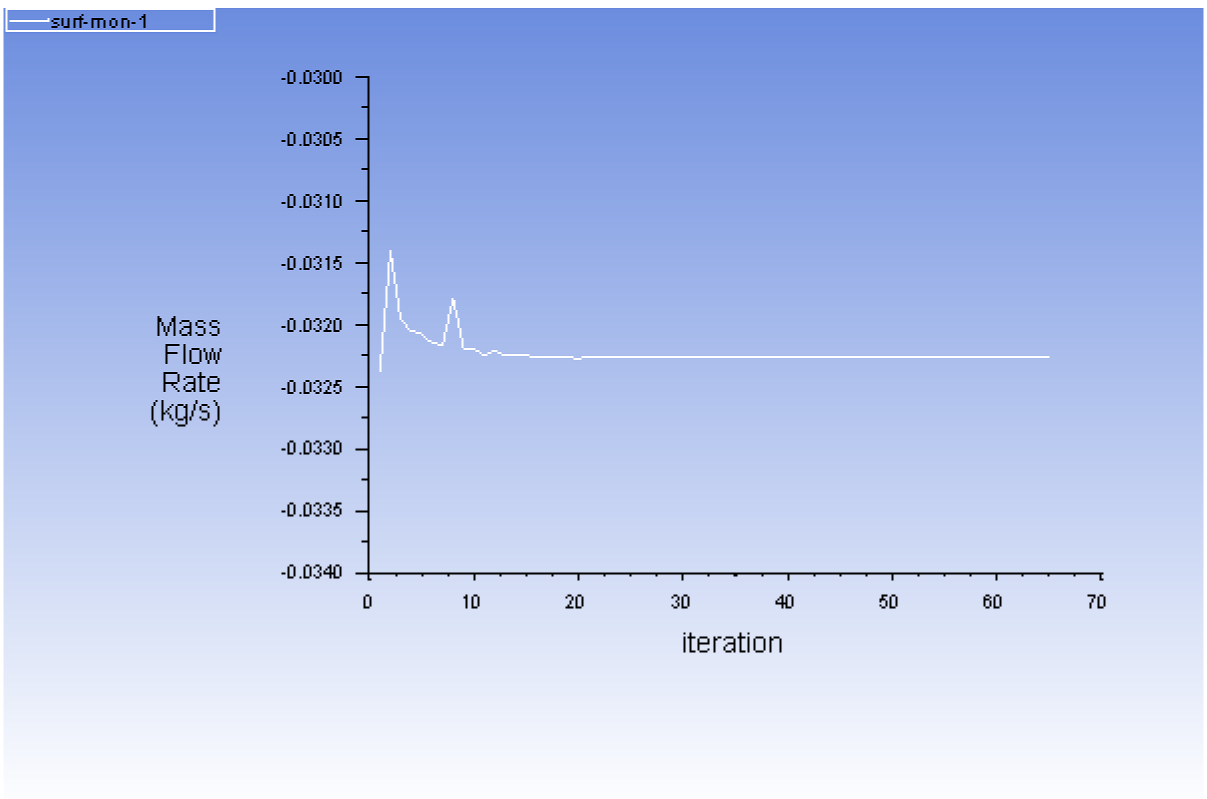
   Solving → Run Calculation

   a. Enter `100` for **No. of Iterations**.

   b. Click **Calculate** to begin the iterations.

   The solution will converge in approximately 65 iterations. The mass flow rate graph flattens out, as seen in Figure 8.3: Mass Flow Rate History (p. 378).

Figure 8.3: Mass Flow Rate History



6. Save the case and data files (`catalytic_converter.cas.gz` and `catalytic_converter.dat.gz`).

   File → Write → Case & Data...

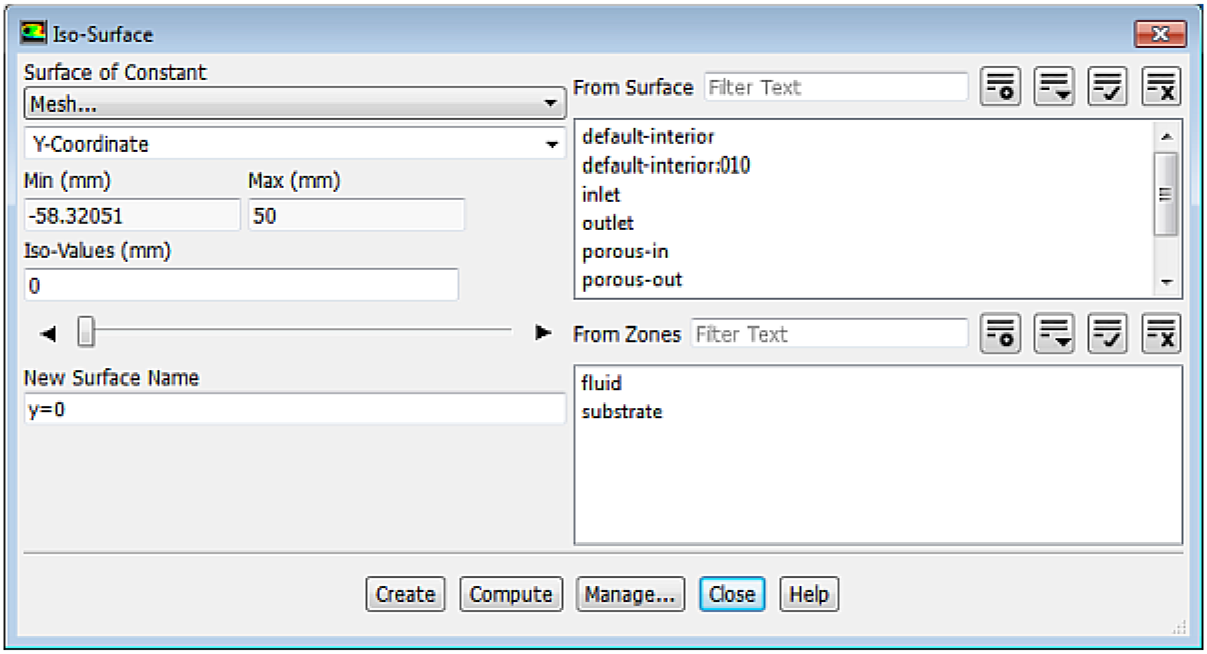
Note

If you choose a filename that already exists in the current folder, ANSYS Fluent will prompt you for confirmation to overwrite the file.

8.4.9. Postprocessing

1. Create a surface passing through the centerline for postprocessing purposes.

   Postprocessing → **Surface** → **Create** → **Iso-Surface...**



   a. Select **Mesh...** and **Y-Coordinate** from the **Surface of Constant** drop-down lists.

   b. Click **Compute** to calculate the **Min** and **Max** values.

   c. Retain the default value of **0** for **Iso-Values**.

   d. Enter `y=0` for **New Surface Name**.

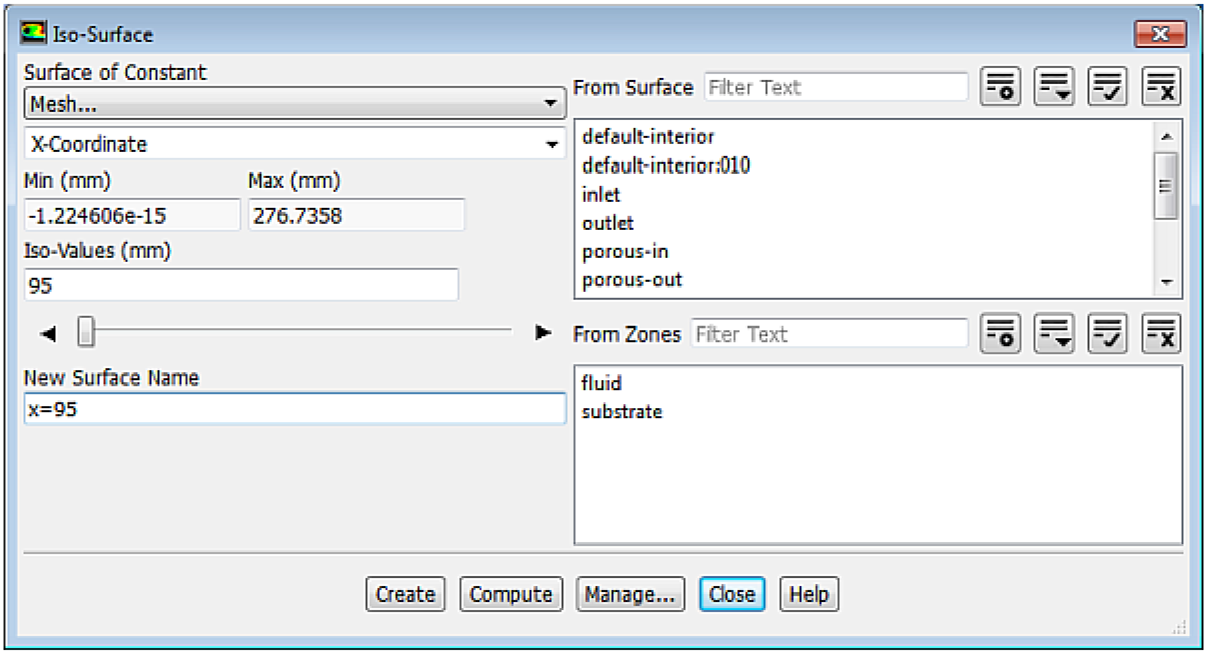
   e. Click **Create**.

   Note

   To interactively place the surface on your mesh, use the slider bar in the **Iso-Surface** dialog box.

2. Create cross-sectional surfaces at locations on either side of the substrate, as well as at its center.

   Postprocessing → **Surface** → **Create** → **Iso-Surface...**



a. Select **Mesh...** and **X-Coordinate** from the **Surface of Constant** drop-down lists.

b. Click **Compute** to calculate the **Min** and **Max** values.

c. Enter 95 for **Iso-Values**.

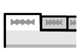
d. Enter `x=95` for the **New Surface Name**.

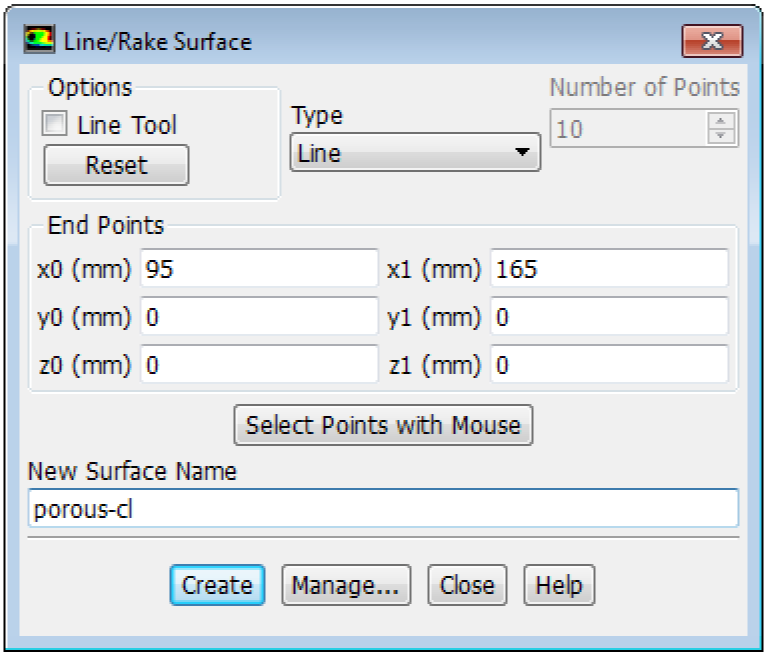
e. Click **Create**.

f. In a similar manner, create surfaces named `x=130` and `x=165` with **Iso-Values** of `130` and `165`, respectively.

g. Close the **Iso-Surface** dialog box after all the surfaces have been created.

3. Create a line surface for the centerline of the porous media.

 **Postprocessing** → **Surface** → **Create** → **Line/Rake...**



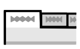
a. Enter the coordinates of the end points of the line in the **End Points** group box as shown.

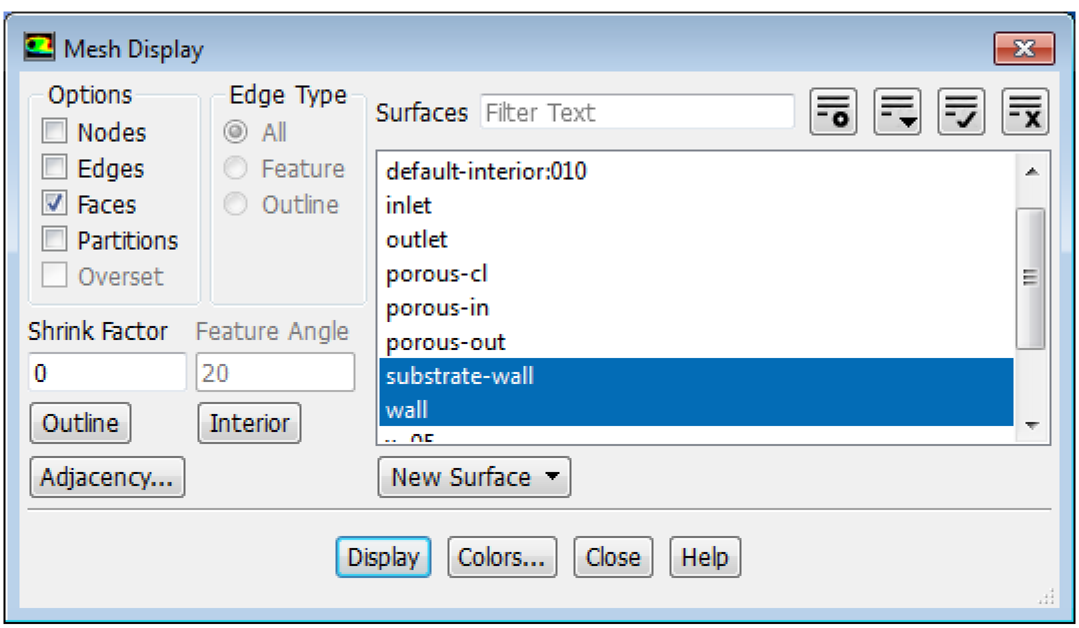
b. Enter `porous-cl` for the **New Surface Name**.

c. Click **Create** to create the surface.

d. Close the **Line/Rake Surface** dialog box.

4. Display the two wall zones (**substrate-wall** and **wall**).

 **Postprocessing → Graphics → Mesh...**



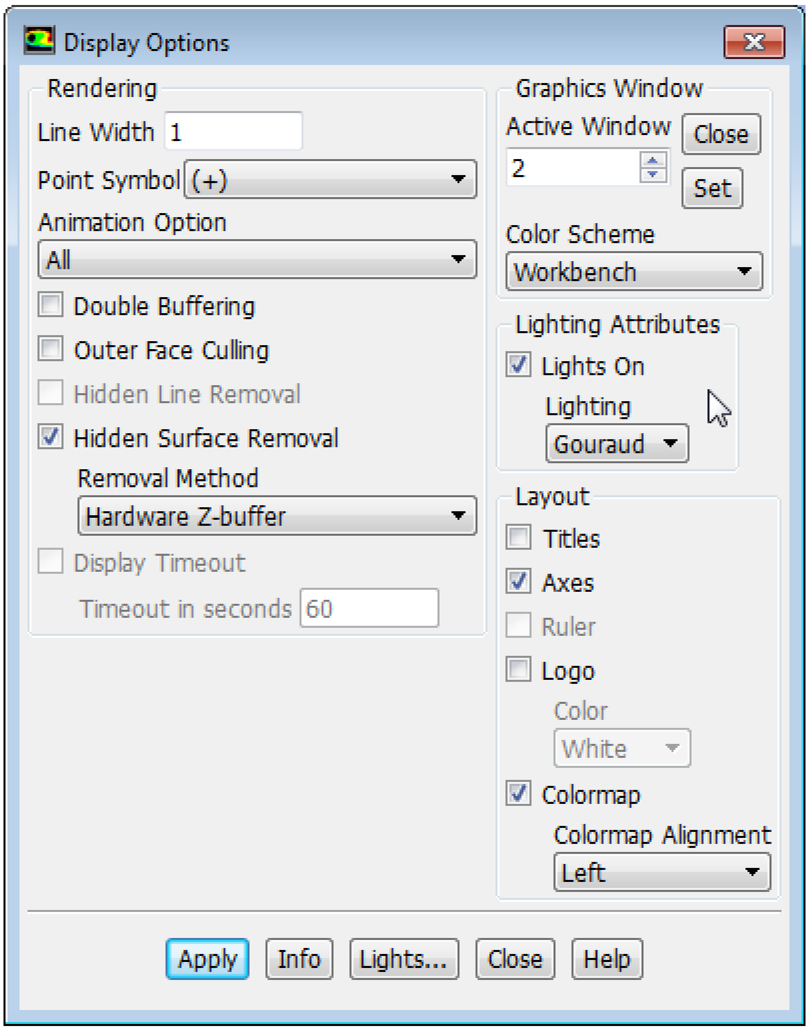
a. Disable **Edges** and make sure **Faces** is enabled in the **Options** group box.

b. Deselect **inlet** and **outlet** in the **Surfaces** selection list, and make sure that only **substrate-wall** and **wall** are selected.

c. Click **Display** and close the **Mesh Display** dialog box.

5. Set the lighting for the display.

Viewing → **Display** → **Options...**



a. Disable **Double Buffering** in the **Rendering** group box.

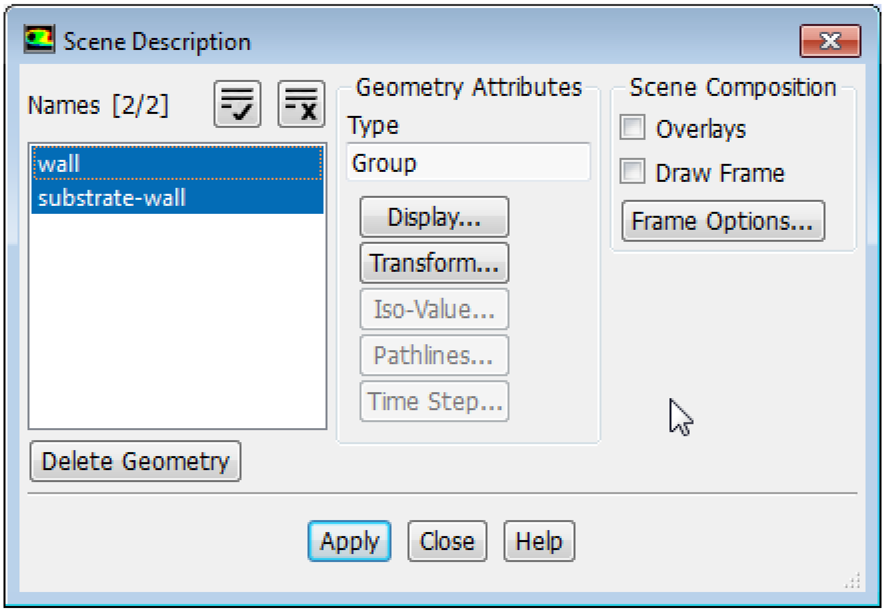
b. Make sure **Lights On** is enabled in the **Lighting Attributes** group box.

c. Select **Gouraud** from the **Lighting** drop-down list.

d. Click **Apply** and close the **Display Options** dialog box.

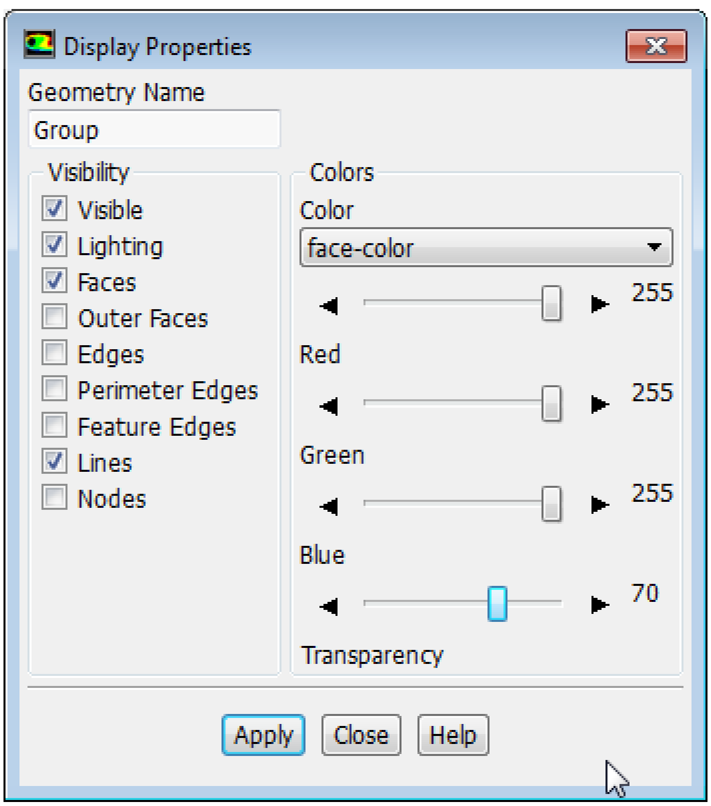
6. Set the transparency parameter for the wall zones (**substrate-wall** and **wall**).

Viewing → **Graphics** → **Compose...**



a. Select **substrate-wall** and **wall** in the **Names** selection list.

b. Click the **Display...** button in the **Geometry Attributes** group box to open the **Display Properties** dialog box.



i. Make sure that the **Red**, **Green**, and **Blue** sliders are set to the maximum position (that is, **255**).

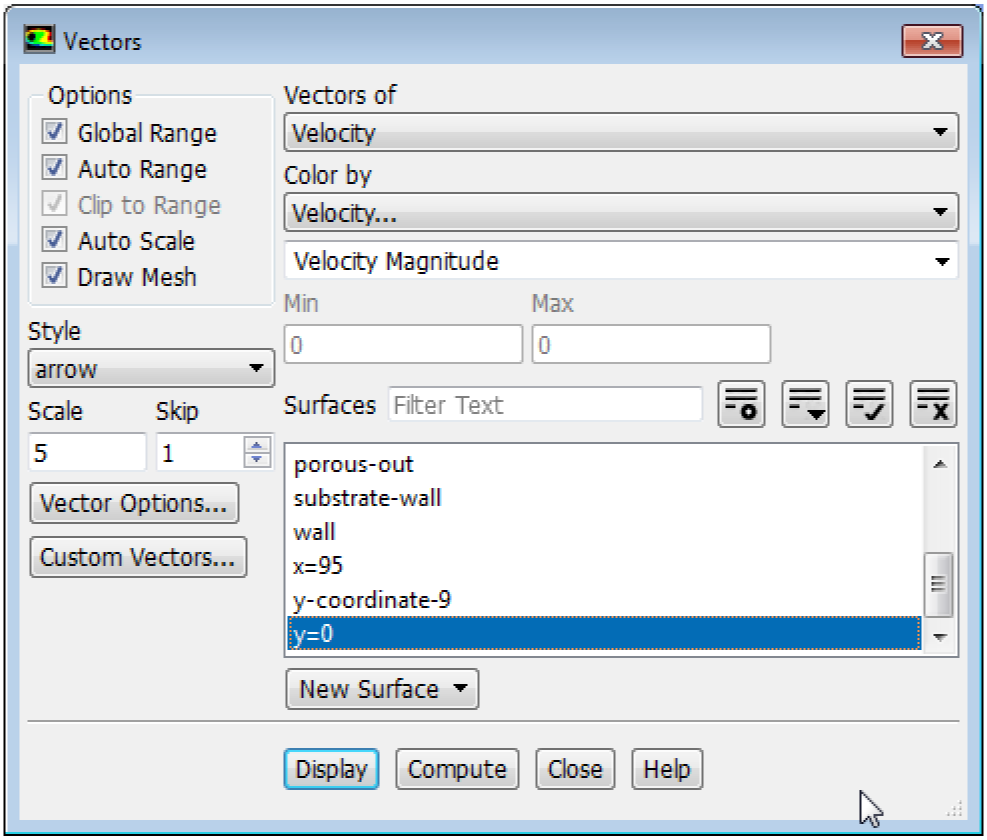
ii. Set the **Transparency** slider to **70**.

iii. Click **Apply** and close the **Display Properties** dialog box.

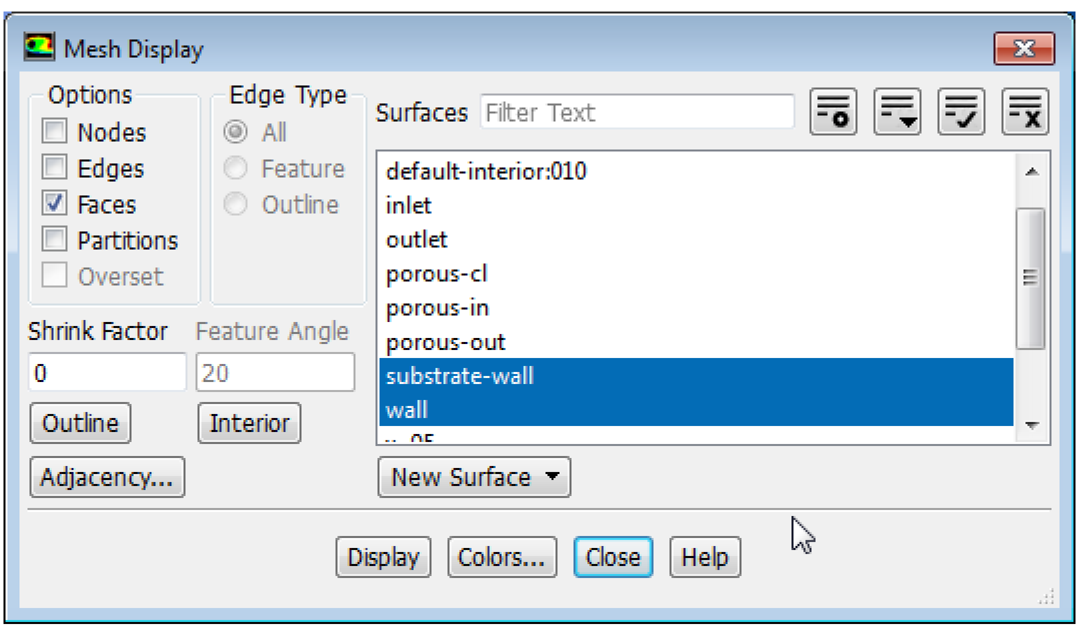
c. Click **Apply** and close the **Scene Description** dialog box.

7. Display velocity vectors on the **y=0** surface (Figure 8.4: Velocity Vectors on the y=0 Plane (p. 385)).

Postprocessing → **Graphics** → **Vectors** → **Edit...**



a. Enable **Draw Mesh** in the **Options** group box to open the **Mesh Display** dialog box.



i. Make sure that **substrate-wall** and **wall** are selected in the **Surfaces** selection list.

ii. Click **Display** and close the **Mesh Display** dialog box.

b. Enter 5 for **Scale**.

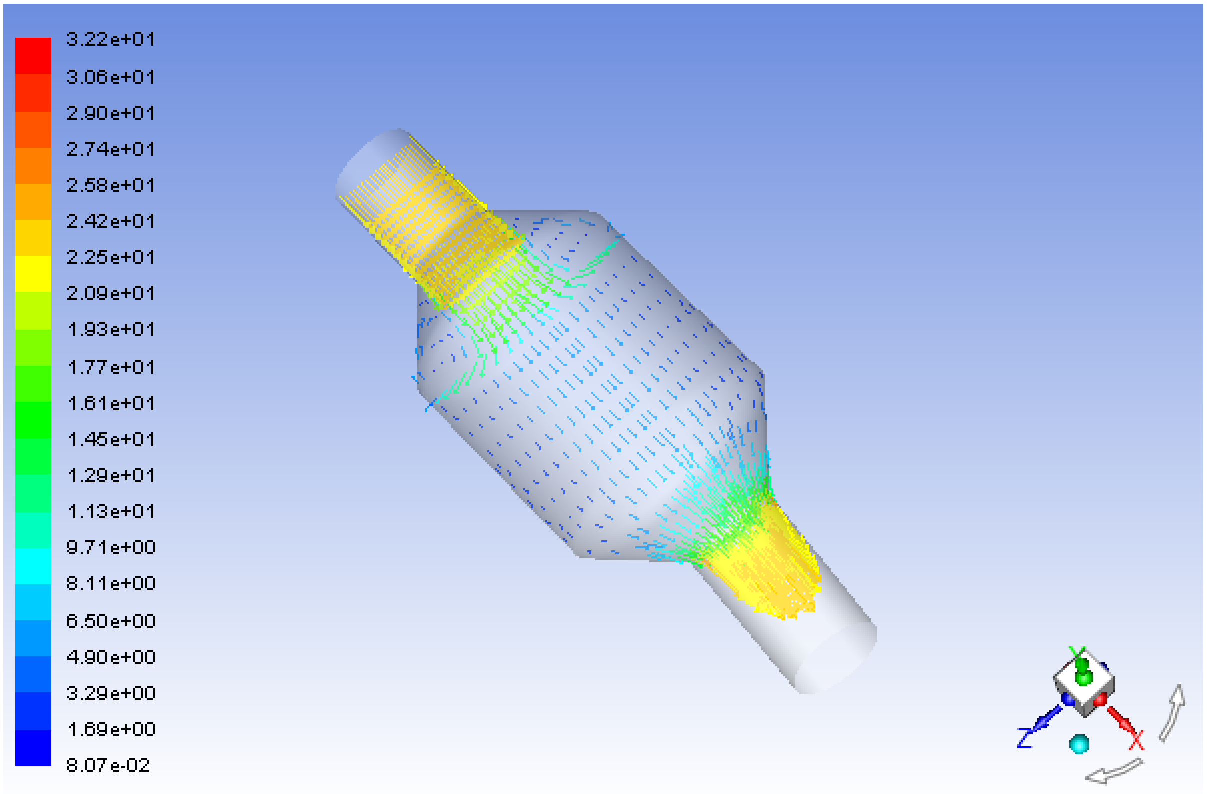
c. Set **Skip** to 1.

d. Select **y=0** in the **Surfaces** selection list.

e. Click **Display** and close the **Vectors** dialog box.

f. Rotate the view and adjust the magnification to get the display shown in Figure 8.4: Velocity Vectors on the y=0 Plane (p. 385).

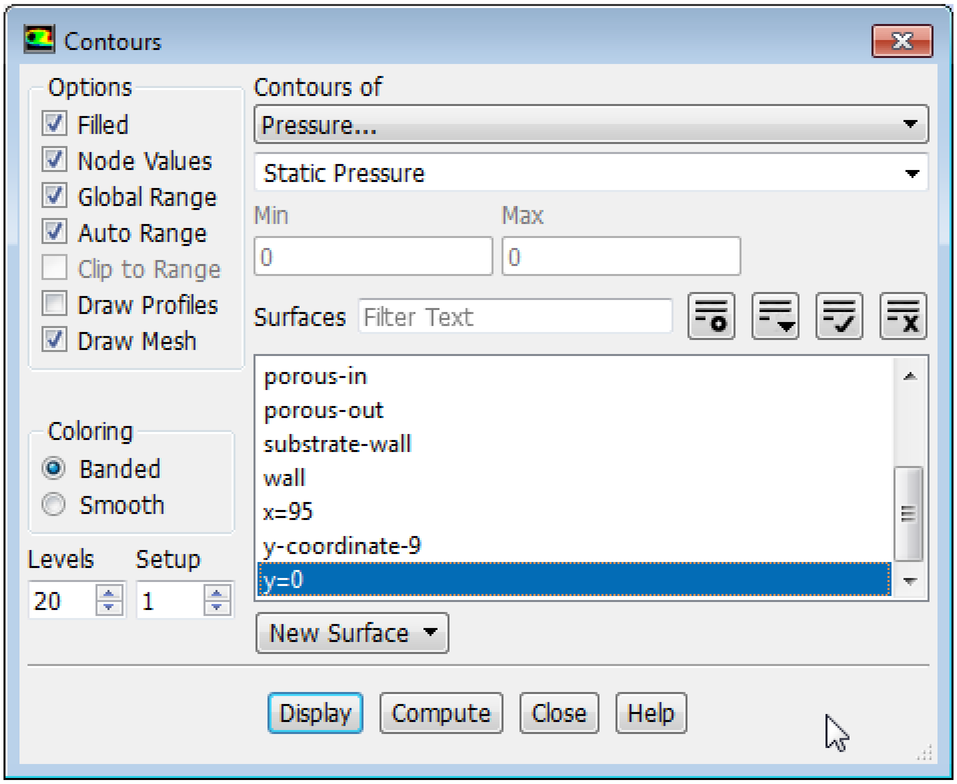
Figure 8.4: Velocity Vectors on the y=0 Plane



The flow pattern shows that the flow enters the catalytic converter as a jet, with recirculation on either side of the jet. As it passes through the porous substrate, it decelerates and straightens out, and exhibits a more uniform velocity distribution. This allows the metal catalyst present in the substrate to be more effective.

8. Display filled contours of static pressure on the **y=0** plane (Figure 8.5: Contours of Static Pressure on the y=0 plane (p. 387)).

Postprocessing → **Graphics** → **Contours** → **Edit...**



a. Enable **Filled** in the **Options** group box.

b. Enable **Draw Mesh** to open the **Mesh Display** dialog box.

   i. Make sure that **substrate-wall** and **wall** are selected in the **Surfaces** selection list.

   ii. Click **Display** and close the **Mesh Display** dialog box.

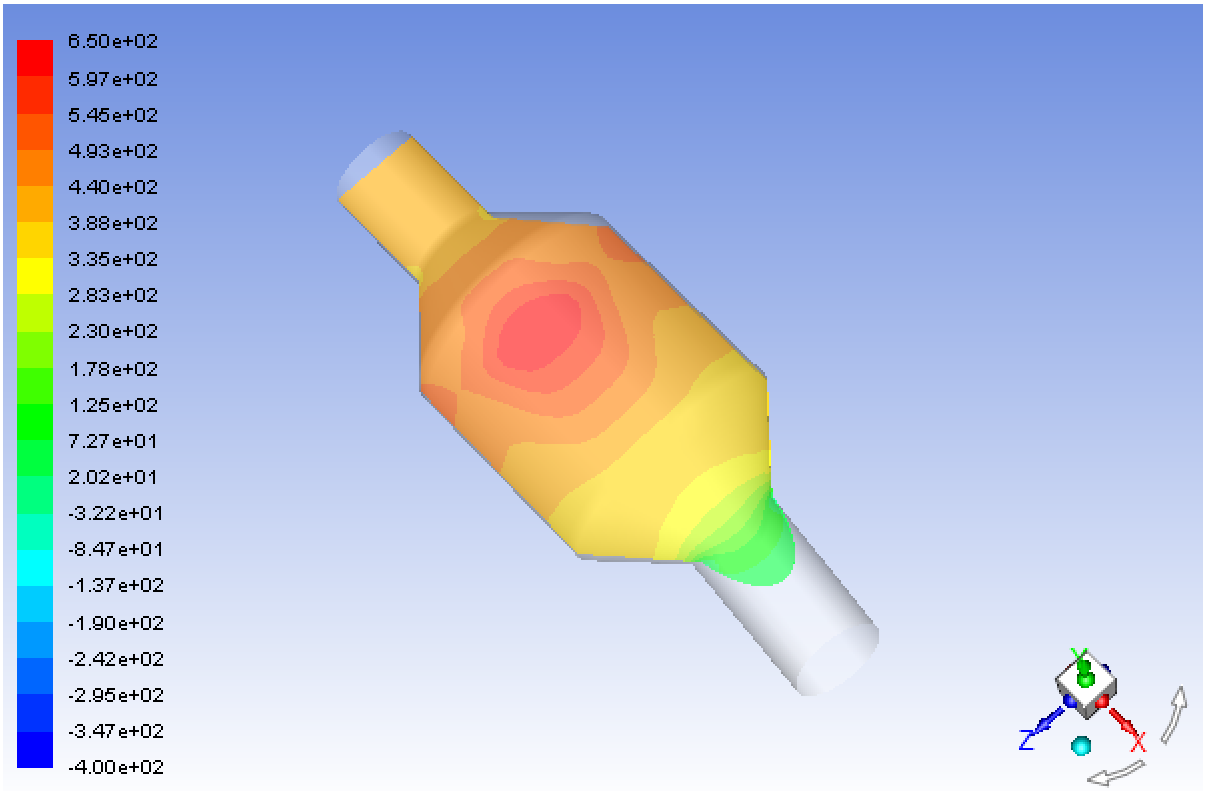
c. Make sure that **Pressure...** and **Static Pressure** are selected from the **Contours of** drop-down lists.

d. Select **y=0** in the **Surfaces** selection list.

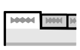
e. Click **Display** and close the **Contours** dialog box.

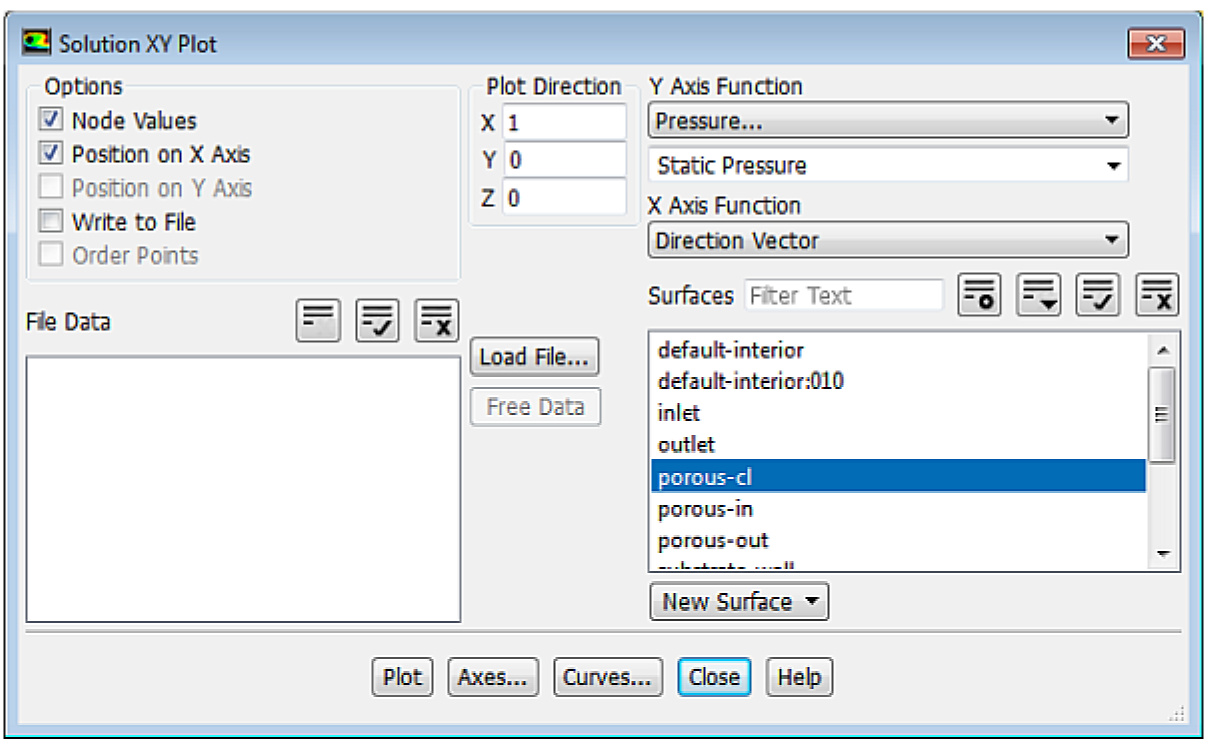
The pressure changes rapidly in the middle section, where the fluid velocity changes as it passes through the porous substrate. The pressure drop can be high, due to the inertial and viscous resistance of the porous media. Determining this pressure drop is one of the goals of the CFD analysis. In the next step, you will learn how to plot the pressure drop along the centerline of the substrate.

Figure 8.5: Contours of Static Pressure on the y=0 plane



9. Plot the static pressure across the line surface **porous-cl** (Figure 8.6: Plot of Static Pressure on the porous-cl Line Surface (p. 388)).

 **Postprocessing → Plots → XY Plot...**

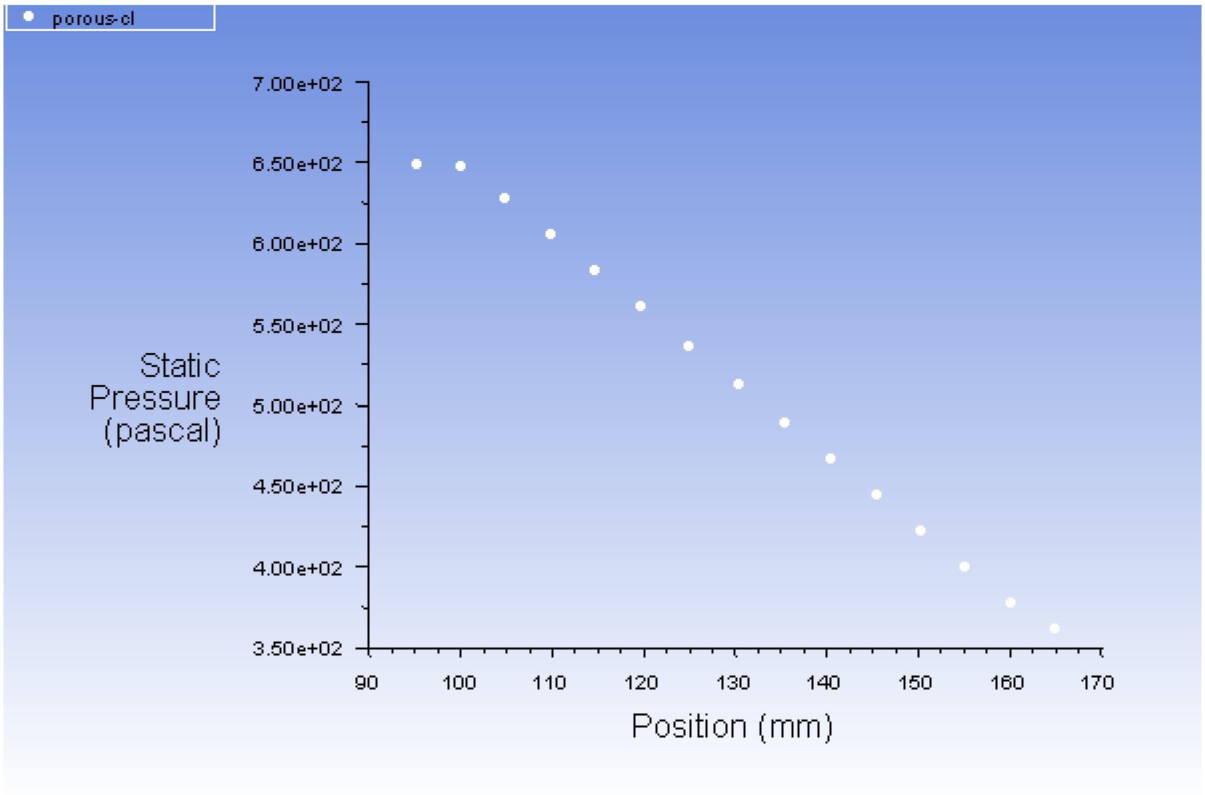


a. Make sure that **Pressure...** and **Static Pressure** are selected from the **Y Axis Function** drop-down lists.

b. Select **porous-cl** in the **Surfaces** selection list.

c. Click **Plot** and close the **Solution XY Plot** dialog box.

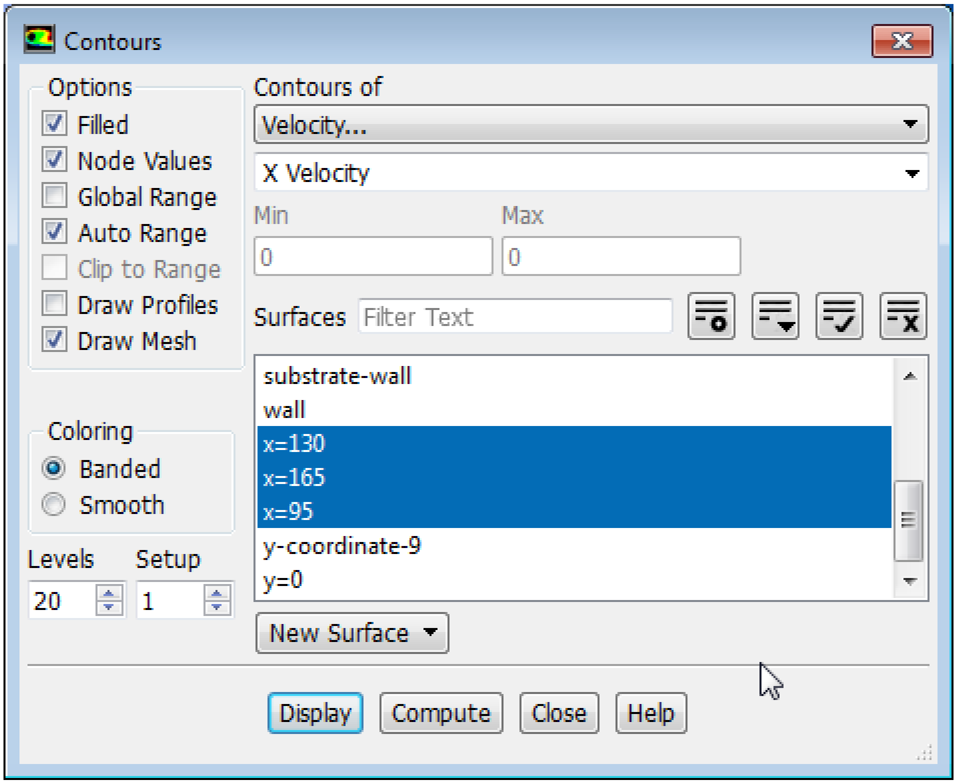
Figure 8.6: Plot of Static Pressure on the porous-cl Line Surface



As seen in Figure 8.6: Plot of Static Pressure on the porous-cl Line Surface (p. 388), the pressure drop across the porous substrate is approximately 300 Pa.

10. Display filled contours of the velocity in the X direction on the **x=95**, **x=130**, and **x=165** surfaces (Figure 8.7: Contours of the X Velocity on the x=95, x=130, and x=165 Surfaces (p. 390)).

Postprocessing → **Graphics** → **Contours** → **Edit...**



a. Make sure that **Filled** and **Draw Mesh** are enabled in the **Options** group box.

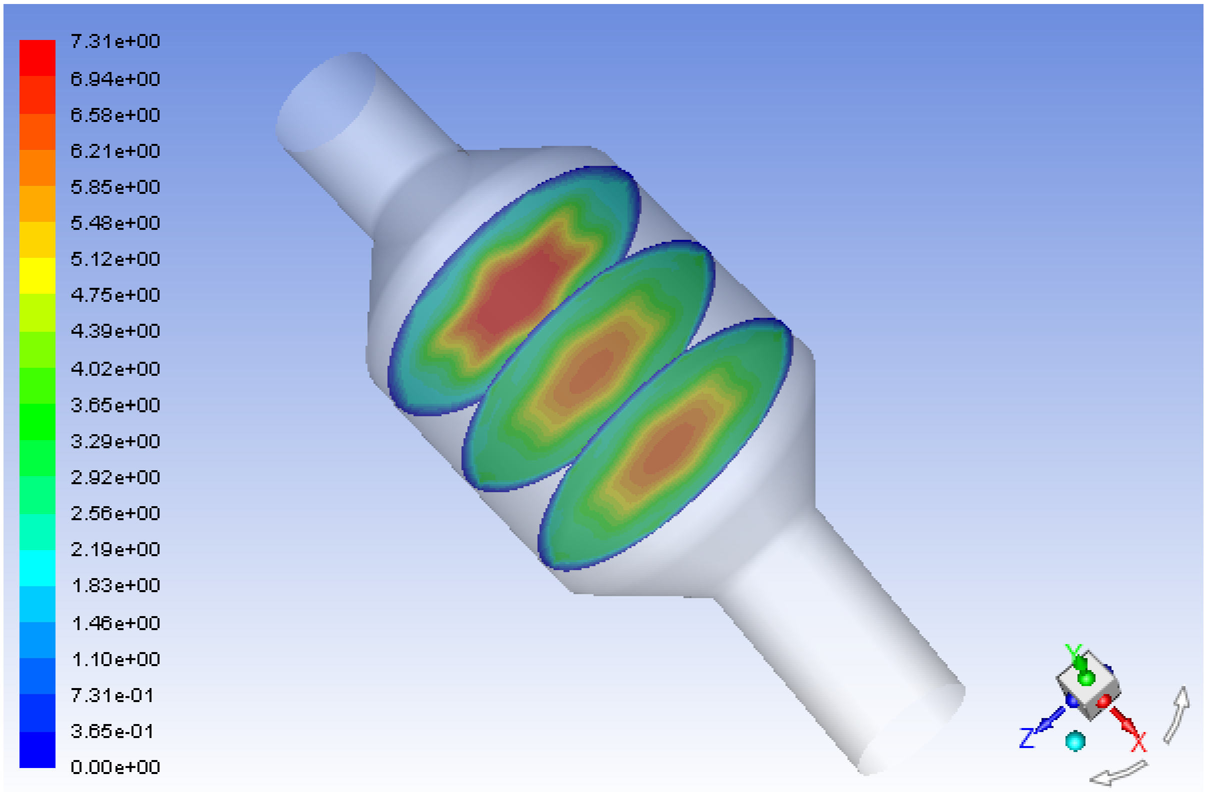
b. Disable **Global Range** in the **Options** group box.

c. Select **Velocity...** and **X Velocity** from the **Contours of** drop-down lists.

d. Select **x=130**, **x=165**, and **x=95** in the **Surfaces** selection list, and deselect **y=0**.

e. Click **Display** and close the **Contours** dialog box.

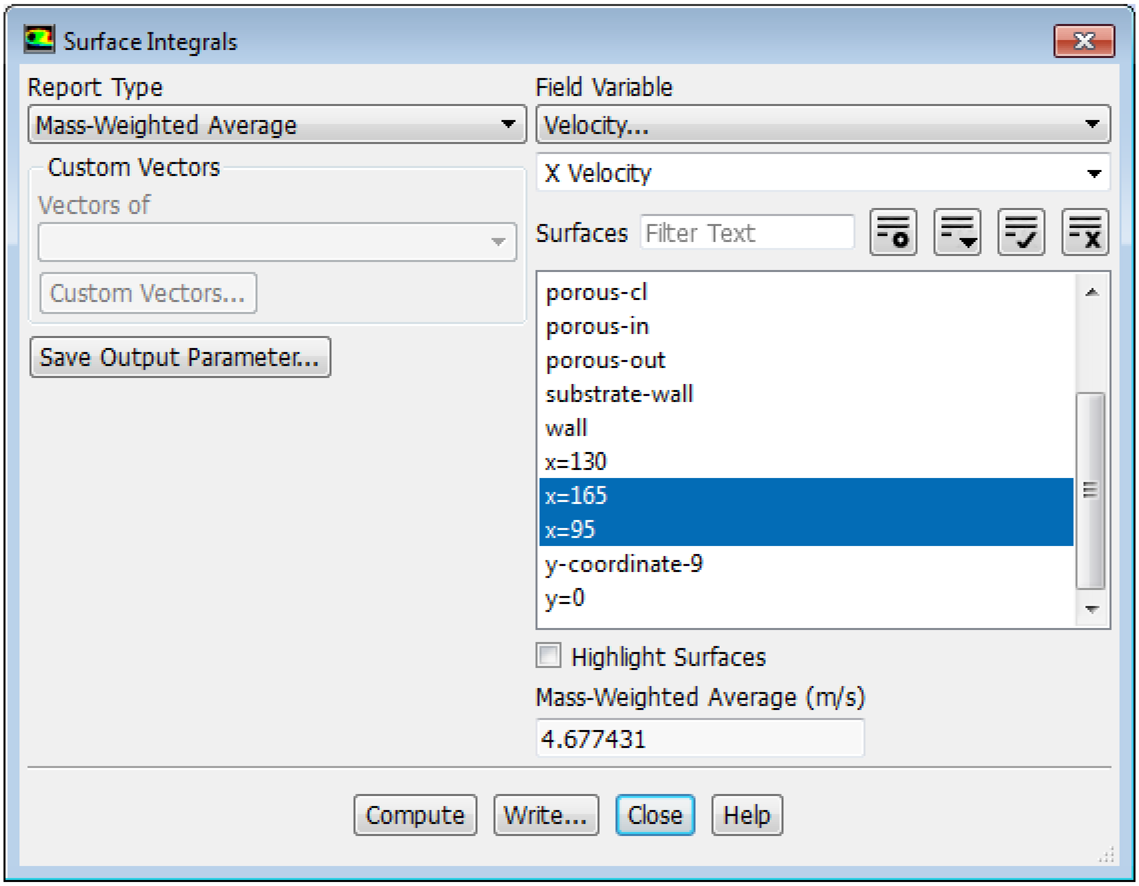
Figure 8.7: Contours of the X Velocity on the x=95, x=130, and x=165 Surfaces



The velocity profile becomes more uniform as the fluid passes through the porous media. The velocity is very high at the center (the area in red) just before the nitrogen enters the substrate and then decreases as it passes through and exits the substrate. The area in green, which corresponds to a moderate velocity, increases in extent.

11. Use numerical reports to determine the average, minimum, and maximum of the velocity distribution before and after the porous substrate.

   Postprocessing → **Reports** → **Surface Integrals...**



a. Select **Mass-Weighted Average** from the **Report Type** drop-down list.

b. Select **Velocity** and **X Velocity** from the **Field Variable** drop-down lists.

c. Select **x=165** and **x=95** in the **Surfaces** selection list.

d. Click **Compute**.

e. Select **Facet Minimum** from the **Report Type** drop-down list and click **Compute**.

f. Select **Facet Maximum** from the **Report Type** drop-down list and click **Compute**.

   The numerical report of average, maximum and minimum velocity can be seen in the main ANSYS Fluent console.

g. Close the **Surface Integrals** dialog box.

The spread between the average, maximum, and minimum values for X velocity gives the degree to which the velocity distribution is non-uniform. You can also use these numbers to calculate the velocity ratio (that is, the maximum velocity divided by the mean velocity) and the space velocity (that is, the product of the mean velocity and the substrate length).

Custom field functions and UDFs can be also used to calculate more complex measures of non-uniformity, such as the standard deviation and the gamma uniformity index.

```
           Mass-Weighted Average
                      X Velocity                 (m/s)
-------------------------------- --------------------
                            x=95            5.3014566
```

```
                 x=165           4.0395238
      ---------------- --------------------
                   Net           4.6774315

         Minimum of Facet Values
                    X Velocity             (m/s)
-------------------------------- --------------------
                          x=95           0.8805235
                         x=165           2.3446729
      ---------------- --------------------
                           Net           0.8805235

         Maximum of Facet Values
                    X Velocity             (m/s)
-------------------------------- --------------------
                          x=95           8.0334597
                         x=165           6.3715858
      ---------------- --------------------
                           Net           8.0334597
```

8.5. Summary

In this tutorial, you learned how to set up and solve a problem involving gas flow through porous media in ANSYS Fluent. You also learned how to perform appropriate postprocessing. Flow non-uniformities were easily identified through images of velocity vectors and pressure contours. Surface integrals and X-Y plots provided purely numeric data.

For additional details about modeling flow through porous media (including heat transfer and reaction modeling), see the Fluent User's Guide, available in the help viewer or on the ANSYS Customer Portal (http://support.ansys.com/documentation).

8.6. Further Improvements

This tutorial guides you through the steps to reach an initial solution. You may be able to obtain a more accurate solution by using an appropriate higher-order discretization scheme and by adapting the mesh. Mesh adaption can also ensure that the solution is independent of the mesh. These steps are demonstrated in Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121).

Chapter 9: Using a Single Rotating Reference Frame

This tutorial is divided into the following sections:

9.1. Introduction

This tutorial considers the flow within a 2D, axisymmetric, co-rotating disk cavity system. Understanding the behavior of such flows is important in the design of secondary air passages for turbine disk cooling.

This tutorial demonstrates how to do the following:

- Set up a 2D axisymmetric model with swirl, using a rotating reference frame.
- Use the standard k- ε and RNG k- ε turbulence models.
- Calculate a solution using the pressure-based solver.
- Display velocity vectors and contours of pressure.
- Set up and display XY plots of radial velocity and wall y^+ distribution.
- Restart the solver from an existing solution.

9.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

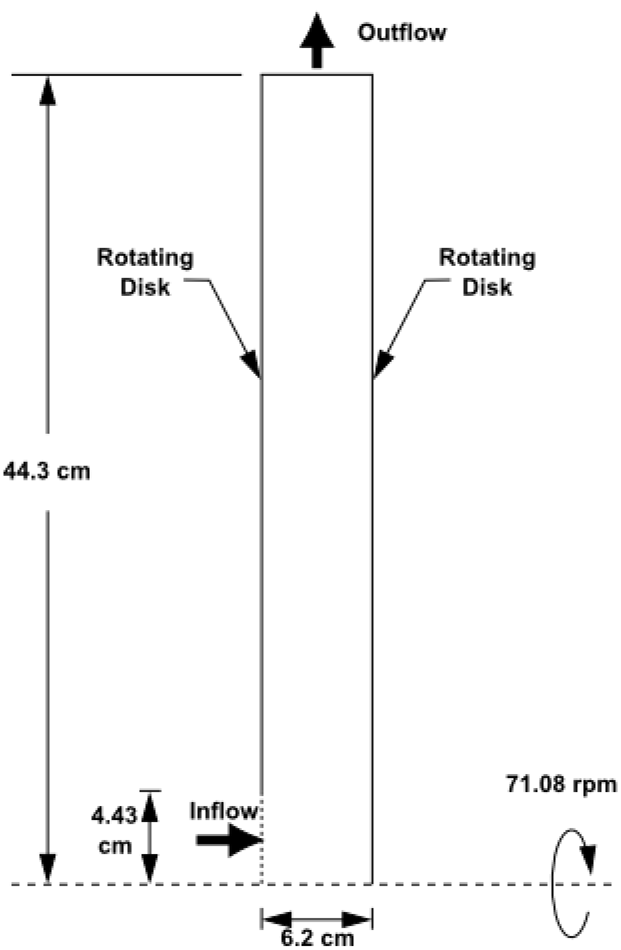
and that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

9.3. Problem Description

The problem to be considered is shown schematically in Figure 9.1: Problem Specification (p. 394). This case is similar to a disk cavity configuration that was extensively studied by Pincombe [1].

Air enters the cavity between two co-rotating disks. The disks are 88.6 cm in diameter and the air enters at 1.146 m/s through a circular bore 8.86 cm in diameter. The disks, which are 6.2 cm apart, are spinning at 71.08 rpm, and the air enters with no swirl. As the flow is diverted radially, the rotation of the disk has a significant effect on the viscous flow developing along the surface of the disk.

Figure 9.1: Problem Specification



As noted by Pincombe [1], there are two nondimensional parameters that characterize this type of disk cavity flow: the volume flow rate coefficient, C_w, and the rotational Reynolds number, Re_φ. These parameters are defined as follows:

$$C_w = \frac{Q}{\nu r_{out}} \tag{9.1}$$

$$Re_\varphi = \frac{\Omega r_{out}^2}{\nu} \tag{9.2}$$

where Q is the volumetric flow rate, Ω is the rotational speed, ν is the kinematic viscosity, and r_{out} is the outer radius of the disks. Here, you will consider a case for which $C_w = 1092$ and $Re_\varphi = 10^5$.

9.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

9.4.1. Preparation

To prepare for running this tutorial:

1. Set up a working folder on the computer you will be using.

2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

   Note

   If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

3. Enter the name of this tutorial into the search bar.

4. Narrow the results by using the filter on the left side of the page.

   a. Click **ANSYS Fluent** under **Product**.

   b. Click **18.0** under **Version**.

5. Select this tutorial from the list.

6. Click the **single_rotating_R180.zip** link to download the input files.

7. Unzip `single_rotating_R180.zip` to your working folder.

   The file `disk.msh` can be found in the `single_rotating` folder created after unzipping the file.

8. Use Fluent Launcher to start the **2D** single precision (disable **Double Precision**) version of ANSYS Fluent.

   Fluent Launcher displays your **Display Options** preferences from the previous session.

   For more information about Fluent Launcher, see starting ANSYS Fluent using the Fluent Launcher in the Fluent Getting Started Guide.

9. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.

10. Run in **Serial** under **Processing Options**.

9.4.2. Mesh

1. Read the mesh file (`disk.msh`).

   File → **Read** → **Mesh...**

   As ANSYS Fluent reads the mesh file, it will report its progress in the console.

   Note

   The Fluent console will display a warning that the current setup for the boundary conditions is not appropriate for a 2D/3D flow problem.

   You will resolve this issue when you modify the solver settings in a subsequent step.

9.4.3. General Settings

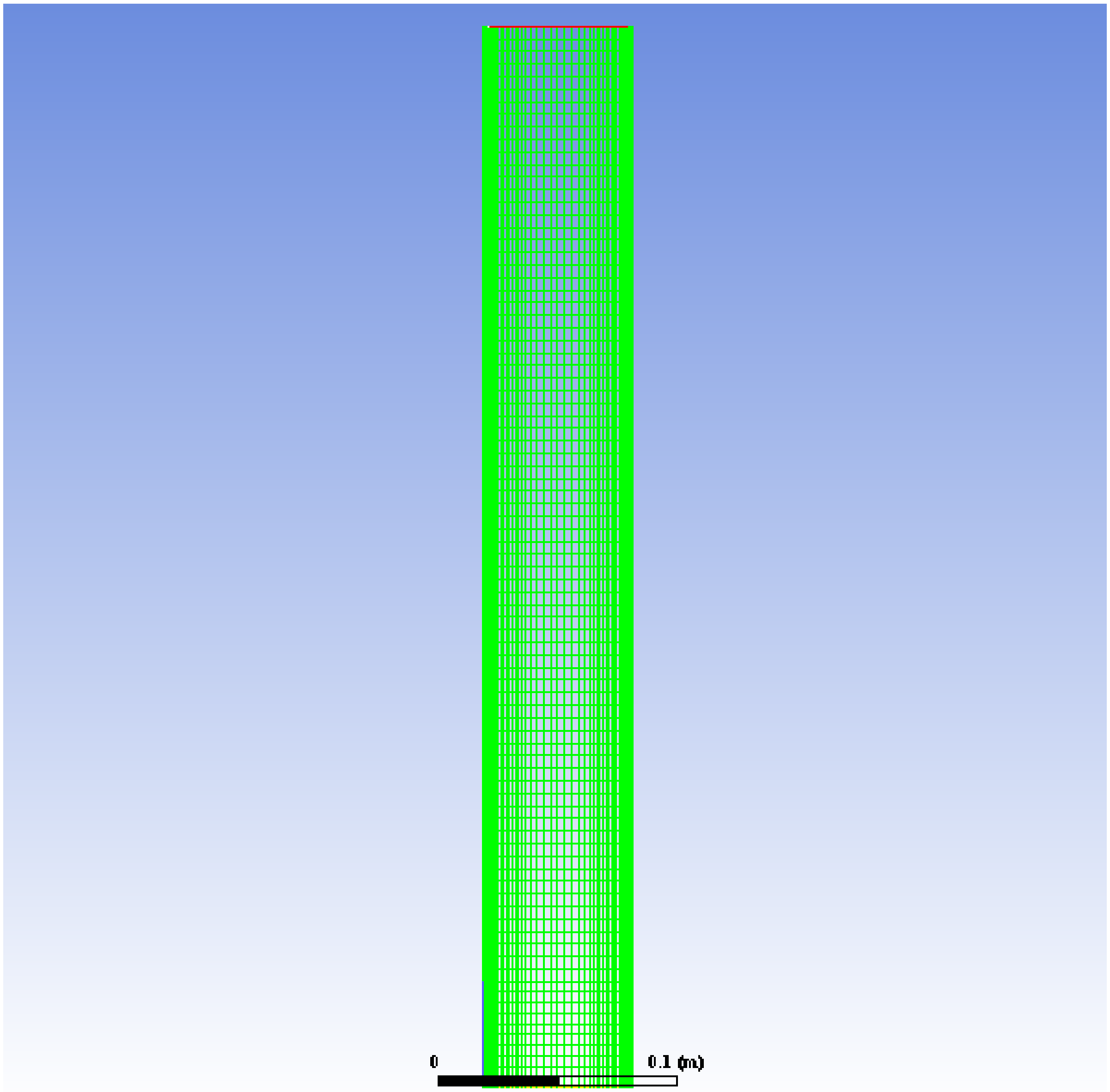
1. Check the mesh.

   Setting Up Domain → **Mesh** → **Check**

   ANSYS Fluent will perform various checks on the mesh and report the progress in the console. Make sure that the reported minimum volume is a positive number.

2. Examine the mesh (Figure 9.2: Mesh Display for the Disk Cavity (p. 397)).

Figure 9.2: Mesh Display for the Disk Cavity



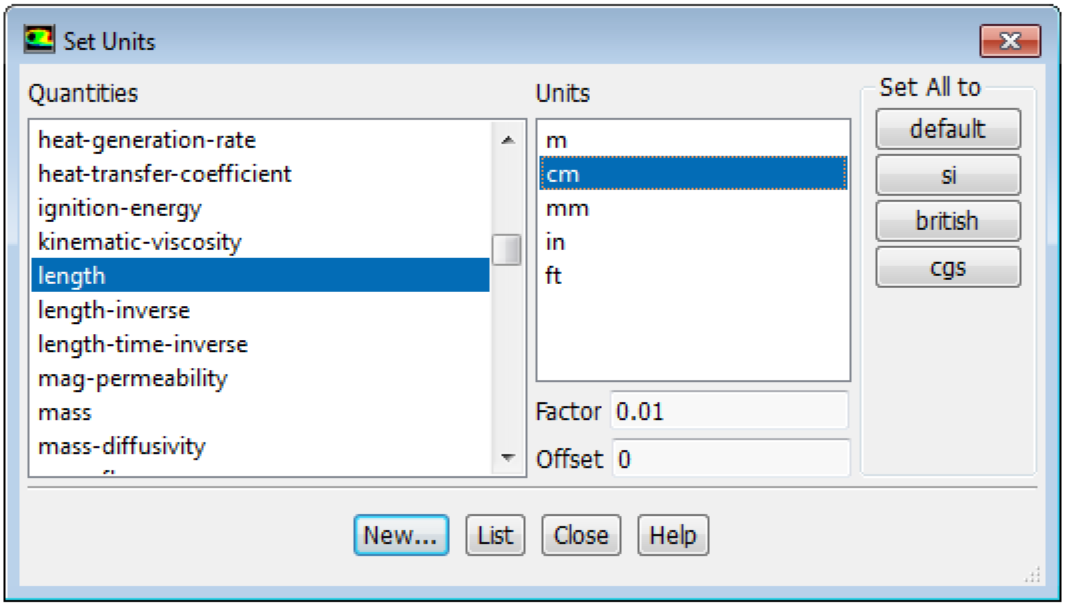
Extra

You can use the right mouse button to check which zone number corresponds to each boundary. If you click the right mouse button on one of the boundaries in the graphics window, information will be displayed in the ANSYS Fluent console about the associated zone, including the name of the zone. This feature is especially useful when you have several zones of the same type and you want to distinguish between them quickly.

3. Define new units for angular velocity and length.

Setting Up Domain → Mesh → Units...

In the problem description, angular velocity and length are specified in rpm and cm, respectively, which is more convenient in this case. These are not the default units for these quantities.



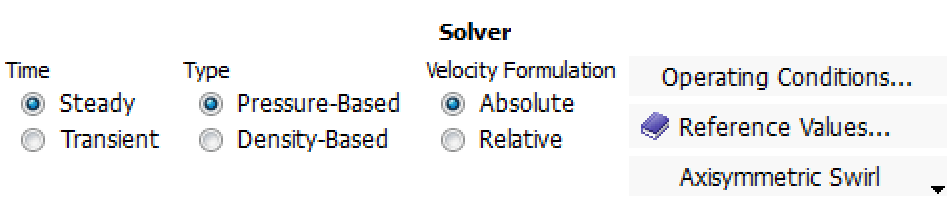
a. Select **angular-velocity** from the **Quantities** list, and **rpm** in the **Units** list.

b. Select **length** from the **Quantities** list, and **cm** in the **Units** list.

c. Close the **Set Units** dialog box.

4. Specify the solver formulation to be used for the model calculation and enable the modeling of axisymmetric swirl.

Setting Up Physics → Solver



a. Retain the default selection of **Pressure-Based** in the **Type** list.

b. Retain the default selection of **Absolute** in the **Velocity Formulation** list.

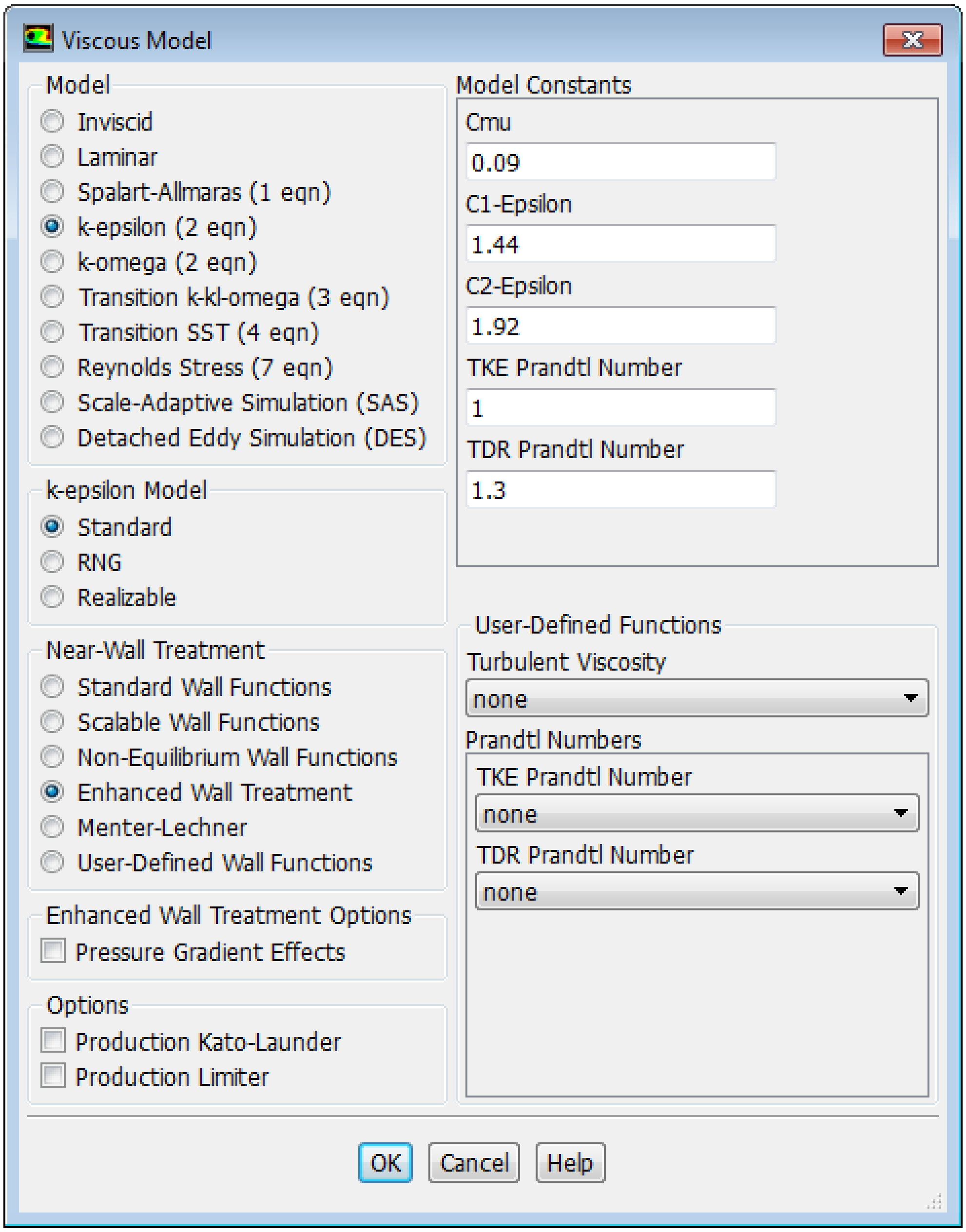
For a rotating reference frame, the absolute velocity formulation has some numerical advantages.

c. Select **Axisymmetric Swirl** from the drop-down list in the **Solver** group box (below **Reference Values...**).

9.4.4. Models

1. Enable the standard k- ε turbulence model with the enhanced near-wall treatment.

Setting Up Physics → Models → Viscous...



a. Select **k-epsilon (2 eqn)** in the **Model** list.

   *The **Viscous Model** dialog box will expand.*

b. Retain the default selection of **Standard** in the **k-epsilon Model** list.

c. Select **Enhanced Wall Treatment** in the **Near-Wall Treatment** list.

d. Click **OK** to close the **Viscous Model** dialog box.

   The ability to calculate a swirl velocity permits the use of a 2D mesh, so the calculation is simpler and more economical to run. This is especially important for problems where the enhanced wall treatment is used. The near-wall flow field is resolved through the viscous sublayer and buffer zones (that is, the first mesh point away from the wall is placed at a y^+ of the order of 1).

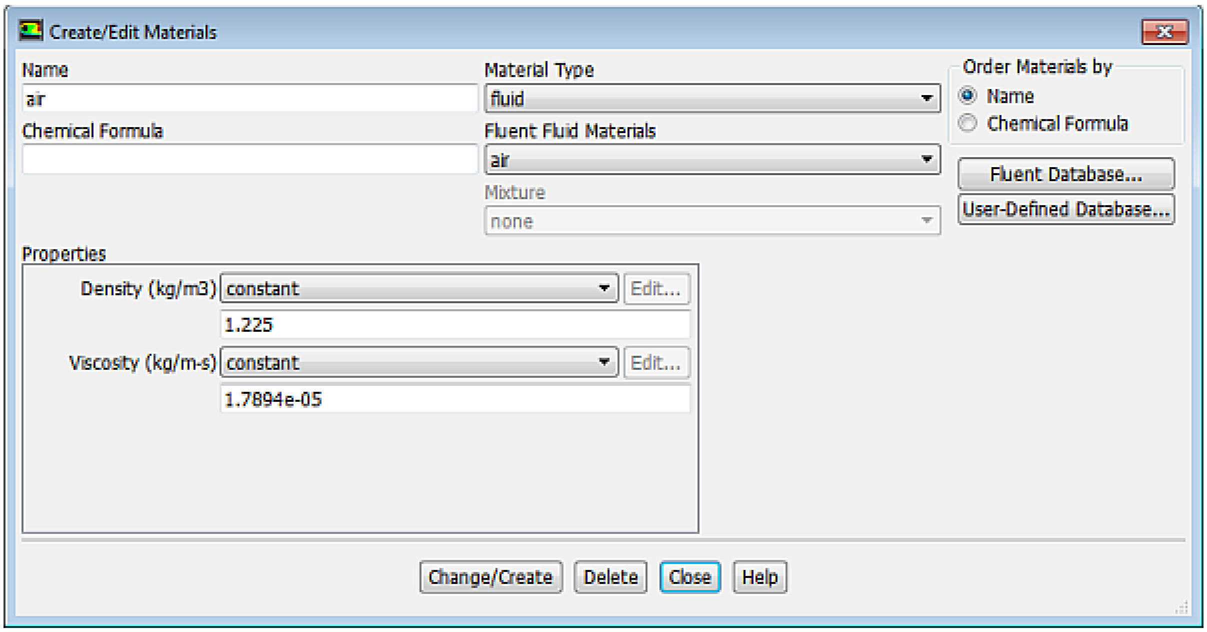
   For details, see enhanced wall treatment in the Fluent Theory Guide.

9.4.5. Materials

For the present analysis, you will model air as an incompressible fluid with a density of 1.225 kg/m^3 *and a dynamic viscosity of 1.7894 ×* 10^{-5} *kg/m-s. Since these are the default values, no change is required in the* ***Create/Edit Materials*** *dialog box.*

1. Retain the default properties for air.

Setting Up Physics → Materials → Create/Edit...



Extra

You can modify the fluid properties for air at any time or copy another material from the database.

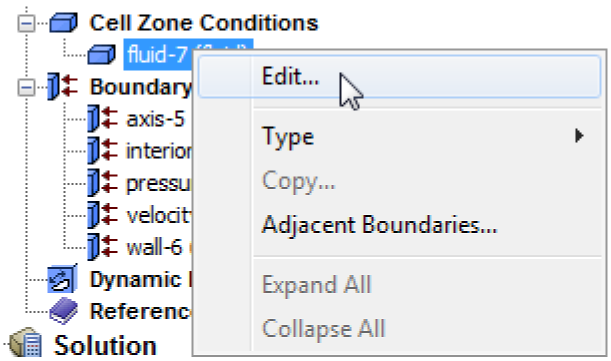
2. Click **Close** to close the **Create/Edit Materials** dialog box.

For details, see physical properties in the Fluent User's Guide.

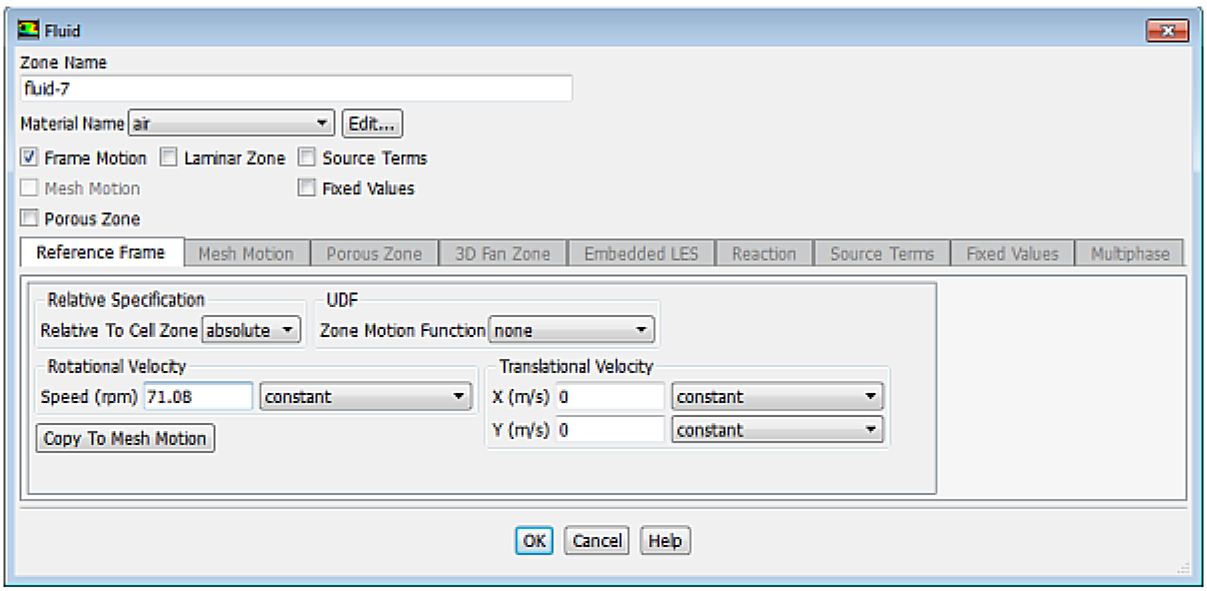
9.4.6. Cell Zone Conditions

Set up the present problem using a rotating reference frame for the fluid. Then define the disk walls to rotate with the moving frame.

Setup → Cell Zone Conditions → fluid-7 Edit...



1. Define the rotating reference frame for the fluid zone (**fluid-7**).



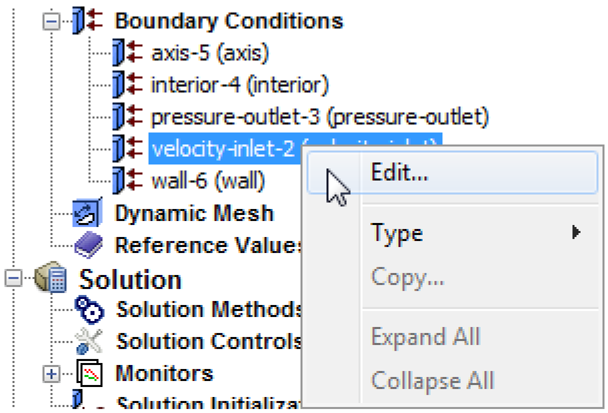
   a. Enable **Frame Motion.**

   b. Enter `71.08` rpm for **Speed** in the **Rotational Velocity** group box.

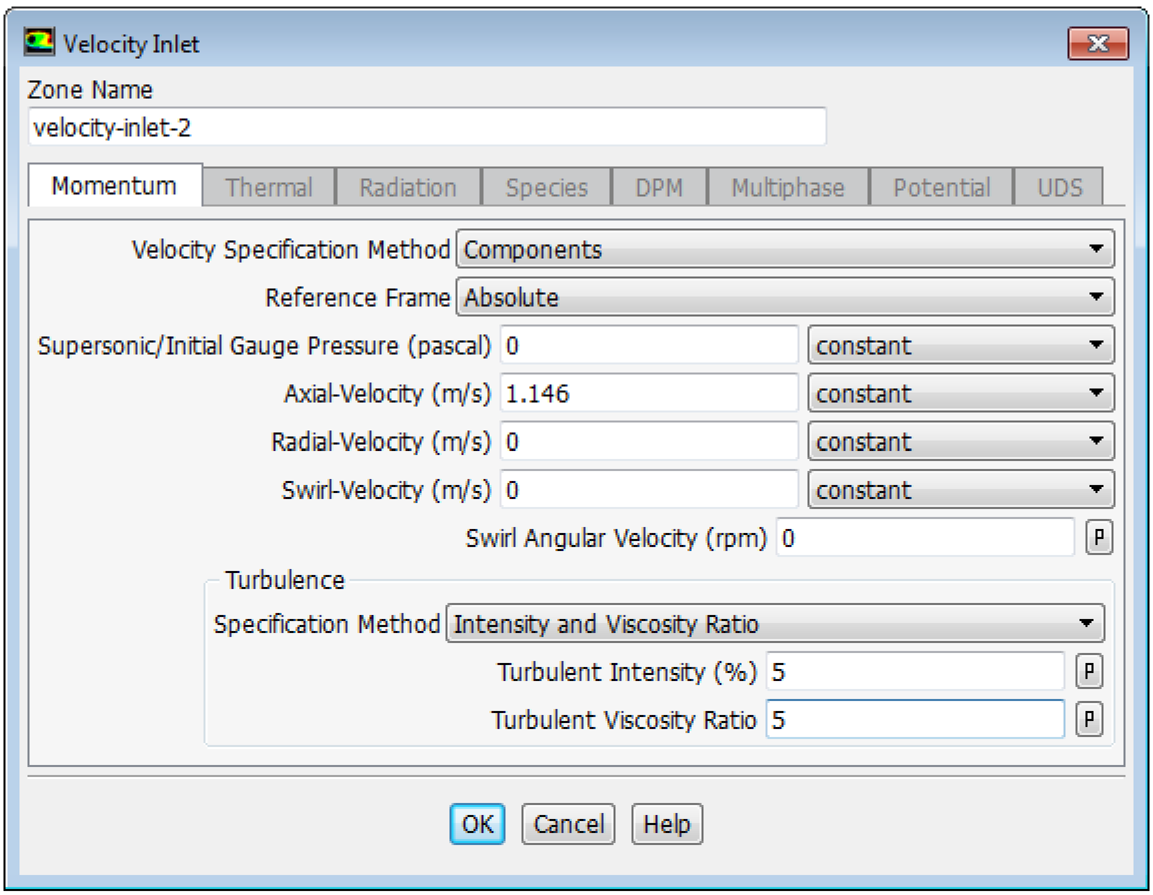
   c. Click **OK** to close the **Fluid** dialog box.

9.4.7. Boundary Conditions

Setup → Boundary Conditions → velocity-inlet-2 Edit...



1. Set the following conditions at the flow inlet (**velocity-inlet-2**).



   a. Select **Components** from the **Velocity Specification Method** drop-down list.

   b. Enter 1.146 m/s for **Axial-Velocity**.

   c. Retain the default selection of **Intensity and Viscosity Ratio** from the **Specification Method** drop-down list in the **Turbulence** group box.

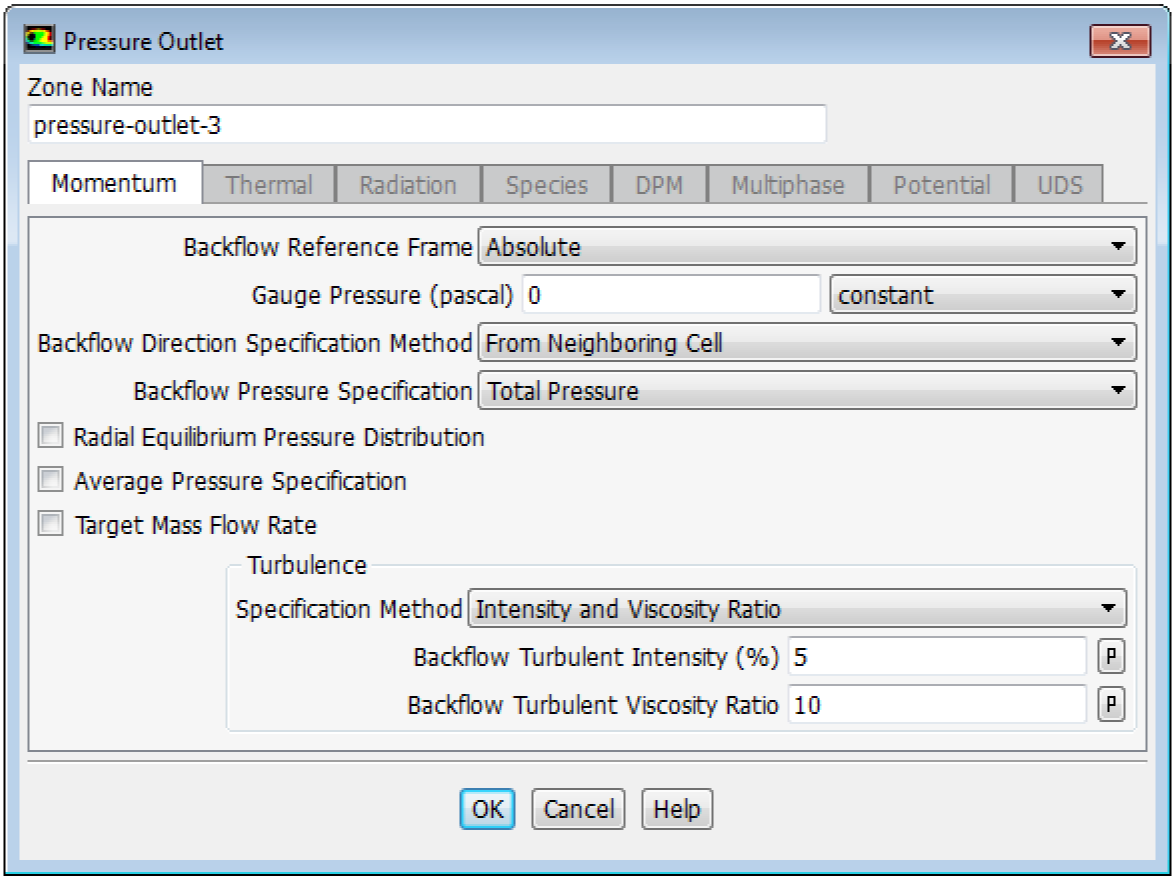
   d. Retain the default value of 5 % for **Turbulent Intensity**.

   e. Enter 5 for **Turbulent Viscosity Ratio**.

   f. Click **OK** to close the **Velocity Inlet** dialog box.

2. Set the following conditions at the flow outlet (**pressure-outlet-3**).

Setup → **Boundary Conditions** → **pressure-outlet-3** **Edit...**



a. Select **From Neighboring Cell** from the **Backflow Direction Specification Method** drop-down list.

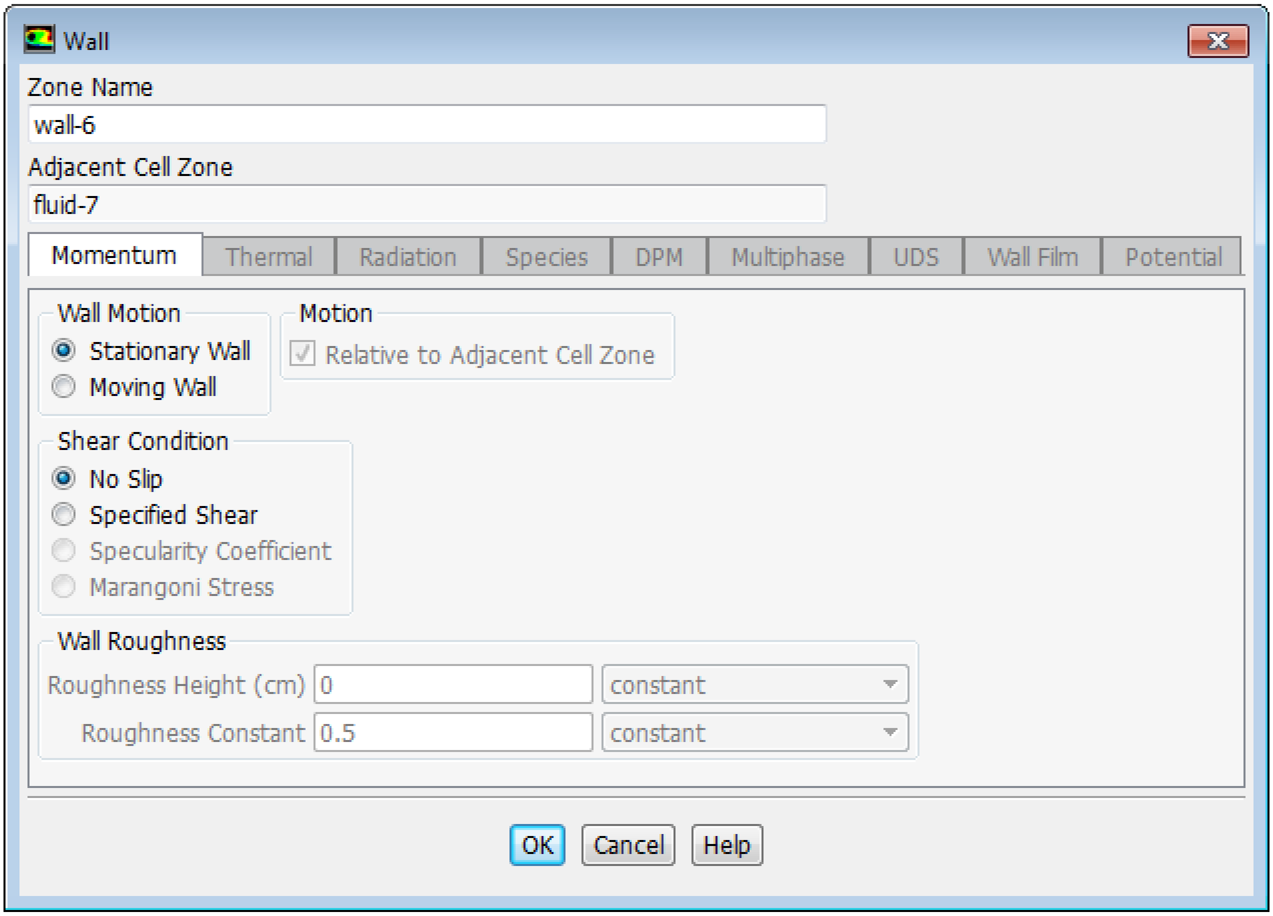
b. Click **OK** to close the **Pressure Outlet** dialog box.

Note

ANSYS Fluent will use the backflow conditions only if the fluid is flowing into the computational domain through the outlet. Since backflow might occur at some point during the solution procedure, you should set reasonable backflow conditions to prevent convergence from being adversely affected.

3. Confirm that the wall boundary condition settings for the disk walls (**wall-6**) are as shown below.

Setup → **Boundary Conditions** → **wall-6** **Edit...**



Note

A Stationary Wall condition implies that the wall is stationary with respect to the adjacent cell zone. Hence, in the case of a rotating reference frame a Stationary Wall is actually rotating with respect to the absolute reference frame. To specify a non-rotating wall in this case you would select Moving Wall (that is, moving with respect to the rotating reference frame). Then you would specify an absolute rotational speed of 0 in the Motion group box.

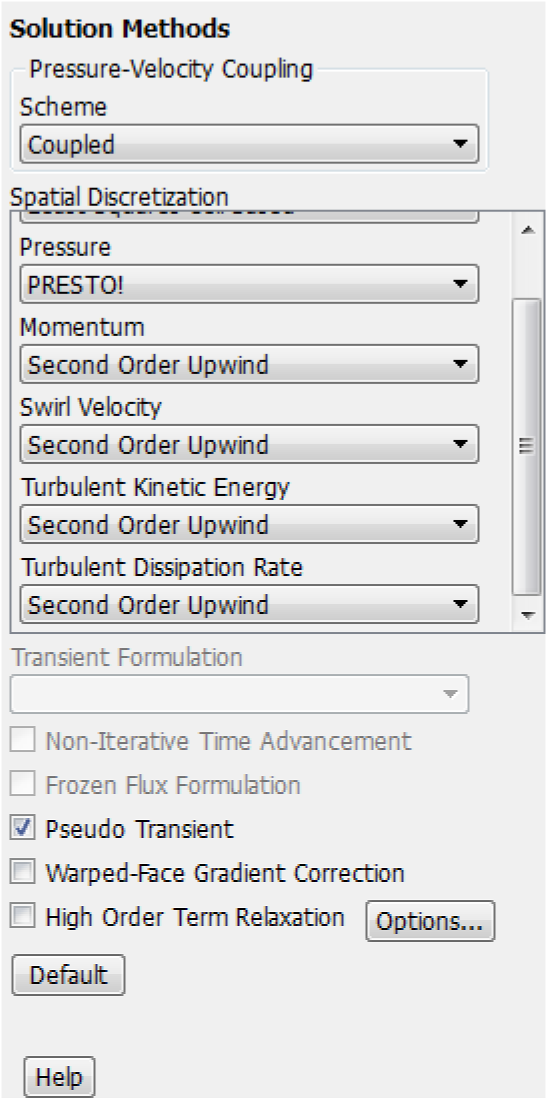
---

a. Click **OK** to close the **Wall** dialog box.

9.4.8. Solution Using the Standard k- ε Model

1. Set the solution parameters.

   Solving → **Solution** → **Methods...**



a. Select **Coupled** from the **Scheme** drop-down list in the **Pressure-Velocity Coupling** group box.

b. Retain the default selection of **Least Squares Cell Based** from the **Gradient** list in the **Spatial Discretization** group box.

c. Select **PRESTO!** from the **Pressure** drop-down list in the **Spatial Discretization** group box.

   The PRESTO! scheme is well suited for steep pressure gradients involved in rotating flows. It provides improved pressure interpolation in situations where large body forces or strong pressure variations are present as in swirling flows.

d. Select **Second Order Upwind** from the **Turbulent Kinetic Energy** and **Turbulent Dissipation Rate** drop-down lists.

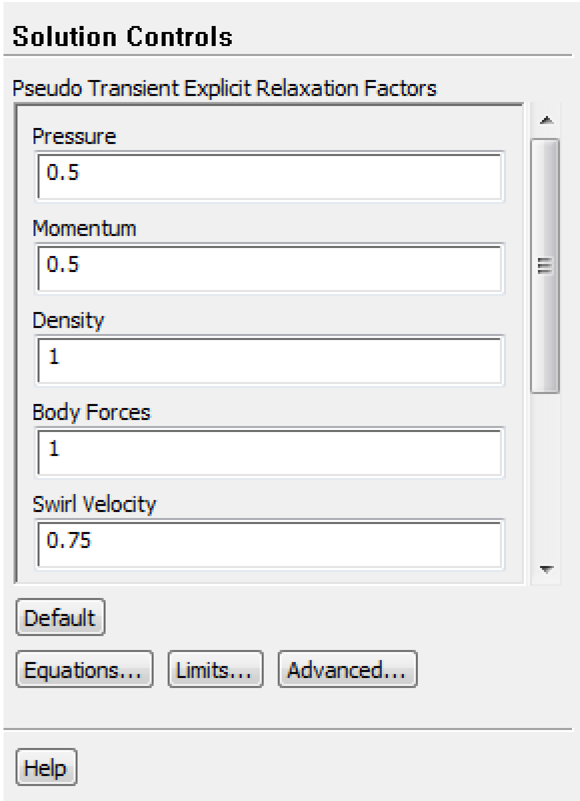
   Use the scroll bar to access the discretization schemes that are not initially visible in the task page.

e. Enable **Pseudo Transient**.

   The Pseudo Transient option enables the pseudo transient algorithm in the coupled pressure-based solver. This algorithm effectively adds an unsteady term to the solution equations in order to improve stability and convergence behavior. Use of this option is recommended for general fluid flow problems.

2. Set the solution controls.

   Solving → Controls → Controls...

   a. Retain the default values in the **Pseudo Transient Explicit Relaxation Factors** group box.

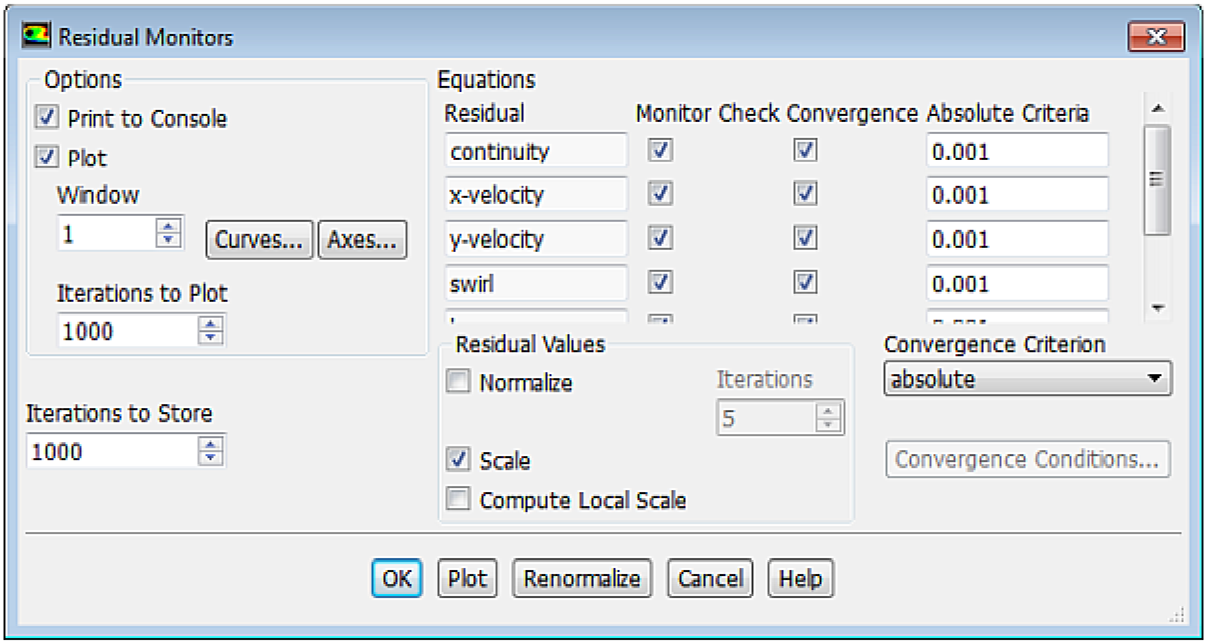
      Note

      For this problem, the default explicit relaxation factors are satisfactory. However, if the solution diverges or the residuals display large oscillations, you may need to reduce the relaxation factors from their default values.

      For tips on how to adjust the explicit relaxation parameters for different situations, see the Fluent User's Guide.

3. Enable the plotting of residuals during the calculation.

   Solving → Reports → Residuals...



a. Ensure that **Plot** is enabled in the **Options** group box.

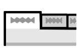
b. Click **OK** to close the **Residual Monitors** dialog box.

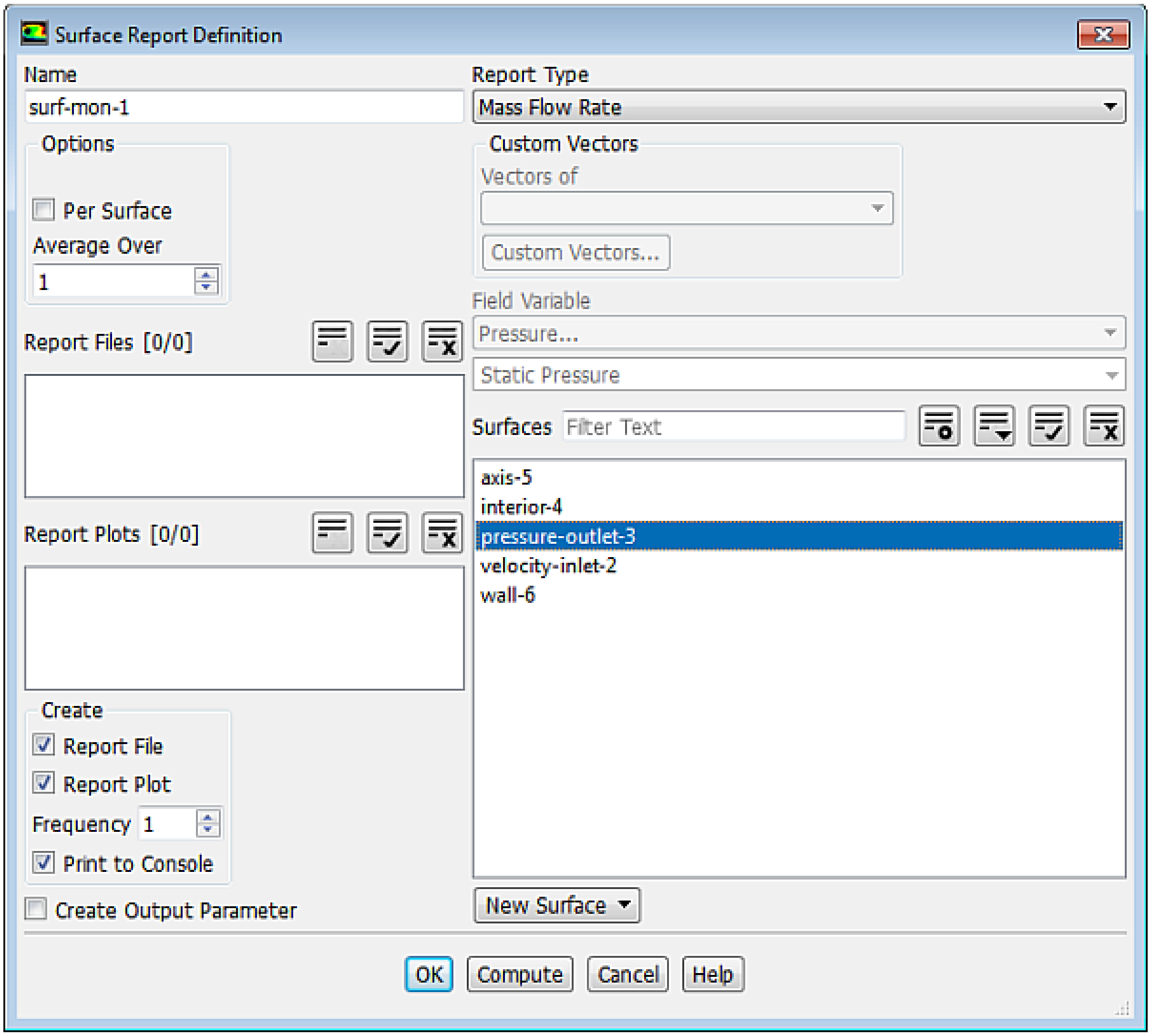
Note

For this calculation, the convergence tolerance on the continuity equation is kept at 0.001. Depending on the behavior of the solution, you can reduce this value if necessary.

---

4. Enable the plotting of mass flow rate at the flow exit.

 **Solving** → **Reports** → **Definitions** → **New** → **Surface Report** → **Mass Flow Rate**



a. Enter **`surf-mon-1`** for the **Name** of the surface report definition.

b. In the **Create** group box, enable **Report File**, **Report Plot** and **Print to Console** for **`surf-mon-1`**.

Note

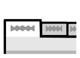
When the **Report File** option is selected in the **Surface Report Definition** dialog box, the mass flow rate history will be written to a file. If you do not enable the **Report File** option, the history information will be lost when you exit ANSYS Fluent.

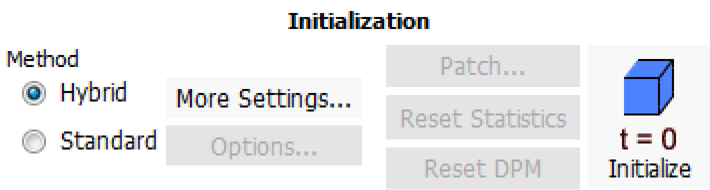
---

c. Select **pressure-outlet-3** from the **Surfaces** selection list.

d. Click **OK** to save the surface report definition settings and close the **Surface Report Definition** dialog box.

5. Initialize the solution.

 **Solving → Initialization**



a. Retain the default selection of the **Hybrid** initialization method.

b. Click **Initialize**.

Note

For flows in complex topologies, hybrid initialization will provide better initial velocity and pressure fields than standard initialization. This in general will help in improving the convergence behavior of the solver.

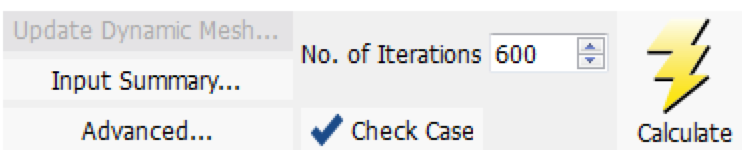
---

6. Save the case file (`disk-ke.cas.gz`).

   File → **Write** → **Case...**

7. Start the calculation by requesting 600 iterations.

   Run Calculation

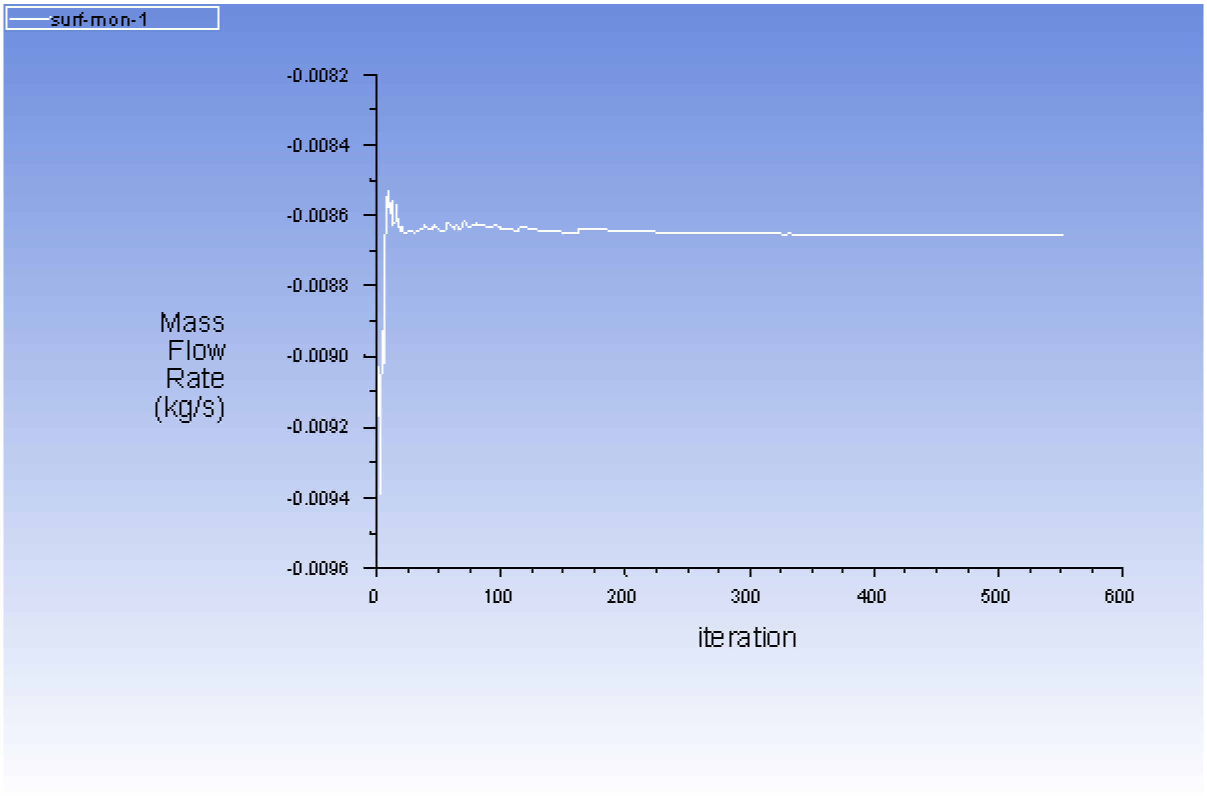


a. Enter `600` for the **Number of Iterations**.

b. Click **Calculate**.

Throughout the calculation, ANSYS Fluent will report reversed flow at the exit. This is reasonable for the current case. The solution should be sufficiently converged after approximately 550 iterations. The mass flow rate history is shown in *Figure 9.3: Mass Flow Rate History (k- ε Turbulence Model) (p. 410)**.*

Figure 9.3: Mass Flow Rate History (k- ε Turbulence Model)

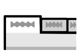


Extra

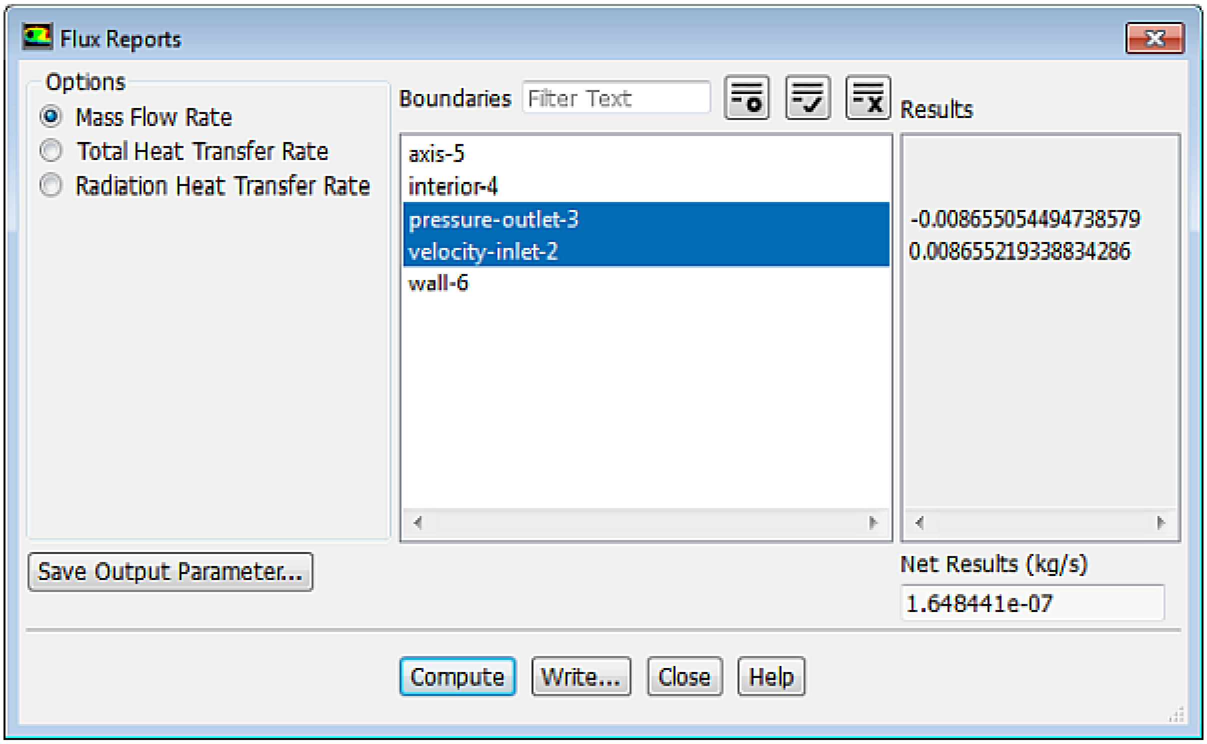
Here we have retained the default **Timescale Factor** of 1 in the **Run Calculation** panel. When performing a Pseudo Transient calculation, larger values of **Timescale Factor** may speed up convergence of the solution. However, setting **Timescale Factor** too large may cause the solution to diverge and fail to complete. As an optional activity, you can re-initialize the solution and try running the calculation with **Timescale Factor** set to 2. Observe the convergence behavior and the number of iterations before convergence. Then try the same again with **Timescale Factor** set to 4. For more information on setting Timescale Factor and the Pseudo Transient solver settings, refer to the Fluent User's Guide.

---

8. Check the mass flux balance.

 **Postprocessing** → **Reports** → **Fluxes...**

> **Warning**
>
> Although the mass flow rate history indicates that the solution is converged, you should also check the net mass fluxes through the domain to ensure that mass is being conserved.



a. Select **velocity-inlet-2** and **pressure-outlet-3** from the **Boundaries** selection list.

b. Retain the default **Mass Flow Rate** option.

c. Click **Compute** and close the **Flux Reports** dialog box.

Warning

The net mass imbalance should be a small fraction (for example, 0.5%) of the total flux through the system. If a significant imbalance occurs, you should decrease the residual tolerances by at least an order of magnitude and continue iterating.

9. Save the case and data files (`disk-ke.cas.gz` and `disk-ke.dat.gz`).

File → Write → Case & Data...

Click **OK** to overwrite **disk-ke.cas.gz**.

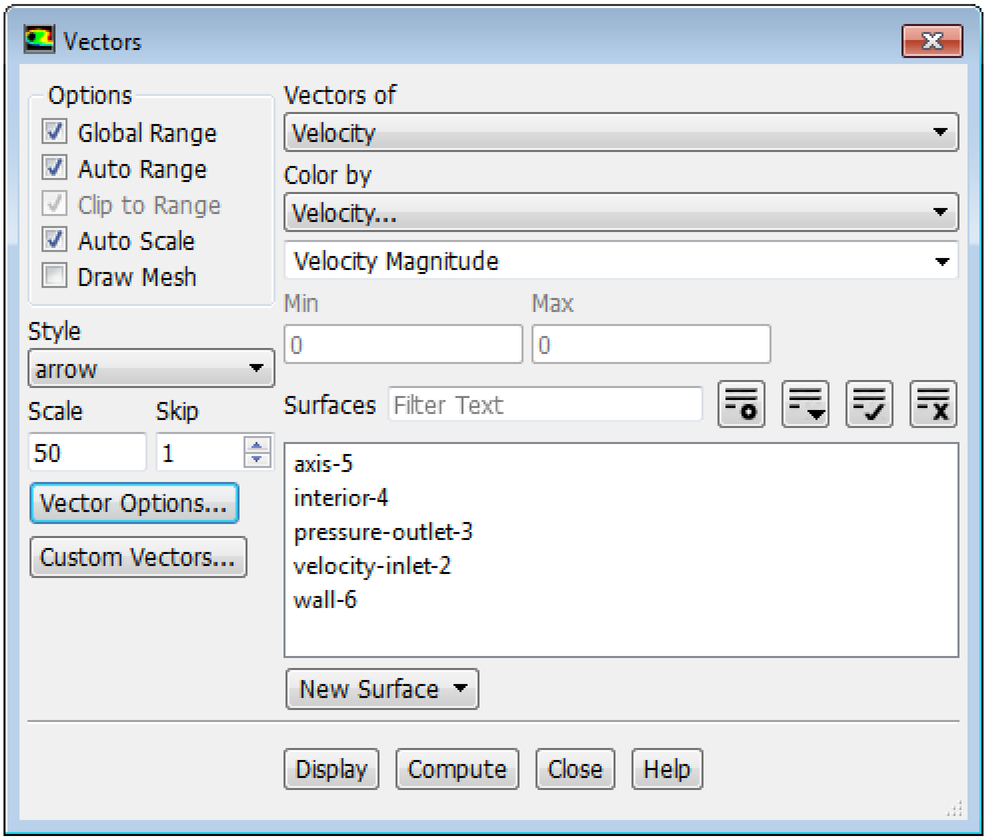
Note

It is always prudent to save both case and data files in case anything has changed.

9.4.9. Postprocessing for the Standard k- ε Solution

1. Display the velocity vectors.

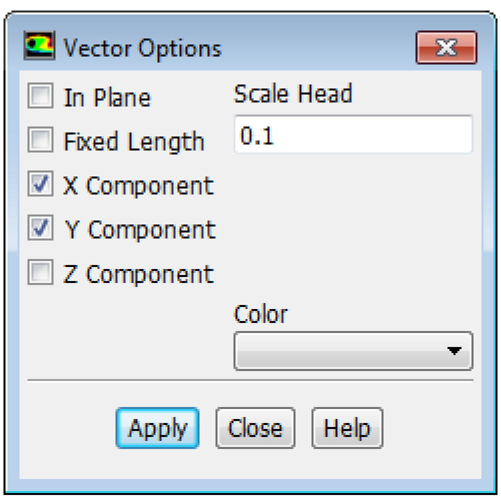
Postprocessing → Graphics → Vectors → Edit...



a. Enter 50 for **Scale**

b. Set **Skip** to 1.

c. Click the **Vector Options...** button to open the **Vector Options** dialog box.



i. Disable **Z Component**.

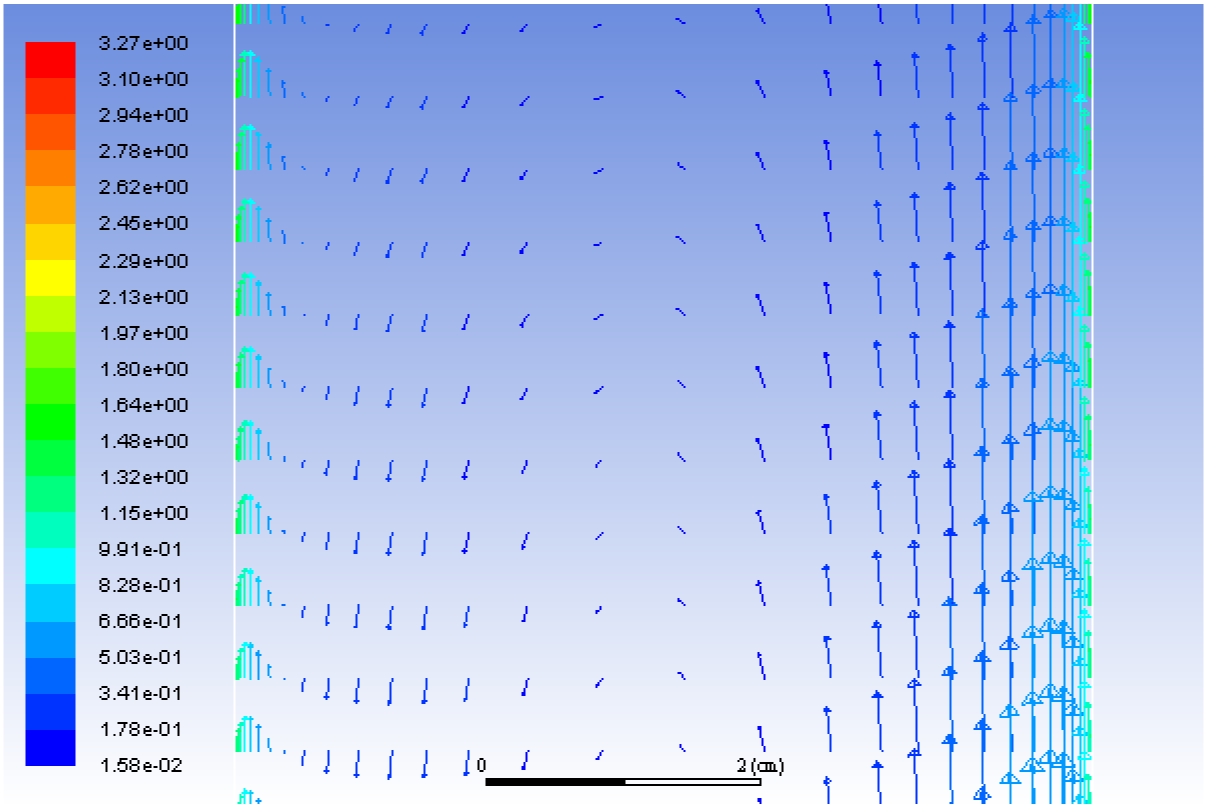
This allows you to examine only the non-swirling components.

ii. Click **Apply** and close the **Vector Options** dialog box.

d. Click **Display** in the **Vectors** dialog box to plot the velocity vectors.

A magnified view of the velocity field displaying a counter-clockwise circulation of the flow is shown in Figure 9.4: Magnified View of Velocity Vectors within the Disk Cavity (p. 413).

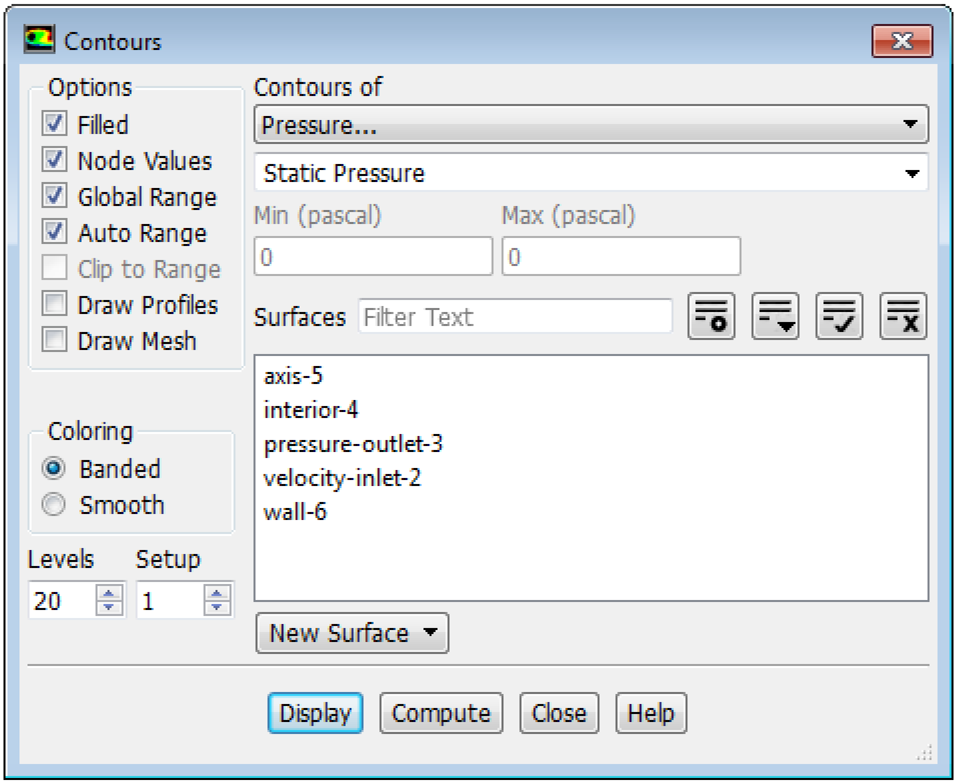
Figure 9.4: Magnified View of Velocity Vectors within the Disk Cavity



e. Close the **Vectors** dialog box.

2. Display filled contours of static pressure.

Postprocessing → **Graphics** → **Contours** → **Edit...**



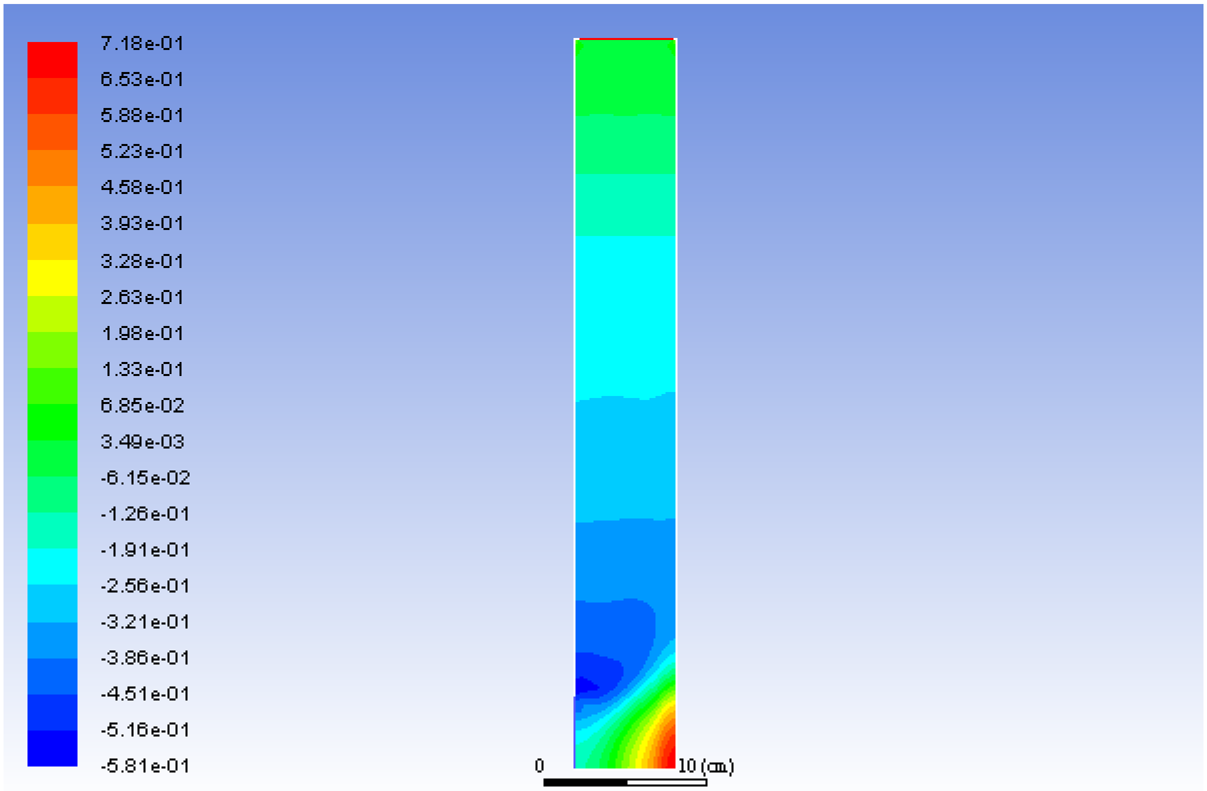
a. Enable **Filled** in the **Options** group box.

b. Retain the selection of **Pressure...** and **Static Pressure** from the **Contours of** drop-down lists.

c. Click **Display** and close the **Contours** dialog box.

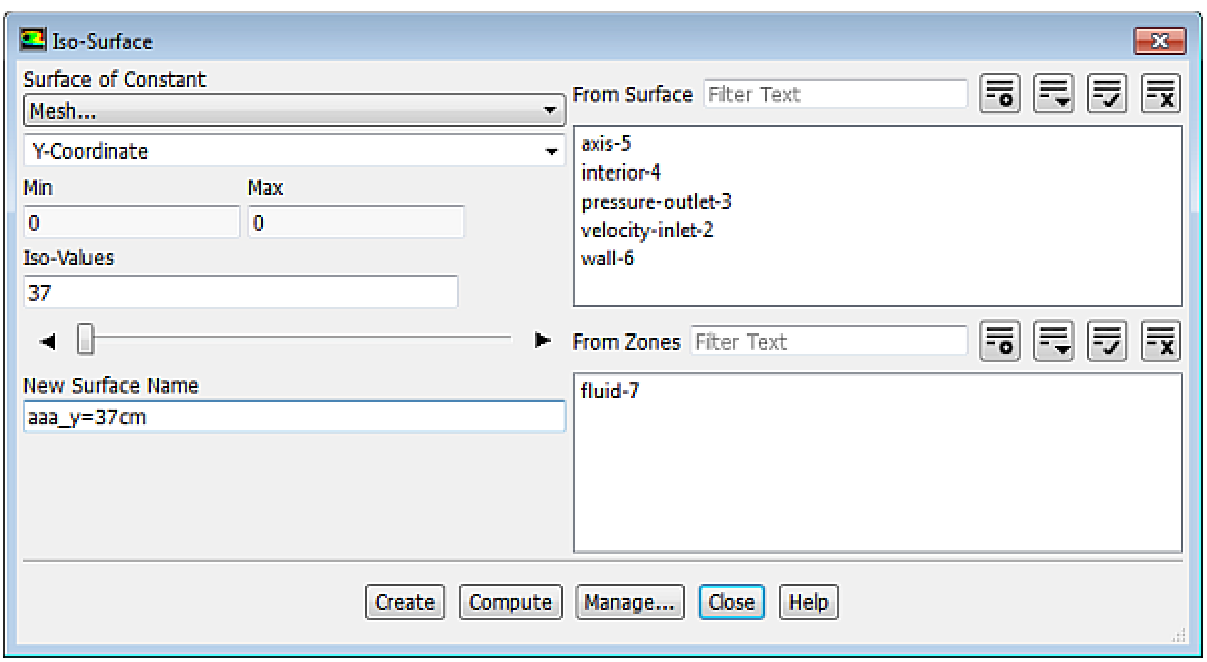
The pressure contours are displayed in Figure 9.5: Contours of Static Pressure for the Entire Disk Cavity (p. 415). Notice the high pressure that occurs on the right disk near the hub due to the stagnation of the flow entering from the bore.

Figure 9.5: Contours of Static Pressure for the Entire Disk Cavity



3. Create a constant y-coordinate line for postprocessing.

Postprocessing → **Surface** → **Create** → **Iso-Surface...**



a. Select **Mesh...** and **Y-Coordinate** from the **Surface of Constant** drop-down lists.

b. Click **Compute** to update the minimum and maximum values.

c. Enter 37 in the **Iso-Values** field.

This is the radial position along which you will plot the radial velocity profile.

d. Enter `aaa_y=37cm` for the **New Surface Name**.

*Using a prefix such as **aaa** or **zzz** allows you to keep all postprocessing surfaces together.*

e. Click **Create** to create the isosurface.

Note

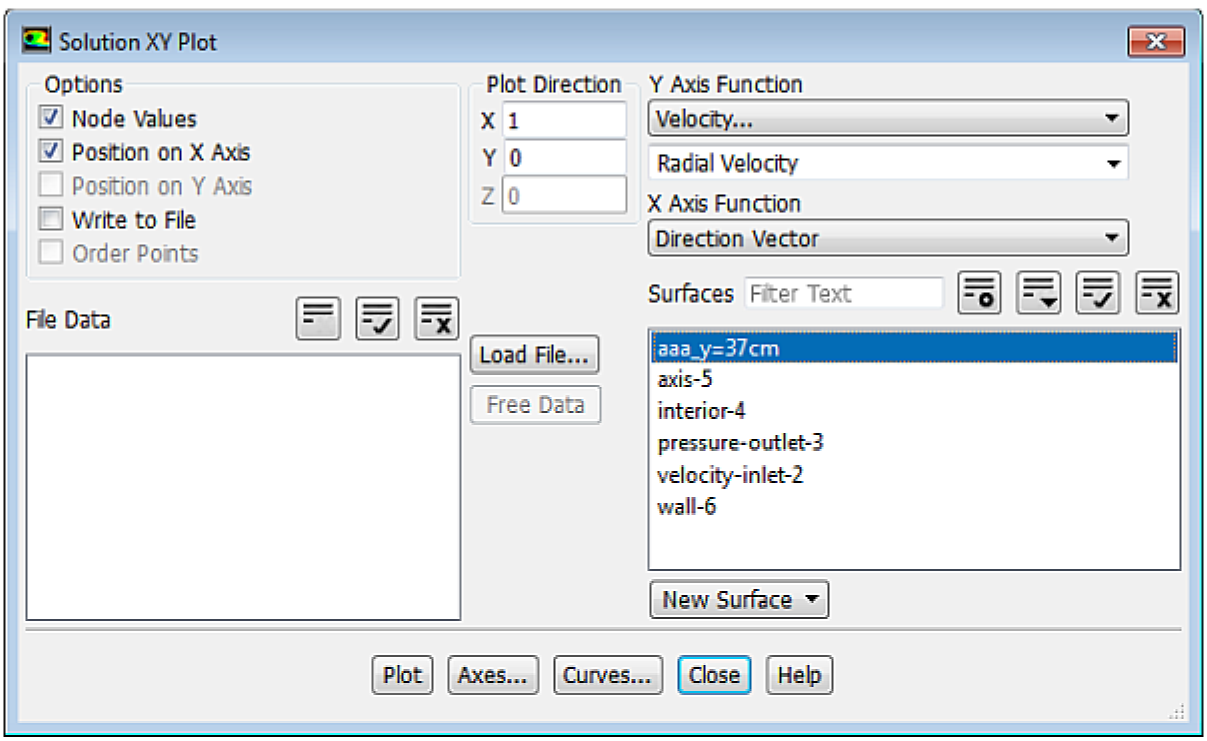
The name you use for an isosurface can be any continuous string of characters (without spaces).

---

f. Close the **Iso-Surface** dialog box.

4. Plot the radial velocity distribution on the surface **aaa_y=37cm**.

Postprocessing → **Plots** → **XY Plot** → **Edit...**



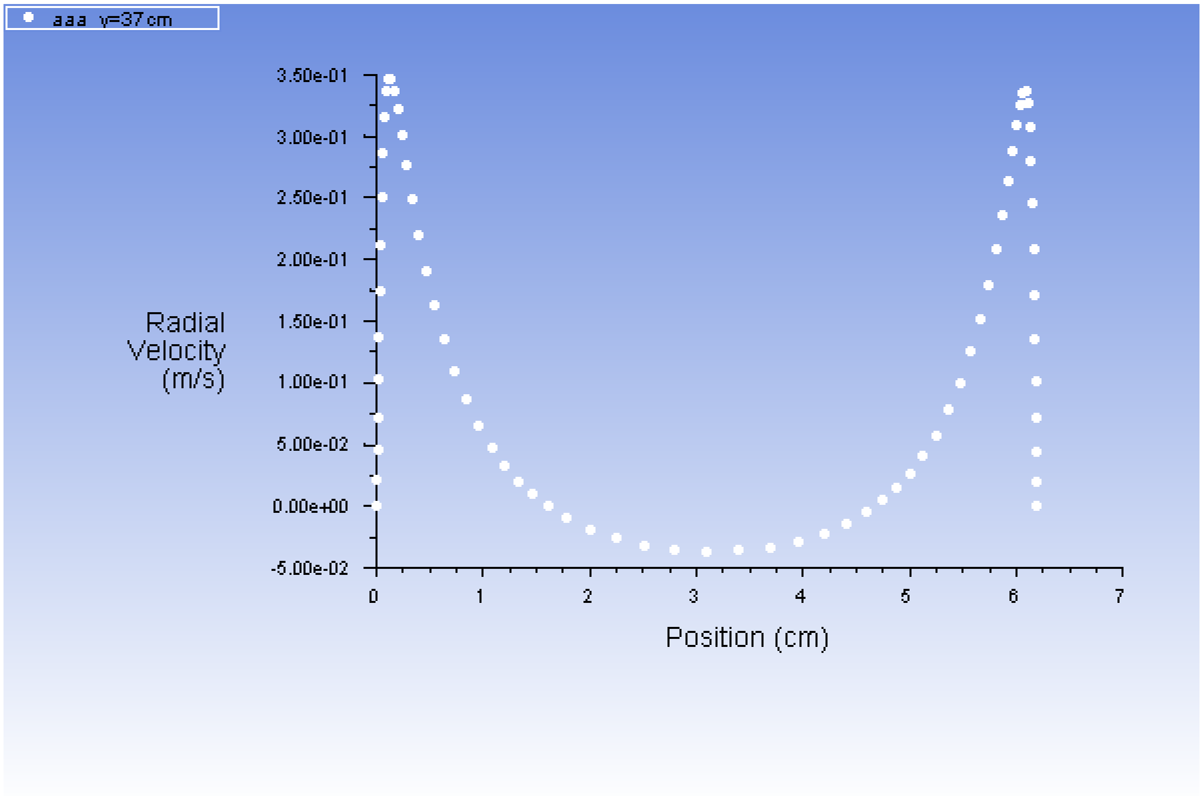
a. Select **Velocity...** and **Radial Velocity** from the **Y Axis Function** drop-down lists.

b. Select the y-coordinate line **aaa_y=37cm** from the **Surfaces** selection list.

c. Click **Plot**.

Figure 9.6: Radial Velocity Distribution—Standard k- ε Solution (p. 417) shows a plot of the radial velocity distribution along y=37cm.

Figure 9.6: Radial Velocity Distribution—Standard k- ε Solution



d. Enable **Write to File** in the **Options** group box to save the radial velocity profile.

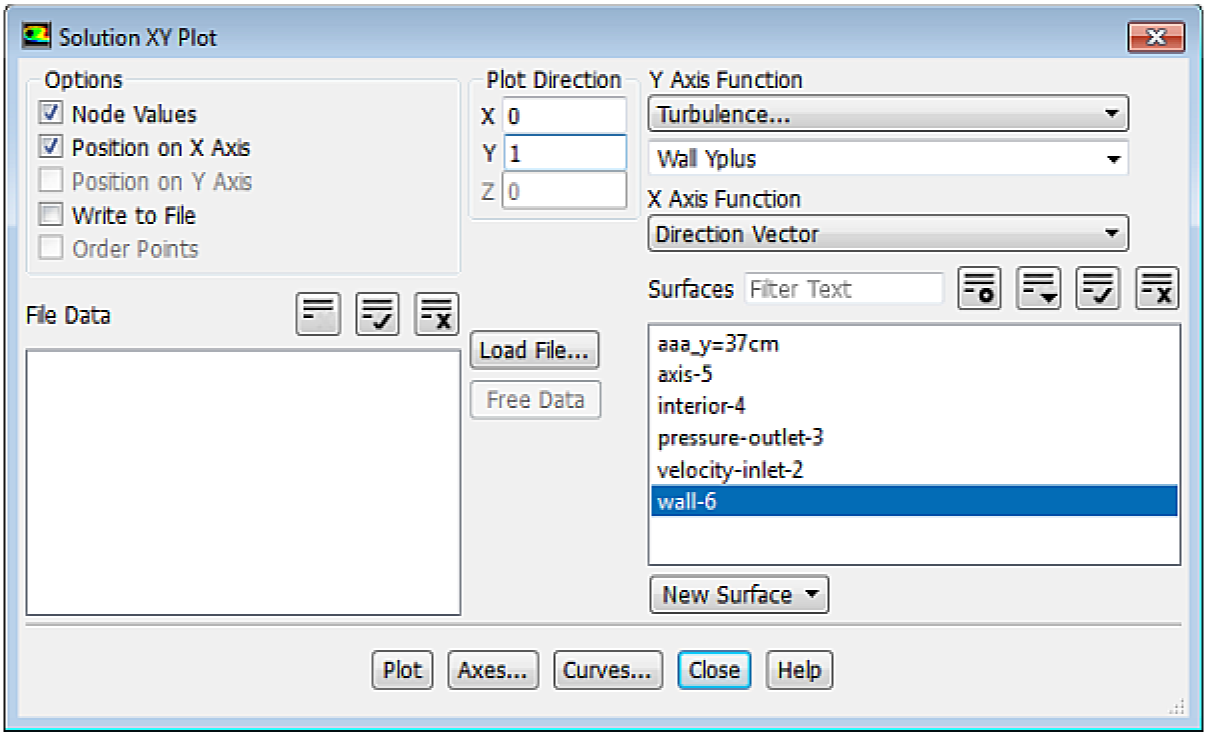
e. Click the **Write...** button to open the **Select File** dialog box.

   i. Enter `ke-data.xy` in the **XY File** text entry box and click **OK**.

      Be sure to double check the location where the files are being saved to ensure they will be saved where you intend.

5. Plot the wall y+ distribution on the rotating disk wall along the radial direction (Figure 9.7: Wall Yplus Distribution on wall-6— Standard k- ε Solution (p. 420)).

   Postprocessing → **Plots** → **XY Plot** → **Edit...**



a. Disable **Write to File** in the **Options** group box.

b. Select **Turbulence...** and **Wall Yplus** from the **Y Axis Function** drop-down lists.

c. Deselect **aaa_y=37cm** and select **wall-6** from the **Surfaces** selection list.

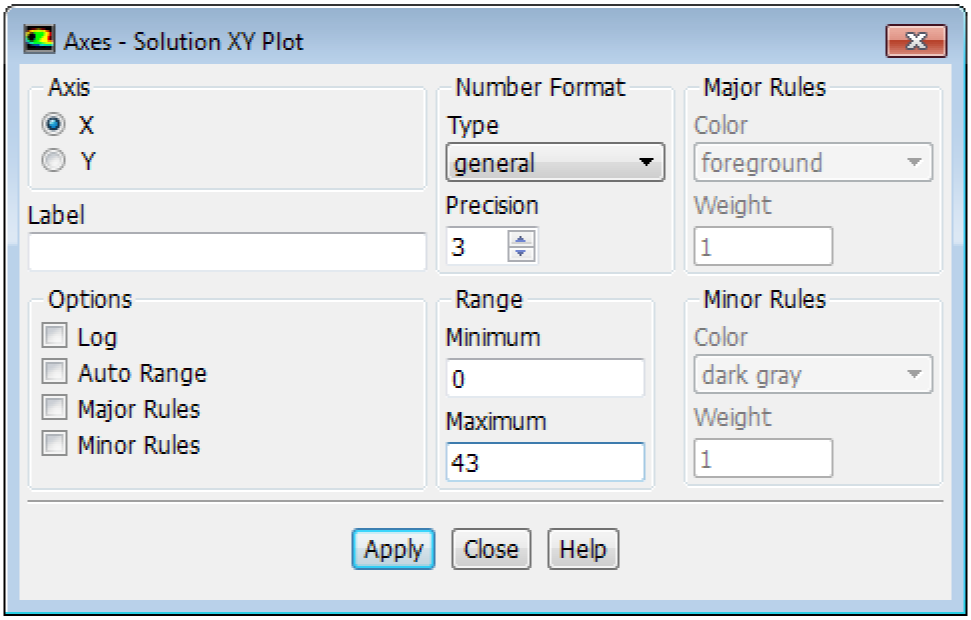
d. Enter `0` and `1` for **X** and **Y** respectively in the **Plot Direction** group box.

Note

The change in Plot Direction is required because we are plotting y+ along the radial dimension of the disk, which is oriented with Y axis.

---

e. Click the **Axes...** button to open the **Axes - Solution XY Plot** dialog box.



i. Retain the default selection of **X** from the **Axis** group box.

ii. Disable **Auto Range** in the **Options** group box.

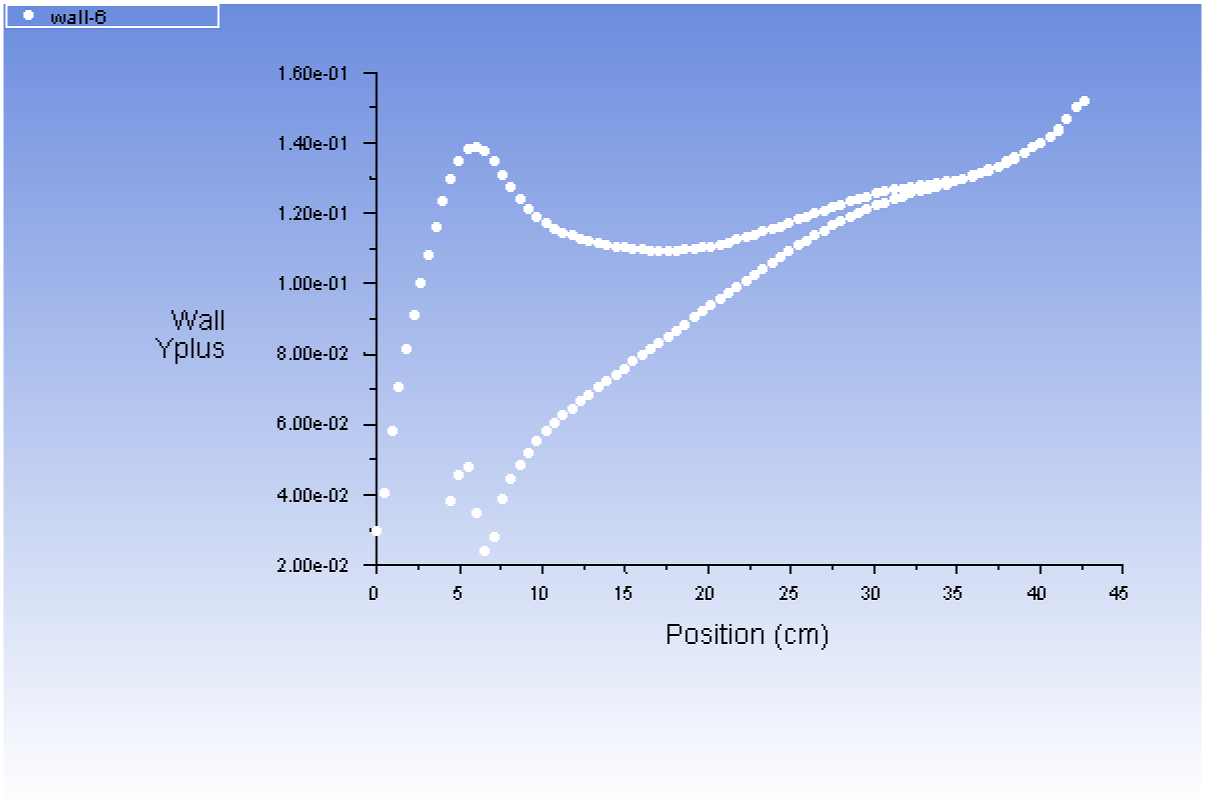
iii. Retain the default value of `0` for **Minimum** and enter 43 for **Maximum** in the **Range** group box.

iv. Click **Apply** and close the **Axes - Solution XY Plot** dialog box.

f. Click **Plot** in the **Solution XY Plot** dialog box.

*Figure 9.7: Wall Yplus Distribution on wall-6— Standard k- ε Solution(p. 420) shows a plot of wall y+ distribution along **wall-6**.*

Figure 9.7: Wall Yplus Distribution on wall-6— Standard k- ε Solution



g. Enable **Write to File** in the **Options** group box to save the wall y+ profile.

h. Click the **Write...** button to open the **Select File** dialog box.

   i. Enter `ke-yplus.xy` in the **XY File** text entry box and click **OK**.

Note

Ideally, while using enhanced wall treatment, the wall y+ should be in the order of 1 (at least less than 5) to resolve the viscous sublayer. The plot justifies the applicability of enhanced wall treatment to the given mesh.

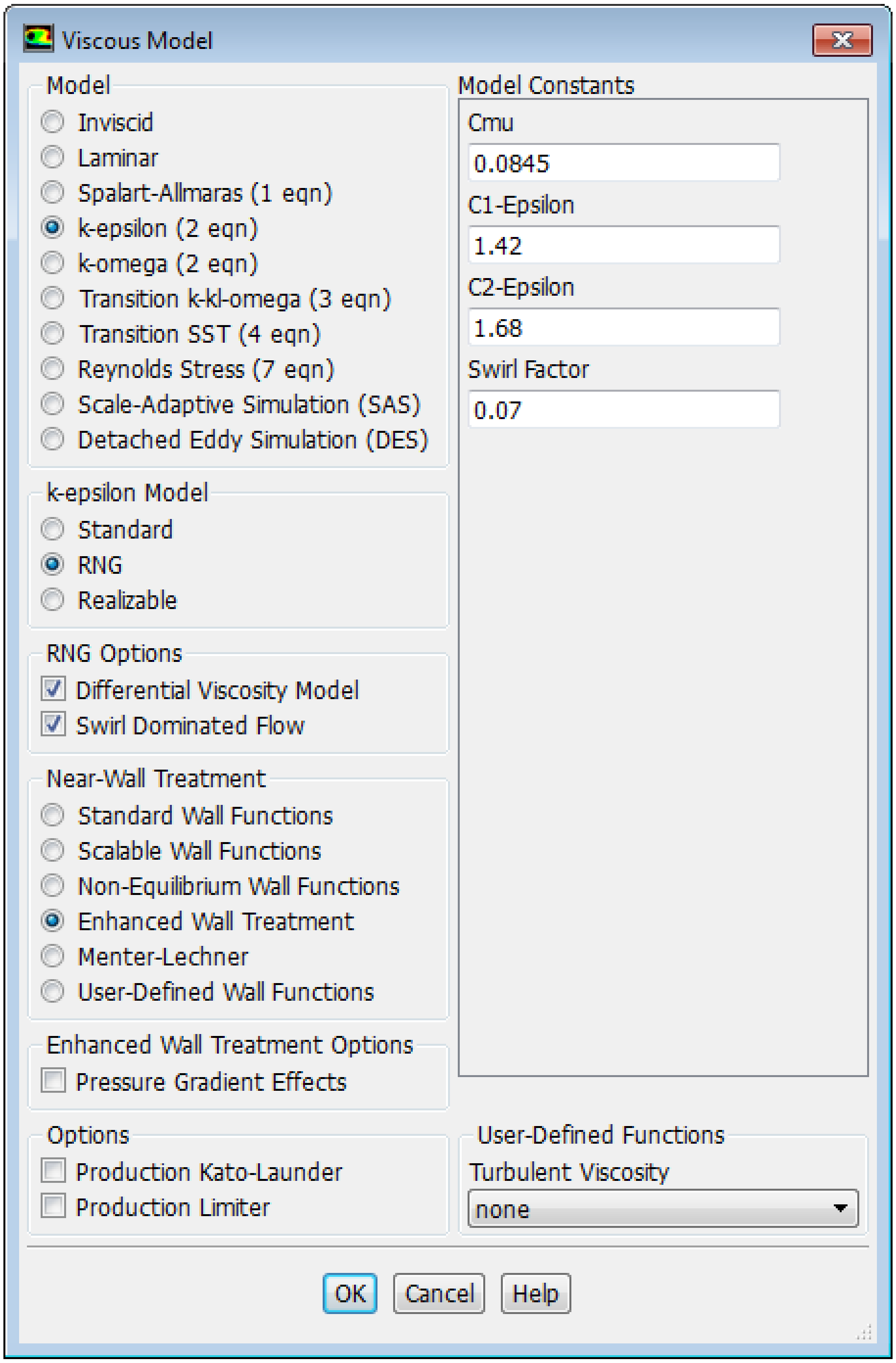
i. Close the **Solution XY Plot** dialog box.

9.4.10. Solution Using the RNG k- ε Model

Recalculate the solution using the RNG k- ε turbulence model.

1. Enable the RNG k- ε turbulence model with the enhanced near-wall treatment.

   Setting Up Physics → **Models** → **Viscous...**



a. Select **RNG** in the **k-epsilon Model** list.

b. Enable **Differential Viscosity Model** and **Swirl Dominated Flow** in the **RNG Options** group box.

   The differential viscosity model and swirl modification can provide better accuracy for swirling flows such as the disk cavity.

   For more information, see RNG swirl modification in the Fluent Theory Guide.

c. Retain **Enhanced Wall Treatment** as the **Near-Wall Treatment**.

d. Click **OK** to close the **Viscous Model** dialog box.

2. Continue the calculation by requesting 300 iterations.

   Solving → **Run Calculation**

   The solution converges after approximately 245 additional iterations.

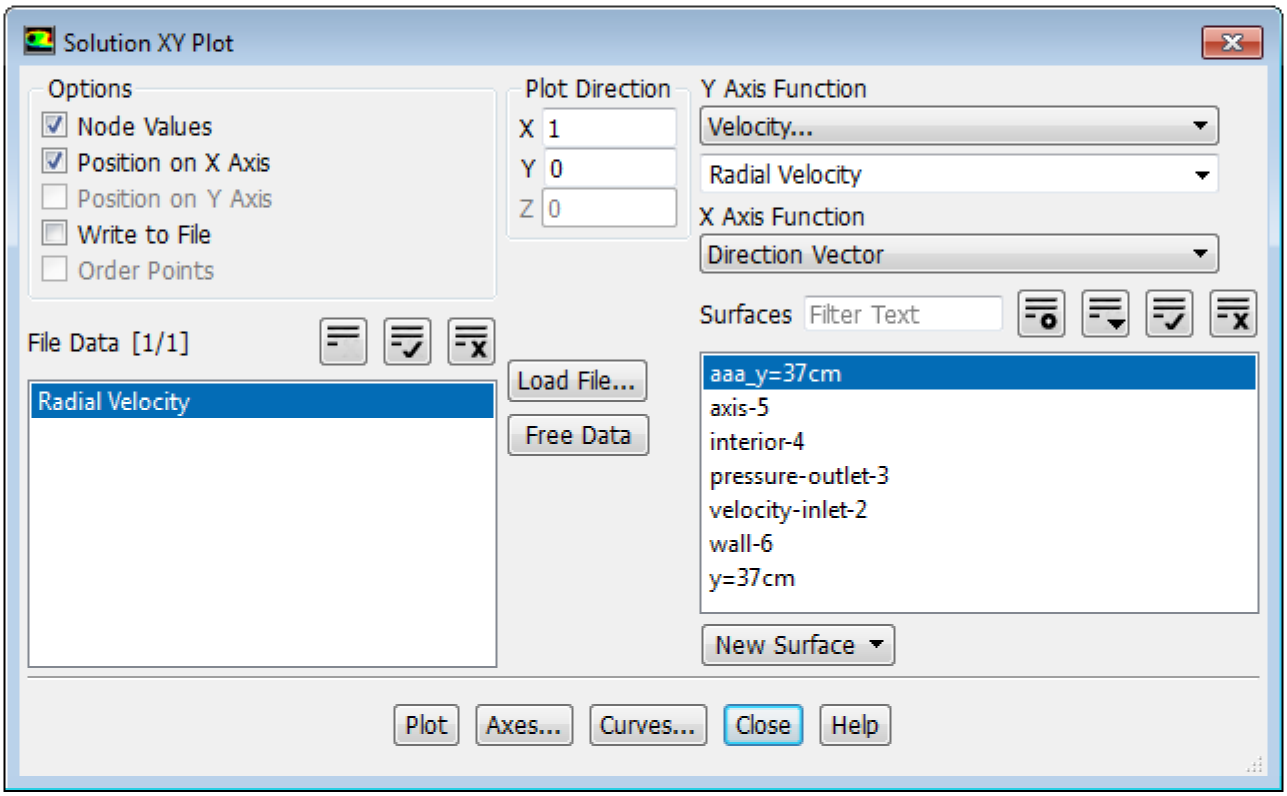
3. Save the case and data files (`disk-rng.cas.gz` and `disk-rng.dat.gz`).

   File → **Write** → **Case & Data...**

9.4.11. Postprocessing for the RNG k- ε Solution

1. Plot the radial velocity distribution for the RNG k- ε solution and compare it with the distribution for the standard k- ε solution.

   Postprocessing → **Plots** → **XY Plot** → **Edit...**



   a. Enter `1` and `0` for **X** and **Y** respectively in the **Plot Direction** group box.

   b. Select **Velocity...** and **Radial Velocity** from the **Y Axis Function** drop-down lists.

   c. Select **aaa_y=37cm** and deselect **wall-6** from the **Surfaces** selection list.

   d. Disable the **Write to File** option.

   e. Click the **Load File...** button to load the k- ε data.

      i. Select the file **ke-data.xy** in the **Select File** dialog box.

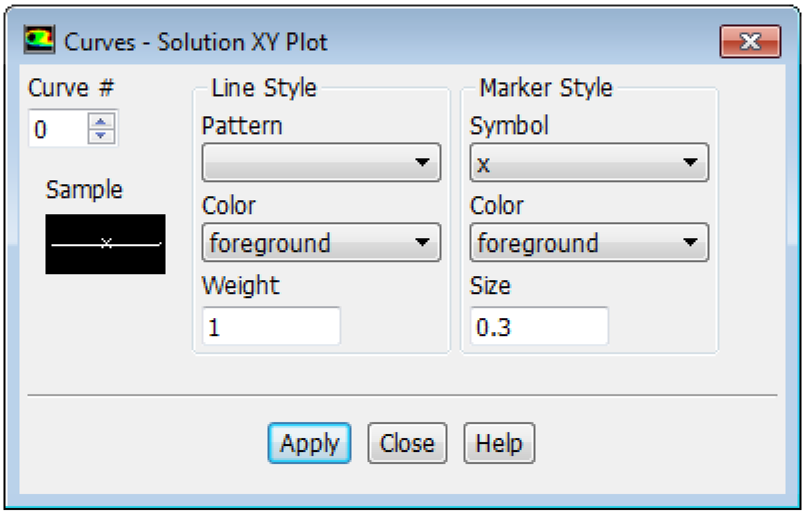
ii. Click **OK**.

f. Click the **Axes...** button to open the **Axes - Solution XY Plot** dialog box.

i. Enable **Auto Range** in the **Options** group box.

ii. Click **Apply** and close the **Axes - Solution XY Plot** dialog box.

g. Click the **Curves...** button to open the **Curves - Solution XY Plot** dialog box, where you will define a different curve symbol for the RNG k- ε data.



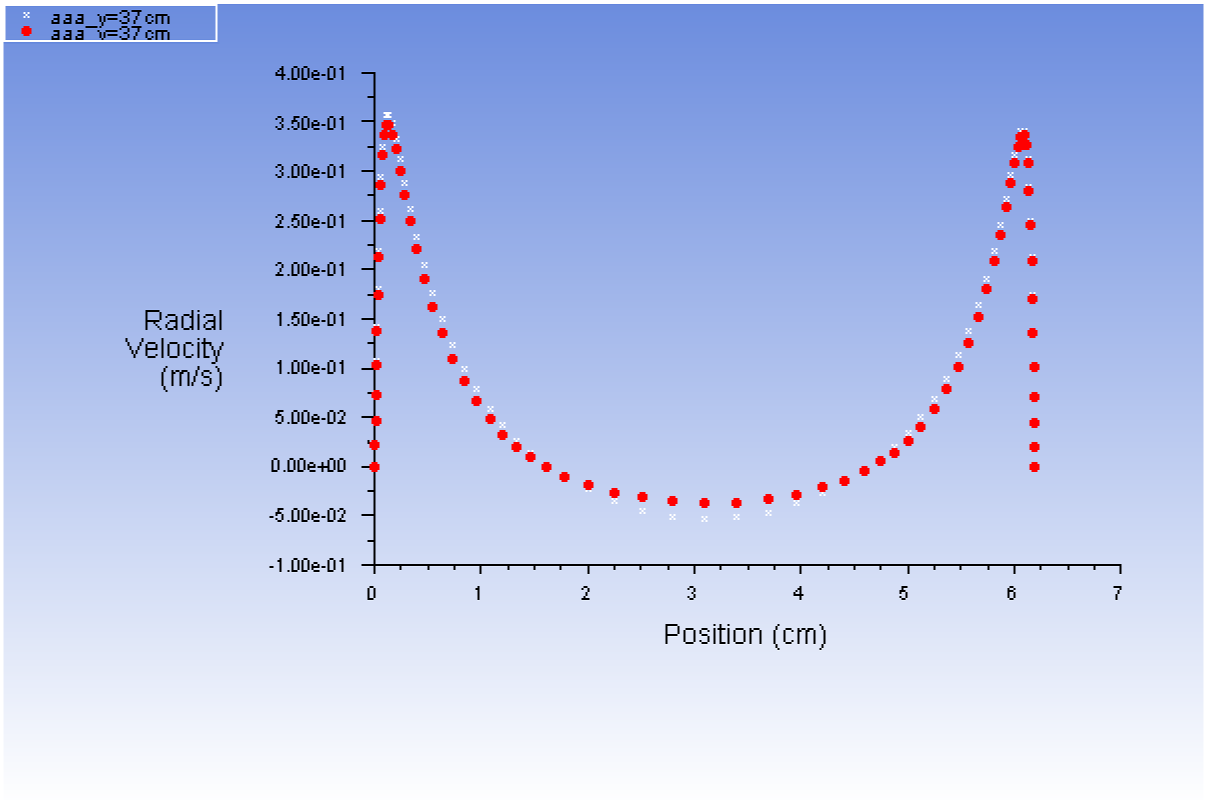
i. Retain `0` for the **Curve #**.

ii. Select **x** from the **Symbol** drop-down list.

iii. Click **Apply** and close the **Curves - Solution XY Plot** dialog box.

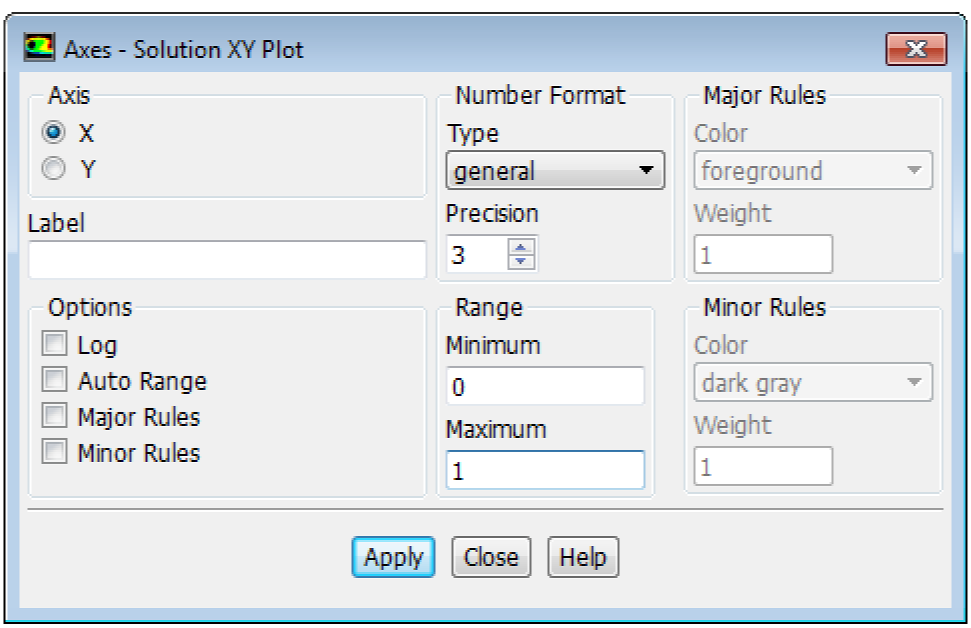
h. Click **Plot** in the **Solution XY Plot** dialog box (Figure 9.8: Radial Velocity Distribution — RNG k- ε and Standard k- ε Solutions (p. 424)).

Figure 9.8: Radial Velocity Distribution — RNG k-ε and Standard k-ε Solutions



The peak velocity predicted by the RNG k-ε solution is higher than that predicted by the standard k-ε solution. This is due to the less diffusive character of the RNG k-ε model. Adjust the range of the x axis to magnify the region of the peaks.

i. Click the **Axes...** button to open the **Axes - Solution XY Plot** dialog box, where you will specify the x-axis range.



i. Disable **Auto Range** in the **Options** group box.

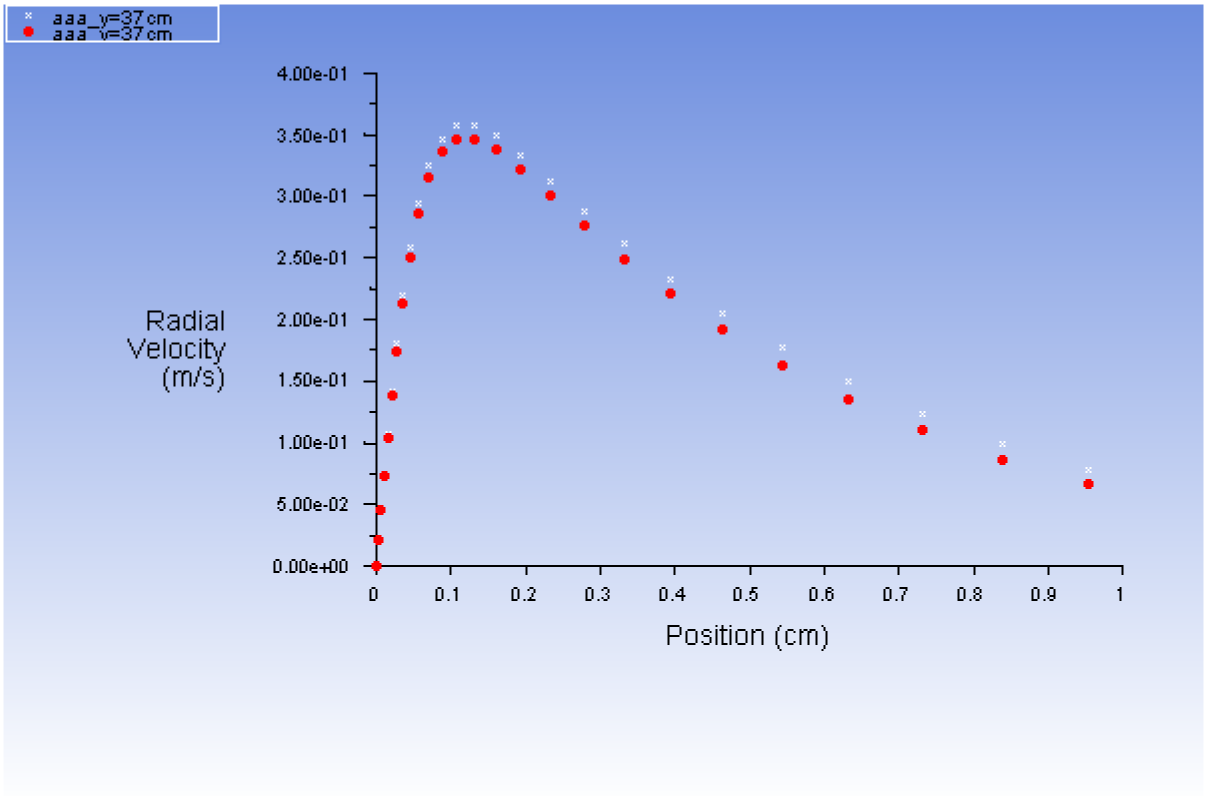
ii. Retain the value of `0` for **Minimum** and enter `1` for **Maximum** in the **Range** dialog box.

iii. Click **Apply** and close the **Axes - Solution XY Plot** dialog box.

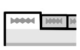
j. Click **Plot**.

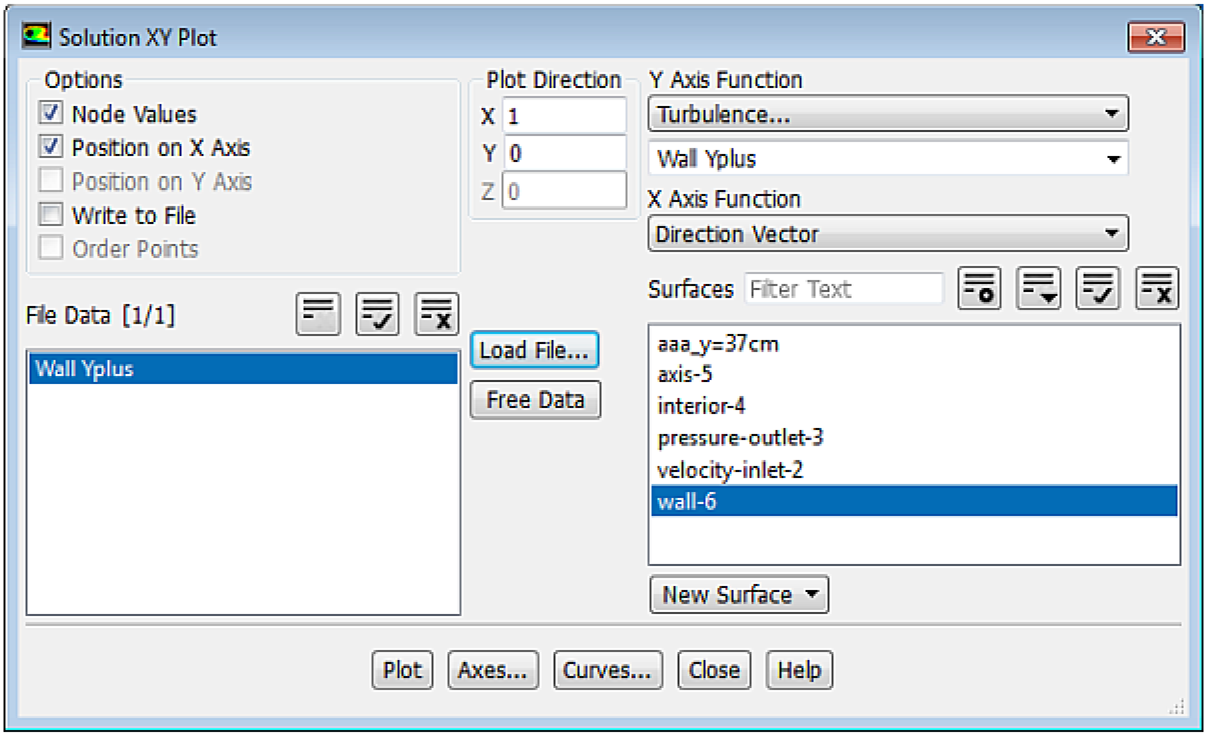
The difference between the peak values calculated by the two models is now more apparent.

Figure 9.9: RNG k- ε and Standard k- ε Solutions (x=0 cm to x=1 cm)



2. Plot the wall y+ distribution on the rotating disk wall along the radial direction Figure 9.10: wall-6 — RNG k- ε and Standard k- ε Solutions (x=0 cm to x=43 cm) (p. 427).

 **Postprocessing** → **Plots** → **XY Plot** → **Edit...**



a. Select **Turbulence...** and **Wall Yplus** from the **Y Axis Function** drop-down lists.

b. Deselect **aaa_y=37cm** and select **wall-6** from the **Surfaces** selection list.

c. Enter `0` and `1` for **X** and **Y** respectively in the **Plot Direction** group box.

d. Select any existing files that appear in the **File Data** selection list and click the **Free Data** button to remove the file.

e. Click the **Load File...** button to load the RNG k- ε data.

   i. Select the file **ke-yplus.xy** in the **Select File** dialog box.

   ii. Click **OK**.

f. Click the **Axes...** button to open the **Axes - Solution XY Plot** dialog box.

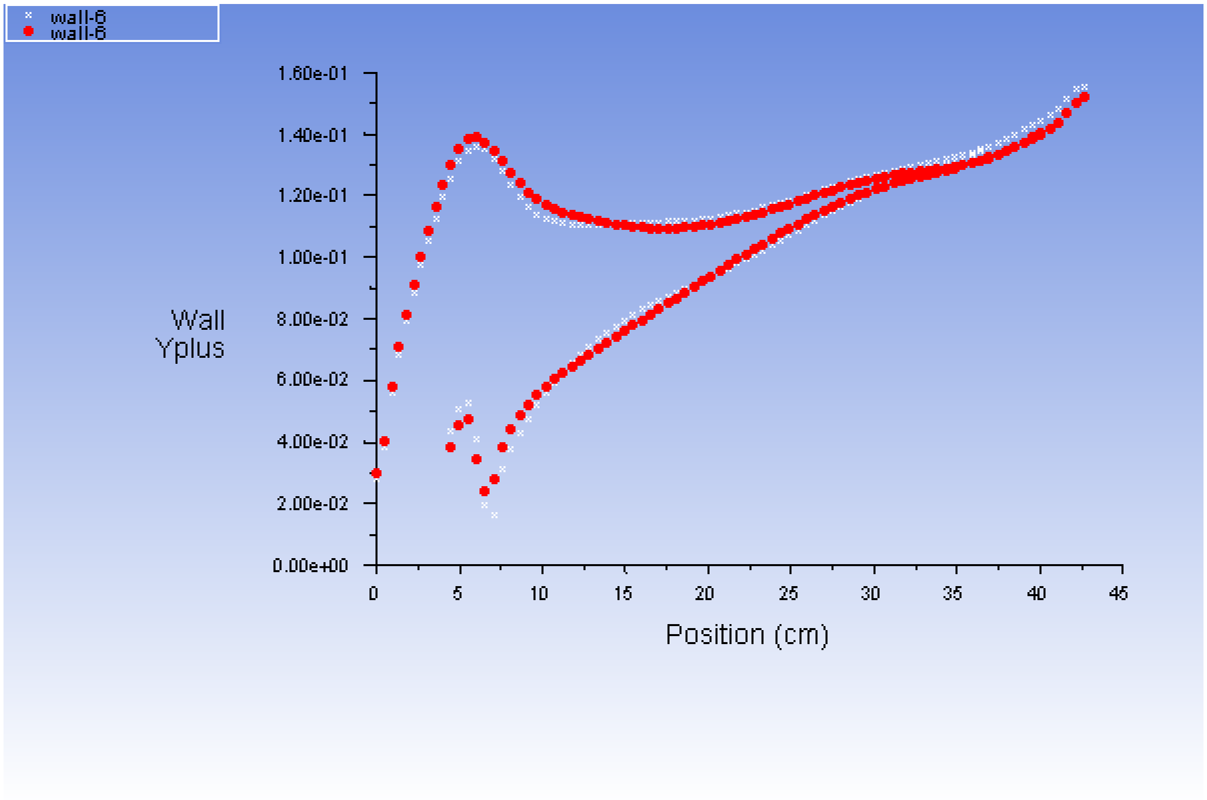
   i. Retain the default selection of **X** from the **Axis** group box.

   ii. Retain the default value of `0` for **Minimum** and enter 43 for **Maximum** in the **Range** group box.

   iii. Click **Apply** and close the **Axes - Solution XY Plot** dialog box.

g. Click **Plot** in the **Solution XY Plot** dialog box.

Figure 9.10: wall-6 — RNG k- ε and Standard k- ε Solutions (x=0 cm to x=43 cm)



9.5. Summary

This tutorial illustrated the setup and solution of a 2D, axisymmetric disk cavity problem in ANSYS Fluent. The ability to calculate a swirl velocity permits the use of a 2D mesh, thereby making the calculation simpler and more economical to run than a 3D model. This can be important for problems where the enhanced wall treatment is used, and the near-wall flow field is resolved using a fine mesh (the first mesh point away from the wall being placed at a y+ on the order of 1).

For more information about mesh considerations for turbulence modeling, see the Fluent User's Guide.

9.6. Further Improvements

The case modeled in this tutorial lends itself to parametric study due to its relatively small size. Here are some things you may want to try:

- Separate **wall-6** into two walls.

  Setting Up Domain → Zones → Separate → Faces...

  Specify one wall to be stationary, and rerun the calculation.

- Use adaption to see if resolving the high velocity and pressure-gradient region of the flow has a significant effect on the solution.

- Introduce a nonzero swirl at the inlet or use a velocity profile for fully-developed pipe flow. This is probably more realistic than the constant axial velocity used here, since the flow at the inlet is typically being supplied by a pipe.
- Model compressible flow (using the ideal gas law for density) rather than assuming incompressible flow text.

This tutorial guides you through the steps to reach an initial solution. You may be able to obtain a more accurate solution by using an appropriate higher-order discretization scheme and by adapting the mesh. Mesh adaption can also ensure that the solution is independent of the mesh. These steps are demonstrated in Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121).

9.7. References

1. Pincombe, J.R., "Velocity Measurements in the Mk II - Rotating Cavity Rig with a Radial Outflow", Thermo-Fluid Mechanics Research Centre, University of Sussex, Brighton, UK, 1981.

Chapter 10: Using Multiple Reference Frames

This tutorial is divided into the following sections:

10.1. Introduction

Many engineering problems involve rotating flow domains. One example is the centrifugal blower unit that is typically used in automotive climate control systems. For problems where all the moving parts (fan blades, hub and shaft surfaces, and so on) are rotating at a prescribed angular velocity, and the stationary walls (for example, shrouds, duct walls) are surfaces of revolution with respect to the axis of rotation, the entire domain can be referred to as a single rotating frame of reference. However, when each of the several parts is rotating about a different axis of rotation, or about the same axis at different speeds, or when the stationary walls are not surfaces of revolution (such as the volute around a centrifugal blower wheel), a single rotating coordinate system is not sufficient to "immobilize" the computational domain so as to predict a steady-state flow field. In such cases, the problem must be formulated using multiple reference frames.

In ANSYS Fluent, the flow features associated with one or more rotating parts can be analyzed using the multiple reference frame (MRF) capability. This model is powerful in that multiple rotating reference frames can be included in a single domain. The resulting flow field is representative of a snapshot of the transient flow field in which the rotating parts are moving. However, in many cases the interface can be chosen in such a way that the flow field at this location is independent of the orientation of the moving parts. That is, if an interface can be drawn on which there is little or no angular dependence, the model can be a reliable tool for simulating time-averaged flow fields. It is therefore very useful in complicated situations where one or more rotating parts are present.

This tutorial illustrates the procedure for setting up and solving a problem using the MRF capability. As an example, the flow field on a 2D section of a centrifugal blower will be calculated. Although this is a general methodology that can be applied to cases where more than one reference frame is moving, this example will be limited to a single rotating reference frame.

This tutorial demonstrates how to do the following:

- Create mesh interfaces from interface-zones defined during meshing.
- Specify different frames of reference for different fluid zones.
- Set the relative velocity of each wall.
- Calculate a solution using the pressure-based solver.

10.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

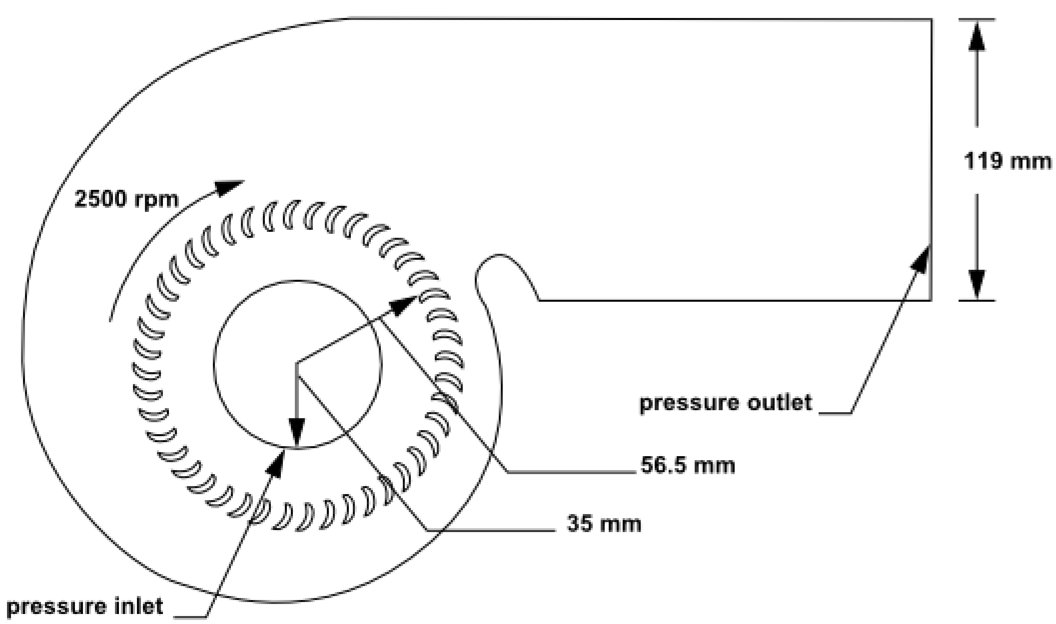
This tutorial also assumes that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

In general, to solve problems using the MRF feature, you should be familiar with the concept of creating multiple fluid zones in your mesh generator.

10.3. Problem Description

This problem considers a 2D section of a generic centrifugal blower. A schematic of the problem is shown in Figure 10.1: Schematic of the Problem (p. 430). The blower consists of 32 blades, each with a chord length of 13.5 mm. The blades are located approximately 56.5 mm (measured from the leading edge) from the center of rotation. The radius of the outer wall varies logarithmically from 80 mm to 146.5 mm. You will simulate the flow under no load, or free-delivery conditions when inlet and outlet pressures are at ambient conditions (0 Pa gauge). This corresponds to the maximum flow-rate of the blower when sitting in free air. The blades are rotating with an angular velocity of 2500 rpm. The flow is assumed to be turbulent.

Figure 10.1: Schematic of the Problem



10.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

10.4.1. Preparation

To access tutorials and their input files on the ANSYS Customer Portal, go to http://support.ansys.com/training.

To prepare for running this tutorial:

1. Set up a working folder on the computer you will be using.
2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

   > **Note**
   >
   > If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

3. Enter the name of this tutorial into the search bar.
4. Narrow the results by using the filter on the left side of the page.
   a. Click **ANSYS Fluent** under **Product**.
   b. Click **18.0** under **Version**.
5. Select this tutorial from the list.
6. Click the **multiple_rotating_R180.zip** link to download the input files.
7. Unzip the `multiple_rotating_R180.zip` file you have downloaded to your working folder.

   The file, `blower-2d.msh` *can be found in the* `multiple_rotating` *directory created after unzipping the file.*

8. Use Fluent Launcher to start the **2D Double Precision** version of ANSYS Fluent.

   Fluent Launcher displays your **Display Options** preferences from the previous session.

For more information about Fluent Launcher, see starting ANSYS Fluent using the Fluent Launcher in the Fluent Getting Started Guide.

9. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.

10. Ensure that the **Serial** processing option is selected.

10.4.2. Reading and Checking the Mesh and Setting the Units

1. Read the mesh file (`blower-2d.msh`).

   File → **Read** → **Mesh...**

   The geometry and mesh are displayed in graphics window (Figure 10.2: Mesh of the 2D Centrifugal Blower (p. 433))

2. Check the mesh.

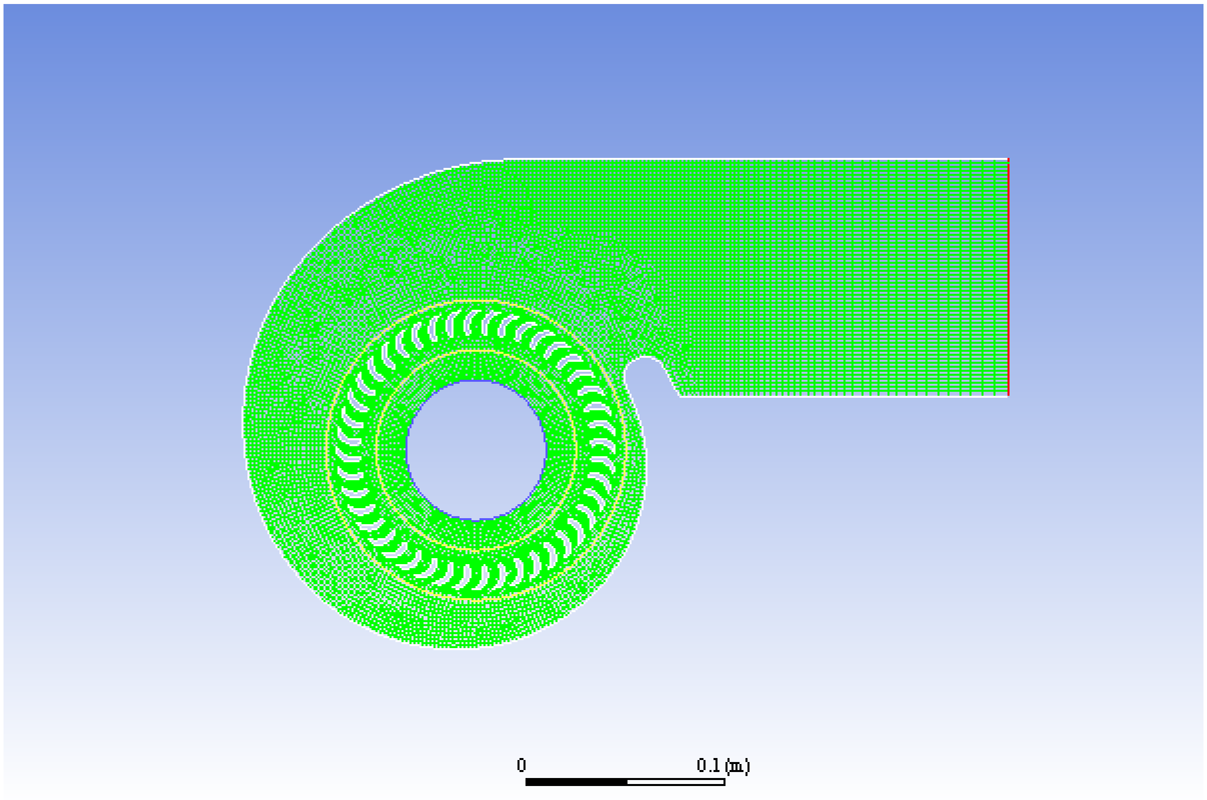
   Setting Up Domain → **Mesh** → **Check**

   ANSYS Fluent will perform various checks on the mesh and will report the progress in the console. Make sure that the reported minimum volume is a positive number. It will also issue warnings about unassigned interface zones. You do not need to take any action now. You will set up the mesh interfaces in a later step.

3. Examine the mesh.

   *The mesh consists of three fluid zones, **fluid-casing**, **fluid-inlet**, and **fluid-rotor**. These are reported in the console when the mesh is read. In the **Mesh Display** dialog box, the fluid zones are reported as interior zones **default-interior**, **default-interior:013**, and **default-interior:015** respectively. The fluid zone containing the blades will be solved in a rotational reference frame.*

Figure 10.2: Mesh of the 2D Centrifugal Blower

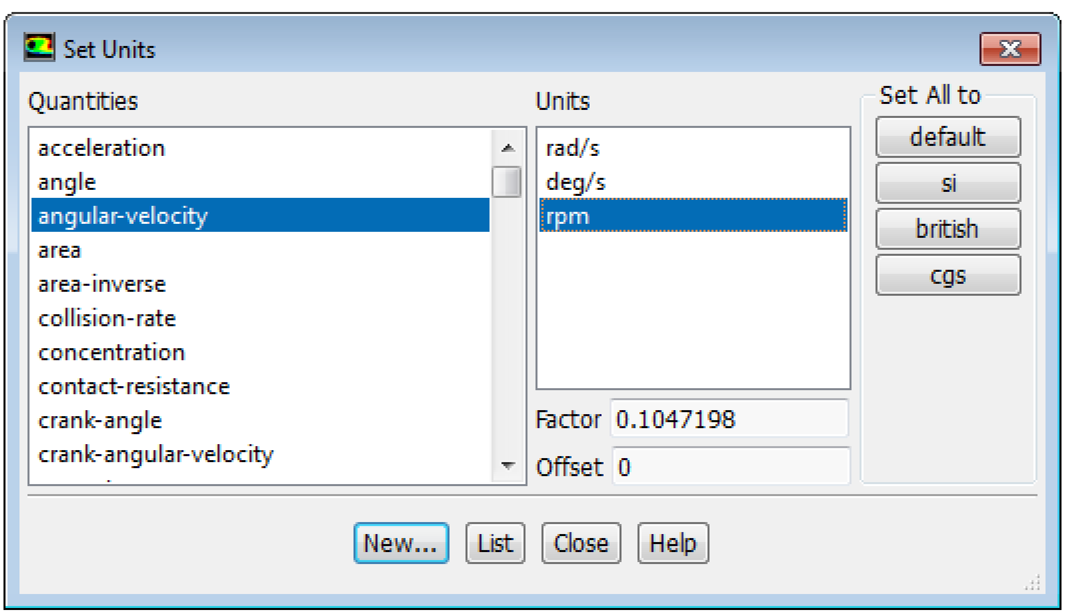


The fluid zones are bounded by interface zones that appear in the mesh display in yellow. These interface boundaries were used in the mesh generator to separate the fluid zones, and will be used to create mesh interfaces between adjacent fluid zones when the boundary conditions are set later in this tutorial.

4. Set the units for angular velocity.

   Setting Up Domain → Mesh → Units...

   In the problem description, angular velocity is specified in rpm rather than in the default unit of rad/s.



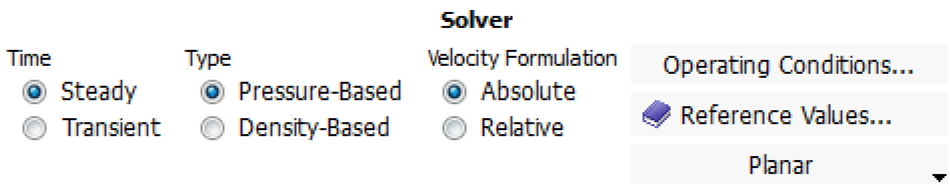
a. Select **angular-velocity** from the **Quantities** list and **rpm** in the **Units** list.

b. Close the **Set Units** dialog box.

10.4.3. Specifying Solver and Analysis Type

1. Retain the default settings of the pressure-based steady-state solver in the **Solver** group.

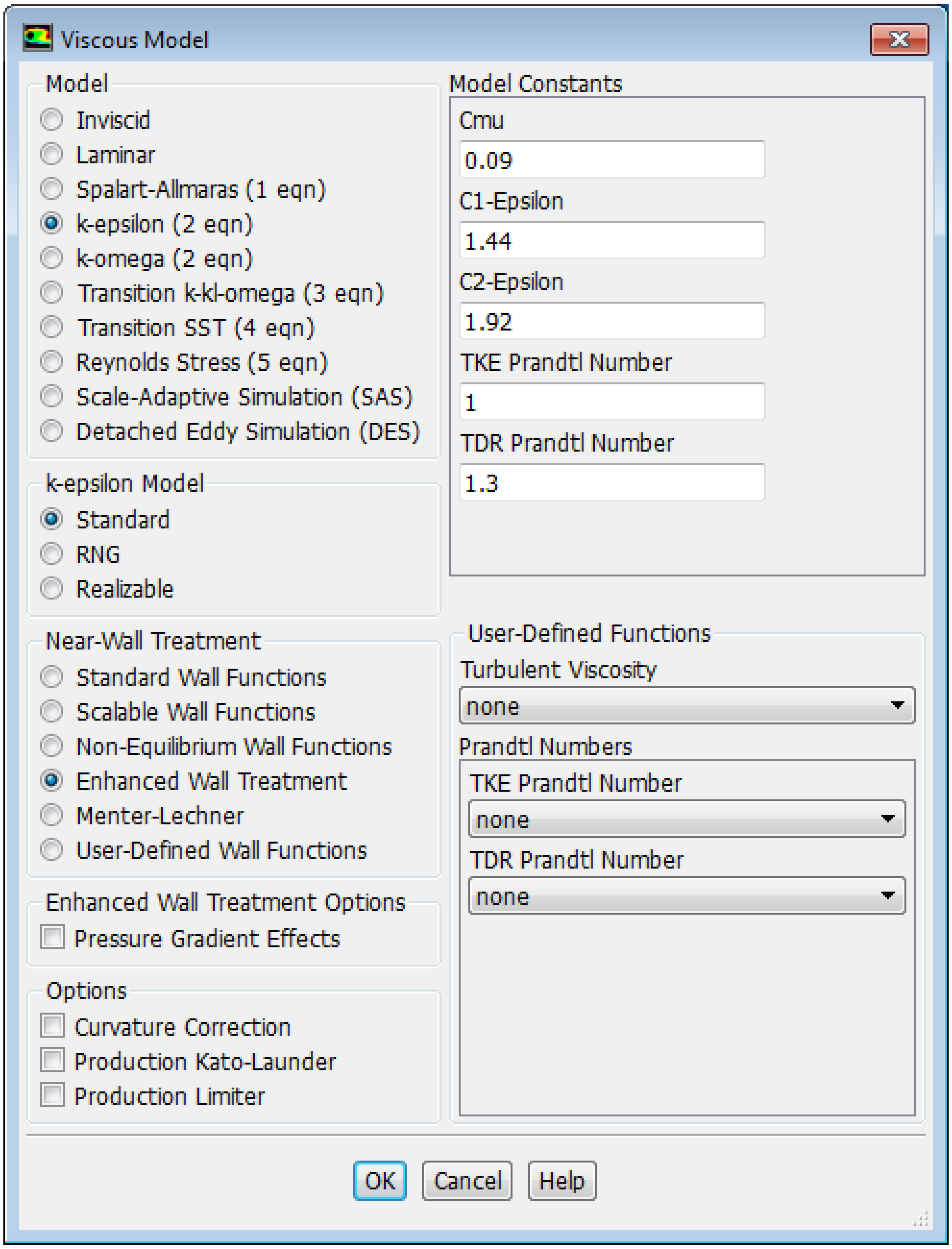
   Setting Up Physics → Solver



10.4.4. Specifying the Models

1. Enable the standard k- ε turbulence model.

   Setup → Models → Viscous Edit...



a. Select **k-epsilon (2eqn)** in the **Model** list.

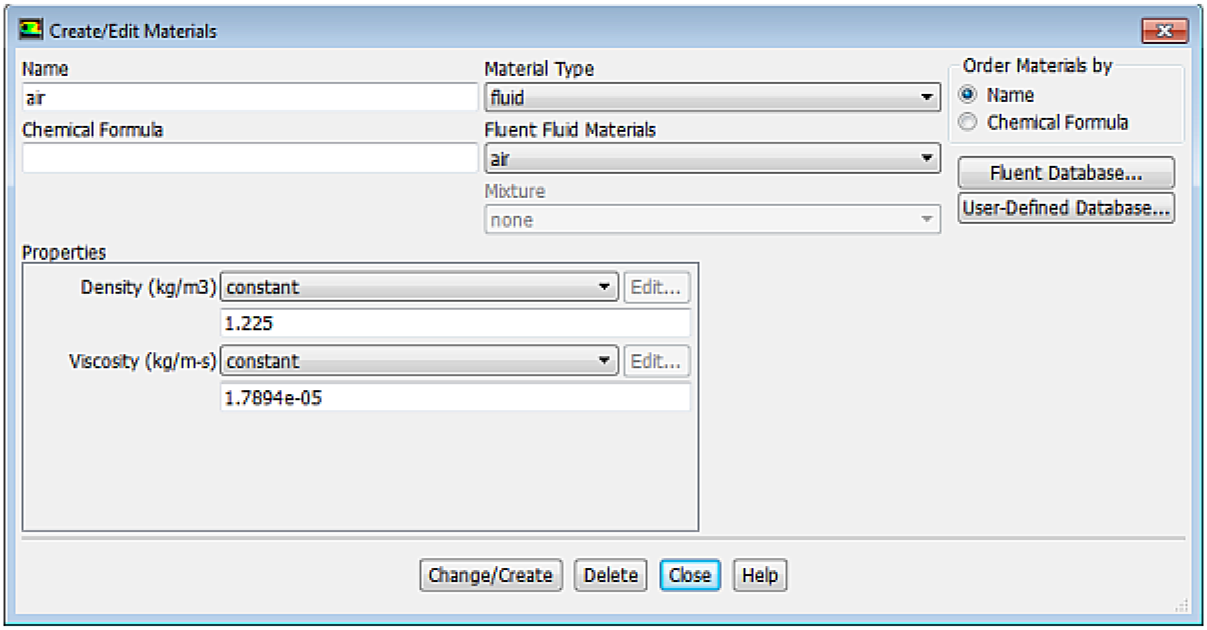
b. Select **Enhanced Wall Treatment** in the **Near-Wall Treatment** list.

c. Click **OK** to close the **Viscous Model** dialog box.

10.4.5. Specifying Materials

1. Retain the default properties for air.

   Setup → **Materials** → **air** → **Create/Edit...**



Tip

If needed, you could modify the fluid properties for air or copy another material from the database.

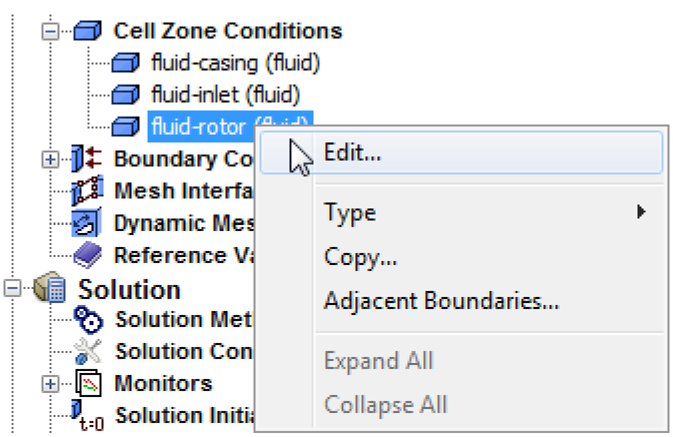
---

2. Click **Close** to close the **Create/Edit Materials** dialog box.

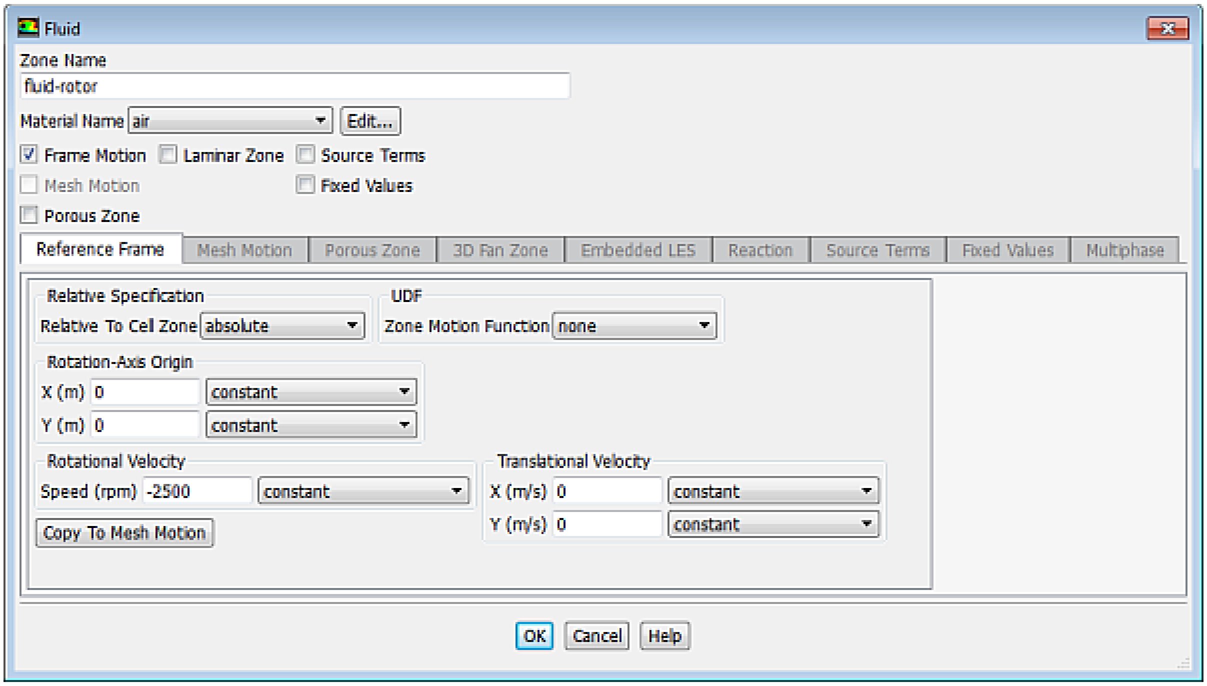
For details, see physical properties in the Fluent User's Guide.

10.4.6. Specifying Cell Zone Conditions

1. Define the boundary conditions for the rotational reference frame (**fluid-rotor**).



Setup → **Cell Zone Conditions** → **fluid-rotor** → **Edit...**



a. Enable **Frame Motion**.

The dialog box will expand to show the relevant inputs.

b. Under the **Reference Frame** tab, retain the **Rotation-Axis Origin** default setting of `(0,0)`.

This is the center of curvature for the circular boundaries of the rotating zone.

c. Enter `-2500` rpm for **Speed** in the **Rotational Velocity** group box.

Note

The speed is entered as a negative value because the rotor is rotating clockwise, which is in the negative sense about the Z axis.

---

d. Click **OK** to close the **Fluid** dialog box.

Note

Since the other fluid zones are stationary, you do not need to set any boundary conditions for them. If one or more of the remaining fluid zones were also rotating, you would need to set the appropriate rotational speed for them.

Tip

In this example, the names of the fluid-zones in the mesh file leave no ambiguity as to which is the rotating fluid zone. In the event that you have a mesh without clear names, you may have difficulty identifying the various fluid-zones. Unlike interior zones, the fluid-zones cannot be individually selected and displayed from the **Mesh Display** dialog box. However, you can use commands in the text interface to display them.

Setting Up Domain → Display

i. Deselect all surfaces except, **default-interior**, **default-interior:013**, and **default-interior:015** in the **Surfaces** selection list to deselect them.

ii. Click **Display**.

Only the domain boundaries and interface zones will be displayed.

iii. Press **Enter** to get the > prompt.

iv. Type the commands, in the console, as shown.

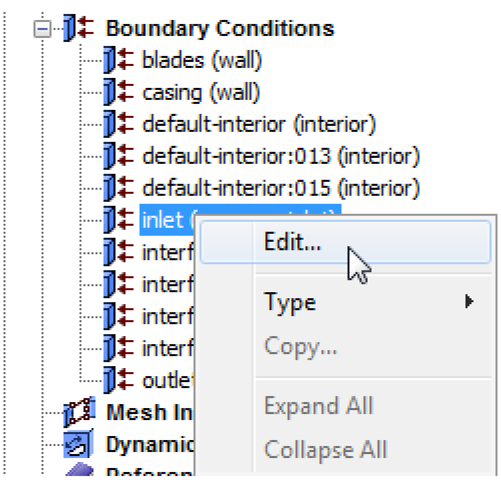
```
>  display

/display>  zone-mesh
()
zone id/name(1) [()]  4
zone id/name(2) [()]  <Enter>
```

The resulting display shows that the zone with ID 4 (in this case ***fluid-rotor****) corresponds to the rotating region.*

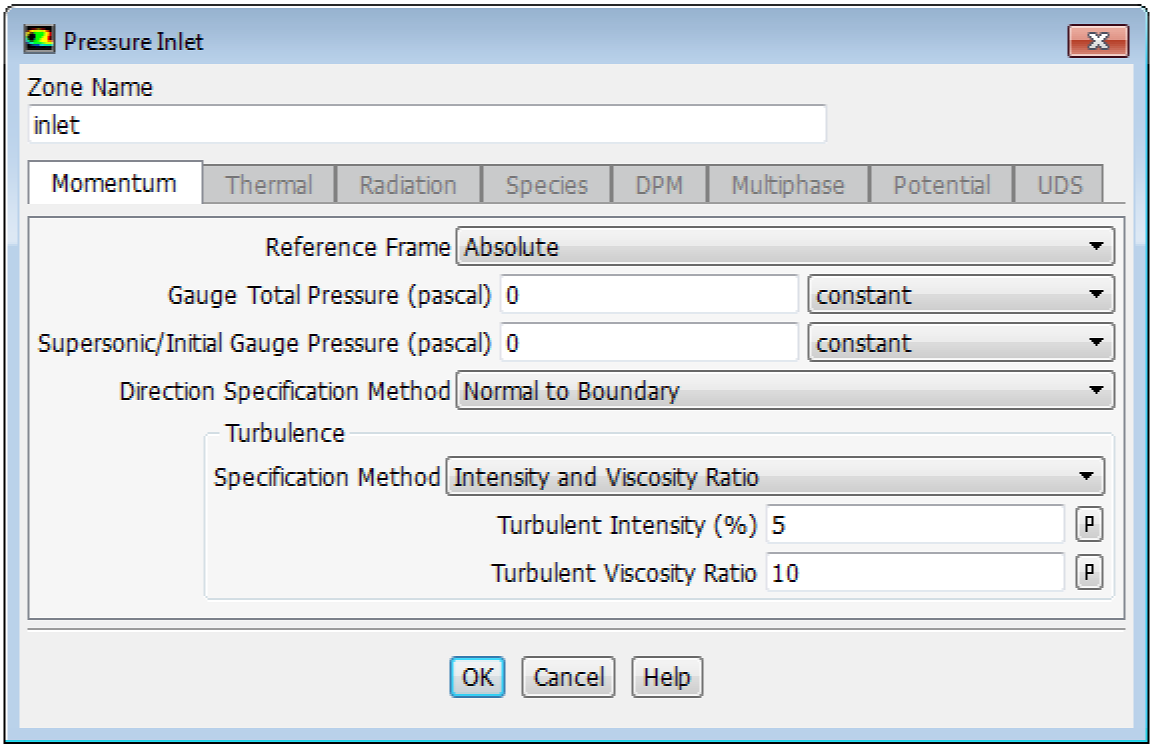
v. Close the **Mesh Display** dialog box.

10.4.7. Setting Boundary Conditions

1. Set the boundary conditions for the flow inlet (**inlet**) as specified in the problem description (see Figure 10.1: Schematic of the Problem (p. 430)).



Setup → Boundary Conditions → inlet Edit...



a. Review the boundary condition definition for the pressure-inlet type. Leave the settings at their defaults.

b. Click **OK** to close the **Pressure Inlet** dialog box.

Note

All pressures that you specify in ANSYS Fluent are gauge pressures, relative to the operating pressure specified in the **Operating Conditions** dialog box. By default, the operating pressure is 101325 Pa.

For details, see operating pressure in the Fluent User's Guide.

2. Review and retain the default values for the boundary conditions for the flow outlet (**outlet**) so that the backflow turbulence parameters for the flow outlet (**outlet**) are set to the same values used for **inlet**.

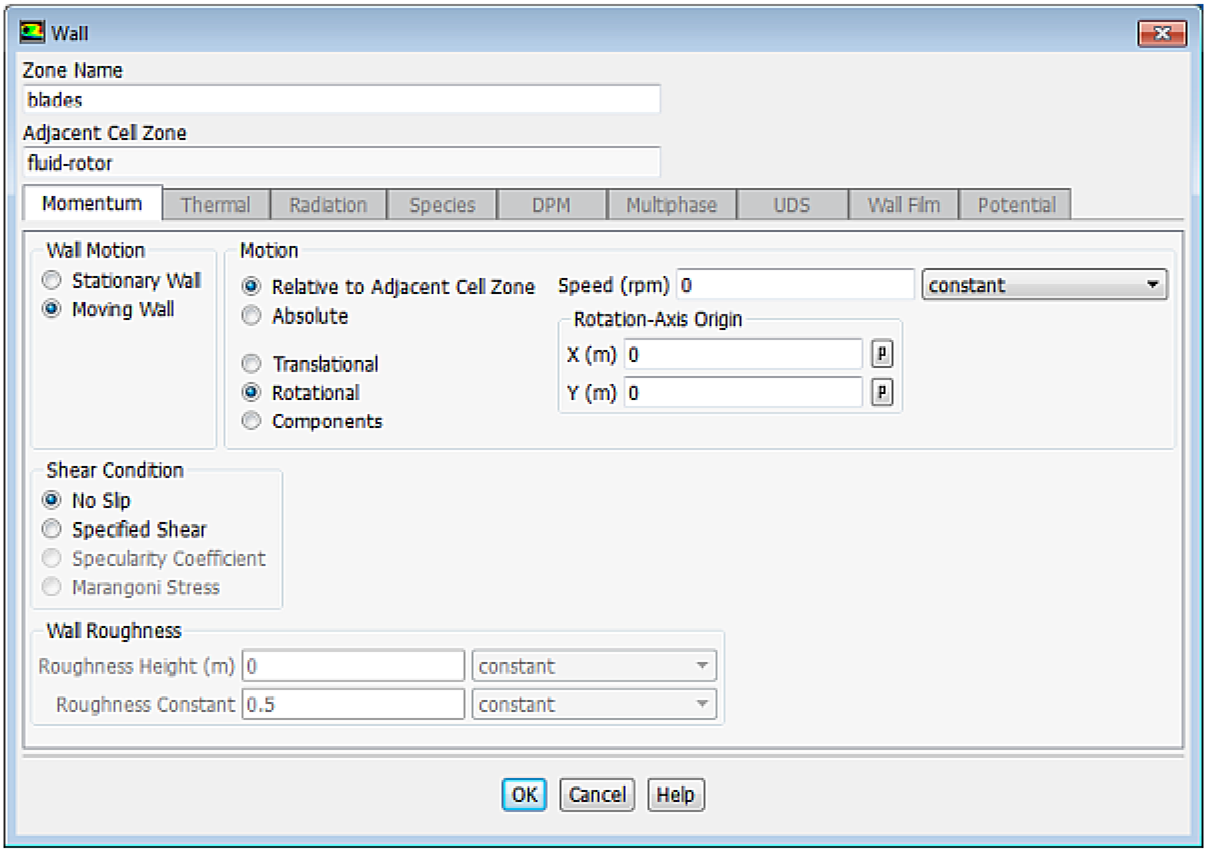
Note

The backflow values are used only if reversed flow occurs at the outlet, but it is a good idea to use reasonable values, even if you do not expect any backflow to occur.

---

3. Define the velocity of the wall zone representing the blades (**blades**) relative to the moving fluid zone.

Setup → **Boundary Conditions** → **blades** → **Edit...**



*With **fluid-rotor** set to a rotating reference frame, **blades** becomes a moving wall.*

a. Select **Moving Wall** in the **Wall Motion** group box.

   *The **Wall** dialog box will expand to show the wall motion parameters.*

b. Retain the default selection of **Relative to Adjacent Cell Zone** and select **Rotational** in the **Motion** group box.

c. Retain the default value of `0` rpm for (relative) **Speed**.

d. Click **OK** to close the **Wall** dialog box.

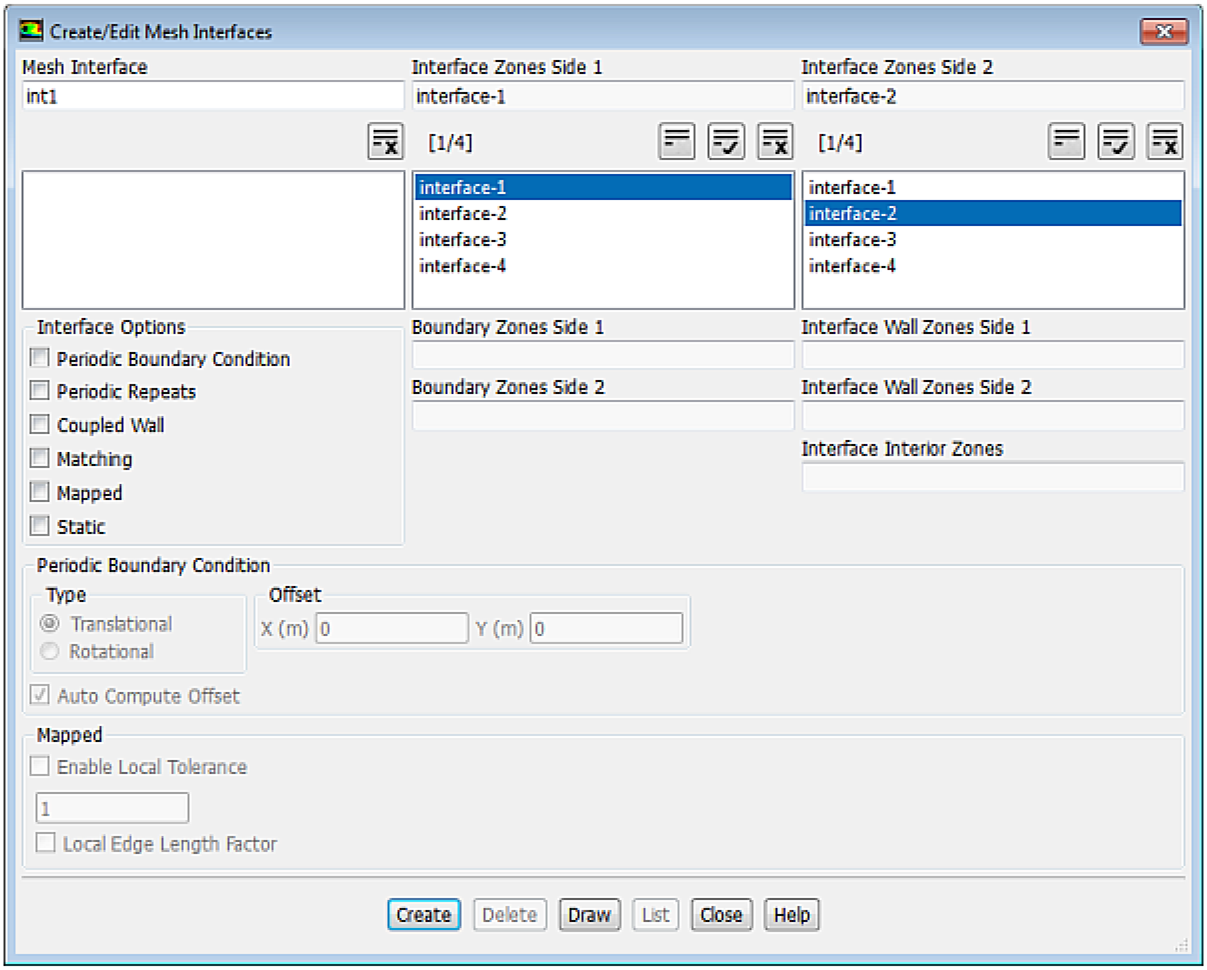
   *The **Rotation-Axis Origin** should be located at x = 0 m and y = 0 m. With these settings, the blades will move at the same speed as the surrounding fluid.*

10.4.8. Defining Mesh Interfaces

Recall that the fluid domain is defined as three distinct fluid zones. You must define mesh interfaces between the adjacent fluid zones so that ANSYS Fluent can solve the flow equations across the interfaces.

1. Set up the mesh interface between **fluid-inlet** and **fluid-rotor**.

   Setup → **Mesh Interfaces** **New...**



a. Enter `int1` under **Mesh Interface** to name this interface definition.

b. Select `interface-1` for **Interface Zone 1** and `interface-2` for **Interface Zone 2**.

   *You can use the **Draw** button to help identify the interface-zones.*

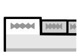
c. Click **Create** in order to create the mesh interface, `int1`.

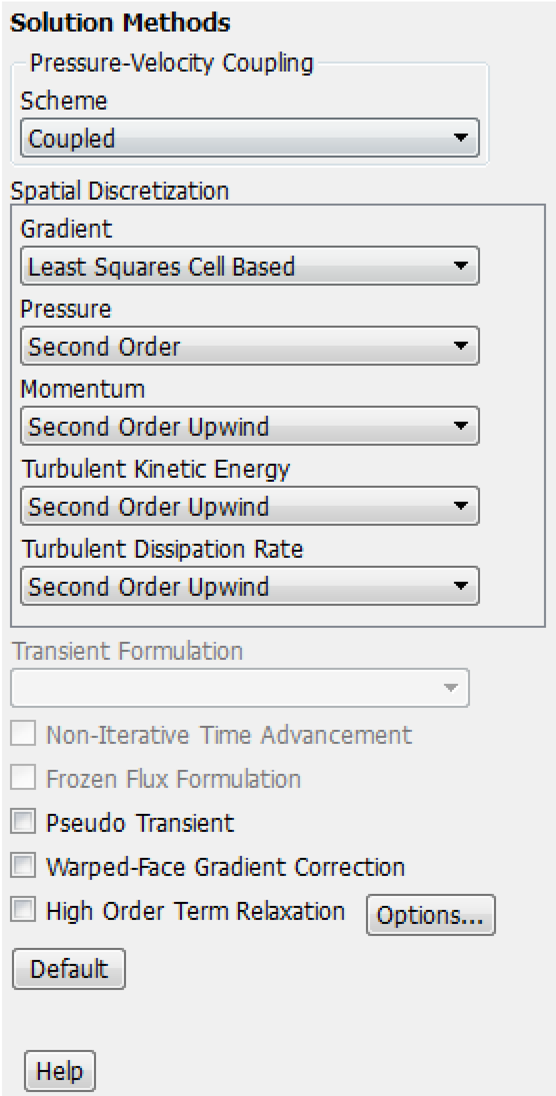
d. In a similar manner, define a mesh interface called `int2` between `interface-3` and `interface-4`.

e. Close the **Create/Edit Mesh Interfaces** dialog box.

10.4.9. Obtaining the Solution

1. Set the solution parameters.

    **Solving → Solution → Methods...**



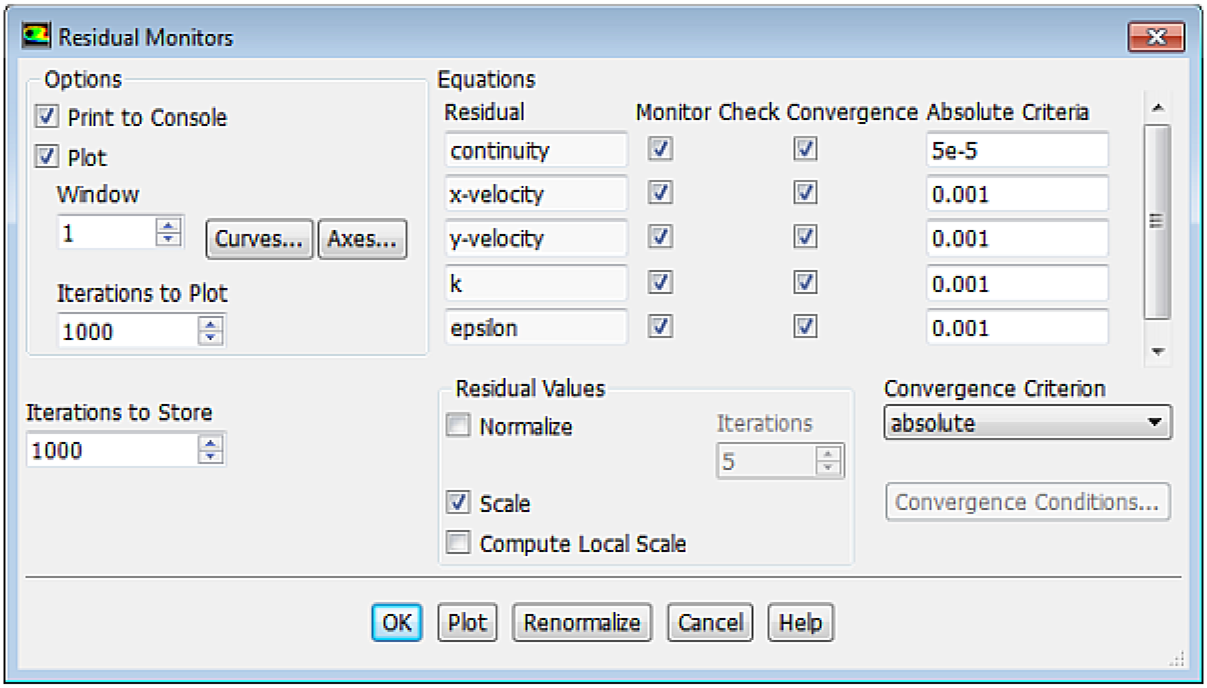
a. Select **Coupled** from the **Scheme** drop-down list in the **Pressure-Velocity Coupling** group box.

b. Select Second Order Upwind for **Turbulent Kinetic Energy** and **Turbulent Dissipation Rate** in the **Spatial Discretization** group box.

   The second-order scheme will provide a more accurate solution.

2. Enable that plotting of residuals during the calculation.

   Solving → **Reports** → **Residuals...**



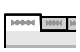
a. Ensure that **Plot** is enabled in the **Options** group box.

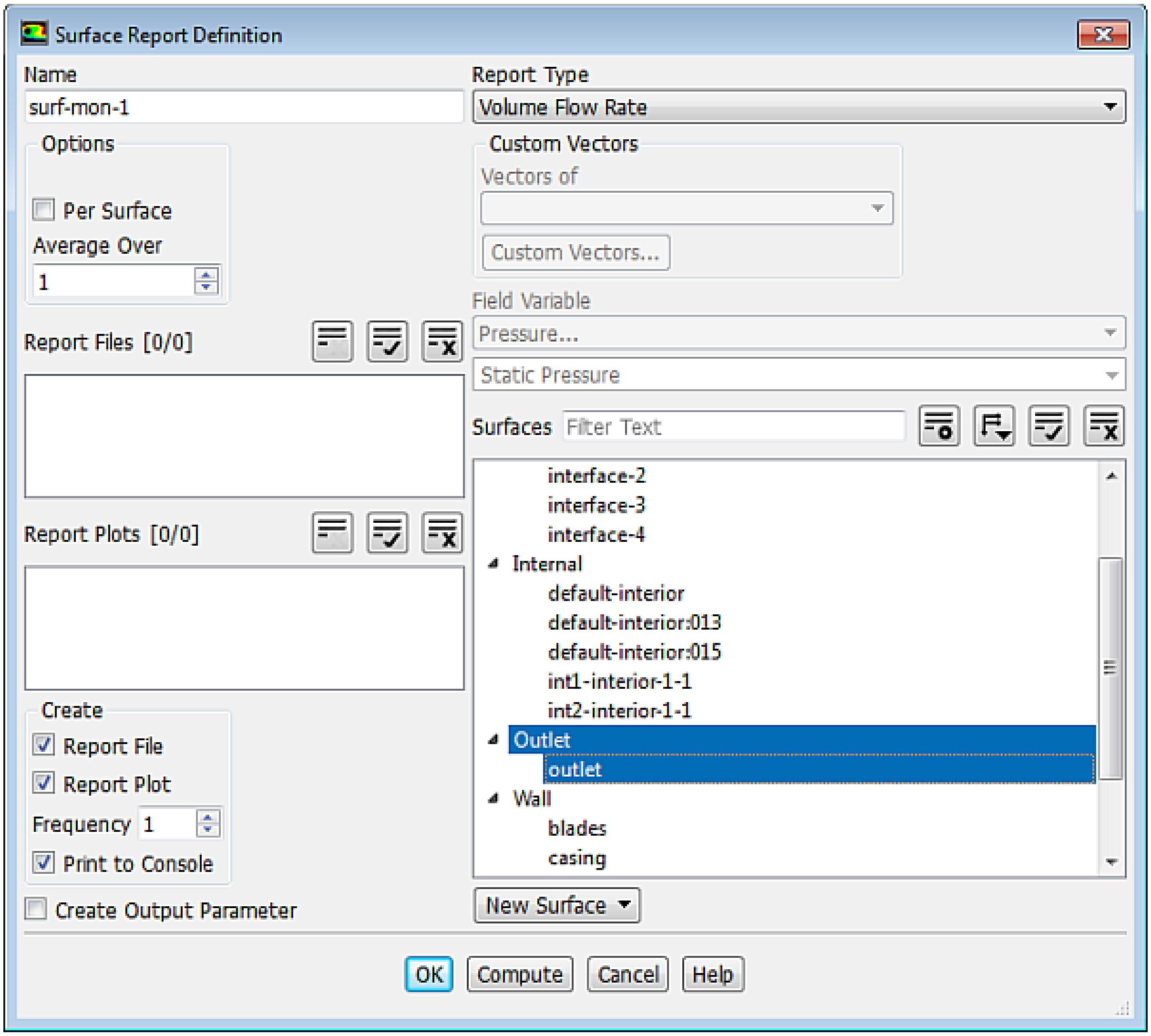
b. Enter `5e-5` under **Absolute Criteria** for the **continuity** equation.

For this problem, the default value of `0.001` is insufficient for the flow rate in the blower to fully converge. All other settings should remain at their default values.

c. Click **OK** to close the **Residual Monitors** dialog box.

3. Create a surface report definition and plot the volume flow rate at the flow outlet.

 **Solving** → **Reports** → **Definitions** → **New** → **Surface Report** → **Volume Flow Rate**



a. Enter **`surf-mon-1`** for the **Name** of the surface report definition.

b. In the **Create** group box, enable **Report File**, **Report Plot** and **Print to Console**.

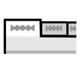
c. **Note**

   When the **Report File** option is selected in the **Surface Report Definition** dialog box, the volume flow rate history will be written to a file. If you do not enable the **Report File** option, the history information will be lost when you exit ANSYS Fluent.

d. Select **outlet** from the **Surfaces** selection list.

e. Click **OK** to save the surface report definition settings and close the **Surface Report Definition** dialog box.

4. Initialize the solution.

 **Solving → Initialization**

a. Retain the **Method** at the default of **Hybrid** in the **Initialization Methods** group.

b. Click **Initialize** to initialize the solution.

Note

For flows in complex topologies, hybrid initialization will provide better initial velocity and pressure fields than standard initialization. This in general will help in improving the convergence behavior of the solver.

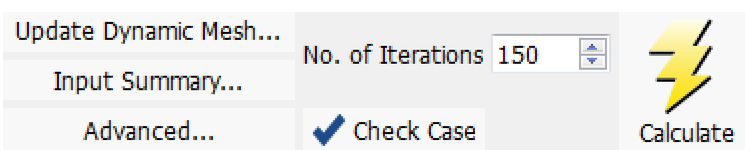
---

5. Save the case file (`blower.cas.gz`).

File → **Write** → **Case...**

6. Start the calculation by requesting 150 iterations.

Solving → **Run Calculation**



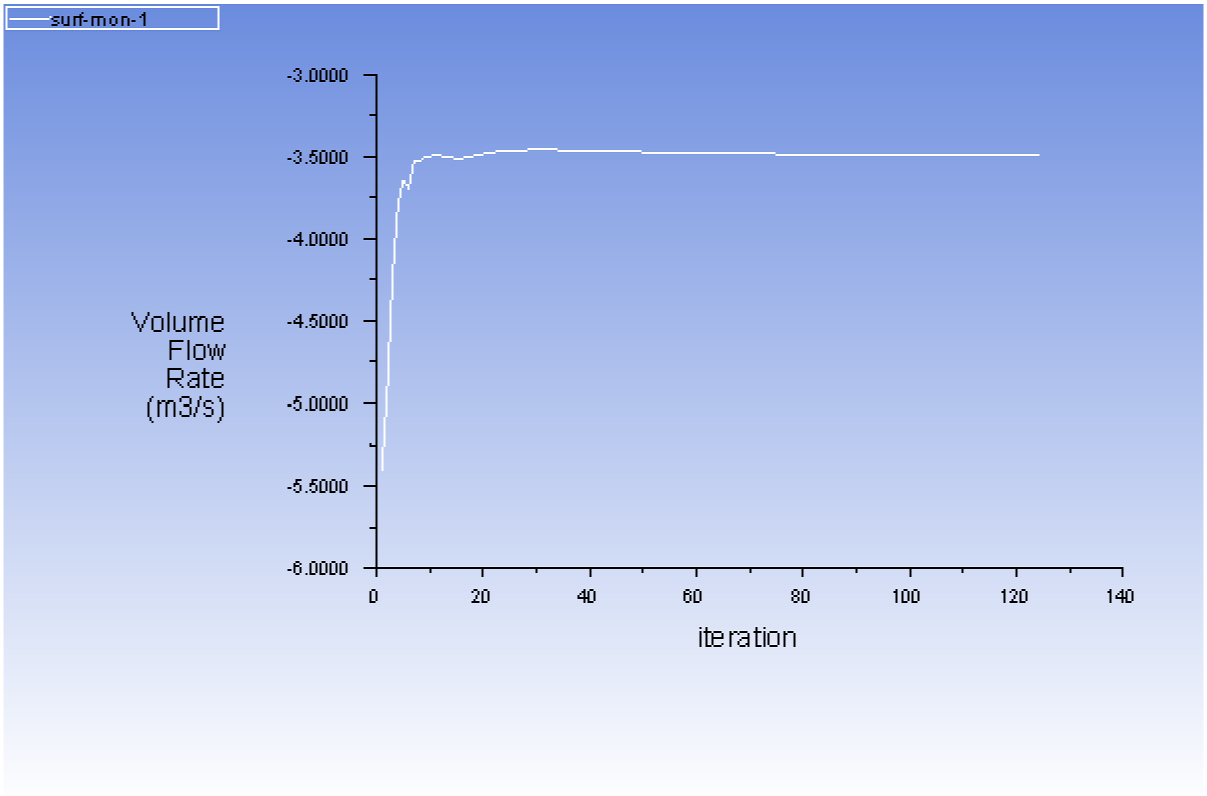
a. Enter `150` for **No. of Iterations**.

b. Click **Calculate**.

Early in the calculation, ANSYS Fluent will report that there is reversed flow occurring at the exit. This is due to the sudden expansion, which results in a recirculating flow near the exit.

The solution will converge in approximately 120 iterations (when all residuals have dropped below their respective criteria).

Figure 10.3: History of Volume Flow Rate the Outlet



The surface report history indicates that the flow rate at the outlet has ceased changing significantly, further indicating that the solution has converged. The volume flow rate is approximately 3.49 m^3*/s.*

Note

You can examine the residuals history in an appropriate graphics tab window.

---

7. Save the case and data files (`blower2.cas.gz` and `blower2.dat.gz`).

   File → **Write** → **Case & Data...**

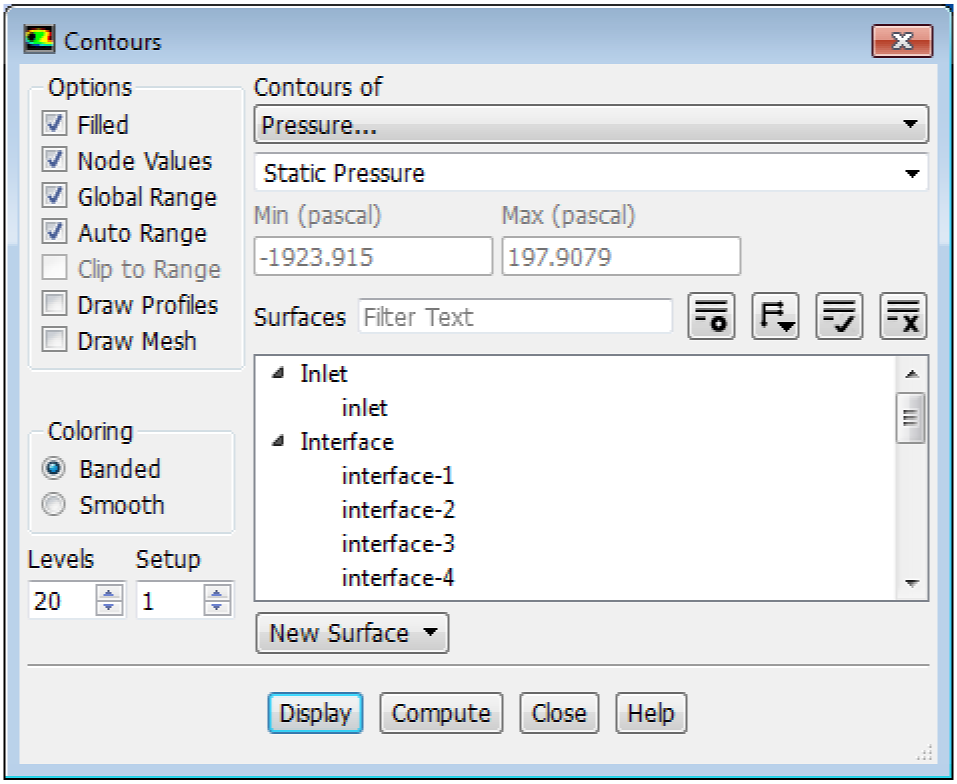
Note

It is good practice to save the case file whenever you are saving the data. This will ensure that the relevant parameters corresponding to the current solution data are saved accordingly.

10.4.10. Step 9: Postprocessing

1. Display filled contours of static pressure (Figure 10.4: Contours of Static Pressure (p. 448)).

   Postprocessing → **Graphics** → **Contours** → **Edit...**



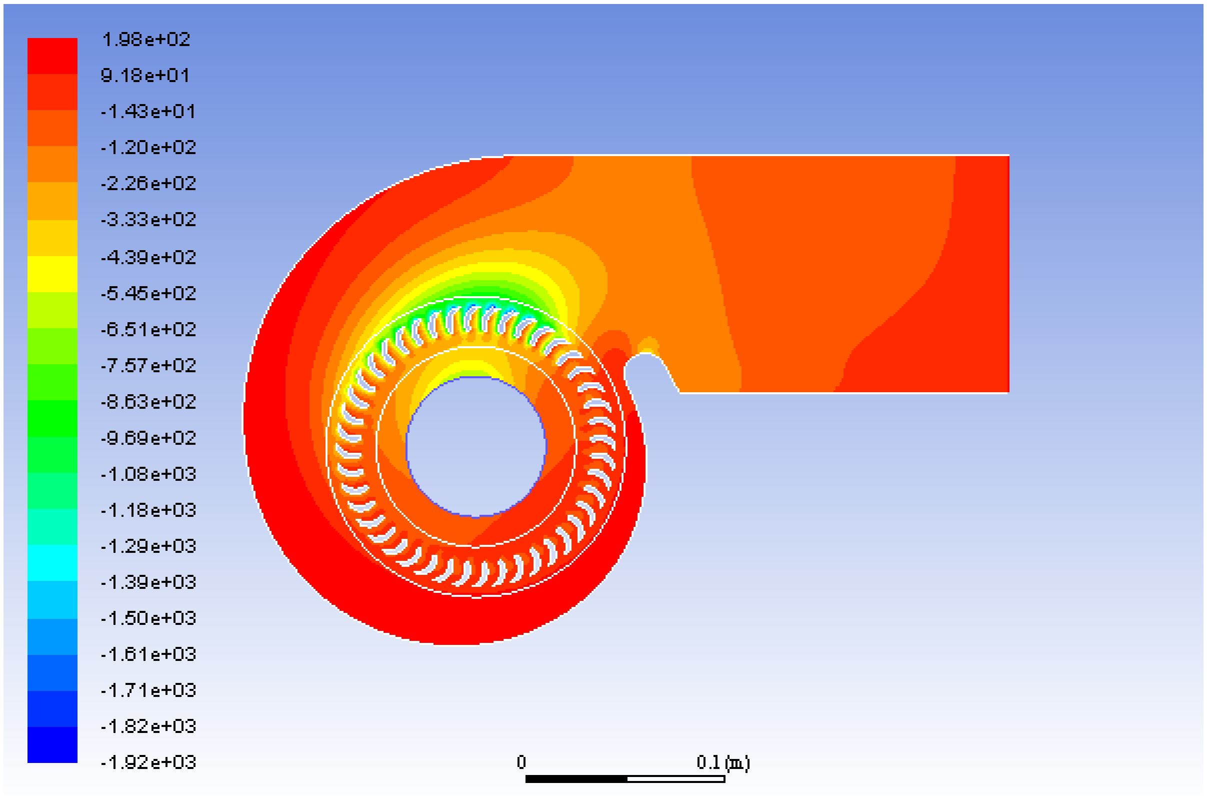
a. Enable **Filled** in the **Options** group box.

b. Ensure **Pressure...** and **Static Pressure** are selected from the **Contours of** drop-down lists.

c. Click **Display** and close the **Contours** dialog box (see Figure 10.4: Contours of Static Pressure (p. 448)).

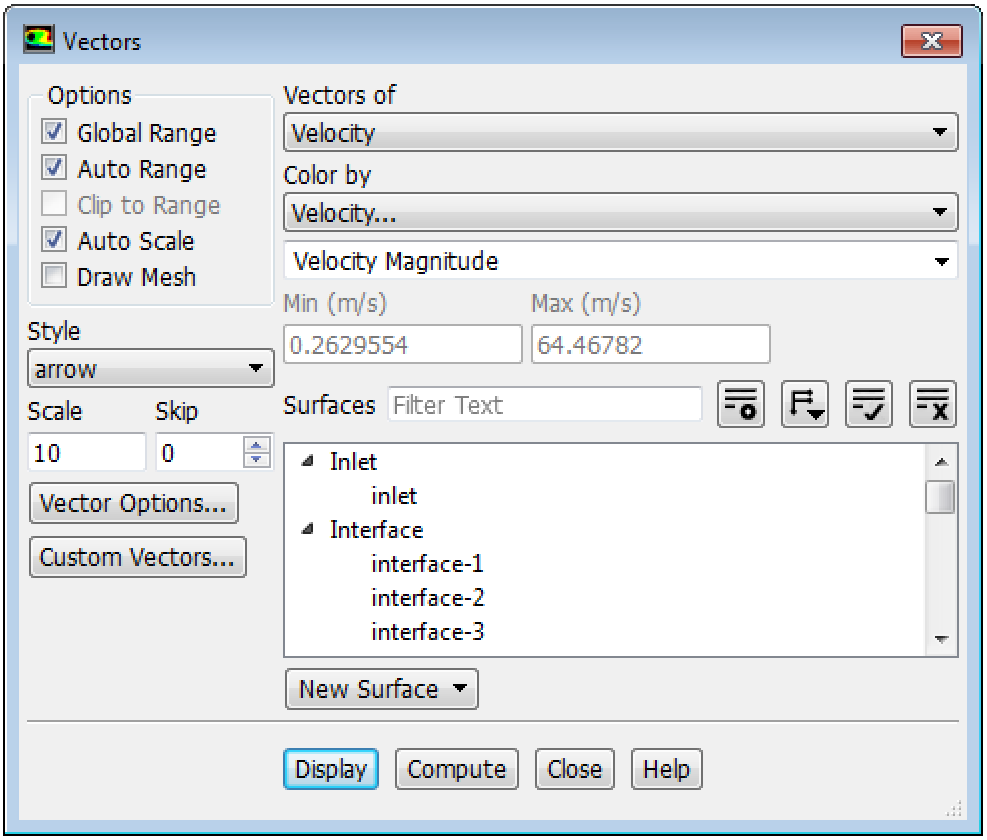
Pressure distribution in the flow domain is plotted in graphics window.

Figure 10.4: Contours of Static Pressure



2. Display absolute velocity vectors (Figure 10.5: Velocity Vectors (p. 450)).

   Postprocessing → **Graphics** → **Vectors** → **Edit...**



a. Enter `10` for **Scale**.

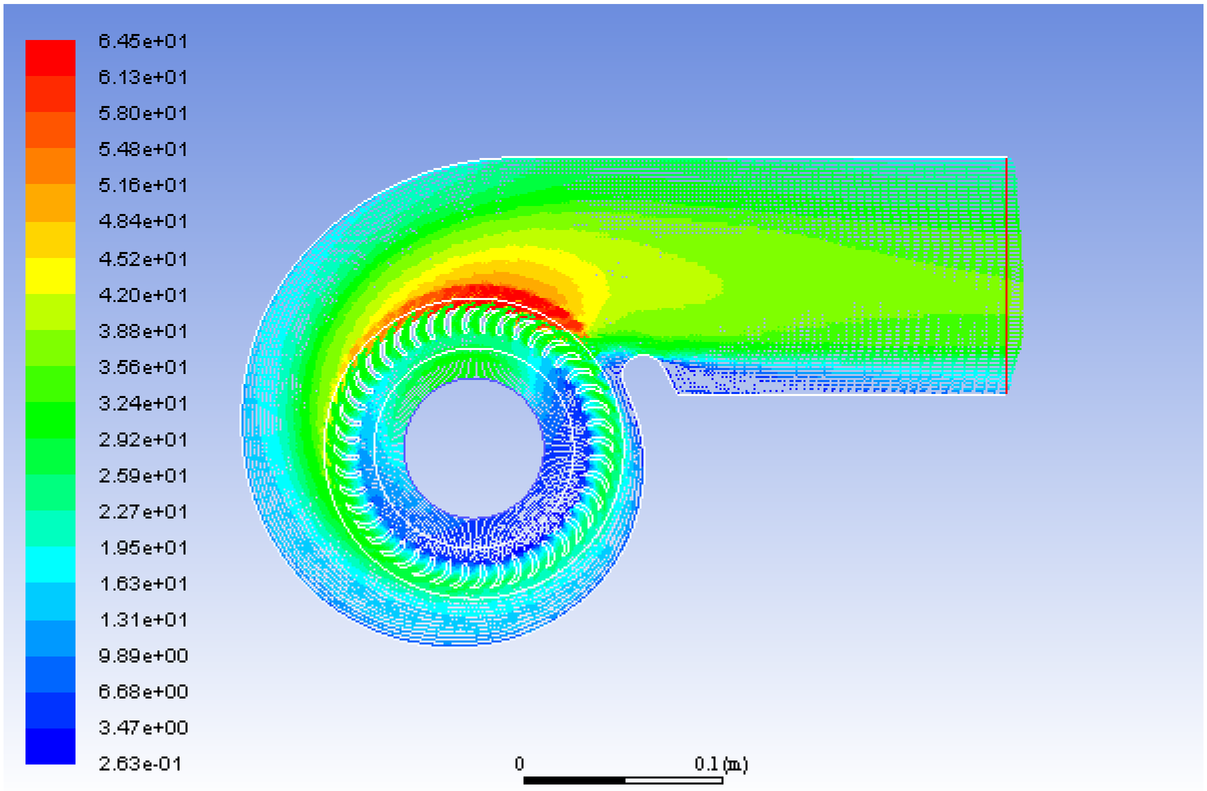
   *By default, **Auto Scale** is chosen. This will automatically scale the length of velocity vectors relative to the size of the smallest cell in the mesh. To increase the length of the "scaled" vectors, set the **Scale** factor to a value greater than* `1`.

b. Retain the default selection of **Velocity** from the **Vectors of** drop-down list.

c. Retain the default selection of **Velocity...** and **Velocity Magnitude** from the **Color by** drop-down list.

d. Click **Display** and close the **Vectors** dialog box (see Figure 10.5: Velocity Vectors (p. 450)).

Figure 10.5: Velocity Vectors



The velocity vectors show an area of flow separation near the bottom of the outlet duct. You can zoom in on this area and see the flow recirculation.

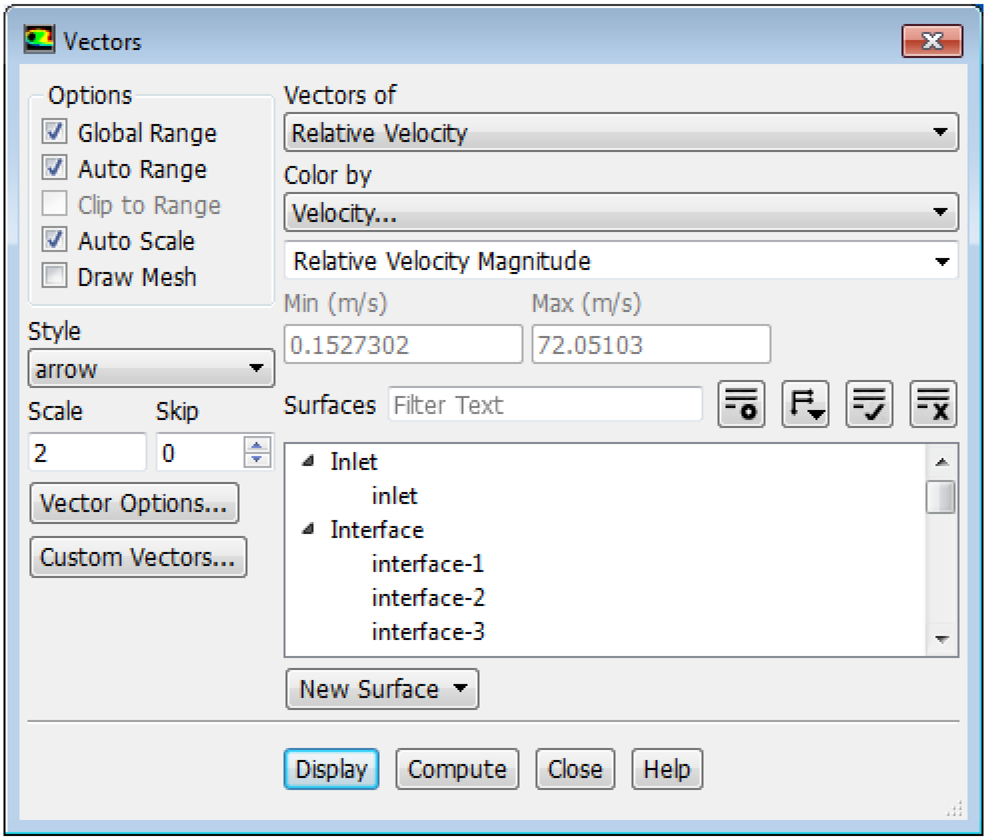
3. Display relative velocity vectors with respect to the rotational reference frame (**fluid-rotor**).

   a. In the **Reference Values** task page, select **fluid-rotor** from the **Reference Zone** drop-down list.

      Setup → **Reference Values**

   b. Open the **Vectors** dialog box.

      Postprocessing → **Graphics** → **Vectors** → **Edit...**



i. Select **Relative Velocity** from the **Vectors of** drop-down list.

ii. Select **Velocity...** and **Relative Velocity Magnitude** from the **Color by** drop-down list.

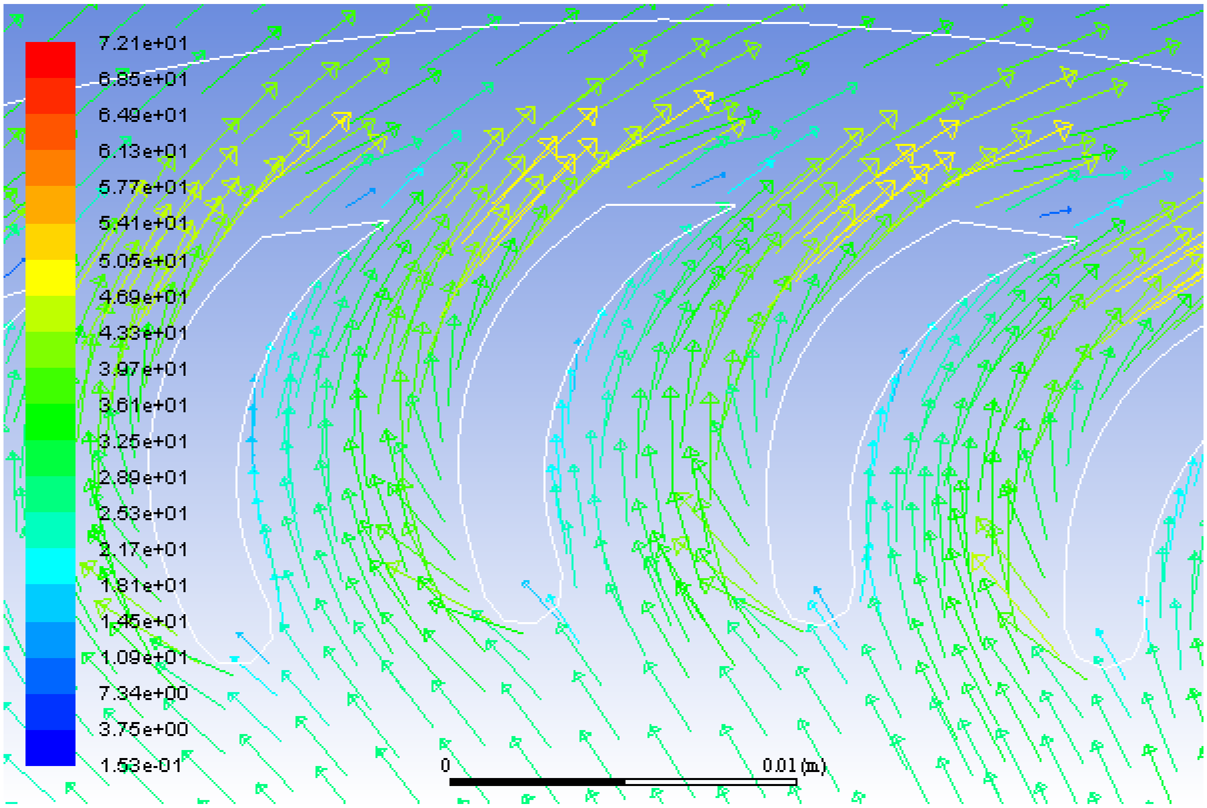
iii. Set **Scale** to 2.

iv. Click **Display** and close the **Vectors** dialog box.

The relative air velocity vectors viewed in the frame of reference rotating with the rotor are displayed.

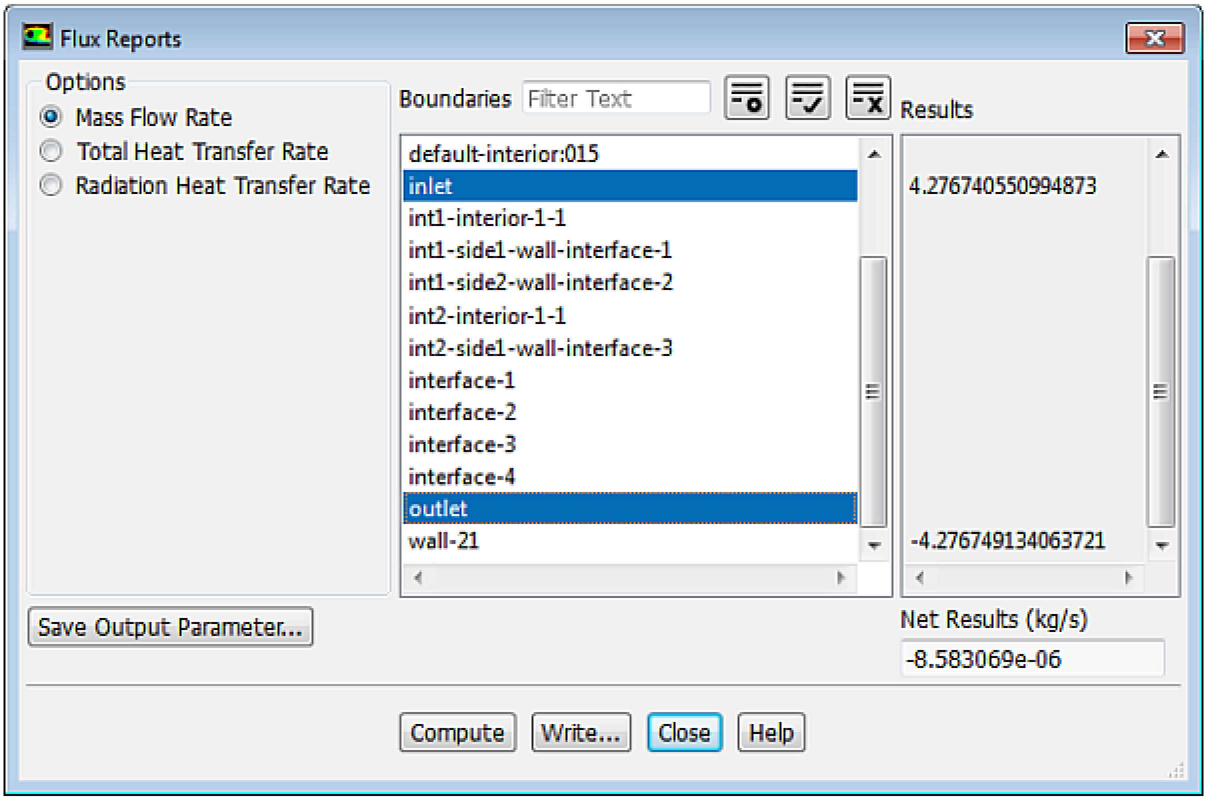
v. Zoom in on the rotor blade region as shown in Figure 10.6: Relative Velocity Vectors (p. 452) and examine the air flow through the rotor blade passages.

Figure 10.6: Relative Velocity Vectors



4. Report the mass flux at **inlet** and **outlet**.

   Postprocessing → **Reports** → **Fluxes...**



a. Retain the selection of **Mass Flow Rate** in the **Options** group box.

b. Select **inlet** and **outlet** in the **Boundaries** selection list.

c. Click **Compute**.

The net mass imbalance should be no more than a small fraction (say, 0.5%) of the total flux through the system. If a significant imbalance occurs, you should decrease your residual tolerances by at least an order of magnitude and continue iterating.

The flux report will compute fluxes only for boundary zones.

d. Close the **Flux Reports** dialog box.

Note

*You can use the **Surface Integrals** option to report fluxes on surfaces or planes.*

Results → **Reports** → **Surface Integrals** **Edit...**

10.5. Summary

This tutorial illustrates the procedure for setting up and solving problems with multiple reference frames using ANSYS Fluent. Although this tutorial considers only one rotating fluid zone, extension to multiple rotating fluid zones is straightforward as long as you delineate each fluid zone.

Note that this tutorial was solved using the default absolute velocity formulation. For some problems involving rotating reference frames, you may want to use the relative velocity formulation. See the Fluent User's Guide for details.

10.6. Further Improvements

This tutorial guides you through the steps to reach an initial solution. You may be able to obtain a more accurate solution by using an appropriate higher-order discretization scheme and by adapting the mesh. Mesh adaption can also ensure that the solution is independent of the mesh. These steps are demonstrated in Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121).

Chapter 11: Using Sliding Meshes

This tutorial is divided into the following sections:

11.1. Introduction

The analysis of turbomachinery often involves the examination of the transient effects due to flow interaction between the stationary components and the rotating blades. In this tutorial, the sliding mesh capability of ANSYS Fluent is used to analyze the transient flow in an axial compressor stage. The rotor-stator interaction is modeled by allowing the mesh associated with the rotor blade row to rotate relative to the stationary mesh associated with the stator blade row.

This tutorial demonstrates how to do the following:

- Create periodic zones.
- Set up the transient solver and cell zone and boundary conditions for a sliding mesh simulation.
- Set up the mesh interfaces for a periodic sliding mesh model.
- Sample the time-dependent data and view the mean value.

11.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

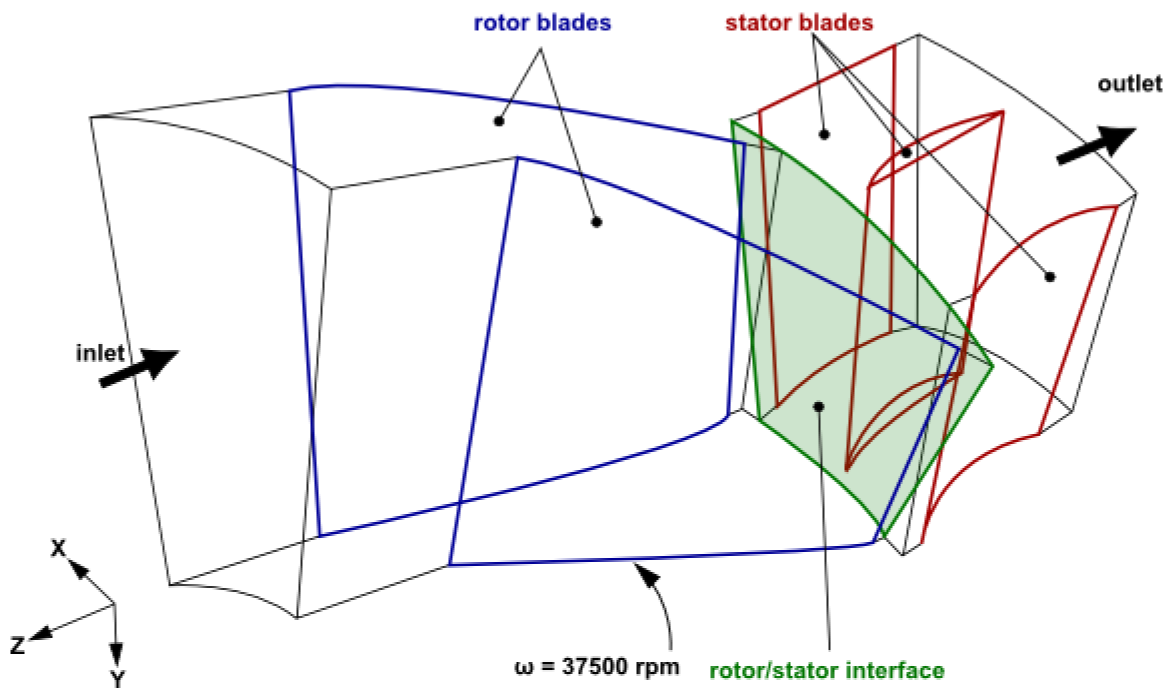
- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

and that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

11.3. Problem Description

The model represents a single-stage axial compressor composed of two blade rows. The first row is the rotor with 16 blades, which is operating at a rotational speed of 37,500 rpm. The second row is the stator with 32 blades. The blade counts are such that the domain is rotationally periodic, with a periodic angle of 22.5 degrees. This enables you to model only a portion of the geometry, namely, one rotor blade and two stator blades. Due to the high Reynolds number of the flow and the relative coarseness of the mesh (both blade rows are composed of only 13,856 cells total), the analysis will employ the inviscid model, so that ANSYS Fluent is solving the Euler equations.

Figure 11.1: Rotor-Stator Problem Description



11.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

11.4.1. Preparation

To prepare for running this tutorial:

1. Set up a working folder on the computer you will be using.

2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

> **Note**
>
> If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

3. Enter the name of this tutorial into the search bar.

4. Narrow the results by using the filter on the left side of the page.

   a. Click **ANSYS Fluent** under **Product**.

   b. Click **18.0** under **Version**.

5. Select this tutorial from the list.

6. Click the **sliding_mesh_R180.zip** link to download the input files.

7. Unzip `sliding_mesh_R180.zip` to your working folder.

   The mesh file `axial_comp.msh` can be found in the `sliding_mesh` directory created after unzipping the file.

8. Use Fluent Launcher to start the **3D** version of ANSYS Fluent.

   Fluent Launcher displays your **Display Options** preferences from the previous session.

   For more information about Fluent Launcher, see starting ANSYS Fluent using the Fluent Launcher in the Fluent Getting Started Guide.

9. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.

10. Enable single precision (disable **Double Precision**).

11. Do not enable **Meshing Mode**.

12. Run in **Serial** under **Processing Options**.

11.4.2. Mesh

1. Read in the mesh file `axial_comp.msh`.

   File → Read → Mesh...

11.4.3. General Settings

1. Check the mesh.

   Setting Up Domain → Mesh → Check

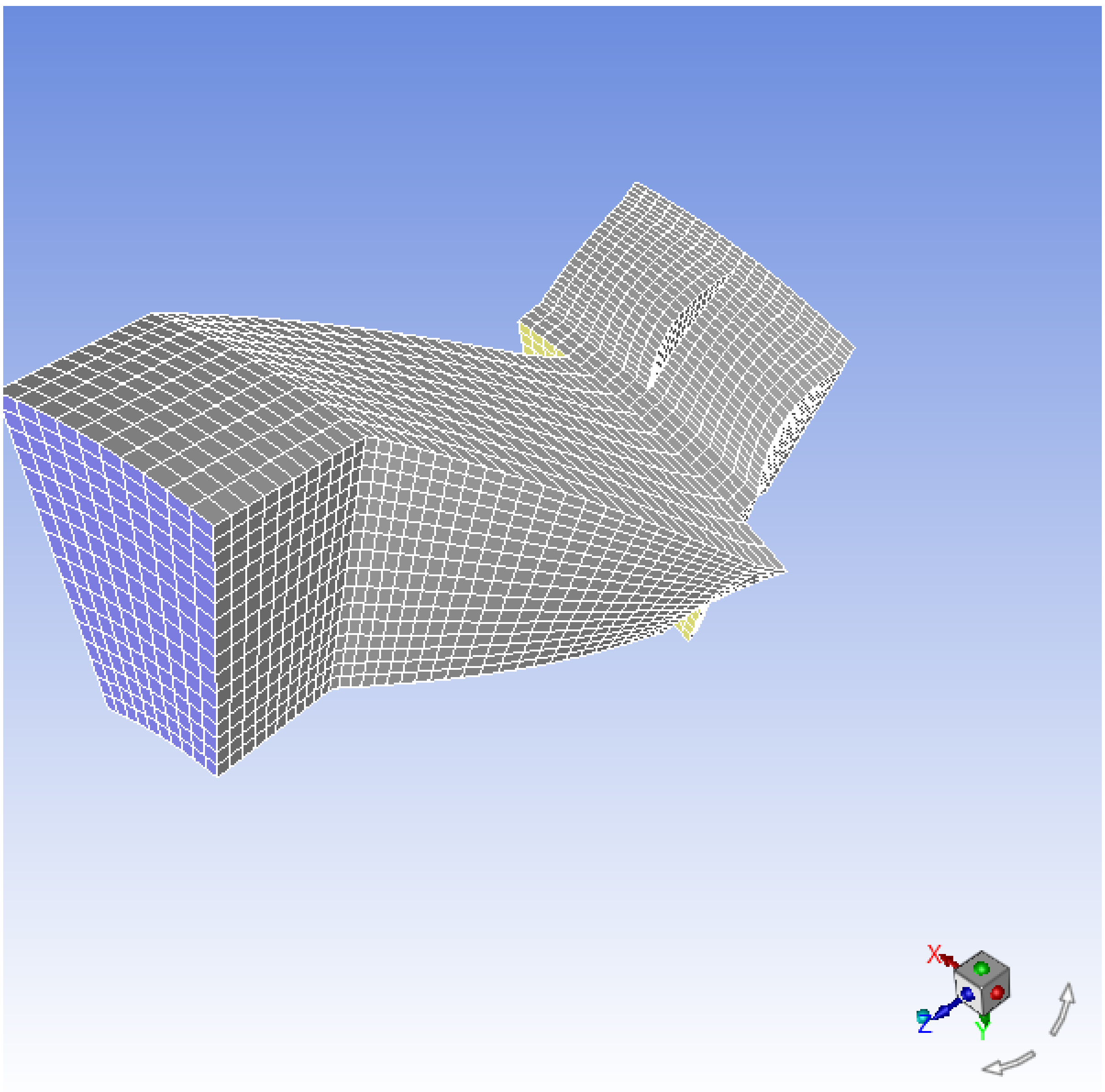
ANSYS Fluent will perform various checks on the mesh and will report the progress in the console. Ensure that the reported minimum volume is a positive number.

Warnings will be displayed regarding unassigned interface zones, resulting in the failure of the mesh check. You do not need to take any action at this point, as this issue will be rectified when you define the mesh interfaces in a later step.

2. Examine the mesh (Figure 11.2: Rotor-Stator Display (p. 458)).

   Orient the view to display the mesh as shown in Figure 11.2: Rotor-Stator Display (p. 458). The inlet of the rotor mesh is colored blue, the interface between the rotor and stator meshes is colored yellow, and the outlet of the stator mesh is colored red.

Figure 11.2: Rotor-Stator Display



3. Use the text user interface to change zones `rotor-per-1` and `rotor-per-3` from wall zones to periodic zones.

a. Press **Enter** in the console to get the command prompt (>).

b. Type the commands as shown below in the console:

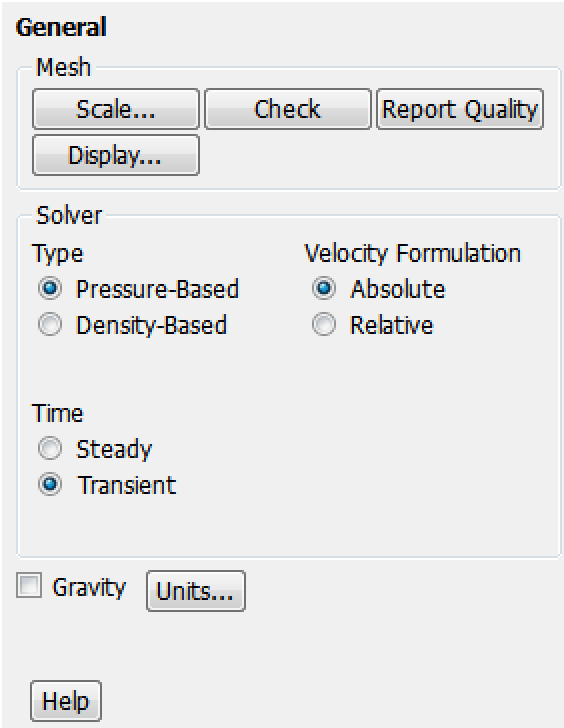
```
> mesh/modify-zones/list-zones
  id  name                       type                material             kind
----  -------------------------  ------------------  -------------------  ----
  13  fluid-rotor                fluid               air                  cell
  28  fluid-stator               fluid               air                  cell
   2  default-interior:0         interior                                 face
  15  default-interior           interior                                 face
   3  rotor-hub                  wall                air                  face
   4  rotor-shroud               wall                air                  face
   7  rotor-blade-1              wall                air                  face
   8  rotor-blade-2              wall                air                  face
  16  stator-hub                 wall                air                  face
  17  stator-shroud              wall                air                  face
  20  stator-blade-1             wall                air                  face
  21  stator-blade-2             wall                air                  face
  22  stator-blade-3             wall                air                  face
  23  stator-blade-4             wall                air                  face
   5  rotor-inlet                pressure-inlet                           face
  19  stator-outlet              pressure-outlet                          face
  10  rotor-per-1                wall                air                  face
  12  rotor-per-2                wall                air                  face
  24  stator-per-2               wall                air                  face
  26  stator-per-1               wall                air                  face
   6  rotor-interface            interface                                face
  18  stator-interface           interface                                face
  11  rotor-per-4                wall                air                  face
   9  rotor-per-3                wall                air                  face
  25  stator-per-4               wall                air                  face
  27  stator-per-3               wall                air                  face

 /mesh/modify-zones> make-periodic
 Periodic zone [()] 10
 Shadow zone [()] 9
 Rotational periodic? (if no, translational) [yes] yes
 Create periodic zones? [yes] yes


     zone 9 deleted

      created periodic zones.
```

4. Similarly, change the following wall zone pairs to periodic zones:

Zone Pairs	Respective Zone IDs
`rotor-per-2` and `rotor-per-4`	12 and 11
`stator-per-1` and `stator-per-3`	26 and 27
`stator-per-2` and `stator-per-4`	24 and 25

5. Define the solver settings.

Setup → **General**

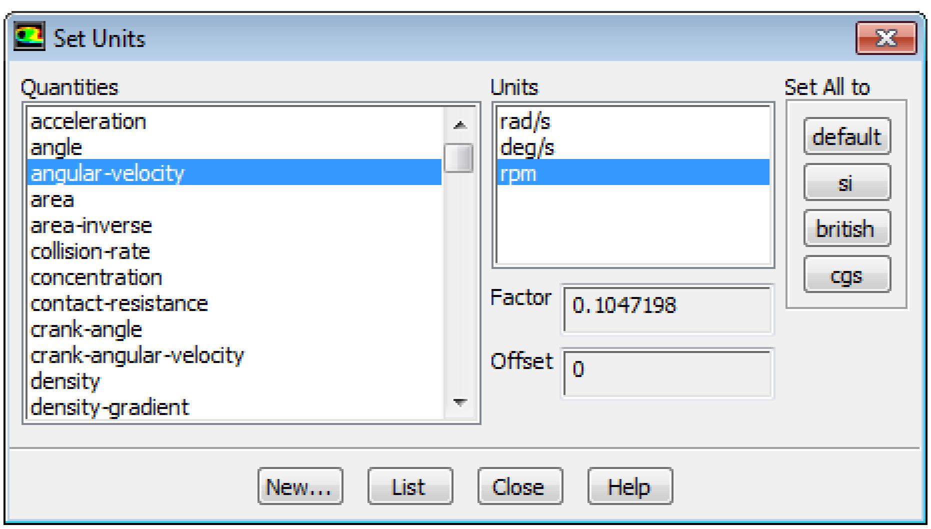


a. Retain the default selection of **Pressure-Based** in the **Type** list.

b. Select **Transient** in the **Time** list.

6. Define the units for the model.

Setup → **General** → **Units...**



a. Select **angular-velocity** from the **Quantities** selection list.

b. Select **rpm** from the **Units** selection list.

c. Select **pressure** from the **Quantities** selection list.

*Scroll down the **Quantities** list to find **pressure**.*

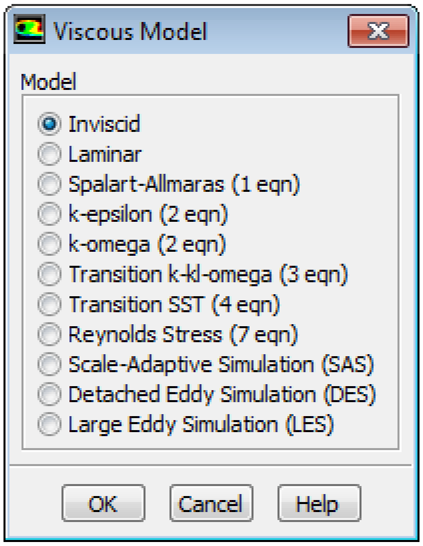
d. Select **atm** from the **Units** selection list.

e. Close the **Set Units** dialog box.

11.4.4. Models

1. Enable the inviscid model.

Setup → **Models** → **Viscous** **Edit...**



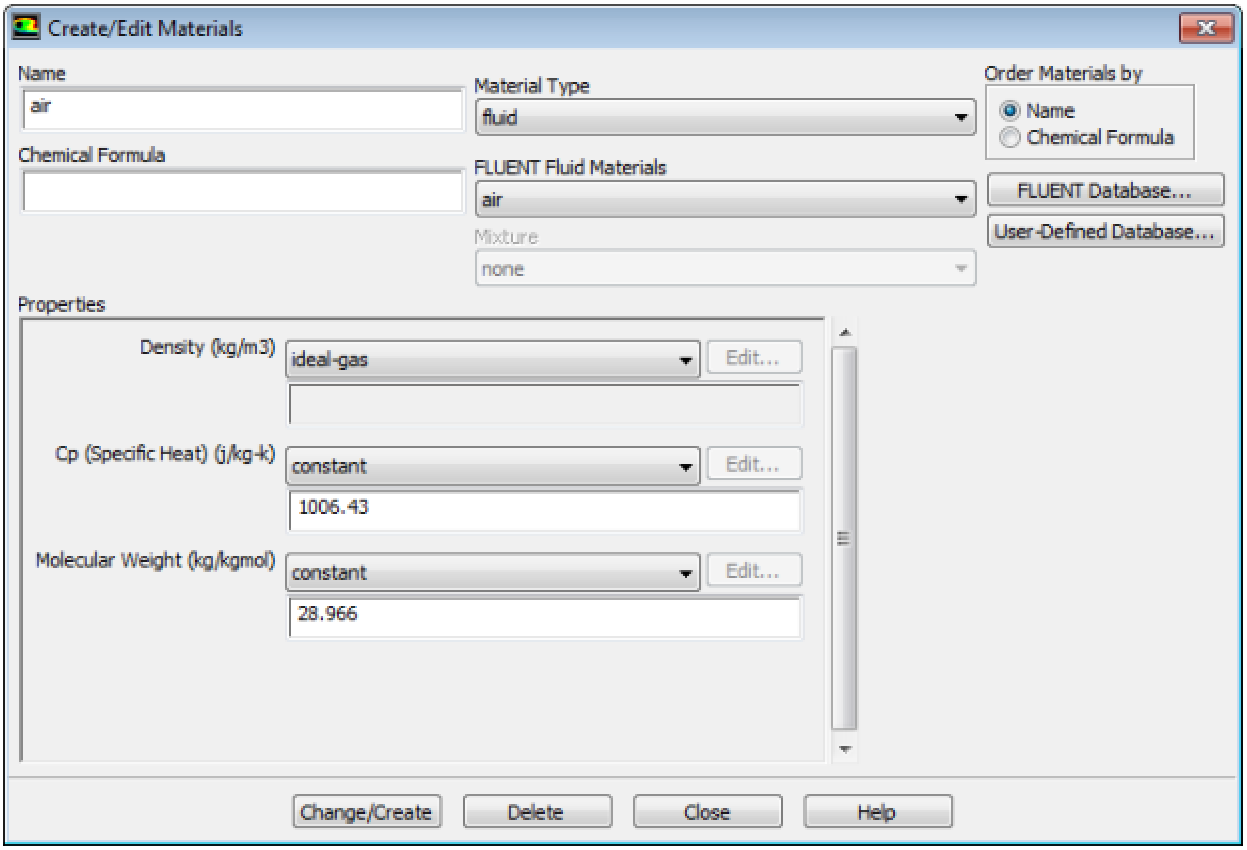
a. Select **Inviscid** in the **Model** list.

b. Click **OK** to close the **Viscous Model** dialog box.

11.4.5. Materials

1. Specify air (the default material) as the fluid material, using the ideal gas law to compute density.

Setup → **Materials** → **Fluids** → **air** → **Edit...**



a. Retain the default entry of **air** in the **Name** text entry field.

b. Select **ideal-gas** from the **Density** drop-down list in the **Properties** group box.

c. Retain the default values for all other properties.

d. Click **Change/Create** and close the **Create/Edit Materials** dialog box.

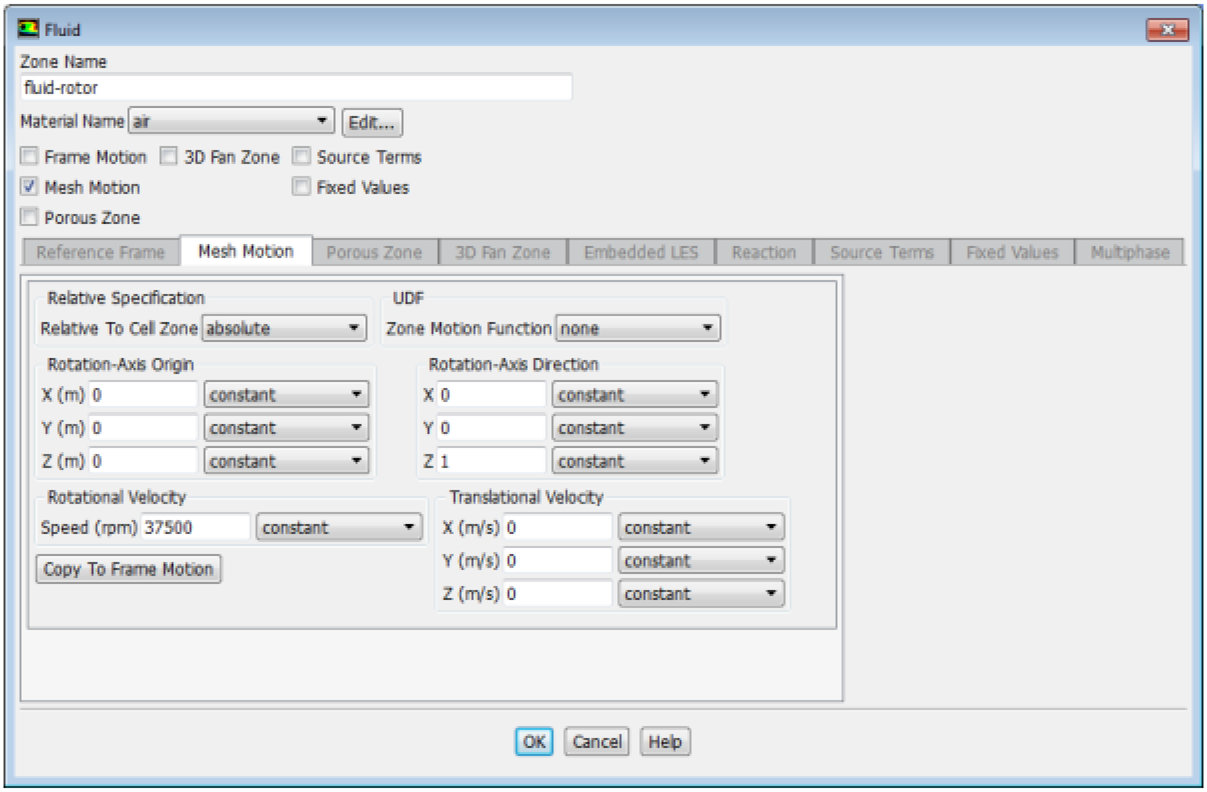
As reported in the console, ANSYS Fluent will automatically enable the energy equation, since this is required when using the ideal gas law to compute the density of the fluid.

11.4.6. Cell Zone Conditions

Setup → **Cell Zone Conditions**

1. Set the cell zone conditions for the fluid in the rotor (**fluid-rotor**).

   Setup → **Cell Zone Conditions** → **fluid-rotor** → **Edit...**



a. Enable **Mesh Motion.**

b. Click the **Mesh Motion** tab.

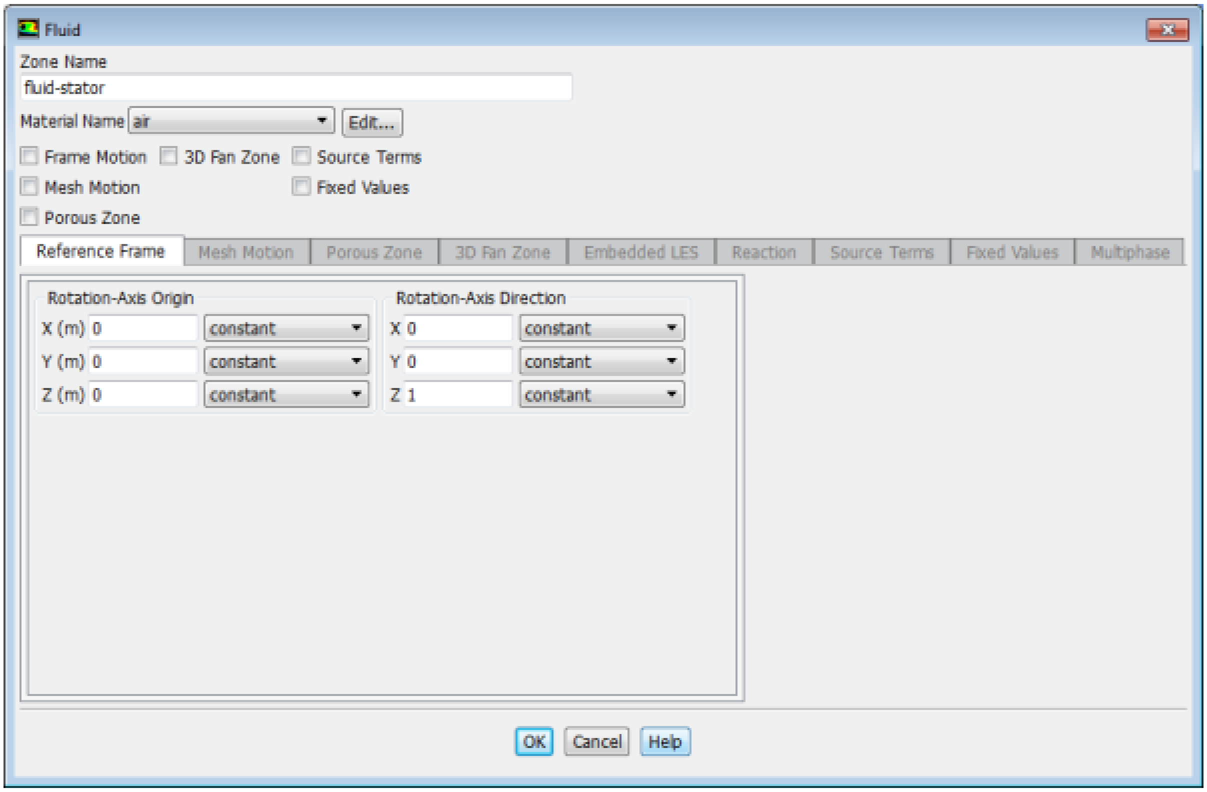
c. Retain the default values of (**0**, **0**, **1**) for **X**, **Y**, and **Z** in the **Rotation-Axis Direction** group box.

d. Enter `37500` rpm for **Speed** in the **Rotational Velocity** group box.

e. Click **OK** to close the **Fluid** dialog box.

2. Set the cell zone conditions for the fluid in the stator (**fluid-stator**).

Setup → **Cell Zone Conditions** → **fluid-stator** → **Edit...**

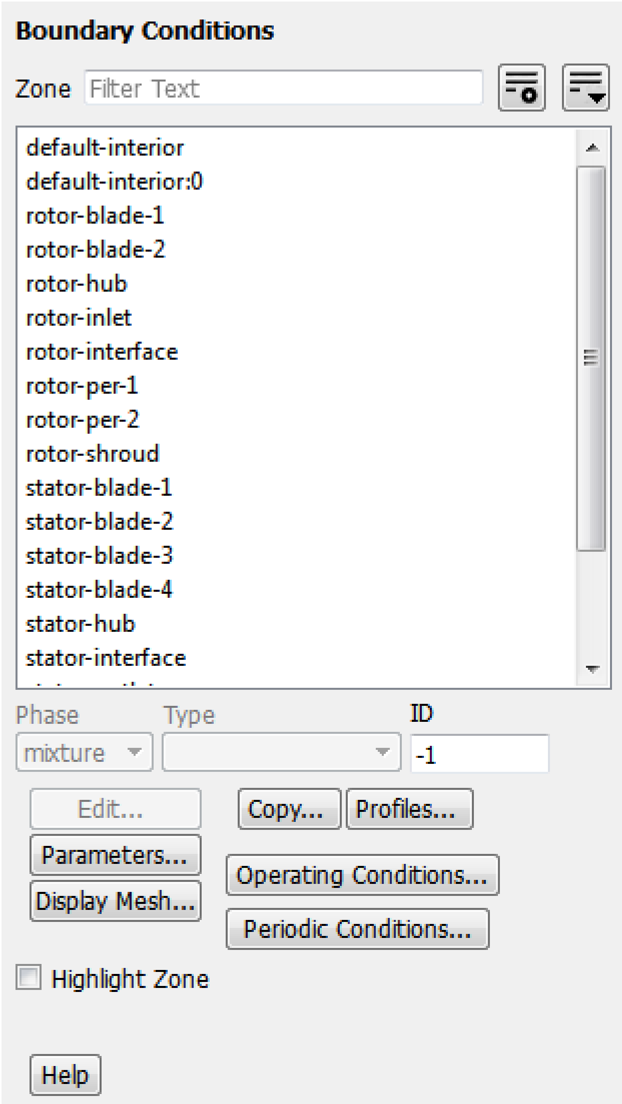


a. Retain the default values of (**0**, **0**, **1**) for **X**, **Y**, and **Z** in the **Rotation-Axis Direction** group box.

b. Click **OK** to close the **Fluid** dialog box.

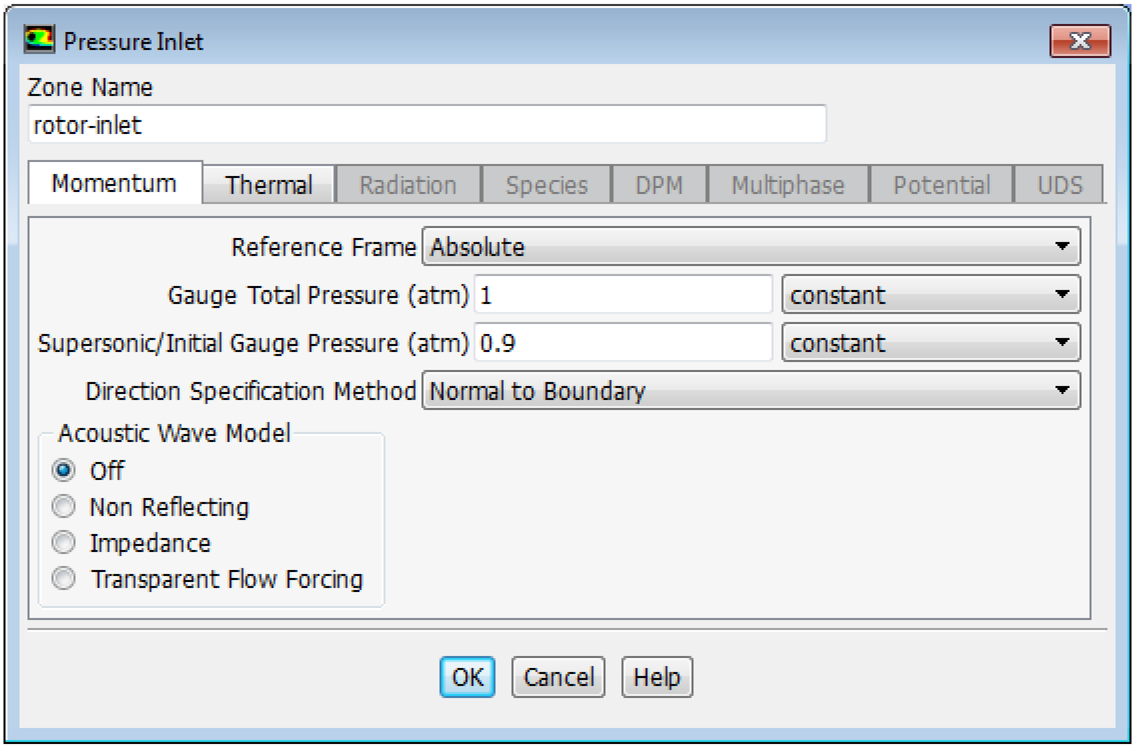
11.4.7. Boundary Conditions

Setup → **Boundary Conditions**



1. Enter `rotor-inlet` into the **Zone** field to filter the zone list.

   Setup → **Boundary Conditions** → **rotor-inlet** → **Edit...**

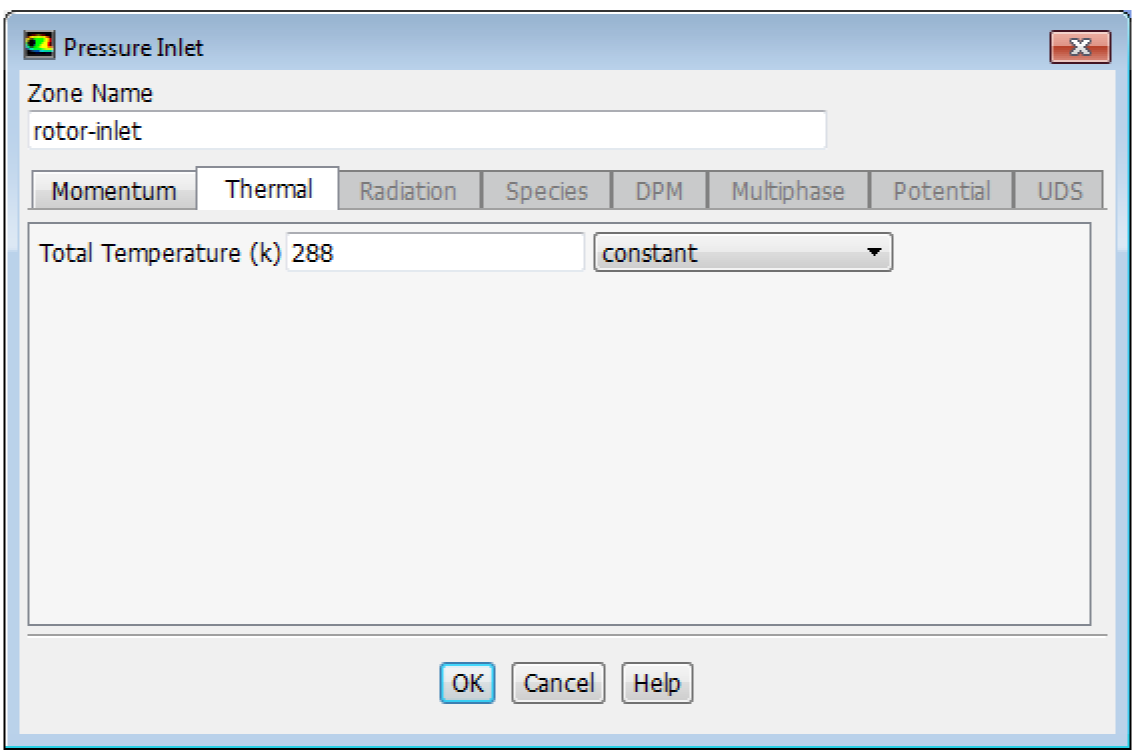


a. Enter `1.0` atm for **Gauge Total Pressure**.

b. Enter `0.9` atm for **Supersonic/Initial Gauge Pressure**.

   For information about the Supersonic/Initial Gauge Pressure, see the Fluent User's Guide.

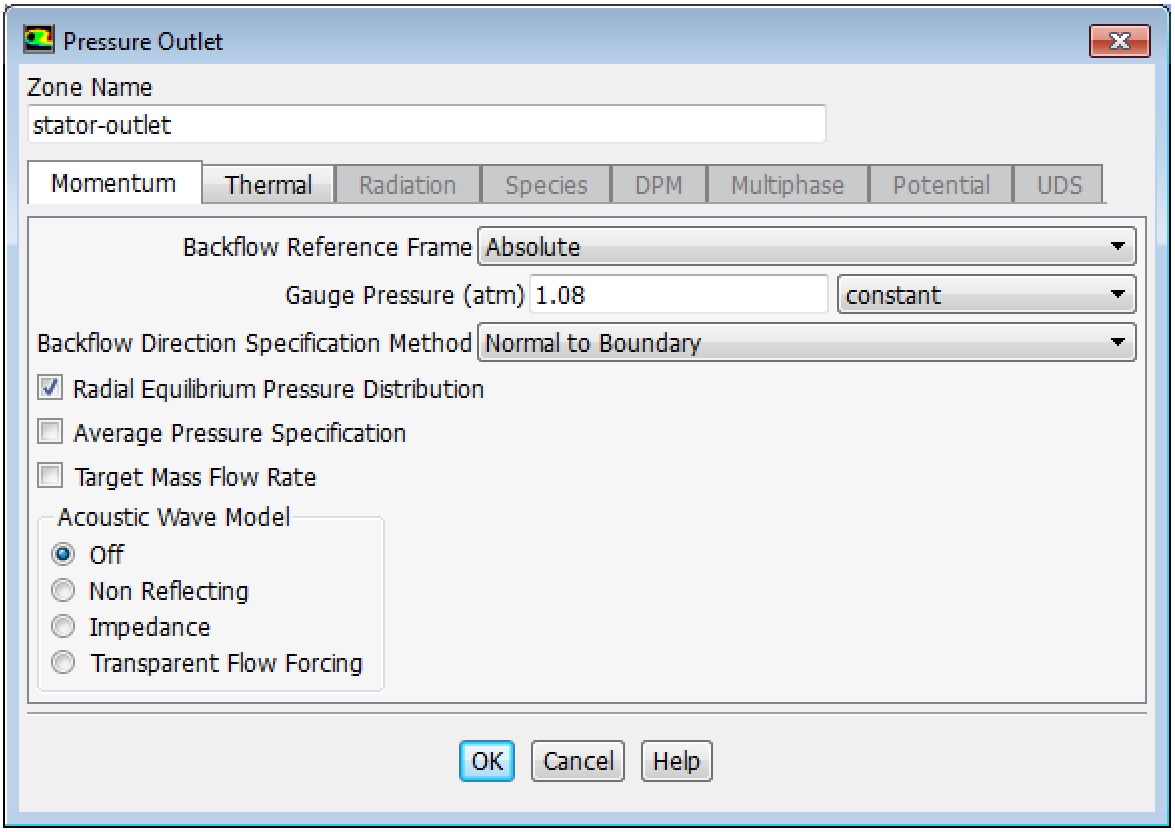
c. Click the **Thermal** tab and enter 288 K for **Total Temperature**.



d. Click **OK** to close the **Pressure Inlet** dialog box.

2. Enter `stator-outlet` into the **Zone** field to filter the zone list.

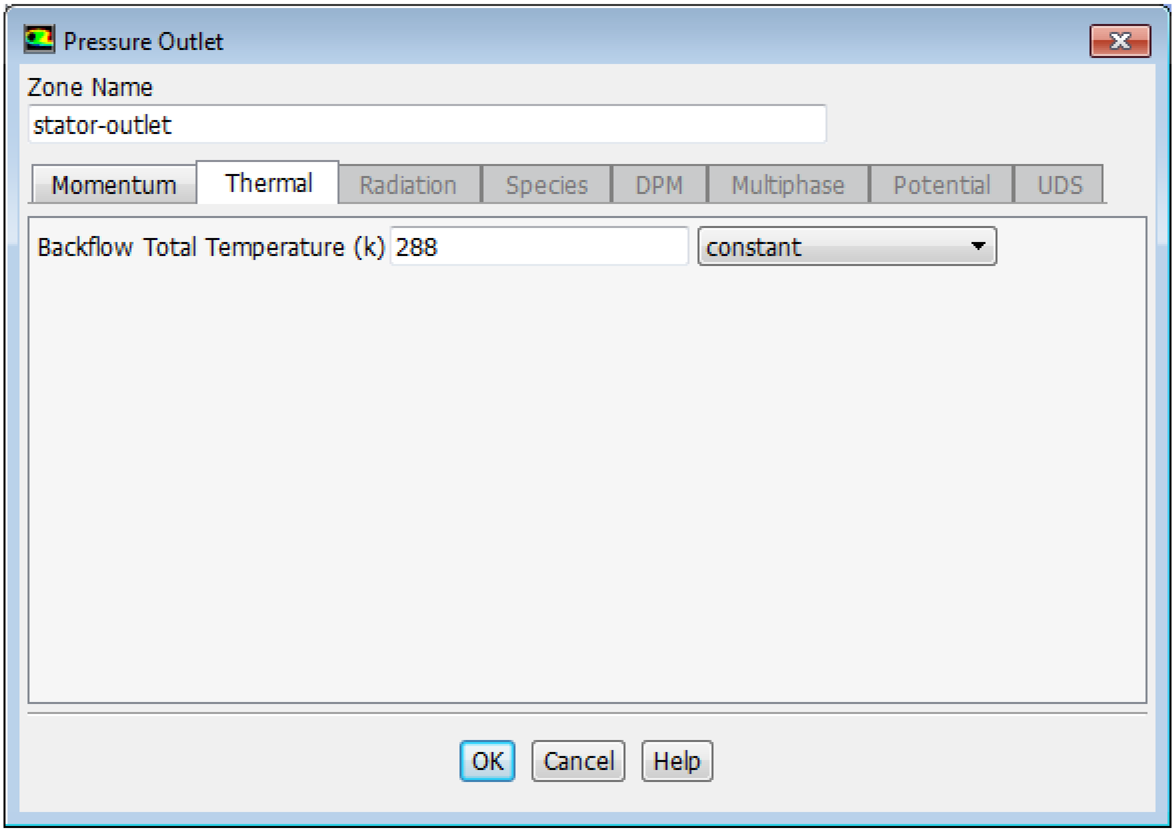
Setup → **Boundary Conditions** → **stator-outlet** → **Edit...**



a. Enter `1.08` atm for **Gauge Pressure**.

b. Enable **Radial Equilibrium Pressure Distribution**.

c. Click the **Thermal** tab and enter 288 K for **Backflow Total Temperature**.



d. Click **OK** to close the **Pressure Outlet** dialog box.

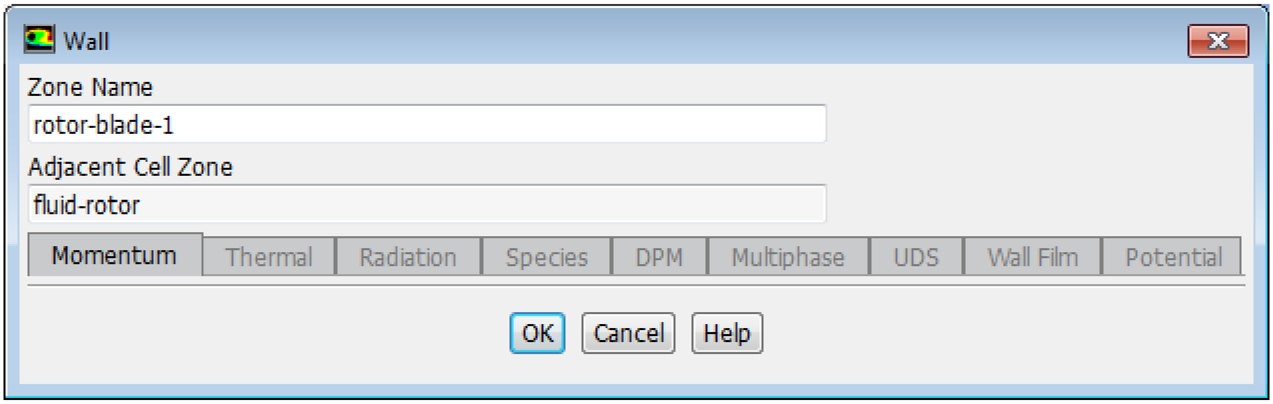
Note

The momentum settings and temperature you input at the pressure outlet will be used only if flow enters the domain through this boundary. It is important to set reasonable values for these downstream scalar values, in case flow reversal occurs at some point during the calculation.

---

3. Retain the default boundary conditions for all wall zones.

Setup → **Boundary Conditions** → **rotor-blade-1** → **Edit...**



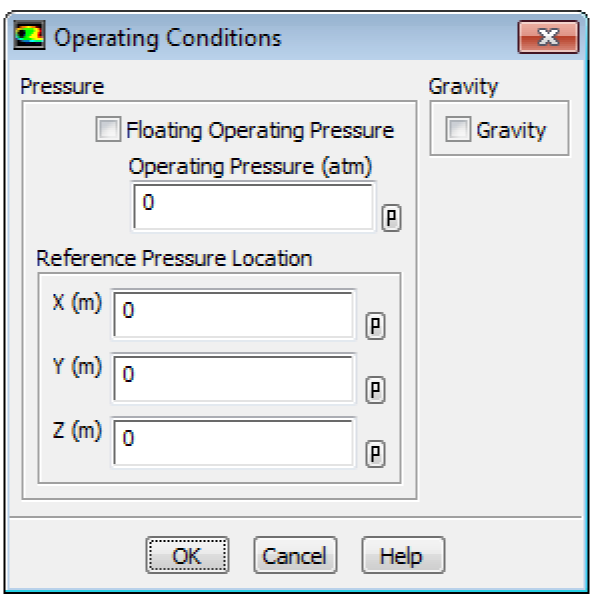
Note

For wall zones, ANSYS Fluent always imposes zero velocity for the normal velocity component, which is required whether or not the fluid zone is moving. This condition is all that is required for an inviscid flow, as the tangential velocity is computed as part of the solution.

11.4.8. Operating Conditions

1. Set the operating pressure.

   Setup → **Boundary Conditions** → **Operating Conditions...**



   a. Enter 0 atm for **Operating Pressure**.

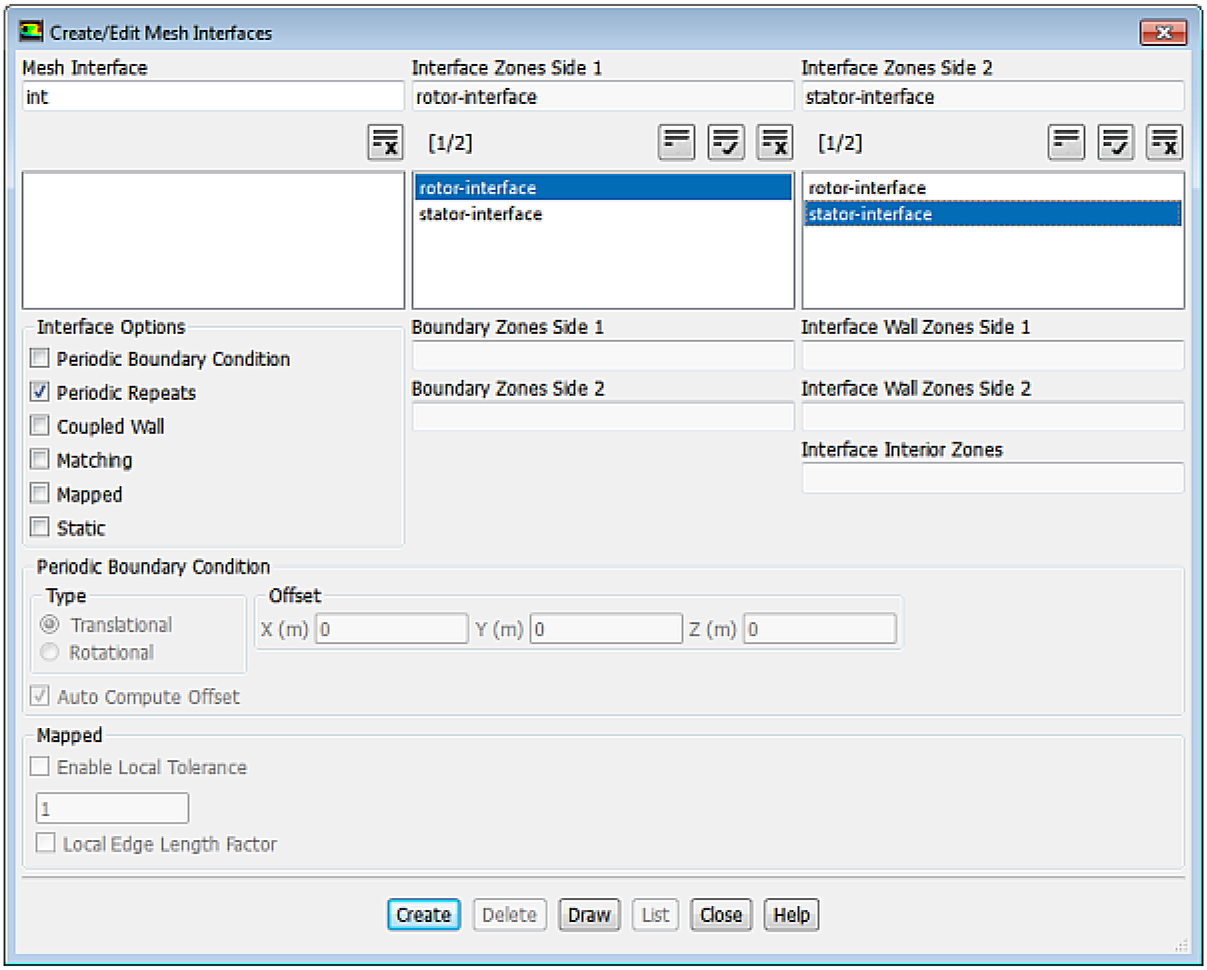
   b. Click **OK** to close the **Operating Conditions** dialog box.

   Since you have specified the boundary condition inputs for pressure in terms of absolute pressures, you have to set the operating pressure to zero. Boundary condition inputs for pressure should always be relative to the value used for operating pressure.

11.4.9. Mesh Interfaces

1. Create a periodic mesh interface between the rotor and stator mesh regions.

   Setup → **Mesh Interfaces** **New...**



   a. Enter `int` for **Mesh Interface**.

   b. Enable **Periodic Repeats** in the **Interface Options** group box.

      Enabling this option, allows ANSYS Fluent to treat the interface between the sliding and non-sliding zones as periodic where the two zones do not overlap.

   c. Select **rotor-interface** from the **Interface Zones Side 1** selection list.

      Note

      In general, when one interface zone is smaller than the other, it is recommended that you choose the smaller zone as **Interface Zone 1**. In this case, since both zones are approximately the same size, the order is not significant.

   d. Select **stator-interface** from the **Interface Zones Side 2** selection list.

e. Click **Create** and close the **Create/Edit Mesh Interfaces** dialog box.

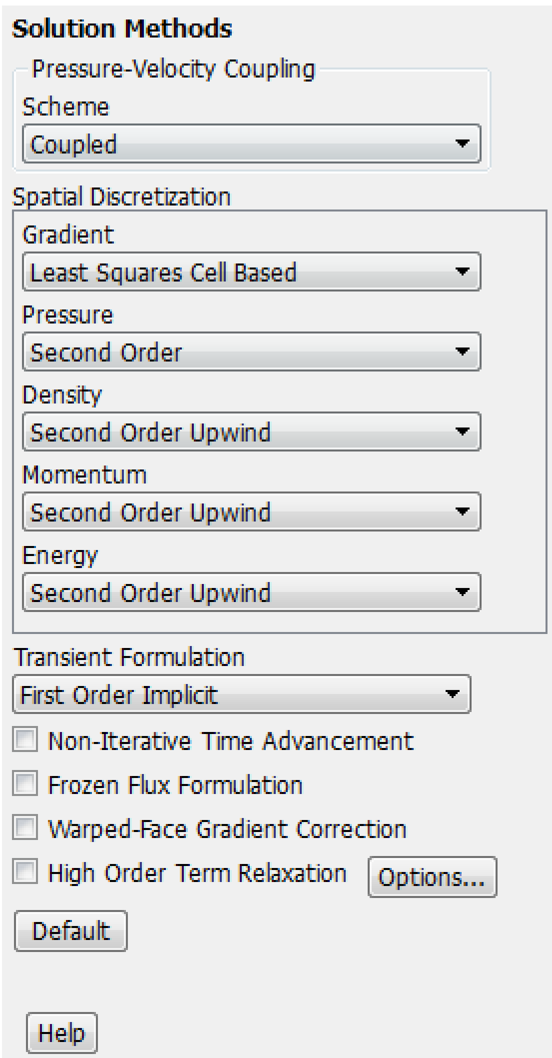
2. Check the mesh again to verify that the warnings displayed earlier have been resolved.

Setting Up Domain → **Mesh** → **Check**

11.4.10. Solution

1. Set the solution parameters.

Solving → **Solution** → **Methods...**

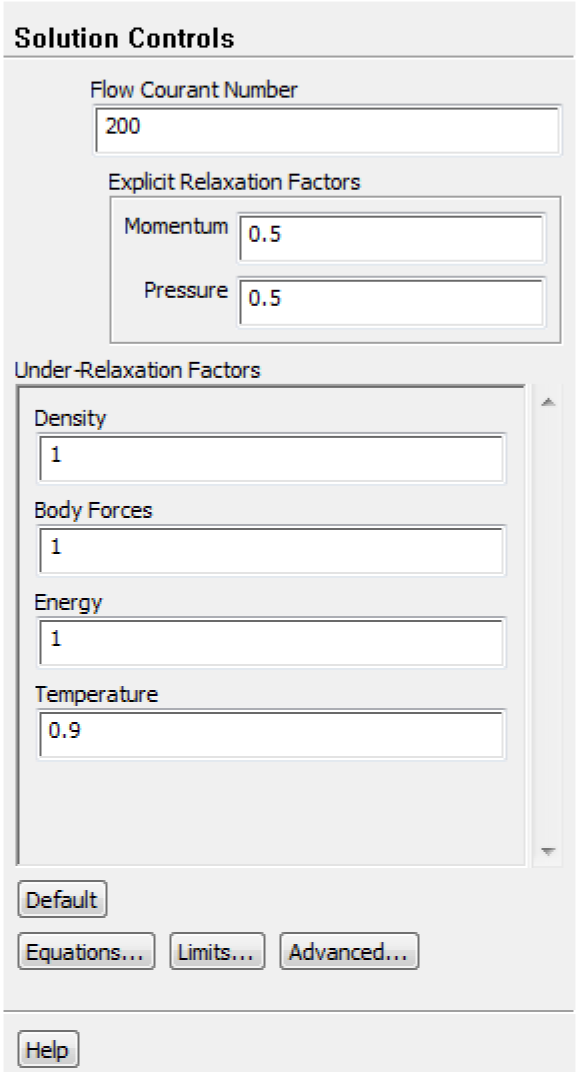


a. Select **Coupled** from the **Pressure-Velocity Coupling** group box.

For many general fluid-flow problems, convergence speed can be improved by using the coupled solver.

2. Change the Solution Controls

Solving → **Controls** → **Controls...**

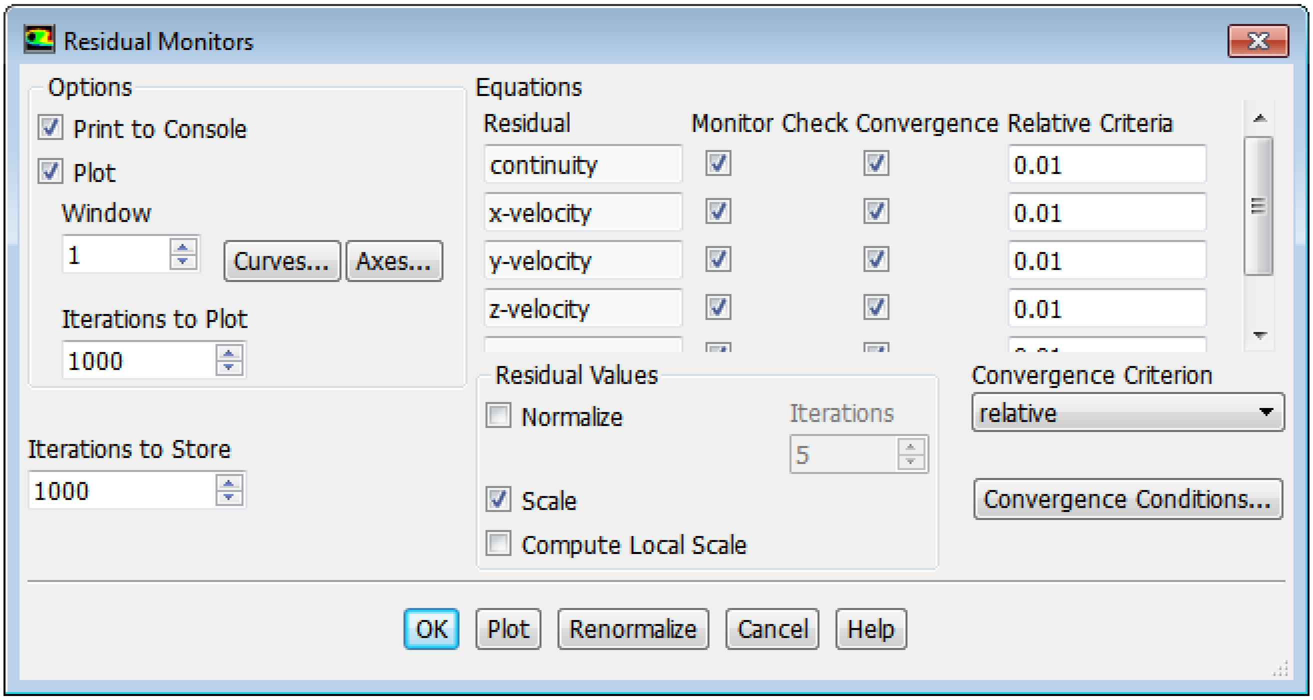


a. Enter `0.5` for **Momentum** and **Pressure** in the **Explicit Relaxation Factors** group box.

b. Enter `0.9` for **Temperature** in the **Under-Relaxation Factors** group box.

3. Enable the plotting of residuals during the calculation.

Solving → Reports → Residuals...



a. Ensure that the **Plot** is selected in the **Options** group box.

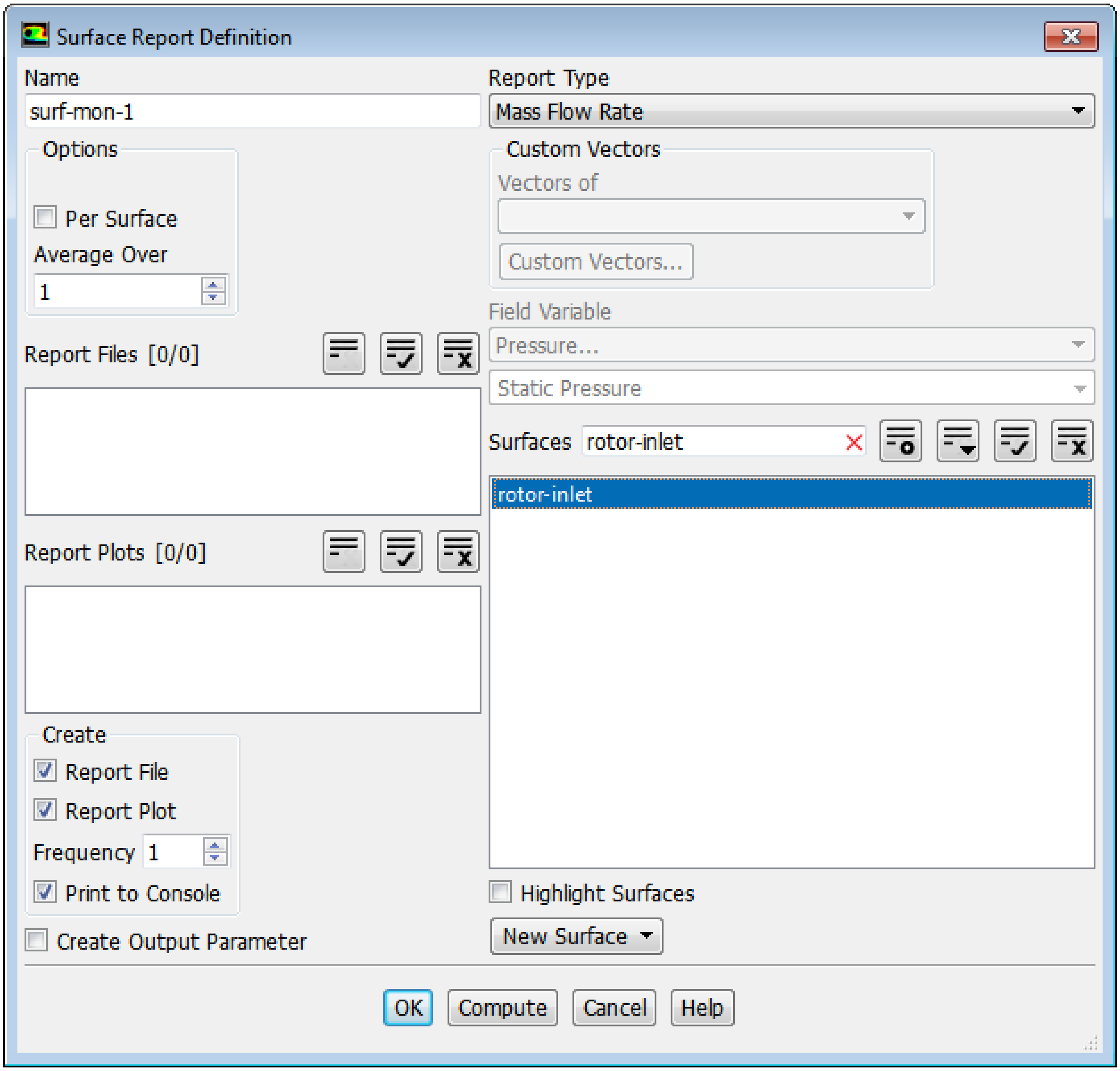
b. Select **relative** from the **Convergence Criterion** drop-down list.

c. Enter `0.01` for **Relative Criteria** for each **Residual** (**continuity**, **x-velocity**, **y-velocity**, **z-velocity**, and **energy**).

d. Click **OK** to close the **Residual Monitors** dialog box.

4. Enable the plotting of mass flow rate at the inlet (**rotor-inlet**).

Solving → **Reports** → **Definitions** → **New** → **Surface Report** → **Mass Flow Rate...**



a. Enter `surf-mon-1` for the **Name** of the surface report definition.

b. In the **Create** group box, enable **Report File**, **Report Plot** and **Print to Console**.

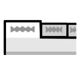
c. Enter `rotor-inlet` in the **Surfaces** field to filter the list.

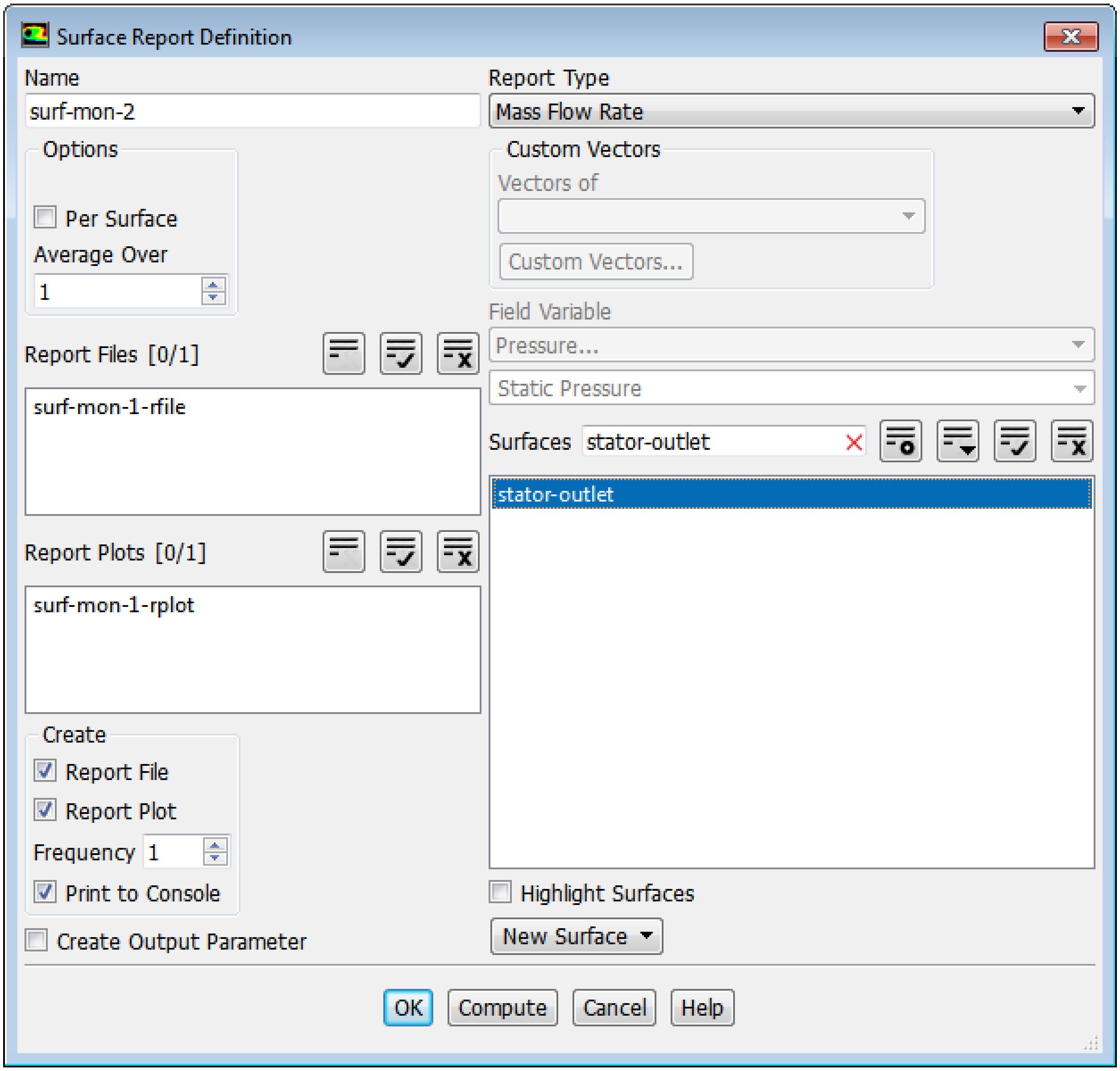
d. Select **rotor-inlet** from the **Surfaces** selection list.

e. Click **OK** to save the surface report definition settings and close the **Surface Report Definition** dialog box.

   surf-mon-1-rplot and **surf-mon-1-rfile** that are automatically generated by Fluent appear in the tree (under **Solution/Monitors/Report Plots** and **Solution/Monitors/Report Files**, respectively).

5. Enable the plotting of mass flow rate at the outlet (**stator-outlet**).

 **Solving** → **Reports** → **Definitions** → **New** → **Surface Report** → **Mass Flow Rate...**



a. Enter `surf-mon-2` for the **Name** of the surface report definition.

b. In the **Create** group box, enable **Report File**, **Report Plot** and **Print to Console**.

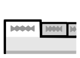
c. Enter `stator-outlet` in the **Surfaces** field to filter the list.

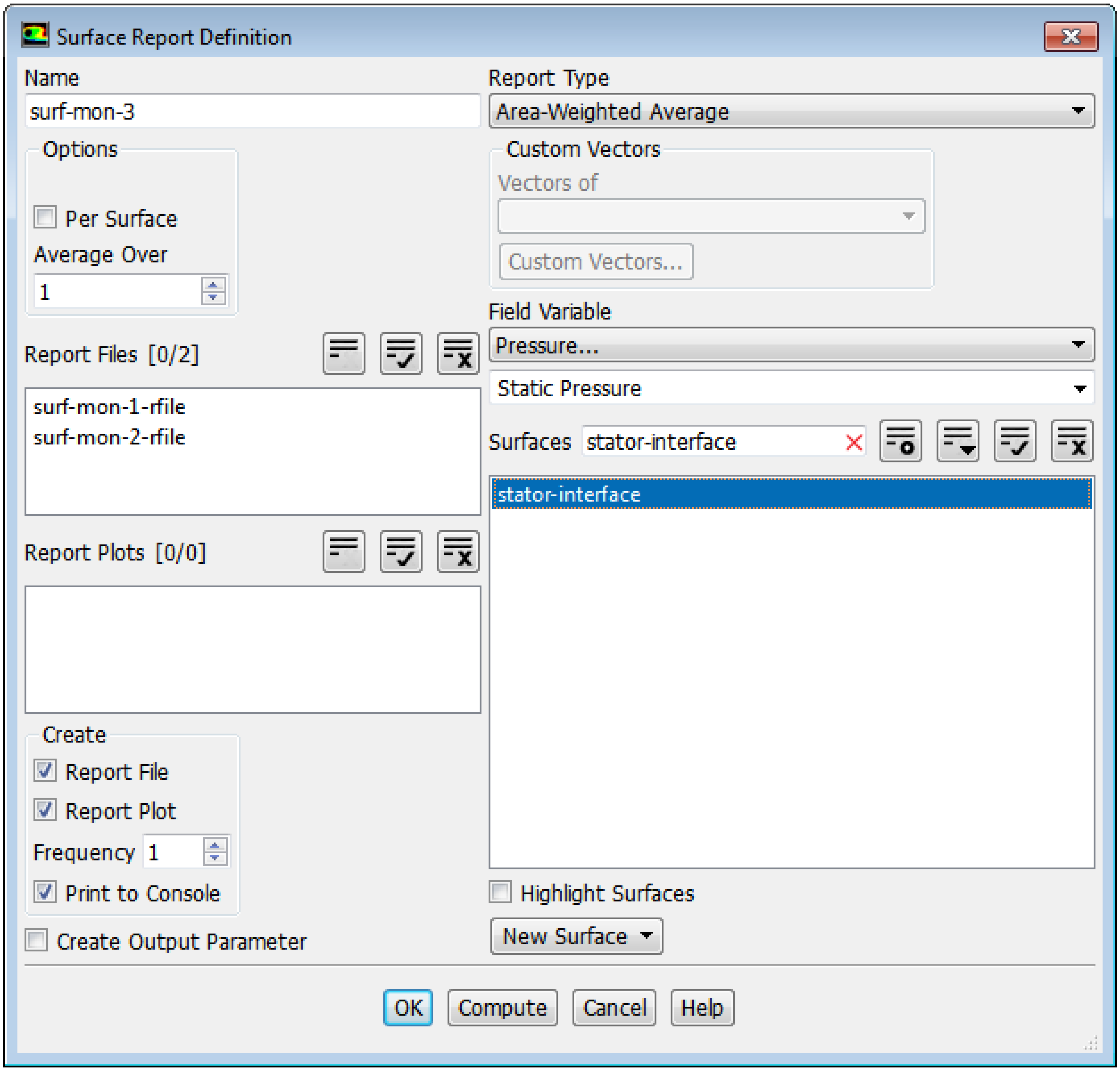
d. Select **stator-outlet** from the **Surfaces** selection list.

e. Click **OK** to save the surface report definition settings and close the **Surface Report Definition** dialog box.

   surf-mon-2-rplot and **surf-mon-2-rfile** that are automatically generated by Fluent appear in the tree (under **Solution/Monitors/Report Plots** and **Solution/Monitors/Report Files**, respectively).

6. Enable the plotting of the area-weighted average of the static pressure at the interface (**stator-interface**).

 **Solving** → **Reports** → **Definitions** → **New** → **Surface Report** → **Area-Weighted Average...**



a. Enter `surf-mon-3` for the **Name** of the surface report definition.

b. In the **Create** group box, enable **Report File**, **Report Plot** and **Print to Console**.

c. Retain the default selection of **Pressure...** and **Static Pressure** from the **Field Variable** drop-down lists.

d. Enter `stator-interface` in the **Surfaces** field to filter the list.

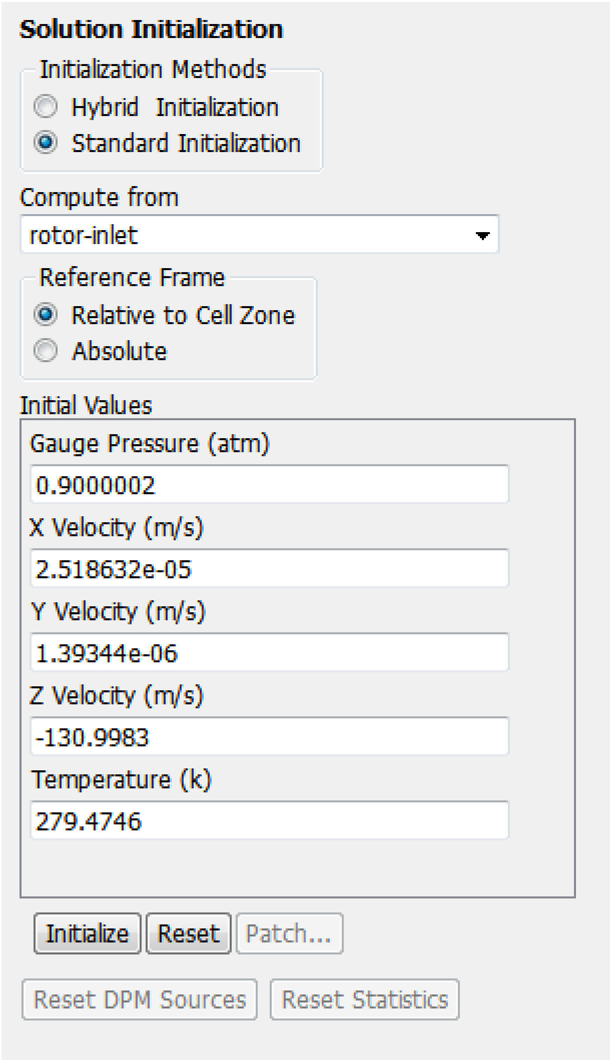
e. Select **stator-interface** from the **Surfaces** selection list.

f. Click **OK** to save the surface report definition settings and close the **Surface Report Definition** dialog box.

   surf-mon-3-rplot and **surf-mon-3-rfile** that are automatically generated by Fluent appear in the tree (under **Solution/Monitors/Report Plots** and **Solution/Monitors/Report Files**, respectively).

7. Initialize the solution using the values at the inlet (**rotor-inlet**).

   Solving → Initialization → Options...



   a. Select **rotor-inlet** from the **Compute from** drop-down list.

   b. Select **Absolute** in the **Reference Frame** list.

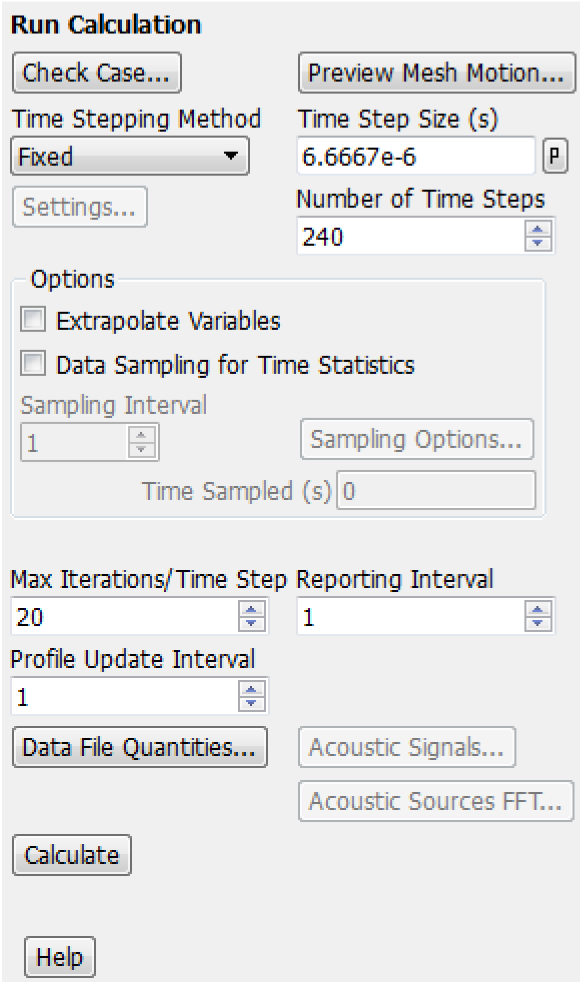
   c. Click **Initialize**.

8. Save the initial case file (`axial_comp.cas.gz`).

   File → Write → Case...

9. Run the calculation for one revolution of the rotor.

   Solving → Run Calculation → Advanced...



a. Enter `6.6667e-6` s for **Time Step Size**.

   The time step is set such that the passing of a single rotor blade is divided into 15 time steps. There are 16 blades on the rotor. Therefore, in each time step the rotor rotates 360/16/15=1.5 degrees. With a rotational speed of 37,500 rpm (225,000 deg/sec), 1.5 degrees of rotation takes 1.5 / 2.25e5 = 6.6667e-6 sec.

b. Enter `240` for **Number of Time Steps**.

   *There are 16 blades on the rotor, and each rotor blade period corresponds to 15 time steps (see above). Therefore, a complete revolution of the rotor will take 16*15=240 time steps.*

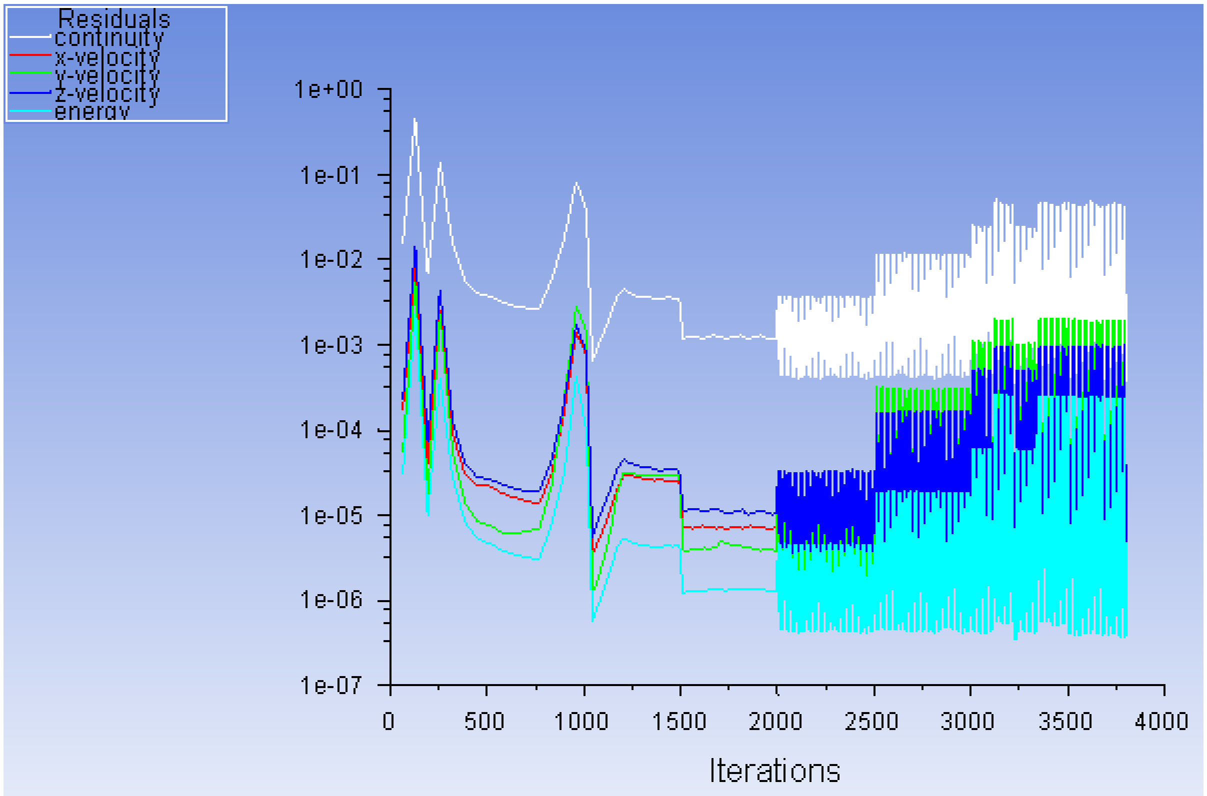
c. Retain the default setting of **20** for **Max Iterations/Time Step**.

d. Click **Calculate**.

The calculation will run for approximately 4,200 iterations.

The residuals jump at the beginning of each time step and then fall at least two to three orders of magnitude. Also, the relative convergence criteria is achieved before reaching the maximum iteration limit (20) for each time step, indicating the limit does not need to be increased.

Figure 11.3: Residual History for the First Revolution of the Rotor



10. Examine the flow variable histories for the first revolution of the rotor (Figure 11.4: Mass Flow Rate at the Inlet During the First Revolution (p. 481), Figure 11.5: Mass Flow Rate at the Outlet During the First Revolution (p. 481), and Figure 11.6: Static Pressure at the Interface During the First Revolution (p. 482)).

Figure 11.4: Mass Flow Rate at the Inlet During the First Revolution

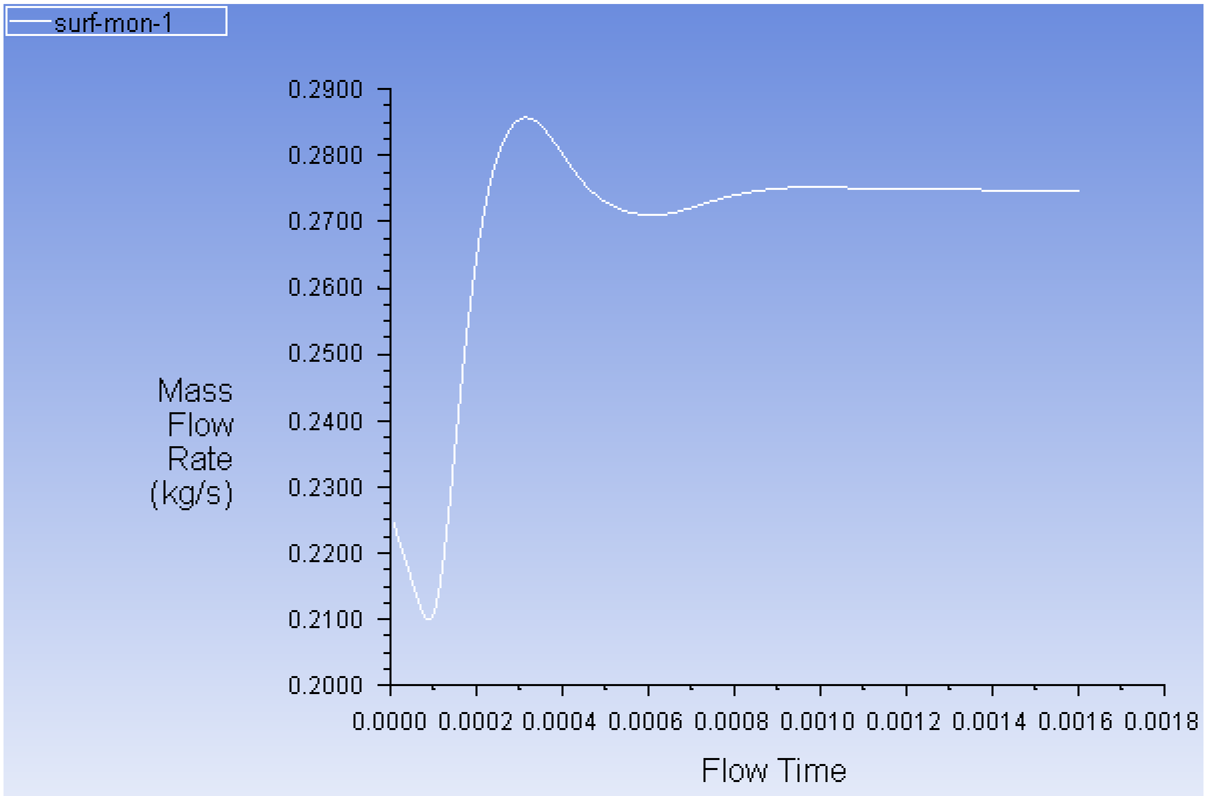


Figure 11.5: Mass Flow Rate at the Outlet During the First Revolution

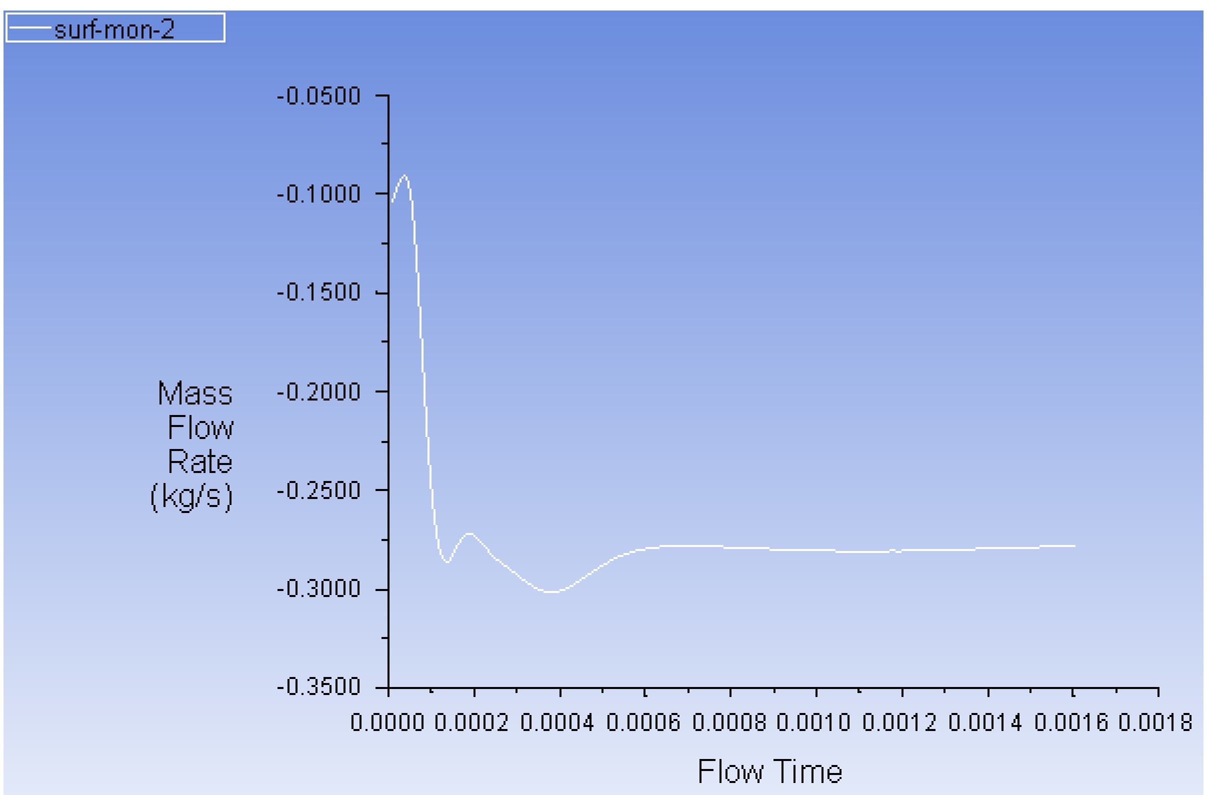
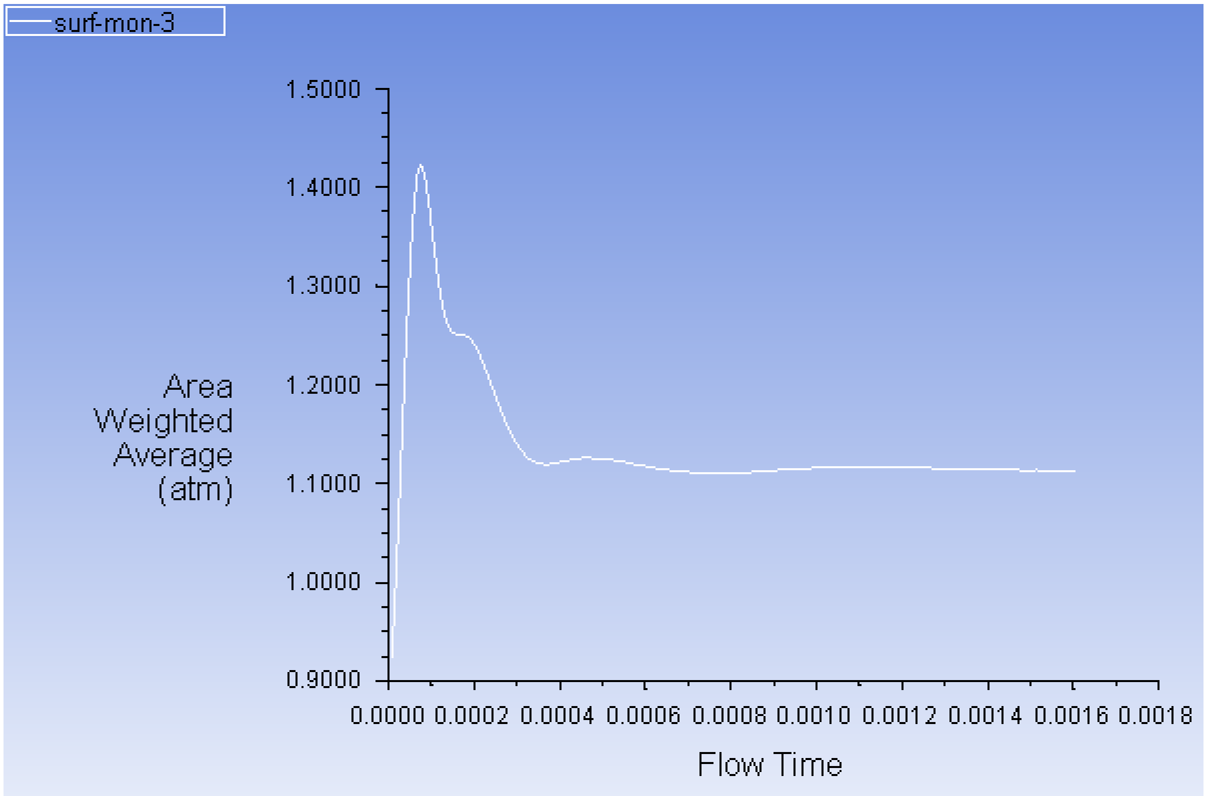


Figure 11.6: Static Pressure at the Interface During the First Revolution



The flow variable histories show that the large variations in flow rate and interface pressure that occur early in the calculation are greatly reduced as time-periodicity is approached.

11. Save the case and data files (`axial_comp-0240.cas.gz` and `axial_comp-0240.dat.gz`).

    File → **Write** → **Case & Data...**

Note

It is a good practice to save the case file whenever you are saving the data file especially for sliding mesh model. This is because the case file contains the mesh information, which is changing with time.

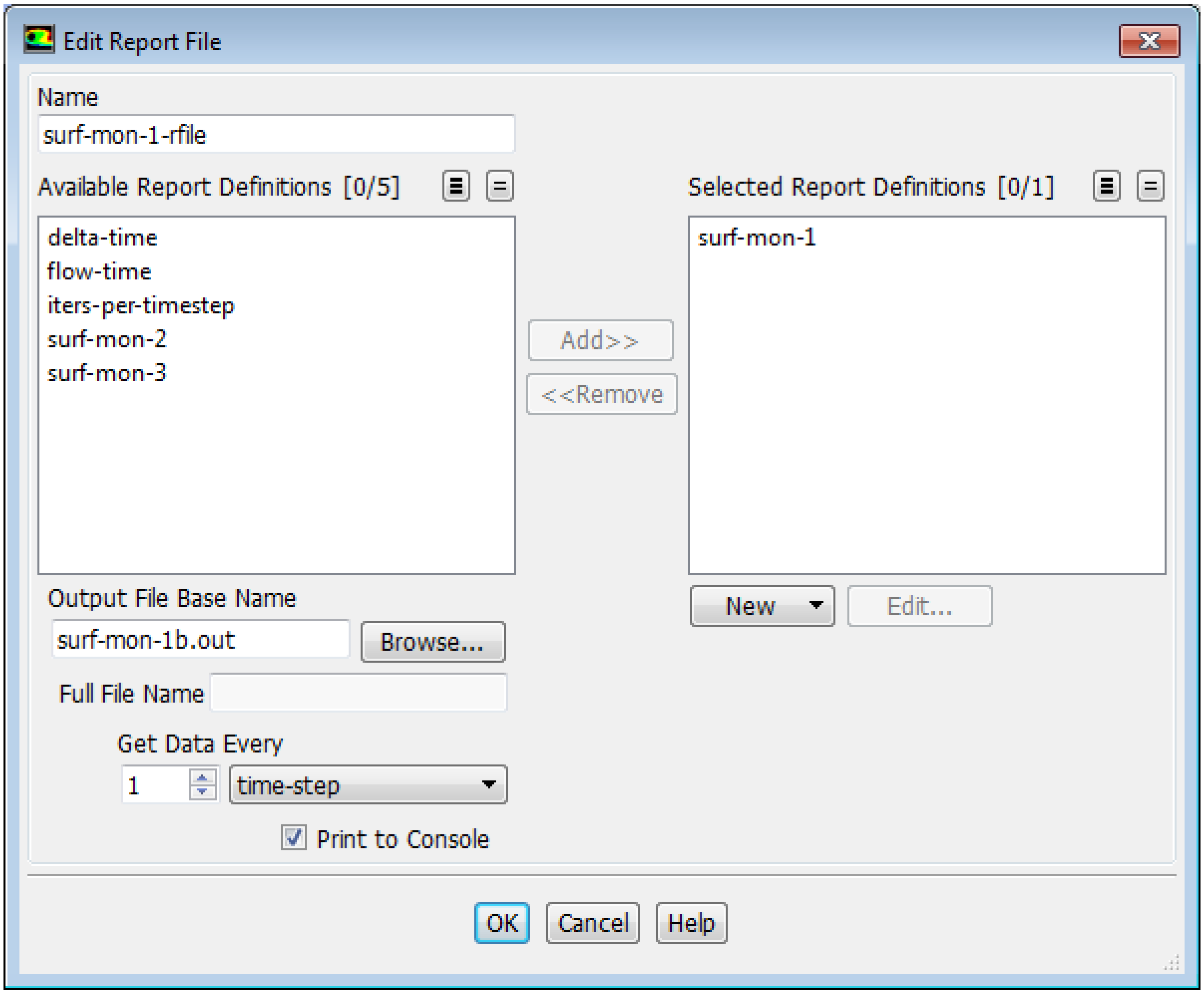
Note

For transient-state calculations, you can add the character string `%t` to the file name so that the iteration number is automatically appended to the name (for example, by entering `axial_comp-%t` for the **File Name** in the **Select File** dialog box, ANSYS Fluent will save files with the names `axial_comp-0240.cas` and `axial_comp-0240.dat`).

---

12. Rename the report output file in preparation for further iterations.

    Solution → **Monitors** → **Report Files** → **surf-mon-1-rfile** **Edit...**

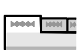


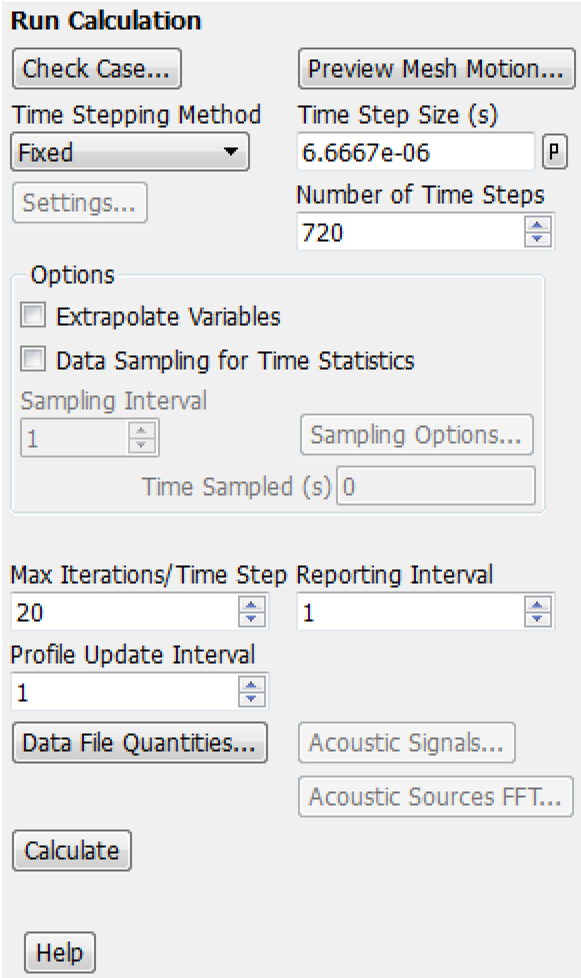
a. Enter `surf-mon-1b.out` for **Output File Base Name**.

b. Click **OK** to close the **Edit Report File** dialog box.

13. Similarly, change the output file names for the **surf-mon-2-rfile** and **surf-mon-3-rfile** report file definitions to `surf-mon-2b.out` and `surf-mon-3b.out`, respectively.

14. Continue the calculation for 720 more time steps to simulate three more revolutions of the rotor.

 **Solving** → **Run Calculation** → **Advanced...**



Note

Calculating three more revolutions will require some additional CPU time. If you choose, instead of calculating the solution, you can read a data file (`axial_comp-0960.dat.gz`) with the precalculated solution for this tutorial. This data file can be found in the `sliding_mesh` directory.

The calculation will run for approximately 11,600 more iterations.

15. Examine the flow variable histories for the next three revolutions of the rotor to verify that the solution is time-periodic (Figure 11.7: Mass Flow Rate at the Inlet During the Next 3 Revolutions (p. 485) Figure 11.8: Mass Flow Rate at the Outlet During the Next 3 Revolutions (p. 486), and Figure 11.9: Static Pressure at the Interface During the Next 3 Revolutions (p. 486)).

Note

If you read the provided data file instead of iterating the solution for three revolutions, the flow variable histories can be displayed by using the **File XY Plot** dialog box.

Results → **Plots** → **File** **Edit...**

Click the **Add** button in the **File XY Plot** dialog box to select one of the flow variable histories from the **Select File** dialog box, click **OK**, and then click **Plot**. To obtain a better view of the data, you may want to manually change the ranges of the axes.

Figure 11.7: Mass Flow Rate at the Inlet During the Next 3 Revolutions

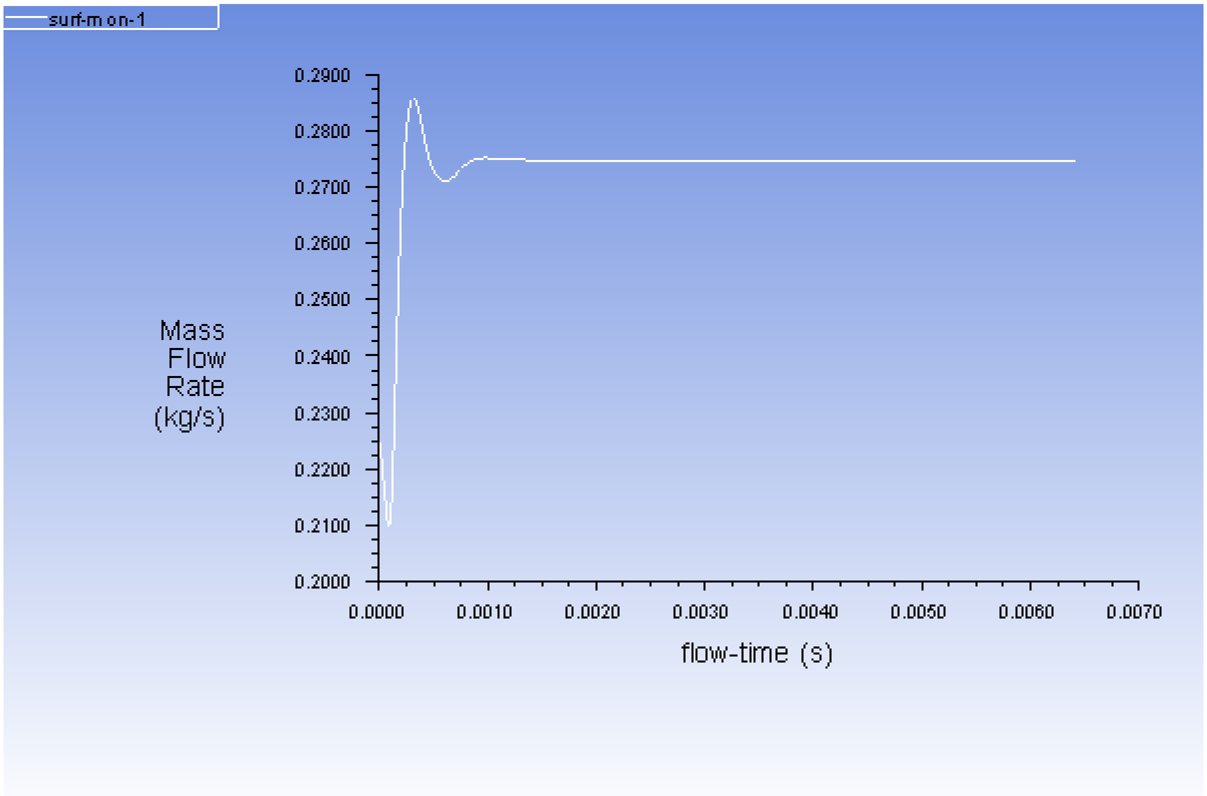


Figure 11.8: Mass Flow Rate at the Outlet During the Next 3 Revolutions

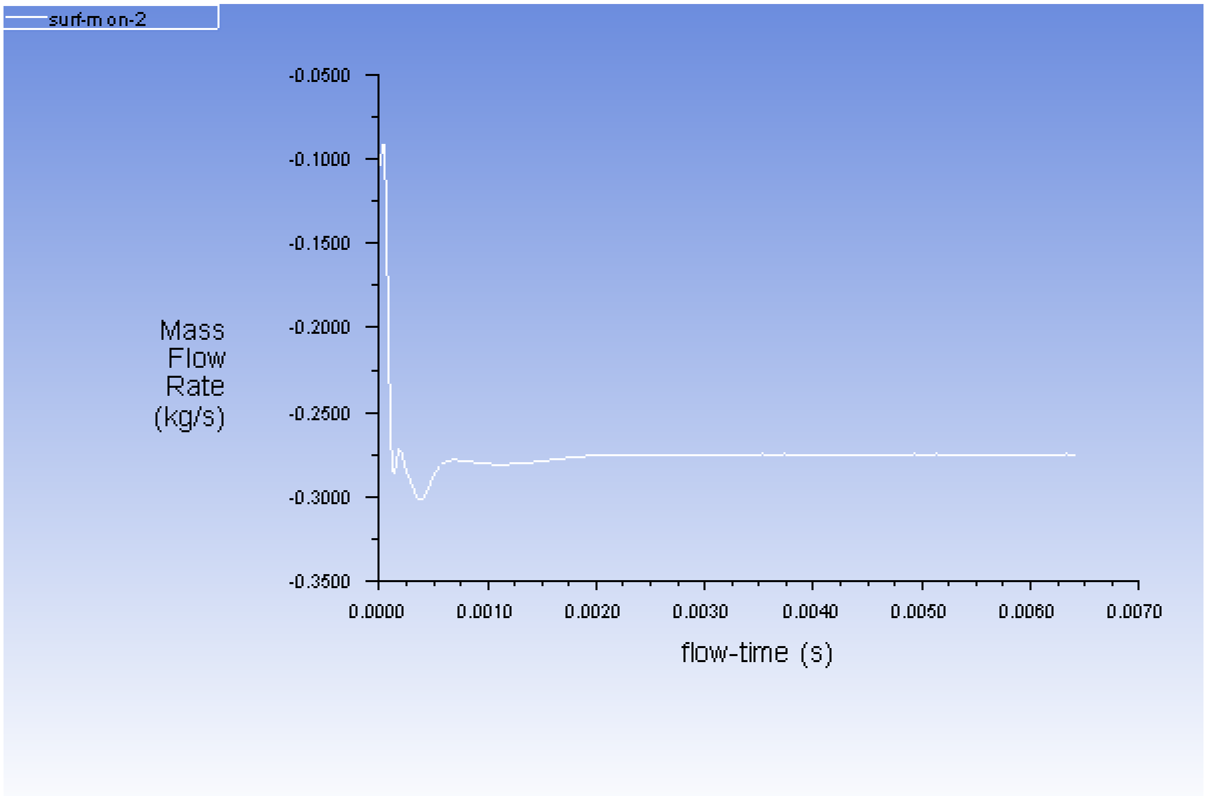
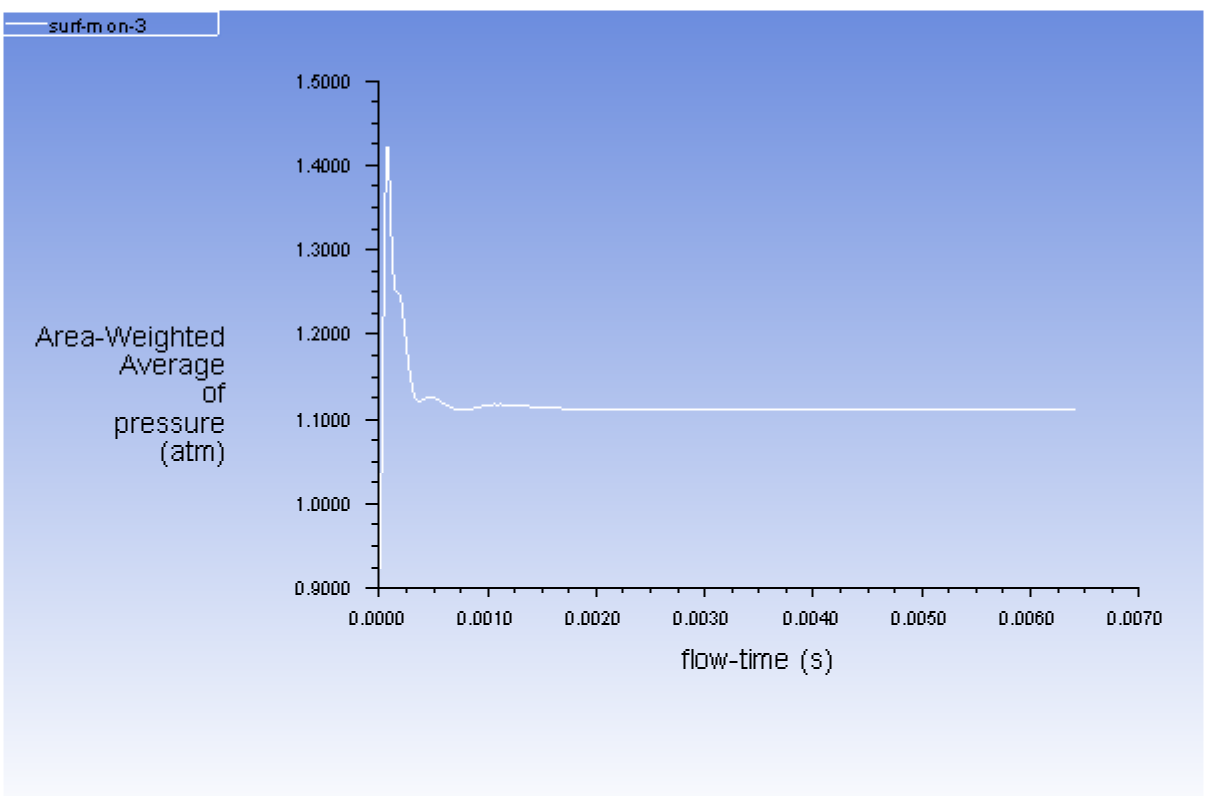


Figure 11.9: Static Pressure at the Interface During the Next 3 Revolutions



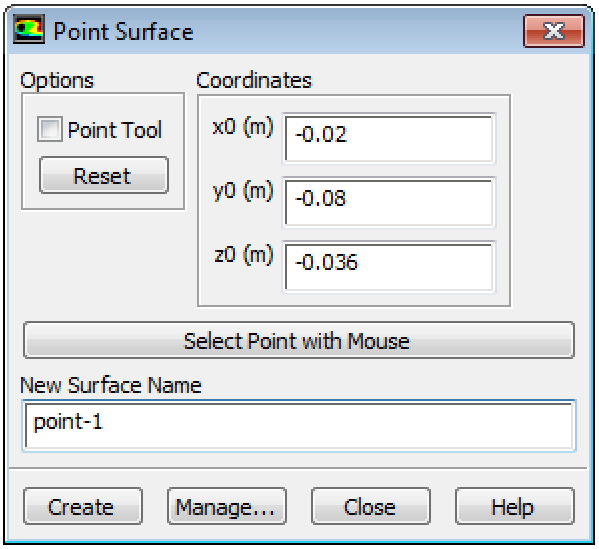
16. Save the case and data files (`axial_comp-0960.cas.gz` and `axial_comp-0960.dat.gz`).

File → Write → Case & Data...

17. Change the file names for **surf-mon-1b.out**, **surf-mon-2b.out**, and **surf-mon-3b.out** to `surf-mon-1c.out`, `surf-mon-2c.out`, and `surf-mon-3c.out`, respectively (as described in a previous step), in preparation for further iterations.

18. Add a point at the interface of the stator.

    Postprocessing → Surface → Create → Point...

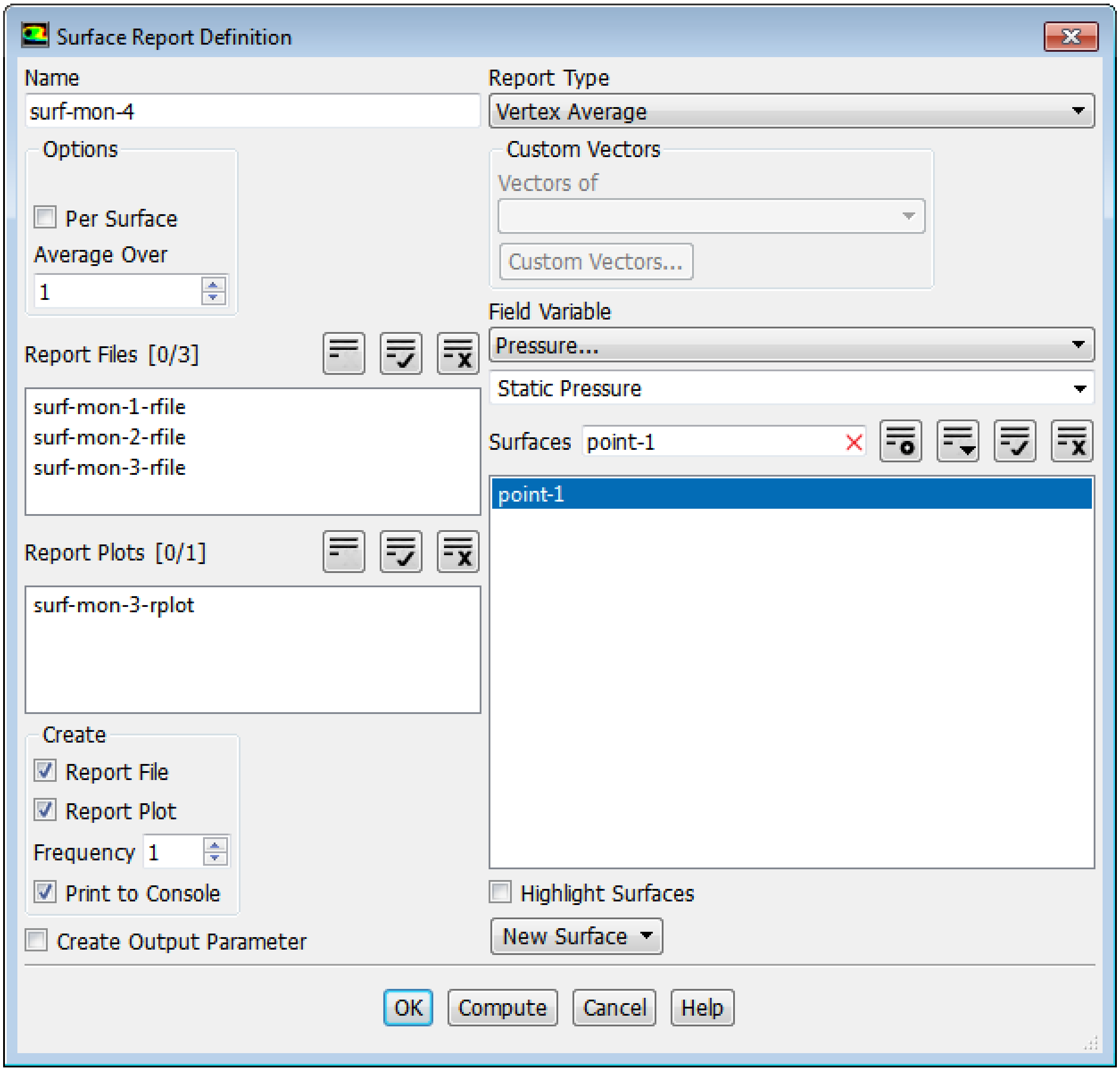
    a. Enter `-0.02` for **x0**, `-0.08` for **y0**, and `-0.036` for **z0** in the **Point Surface** dialog box.

    b. Retain the default, `point-1` for **New Surface Name**.

    c. Click **Create** and close the **Point Surface** dialog box.

19. Enable plotting of the static pressure at a point on the stator interface (**point-1**).

    Solving → Reports → Definitions → New → Surface Report → Vertex Average...



a. Enter **surf-mon-4** for the **Name** of the surface report definition.

b. In the **Create** group box, enable **Report File**, **Report Plot** and **Print to Console**.

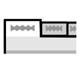
c. Retain the defaults of **Pressure** and **Static Pressure** for **Field Variable**.

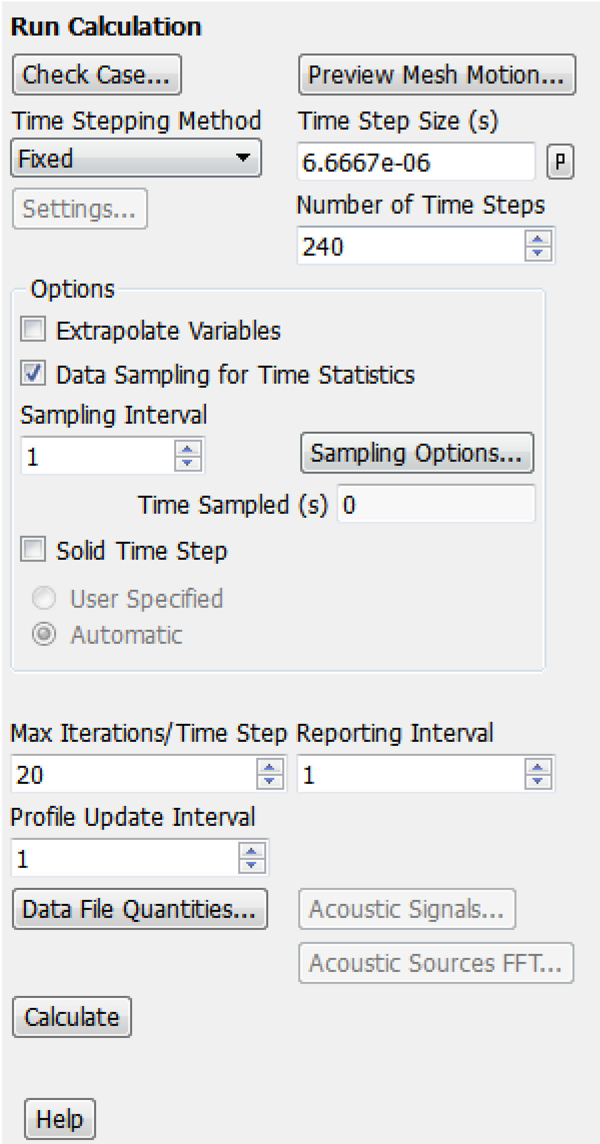
d. Enter `point-1` in the **Surfaces** field to filter the list.

e. Select **point-1** from the **Surfaces** selection list.

f. Click **OK** to save the surface report definition settings and close the **Surface Report Definition** dialog box.

20. Continue the calculation for one final revolution of the rotor, while saving data samples for the postprocessing of the time statistics.

 **Solving → Run Calculation → Advanced...**



a. Enter `240` for **Number of Time Steps**.

b. Enable **Data Sampling for Time Statistics** in the **Options** group box.

   *Enabling **Data Sampling for Time Statistics** causes ANSYS Fluent to calculate and store mean and root-mean-square (RMS) values of various quantities and field functions over the calculation interval.*

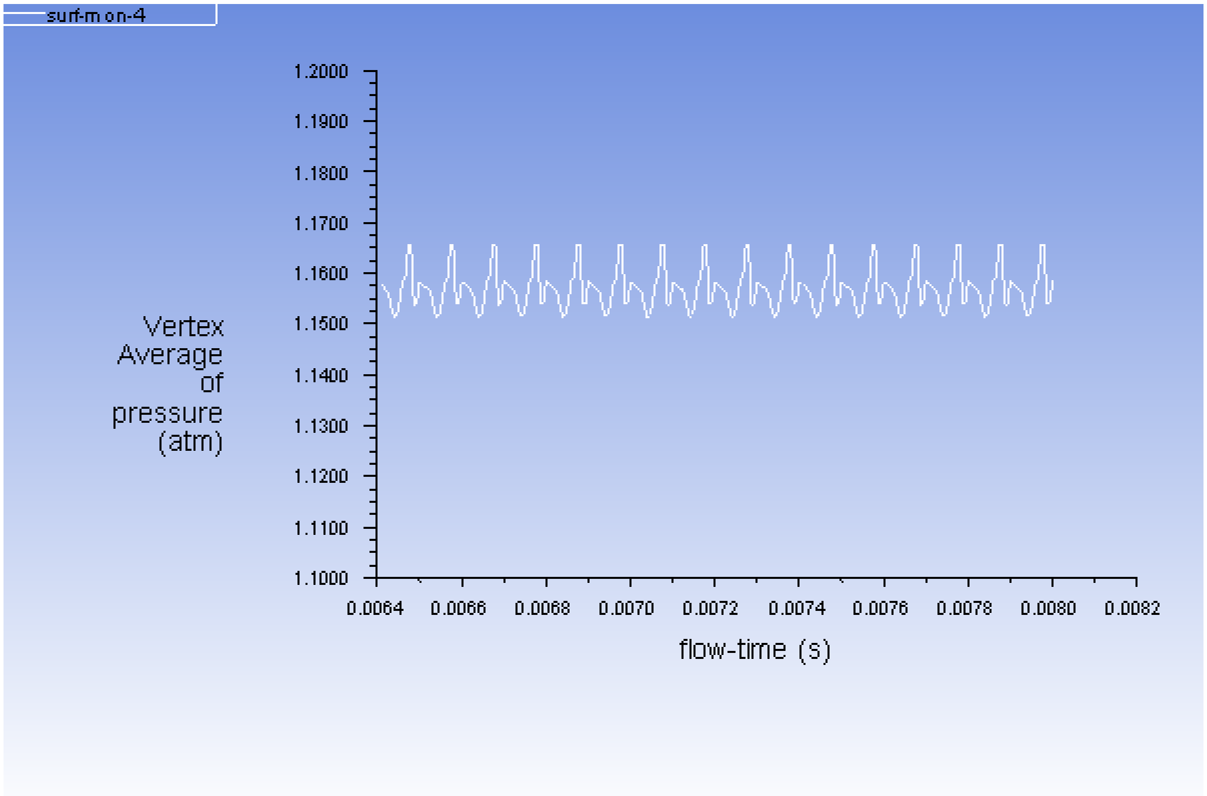
c. Click **Calculate**.

The calculation will run for approximately 3,800 more iterations.

21. Save the case and data files (`axial_comp-1200.cas.gz` and `axial_comp-1200.dat.gz`).

    File → **Write** → **Case & Data...**

Figure 11.10: Static Pressure at a Point on The Stator Interface During the Final Revolution

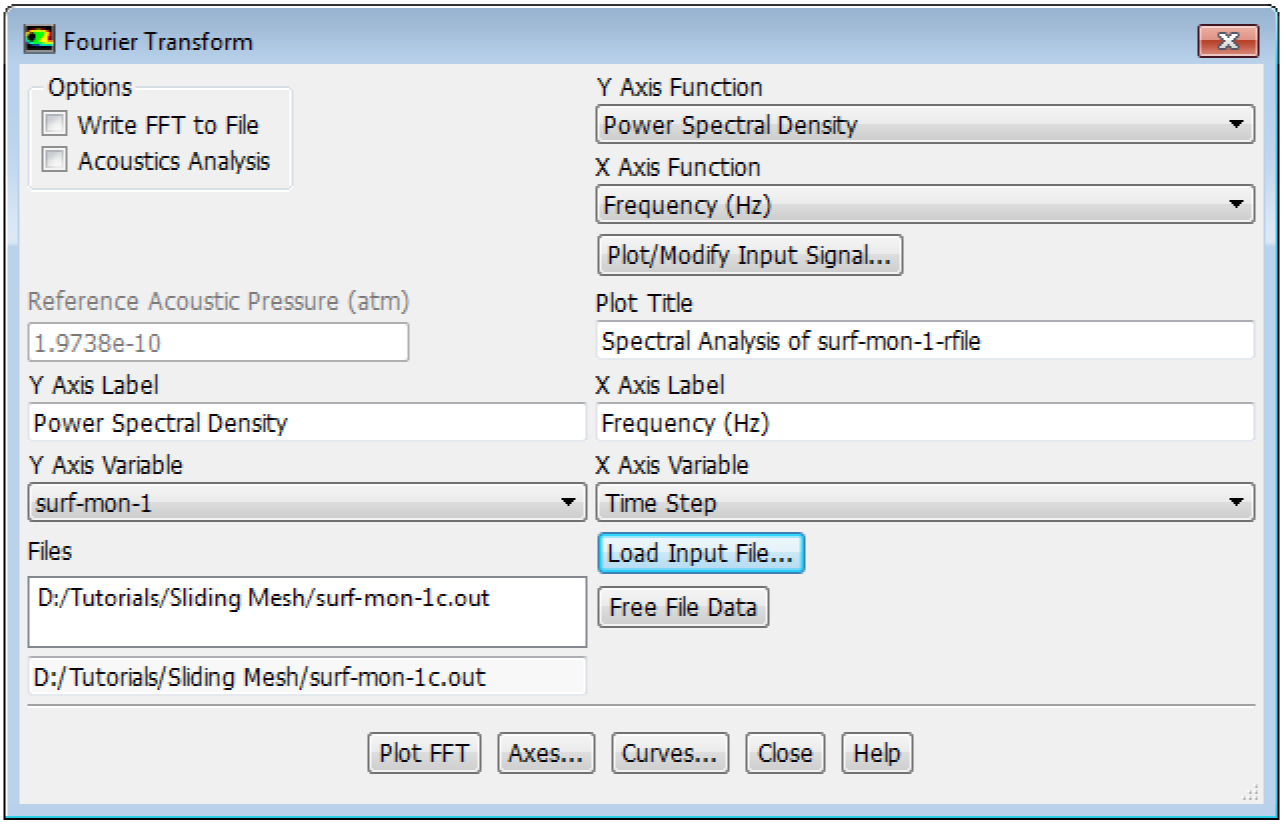


11.4.11. Postprocessing

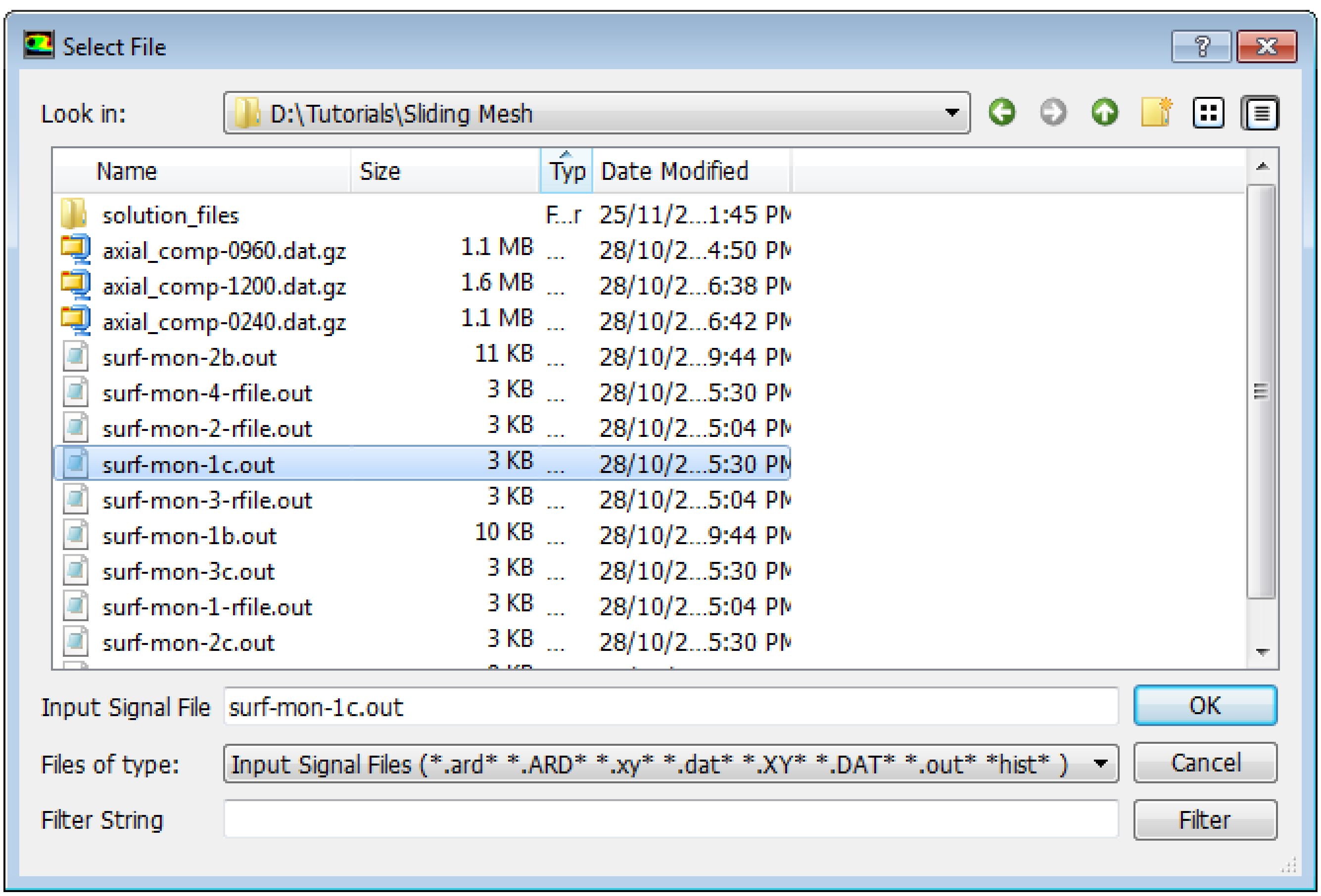
In the next three steps you will examine the time-averaged values for the mass flow rates at the inlet and the outlet during the final revolution of the rotor. By comparing these values, you will verify the conservation of mass on a time-averaged basis for the system over the course of one revolution.

1. Examine the time-averaged mass flow rate at the inlet during the final revolution of the rotor (as calculated from `surf-mon-1c.out`).

   Postprocessing → **Plots** → **FFT...**



a. Click the **Load Input File...** button to open the **Select File** dialog box.

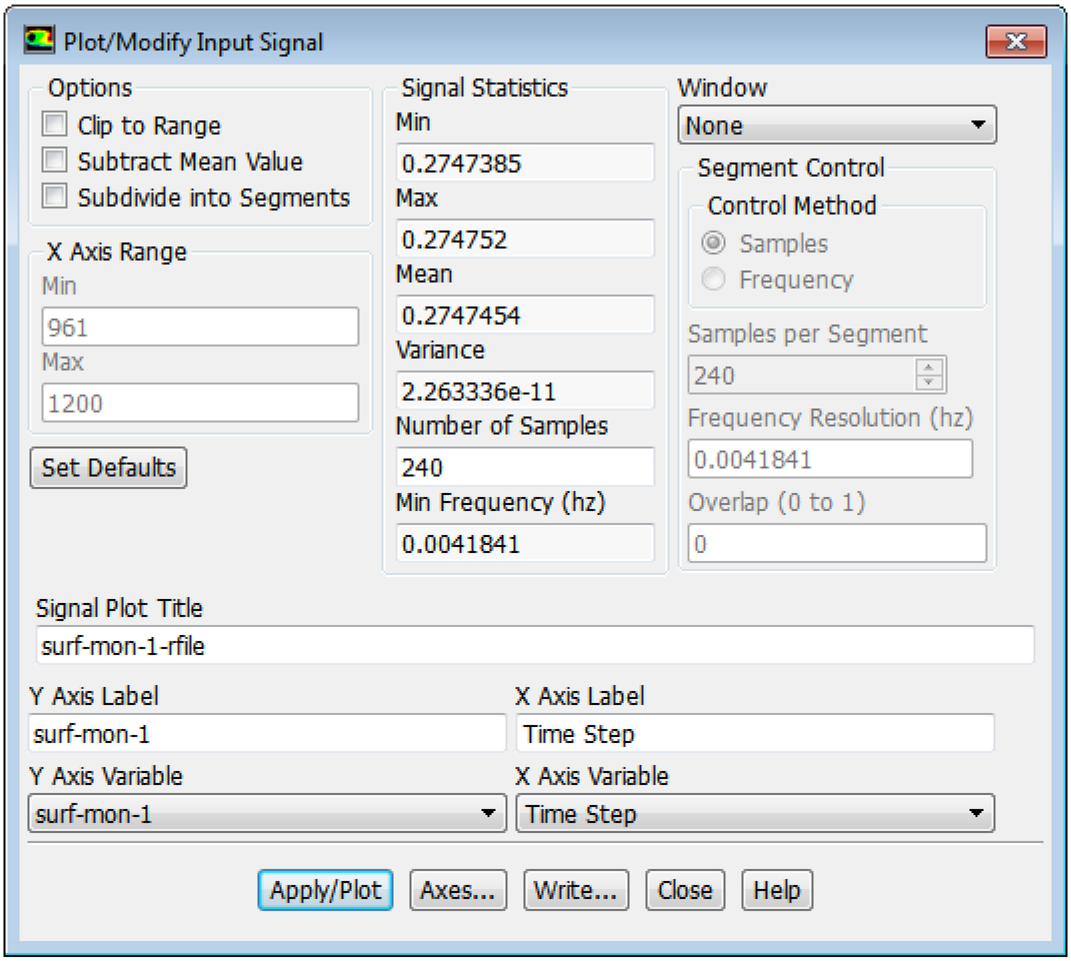


i. Select **All Files** from the **files of type:** drop-down list.

ii. Select `surf-mon-1c.out` from the list of files.

iii. Click **OK** to close the **Select File** dialog box.

b. Click the **Plot/Modify Input Signal...** button to open the **Plot/Modify Input Signal** dialog box.



i. Examine the values for **Min**, **Max**, **Mean**, and **Variance** in the **Signal Statistics** group box.

ii. Close the **Plot/Modify Input Signal** dialog box.

c. Select the directory path ending in **surf-mon-1c.out** from the **Files** selection list.

d. Click the **Free File Data** button.

2. Examine the time-averaged mass flow rate at the outlet during the final revolution of the rotor (as calculated from `surf-mon-2c.out`), and plot the data.

Postprocessing → **Plots** → **FFT...**

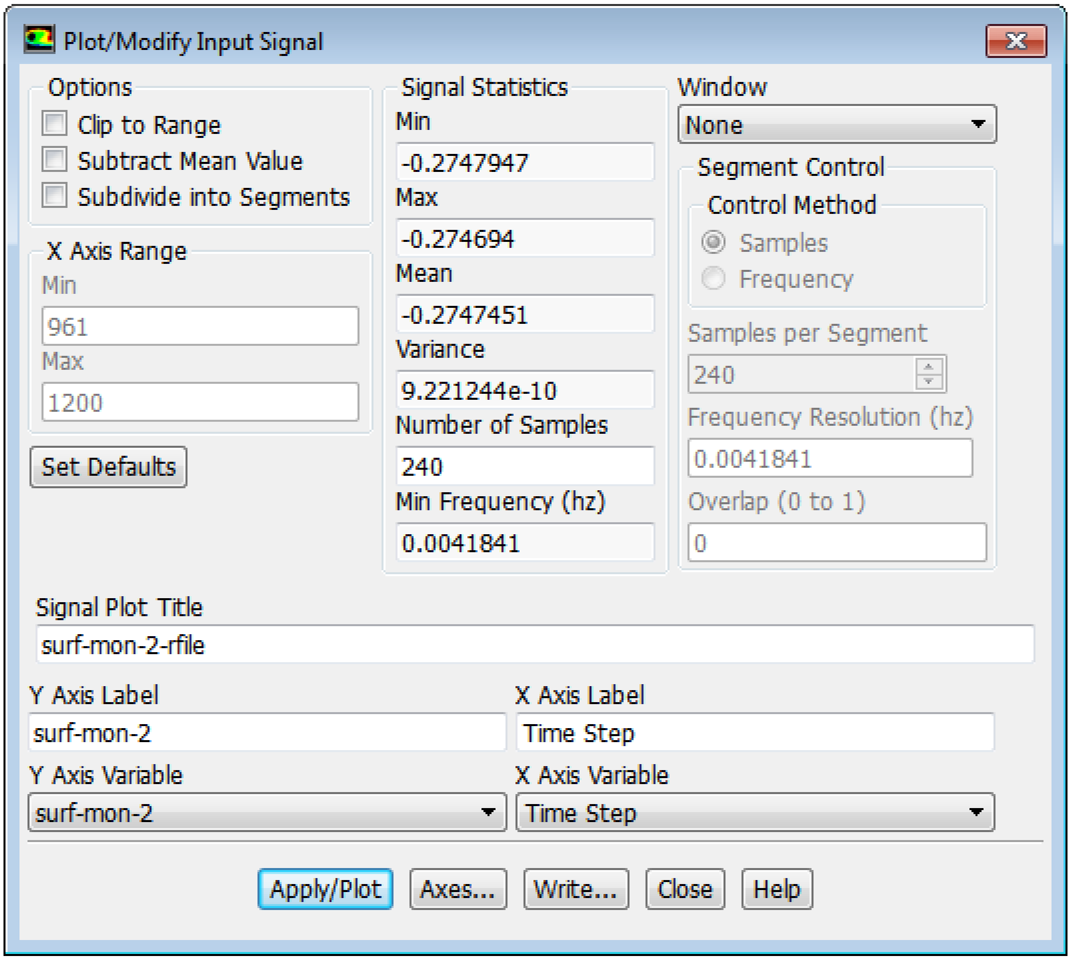
a. Click the **Load Input File...** button to open the **Select File** dialog box.

i. Select **All Files** from the **files of type:** drop-down list.

ii. Select `surf-mon-2c.out` from the list of files.

iii. Click **OK** to close the **Select File** dialog box.

b. Click the **Plot/Modify Input Signal...** button to open the **Plot/Modify Input Signal** dialog box.

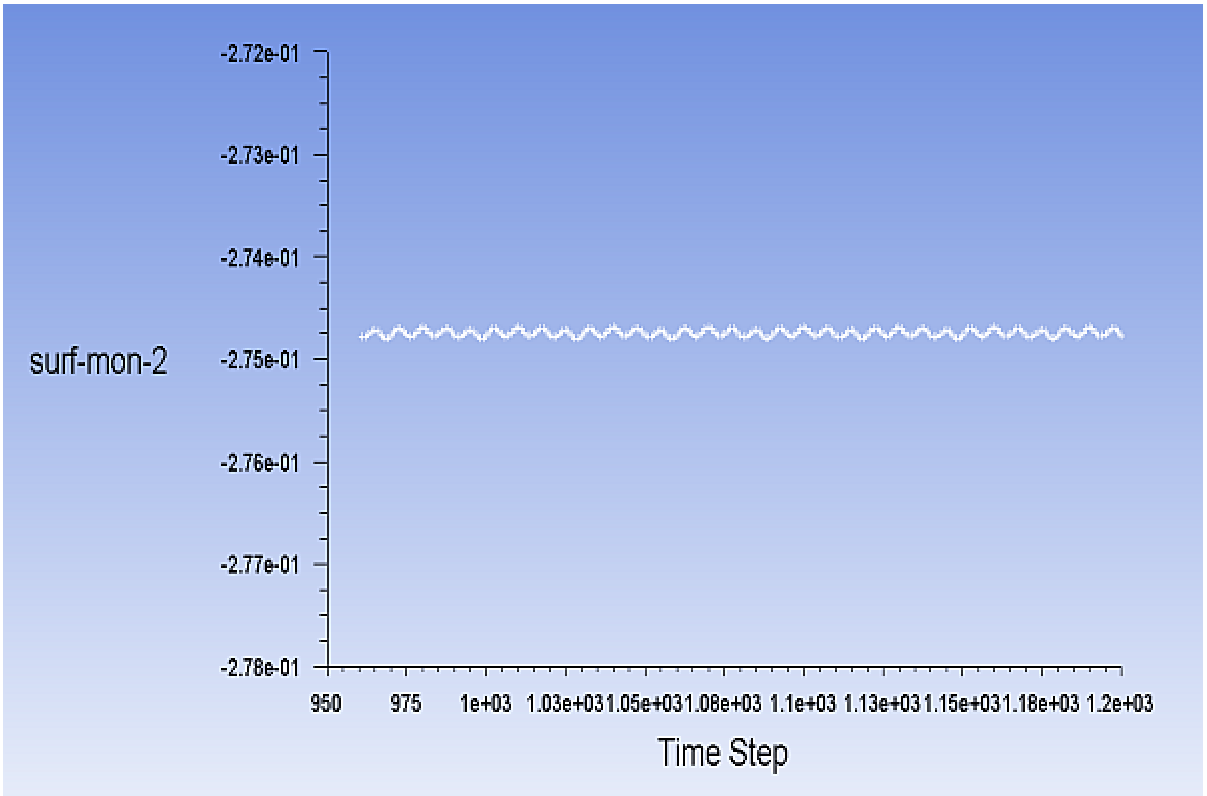


i. Examine the values for **Min**, **Max**, and **Variance** in the **Signal Statistics** group box.

ii. Click **Set Defaults**.

iii. Click **Apply/Plot** to display the area-weighted average of mass flow rate at the outlet (Figure 11.11: Area-Weighted Average Mass Flow Rate at the Outlet During the Final Revolution (p. 494)).

Figure 11.11: Area-Weighted Average Mass Flow Rate at the Outlet During the Final Revolution



iv. Close the **Plot/Modify Input Signal** dialog box.

3. Examine the vertex-averaged static pressure at the stator during the final revolution of the rotor (as calculated from `surf-mon-4.out`), and plot the data.

   Postprocessing → **Plots** → **FFT...**

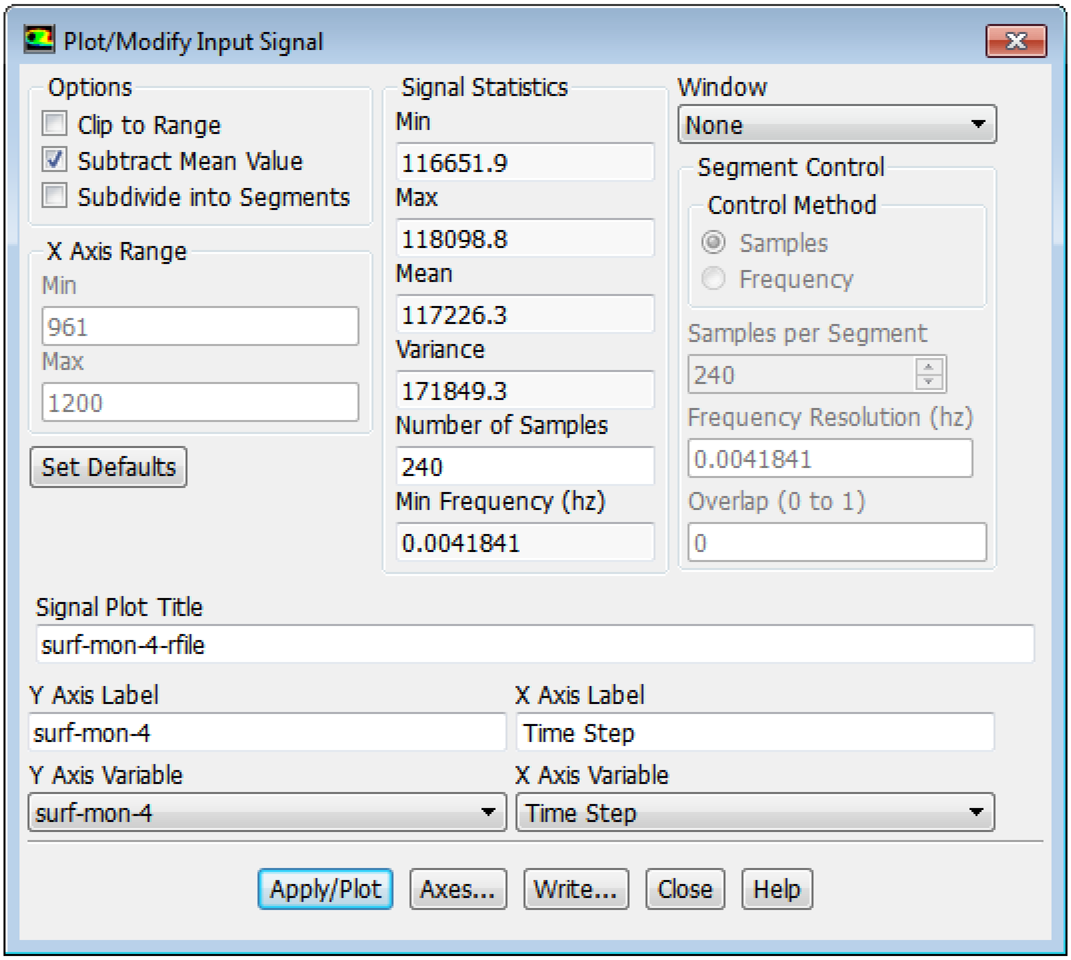
   a. Click the **Load Input File...** button to open the **Select File** dialog box.

      i. Select **All Files** from the **Files of type:** drop-down list.

      ii. Select `surf-mon-4-rfile.out` from the list of files.

      iii. Click **OK** to close the **Select File** dialog box.

   b. Click the **Plot/Modify Input Signal...** button to open the **Plot/Modify Input Signal** dialog box.



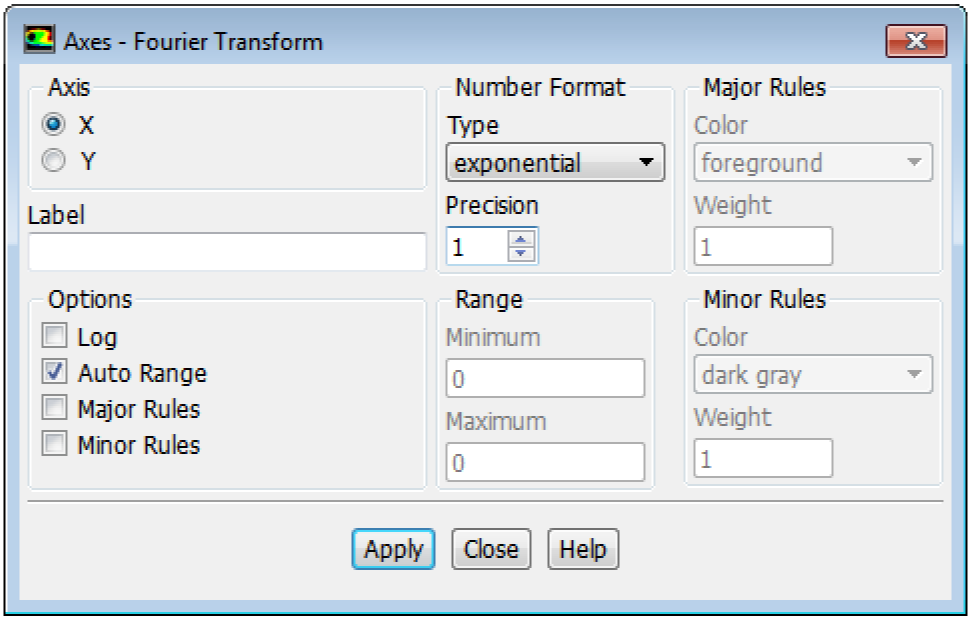
i. Enable **Subtract Mean Value** in the **Options** group box.

ii. Click **Apply/Plot**.

iii. Close the **Plot/Modify Input Signal** dialog box.

c. Click **Plot FFT** in the **Fourier Transform** dialog box.

d. Click **Axes...** to open the **Axes - Fourier Transform** dialog box.

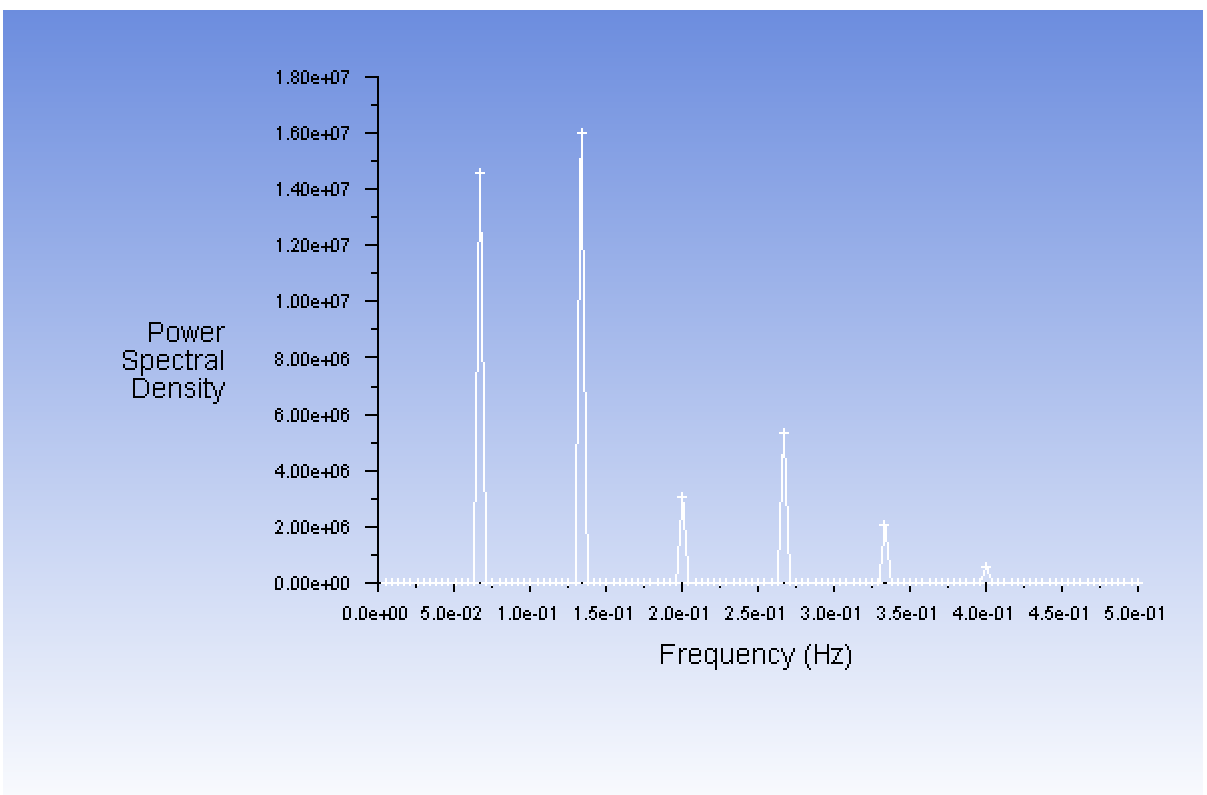


e. Select **exponential** from the **Type** drop-down list, and set **Precision** to 1 in the **Number Format** group box.

f. Click **Apply** and close the **Axes - Fourier Transform** dialog box.

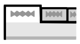
g. Click **Plot FFT** and close the **Fourier Transform** dialog box.

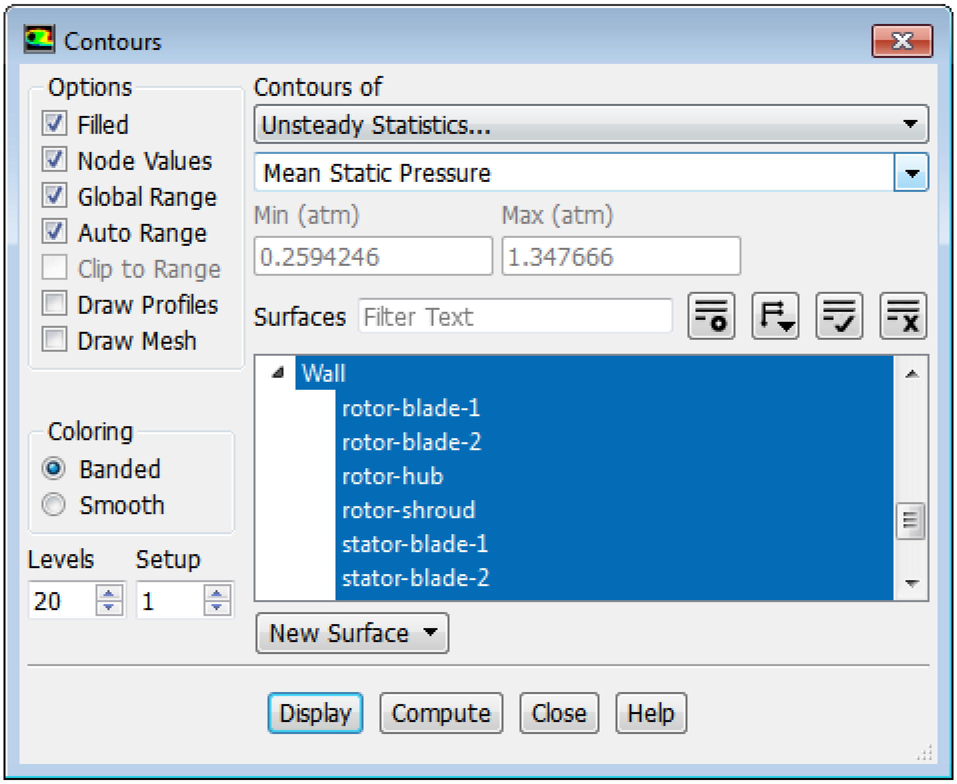
Figure 11.12: FFT of Static Pressure at the Stator



The FFT plot clearly shows that the pressure fluctuations due to interaction at the interface are dominated by the rotor and stator blade passing frequencies (which are 10 kHz and 20 kHz, respectively) and their higher harmonics.

4. Display contours of the mean static pressure on the walls of the axial compressor.

 **Postprocessing** → **Graphics** → **Contours** → **Edit**



a. Enable **Filled** in the **Options** group box.

b. Select **Unsteady Statistics...** and **Mean Static Pressure** from the **Contours of** drop-down lists.

c. Select **wall** from the **Surface Types** selection list.

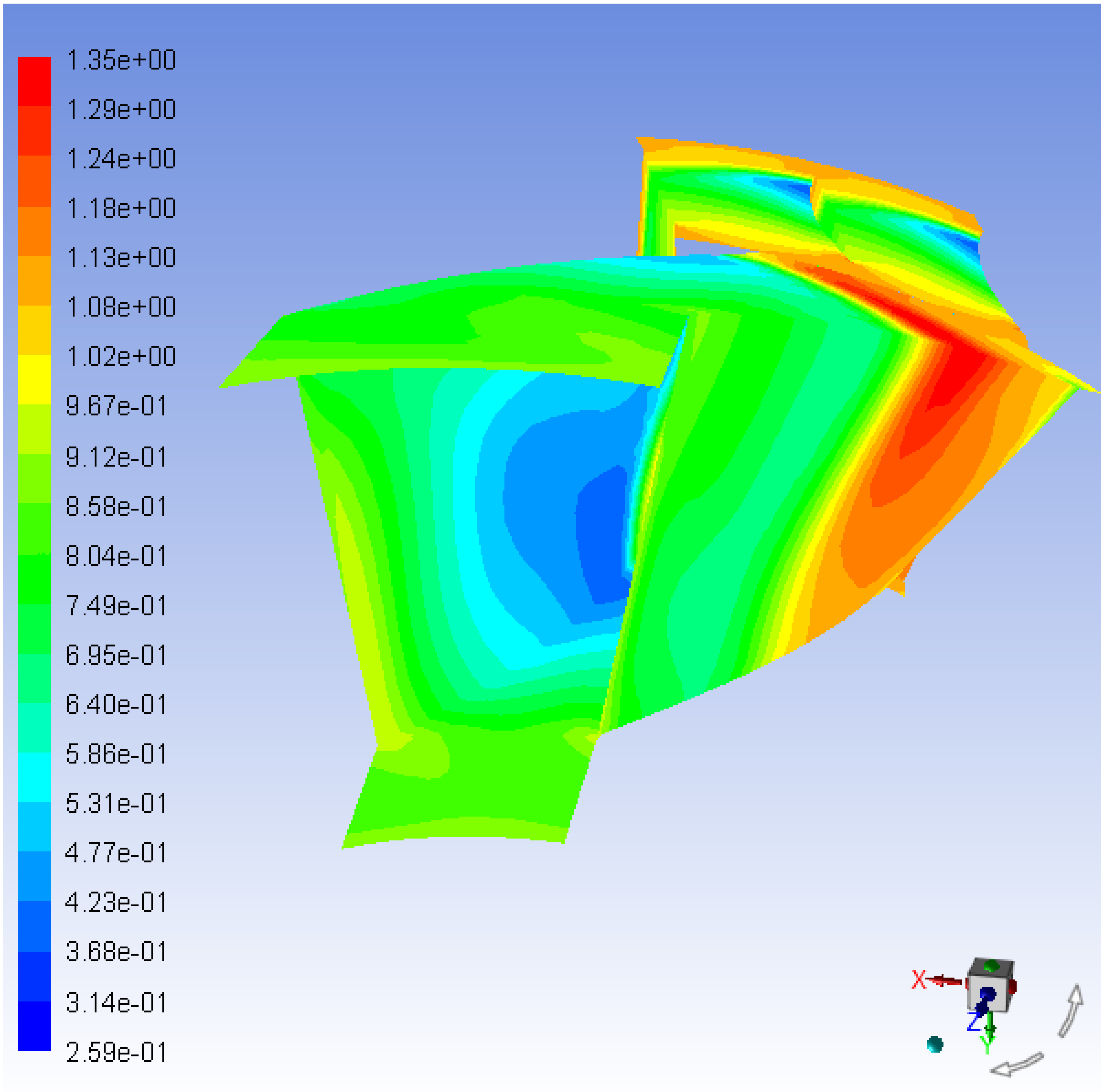
   *Scroll down the **Surfaces** selection list to find **wall**.*

d. Click **Display** and close the **Contours** dialog box.

e. Rotate the view to get the display as shown in Figure 11.13: Mean Static Pressure on the Outer Shroud of the Axial Compressor (p. 498).

Shock waves are clearly visible in the flow near the outlets of the rotor and stator, as seen in the areas of rapid pressure change on the outer shroud of the axial compressor.

Figure 11.13: Mean Static Pressure on the Outer Shroud of the Axial Compressor



11.5. Summary

This tutorial has demonstrated the use of the sliding mesh model for analyzing transient rotor-stator interaction in an axial compressor stage. The model utilized the coupled pressure-based solver in conjunction with the transient algorithm to compute the inviscid flow through the compressor stage. The solution was calculated over time until the reported variables displayed time-periodicity (which required several revolutions of the rotor), after which time-averaged data was collected while running the case for the equivalent of one additional rotor revolution (240 time steps).

The Fast Fourier Transform (FFT) utility in ANSYS Fluent was employed to determine the time averages from stored flow variable report data. You also used the FFT utility to examine the frequency content of the transient report data. The observed peak corresponds to the passing frequency and the higher harmonics of the passing frequency, which occurred at approximately 10,000 Hz.

11.6. Further Improvements

This tutorial guides you through the steps to reach a second-order solution. You may be able to obtain a more accurate solution by adapting the mesh. Adapting the mesh can also ensure that your solution is independent of the mesh. These steps are demonstrated in Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121).

Chapter 12: Using Dynamic Meshes

This tutorial is divided into the following sections:

12.1. Introduction

In ANSYS Fluent, the dynamic mesh capability is used to simulate problems with boundary motion, such as check valves and store separations. The building blocks for dynamic mesh capabilities within ANSYS Fluent are three dynamic mesh schemes, namely, smoothing, layering, and remeshing. A combination of these three schemes is used to tackle the most challenging dynamic mesh problems. However, for simple dynamic mesh problems involving linear boundary motion, the layering scheme is often sufficient. For example, flow around a check valve can be simulated using only the layering scheme, as is demonstrated in this tutorial.

Check valves are commonly used to allow unidirectional flow. For instance, they are often used to act as a pressure-relieving device by only allowing fluid to leave the domain when the pressure is higher than a certain level. In such a case, a ball is connected to a spring that acts to push the ball to the valve seat and to shut the flow. But when the pressure force on the ball is greater than the spring force, the ball will move away from the valve seat and allow fluid to leave, thus reducing the pressure upstream. Gravity could be another factor in the force balance, and can be considered in ANSYS Fluent. The deformation of the ball is typically neglected, thus allowing for a rigid body Fluid Structure Interaction (FSI) calculation, which can be handled by Fluent's six degrees of freedom (six DOF) solver.

This tutorial provides information for performing basic dynamic mesh calculations by demonstrating how to do the following:

- Use the dynamic mesh capability of ANSYS Fluent to solve a simple flow-driven rigid-body motion problem.
- Set boundary conditions for internal flow.
- Set up the rigid-body motion of the ball, by specifying a set of six DOF properties that define the translation, spring loading, and range.
- Calculate a solution using the pressure-based solver.

12.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)

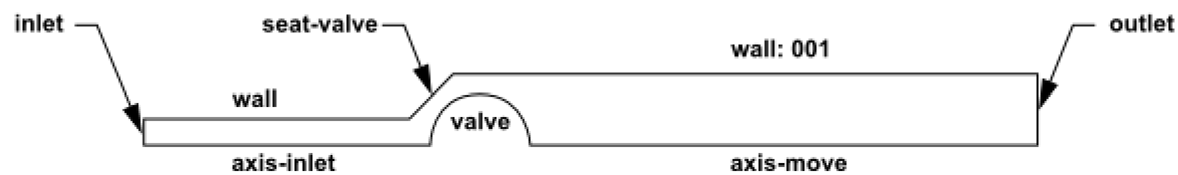
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

and that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

12.3. Problem Description

The check valve problem to be considered is shown schematically in Figure 12.1: Problem Specification (p. 502). A 2D axisymmetric valve geometry is used, consisting of a mass-flow inlet on the left, and a pressure outlet on the right, affecting the motion of a valve ball. In this case, the transient motion of the valve ball due to spring force and hydrodynamic force is studied.

Figure 12.1: Problem Specification



12.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

12.4.1. Preparation

To prepare for running this tutorial:

1. Set up a working folder on the computer you will be using.
2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

   Note

   If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

3. Enter the name of this tutorial into the search bar.

4. Narrow the results by using the filter on the left side of the page.

   a. Click **ANSYS Fluent** under **Product**.

   b. Click **18.0** under **Version**.

5. Select this tutorial from the list.

6. Click the **dynamic_mesh_R180.zip** link to download the input files.

7. Unzip `dynamic_mesh_R180.zip` to your working folder.

   The mesh file `valve.msh` *can be found in the* `dynamic_mesh` *folder created after unzipping the file.*

8. Use Fluent Launcher to start the **2D** version of ANSYS Fluent.

   Fluent Launcher displays your **Display Options** *preferences from the previous session.*

9. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.

10. Ensure that **Double Precision** is disabled if you want to match the tutorial setup exactly.

11. Select **Serial** under **Processing Options**.

    If you decide to instead run this tutorial in parallel, make sure to use **Principal Axes** *as the partitioning method.*

For more information about the Fluent Launcher, see starting ANSYS Fluent using the Fluent Launcher in the Fluent Getting Started Guide.

12.4.2. Mesh

1. Read the mesh file `valve.msh`.

   File → Read → Mesh...

12.4.3. General Settings

1. Check the mesh.

   Setting Up Domain → Mesh → Check

   Note

   You should always make sure that the cell minimum volume is not negative, since ANSYS Fluent cannot begin a calculation if this is the case.

   ---

2. Change the display units for length to millimeters in the **Set Units** dialog box.

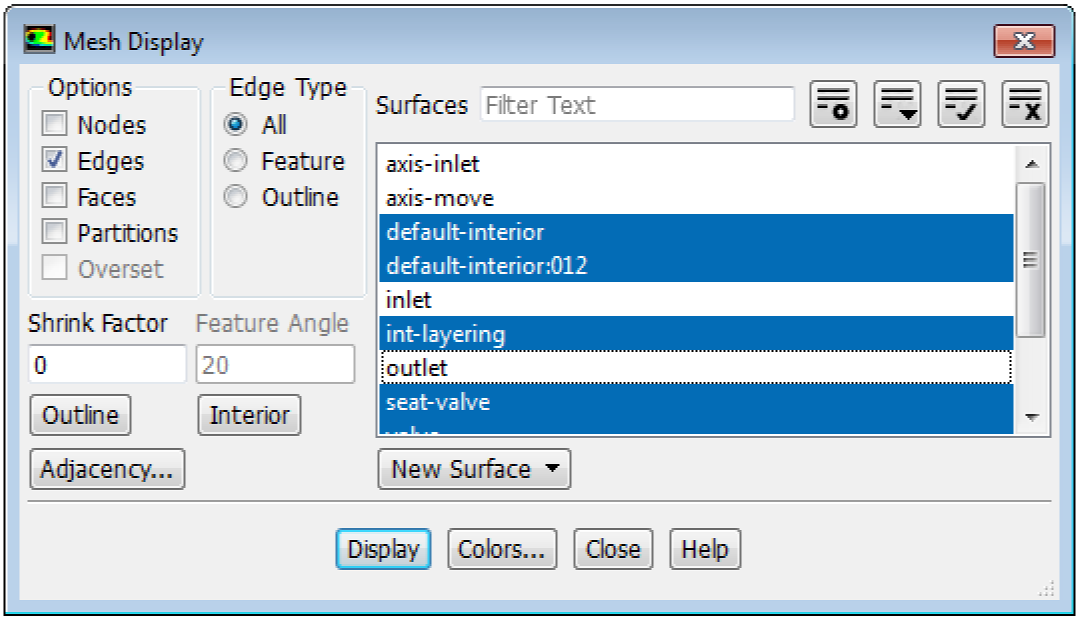
   Setting Up Domain → Mesh → Units...

a. Select **length** under **Quantities** and **mm** under **Units**.

b. Close the **Set Units** dialog box.

3. Display the mesh (Figure 12.2: Initial Mesh for the Valve (p. 505)).

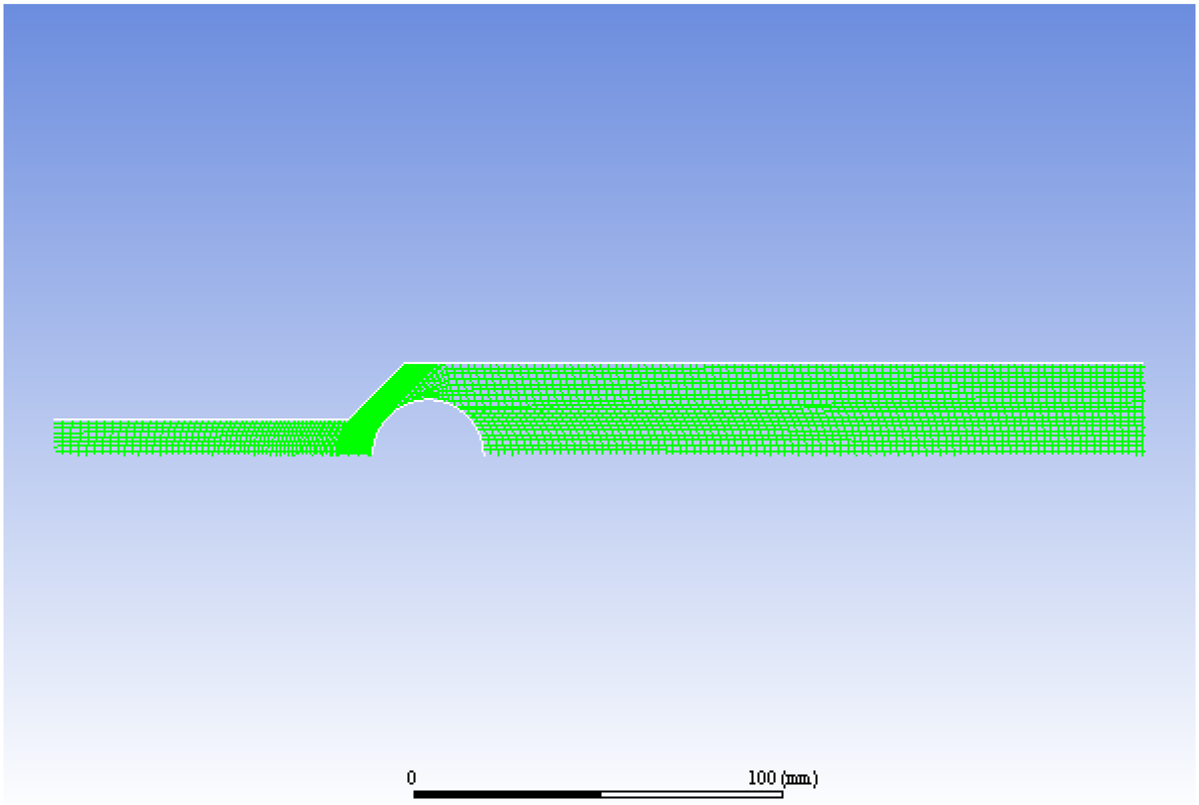
Setting Up Domain → **Mesh** → **Display...**



a. Deselect **axis-inlet**, **axis-move**, **inlet**, and **outlet** from the **Surfaces** selection list.

b. Click **Display**.

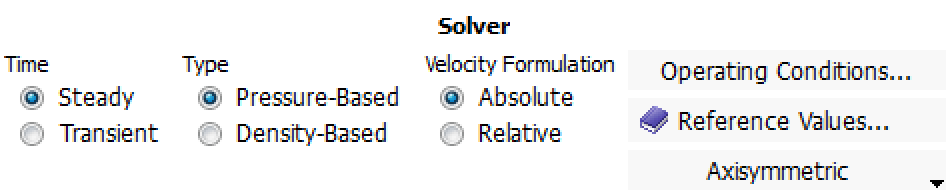
Figure 12.2: Initial Mesh for the Valve



c. Close the **Mesh Display** dialog box.

4. Enable an axisymmetric steady-state calculation.

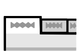
 **Setting Up Physics → Solver**

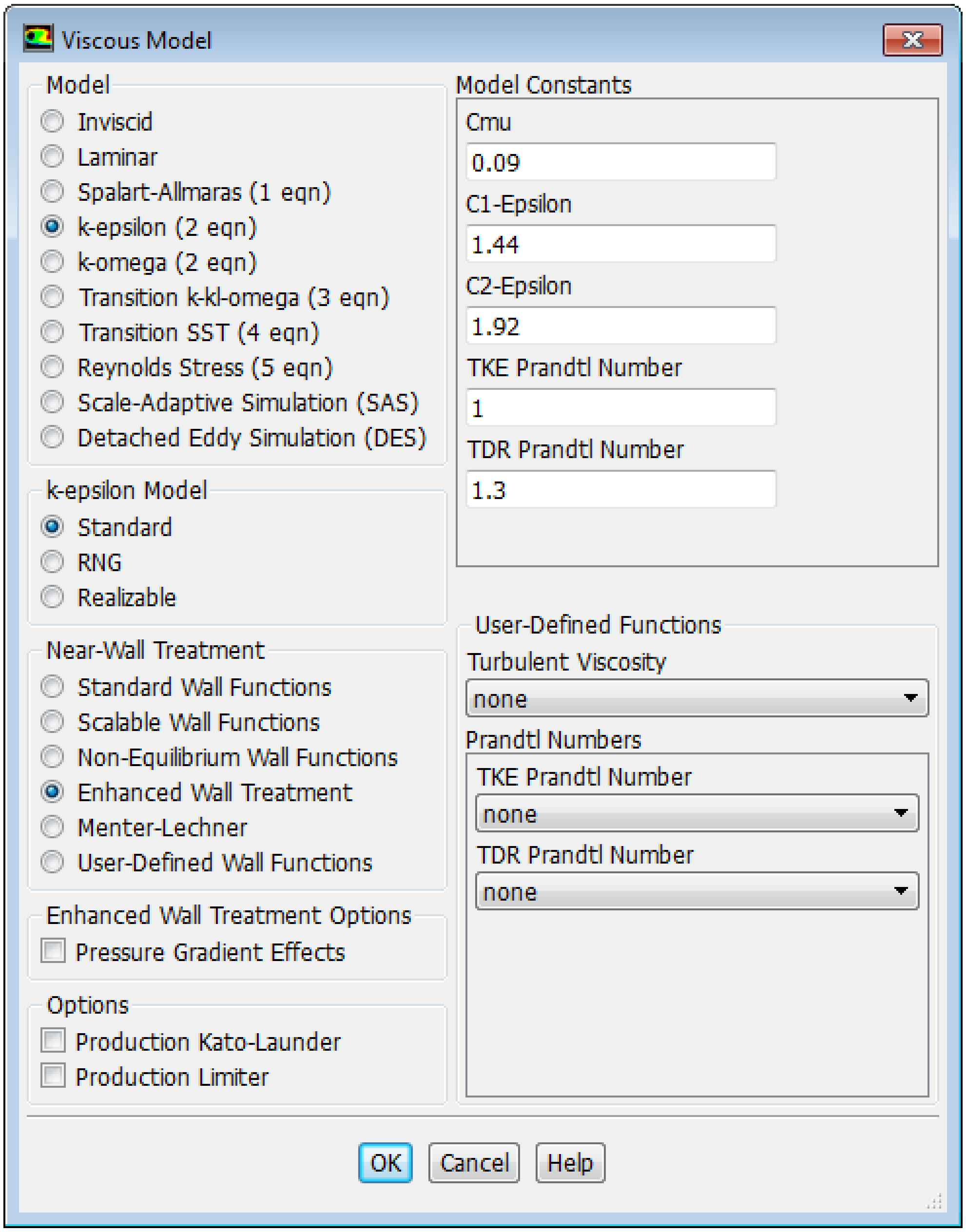


Select **Axisymmetric** from the drop-down list in the **Solver** group box (below **Reference Values...**).

12.4.4. Models

1. Enable the standard k- ε turbulence model.

 **Setting Up Physics → Models → Viscous...**



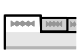
a. Select **k-epsilon (2 eqn)** from the **Model** list and retain the default selection of **Standard** in the **k-epsilon Model** group box.

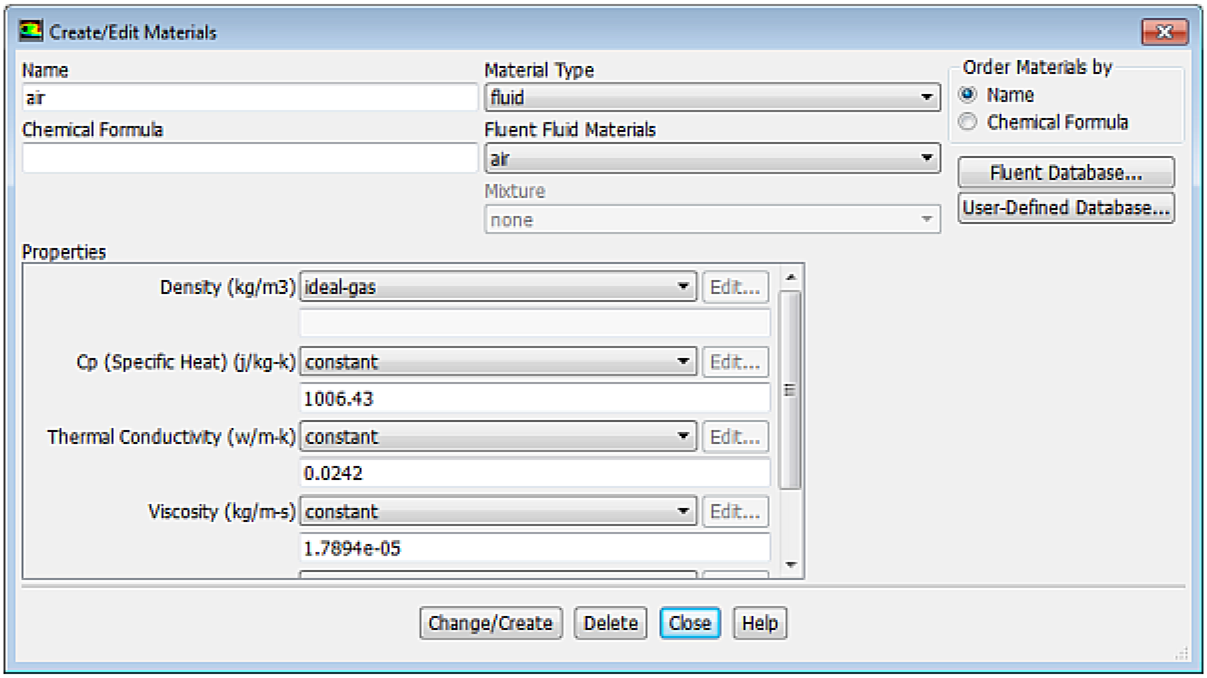
b. Select **Enhanced Wall Treatment** for the **Near-Wall Treatment**.

c. Click **OK** to close the **Viscous Model** dialog box.

12.4.5. Materials

1. Apply the ideal gas law for the incoming air stream.

 **Setting Up Physics** → **Materials** → **Create/Edit...**



a. Ensure **air** is selected from the **Fluent Fluid Materials** drop-down list.

b. Select **ideal-gas** from the **Density** drop-down list.

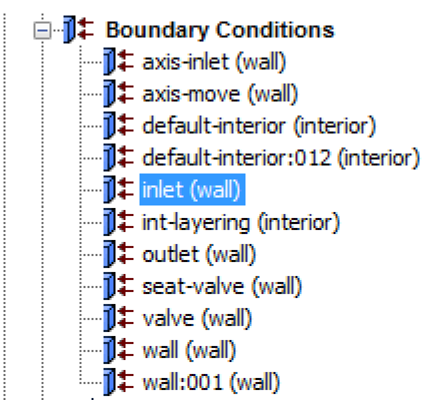
c. Click **Change/Create**.

d. Close the **Create/Edit Materials** dialog box.

Note that the energy equation is automatically enabled based on your change to the material density method.

12.4.6. Boundary Conditions

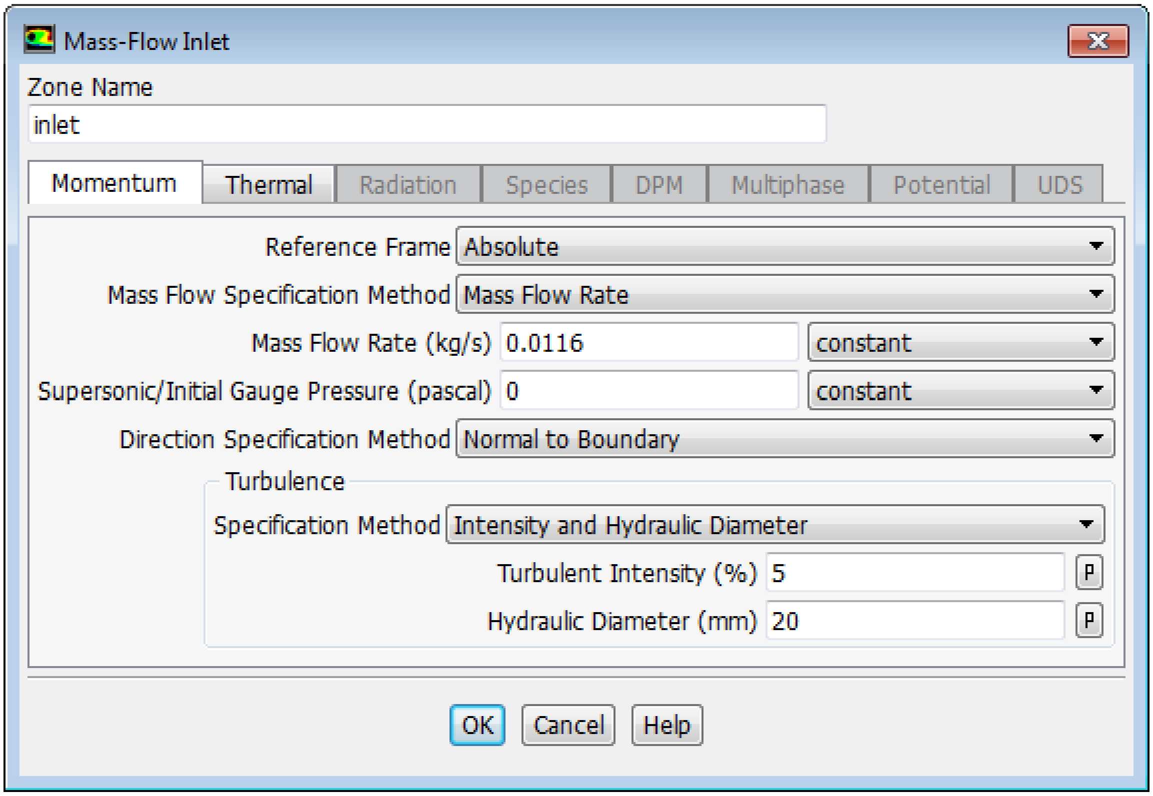
Dynamic mesh motion and all related parameters are specified using the items in the **Dynamic Mesh** *task page, not through the* **Boundary Conditions** *task page. You will set these conditions in a later step.*



1. Set the conditions for the mass-flow inlet (**inlet**).

   Since the **inlet** *boundary is assigned to a wall boundary type in the original mesh, you will need to explicitly assign the inlet boundary to a mass-flow inlet boundary type in ANSYS Fluent.*

Setup → **Boundary Conditions** → **inlet** **Type** → **mass-flow-inlet**



a. Enter `0.0116` kg/s for **Mass Flow Rate**.

b. Select **Normal to Boundary** from the **Direction Specification Method** drop-down list.

c. Select **Intensity and Hydraulic Diameter** from the **Specification Method** drop-down list in the **Turbulence** group box.

d. Retain 5% for **Turbulent Intensity**.

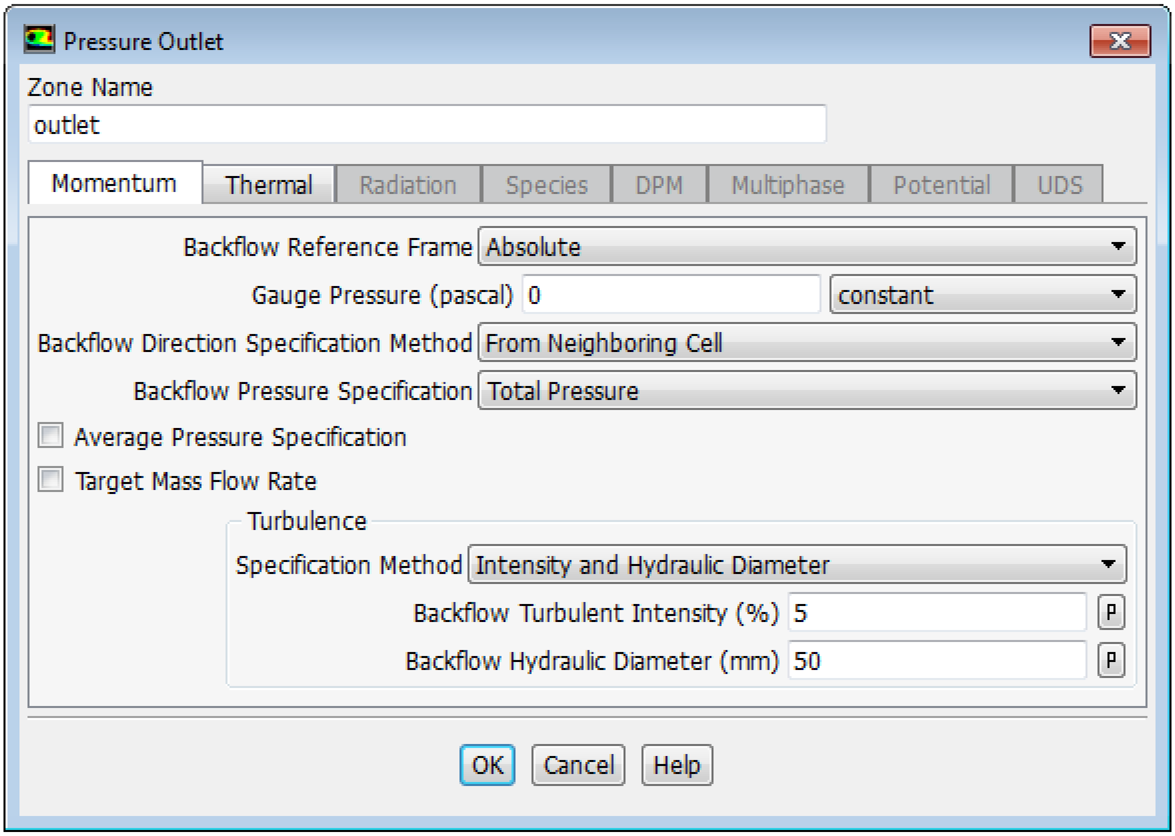
e. Enter `20` mm for the **Hydraulic Diameter**.

f. Click **OK** to close the **Mass-Flow Inlet** dialog box.

2. Set the conditions for the exit boundary (**outlet**).

Since the **outlet** *boundary is assigned to a wall boundary type in the original mesh, you will need to explicitly assign the outlet boundary to a pressure outlet boundary type in ANSYS Fluent.*

Setup → **Boundary Conditions** → **outlet** **Type** → **pressure-outlet**



a. Select **From Neighboring Cell** from the **Backflow Direction Specification Method** drop-down list.

b. Select **Intensity and Hydraulic Diameter** from the **Specification Method** drop-down list in the **Turbulence** group box.

c. Retain 5% for **Backflow Turbulent Intensity**.

d. Enter `50` mm for **Backflow Hydraulic Diameter**.

e. Click **OK** to close the **Pressure Outlet** dialog box.

3. Set the boundary type to **axis** for the **axis-inlet** boundary.

   Since the **axis-inlet** *boundary is assigned to a wall boundary type in the original mesh, you will need to explicitly assign this boundary to an axis boundary type in ANSYS Fluent.*

   Setup → **Boundary Conditions** → **axis-inlet** **Type** → **axis**

4. Set the boundary type to **axis** for the **axis-move** boundary.

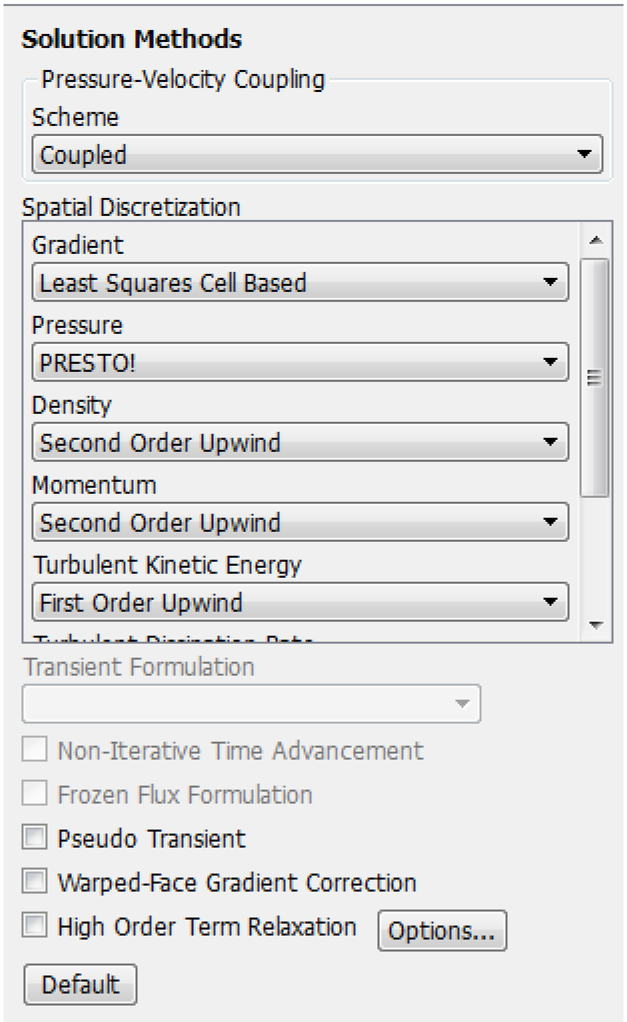
   Setup → **Boundary Conditions** → **axis-move** **Type** → **axis**

12.4.7. Solution: Steady Flow

In this step, you will generate a steady-state flow solution that will be used as an initial condition for the time-dependent solution.

1. Set the solution parameters.

   Solving → **Solution** → **Methods...**



   a. Select **Coupled** from the **Scheme** drop-down list.

   b. Select **PRESTO!** from the **Pressure** drop-down list.

   c. Retain the default of **Second Order Upwind** in the **Density** drop-down list.

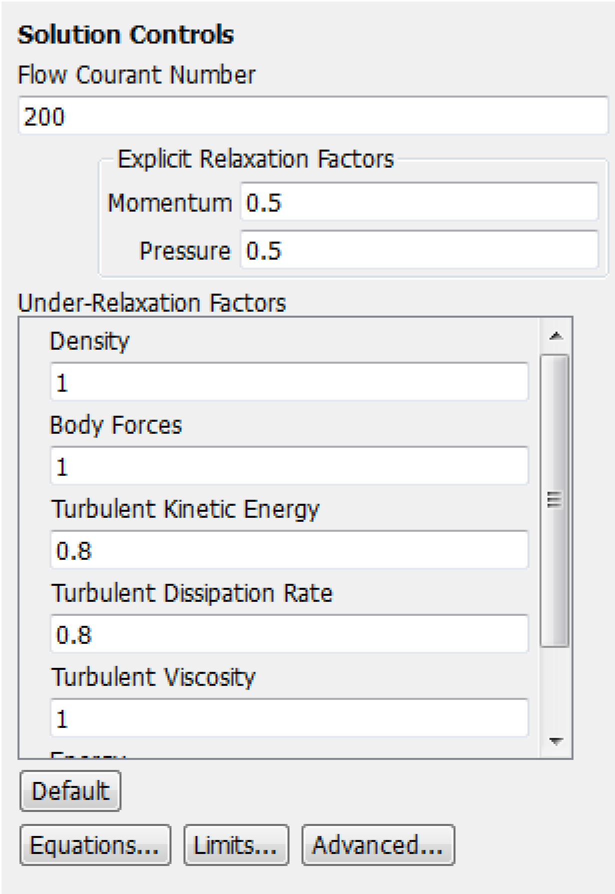
   d. Retain the default of **Second Order Upwind** in the **Momentum** drop-down list.

   e. Retain the defaults of **First Order Upwind** in the **Turbulent Kinetic Energy** and **Turbulent Dissipation Rate** drop-down lists.

   f. Retain the default of **Second Order Upwind** in the **Energy** drop-down list.

2. Set the relaxation factors.

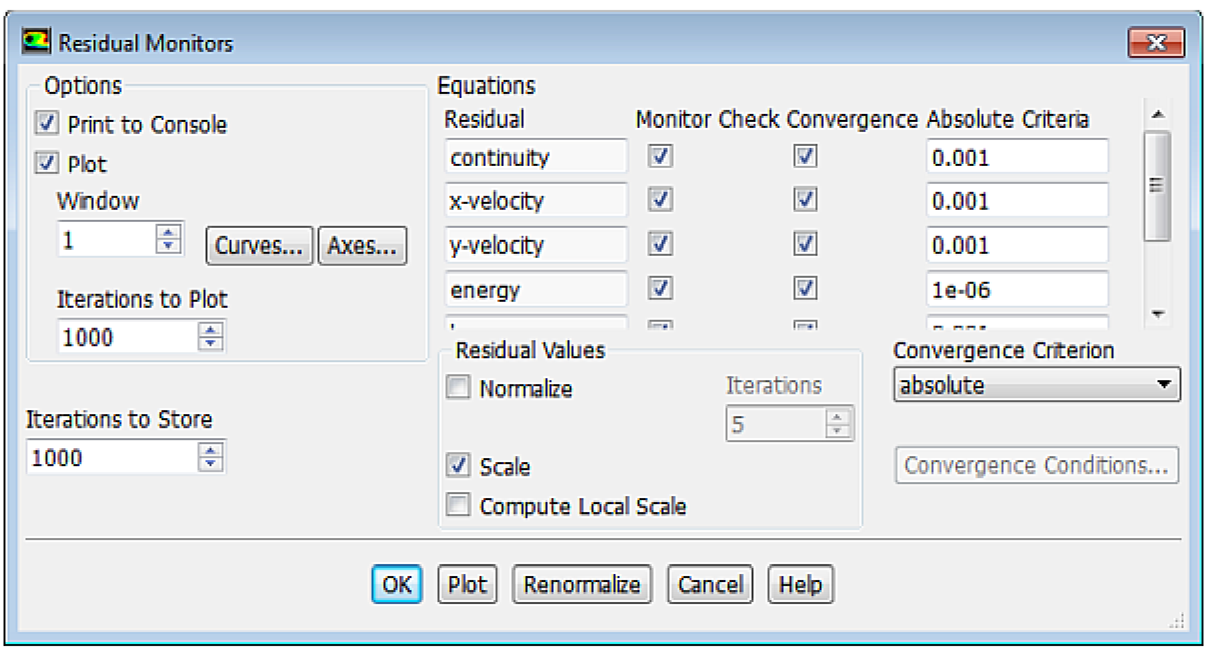
   Solving → **Controls** → **Controls...**



Retain the default values for **Under-Relaxation Factors** in the **Solution Controls** task page.

3. Enable the plotting of residuals during the calculation.

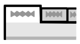
Solving → **Reports** → **Residuals...**

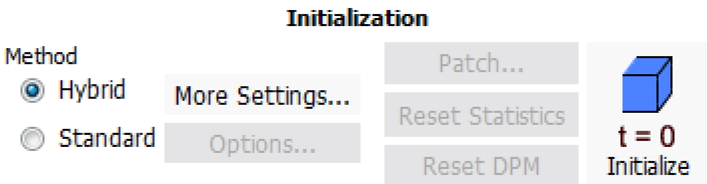


a. Ensure that **Plot** is enabled in the **Options** group box.

b. Click **OK** to close the **Residual Monitors** dialog box.

4. Initialize the solution.

 **Solving** → **Initialization**



a. Retain the default selection of **Hybrid** from the **Method** list.

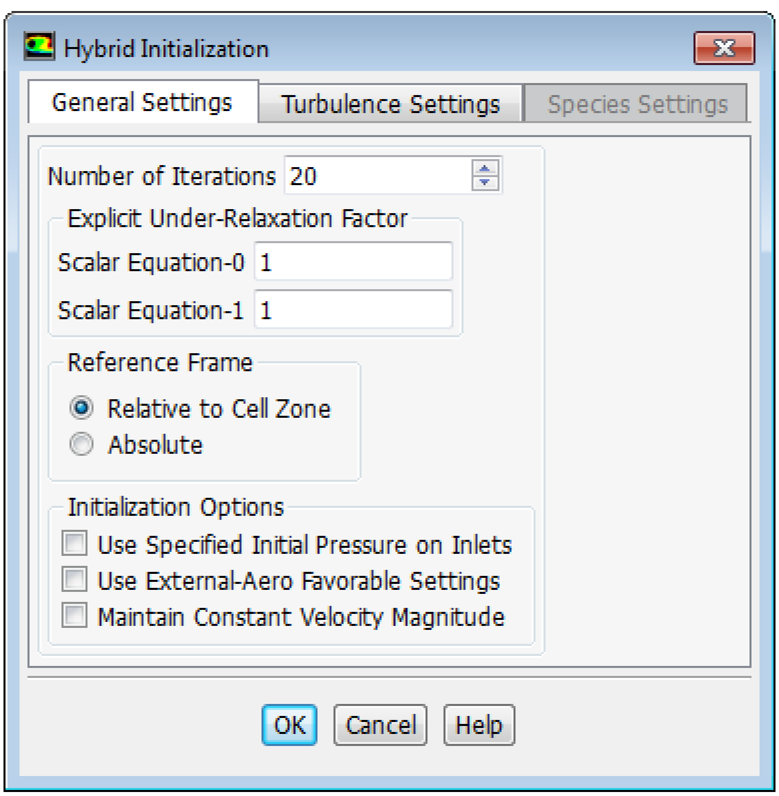
b. Click **Initialize**.

Note

A warning is displayed in the console stating that the convergence tolerance of 1.000000e-06 was not reached during hybrid initialization. This means that the default number of iterations is not enough. You will increase the number of iterations and re-initialize the flow. For more information refer to hybrid initialization in the Fluent User's Guide.

---

c. Click **More Settings...**.



i. Increase the **Number of Iterations** to `20`.

ii. Click **OK** to close the **Hybrid Initialization** dialog box.

d. Click **Initialize** again.

Note

Click **OK** in the **Question** dialog box that asks if it is OK to discard the current data. The console will then display that hybrid initialization is done.

Note

For flows in complex topologies, hybrid initialization will provide better initial velocity and pressure fields than standard initialization. This will help to improve the convergence behavior of the solver.

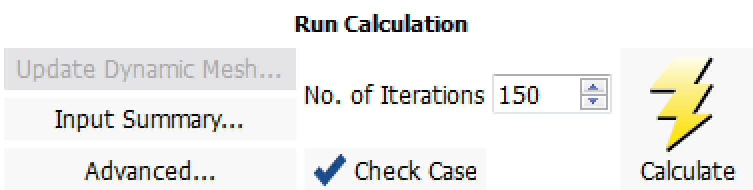
---

5. Save the case file (`valve_init.cas.gz`).

   File → Write → Case...

6. Start the calculation by requesting 150 iterations.

   Solving → Run Calculation



   a. Enter `150` for **No. of Iterations**.

   b. Click **Calculate** to start the steady flow simulation.

   The solution converges in approximately 132 iterations.

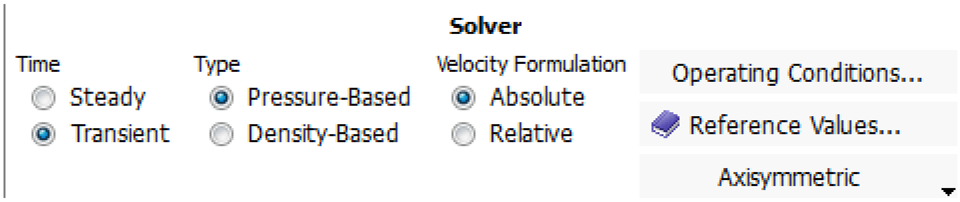
7. Save the case and data files (`valve_init.cas.gz` and `valve_init.dat.gz`).

   File → Write → Case & Data...

12.4.8. Time-Dependent Solution Setup

1. Enable a time-dependent calculation.

   Setting Up Physics → Solver

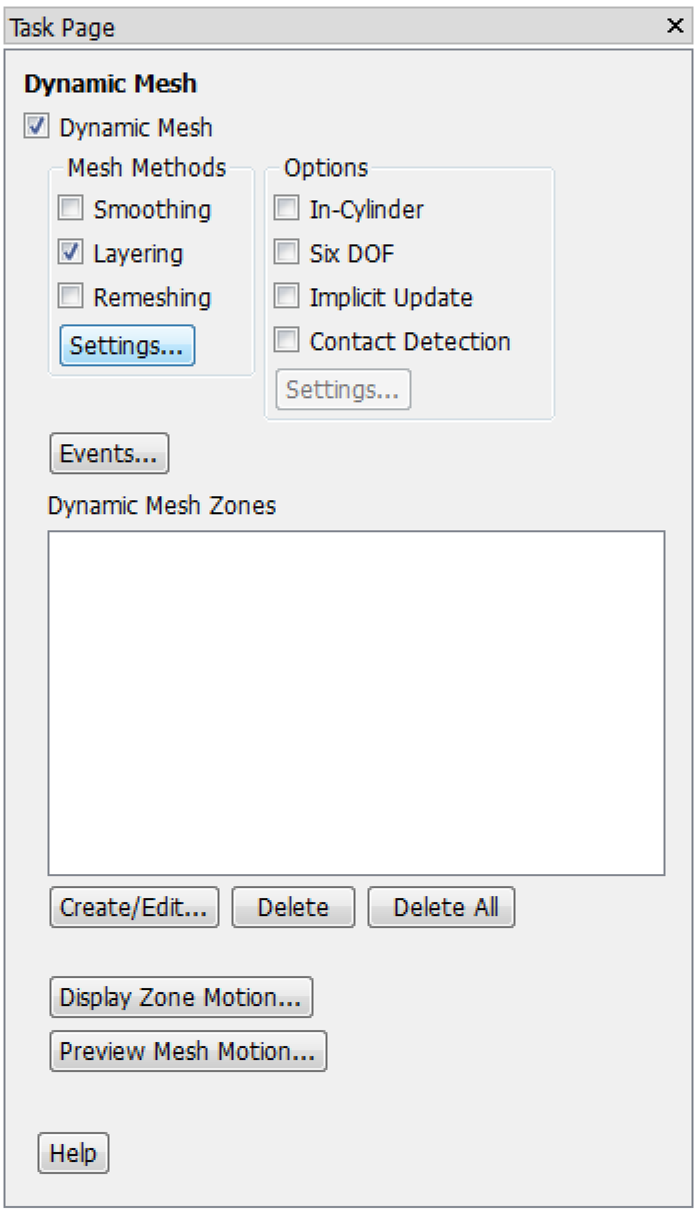


Select **Transient** from the **Time** list.

12.4.9. Mesh Motion

1. Enable dynamic mesh motion and define the layering settings.

   Setting Up Domain → **Mesh Models** → **Dynamic Mesh...**



   a. Enable **Dynamic Mesh** in the **Dynamic Mesh** task page.

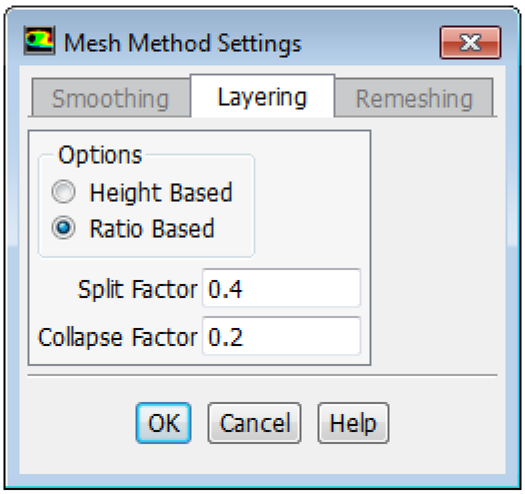
For more information on the available models for moving and deforming zones, see the Fluent User's Guide.

b. Disable **Smoothing** and enable **Layering** in the **Mesh Methods** group box.

Note

ANSYS Fluent will automatically flag the existing mesh zones for use of the different dynamic mesh methods where applicable.

c. Click the **Settings...** button in the **Mesh Methods** group box to open the **Mesh Method Settings** dialog box.



i. Select **Ratio Based** in the **Options** group box.

ii. Retain the default settings of `0.4` and `0.2` for **Split Factor** and **Collapse Factor**, respectively.

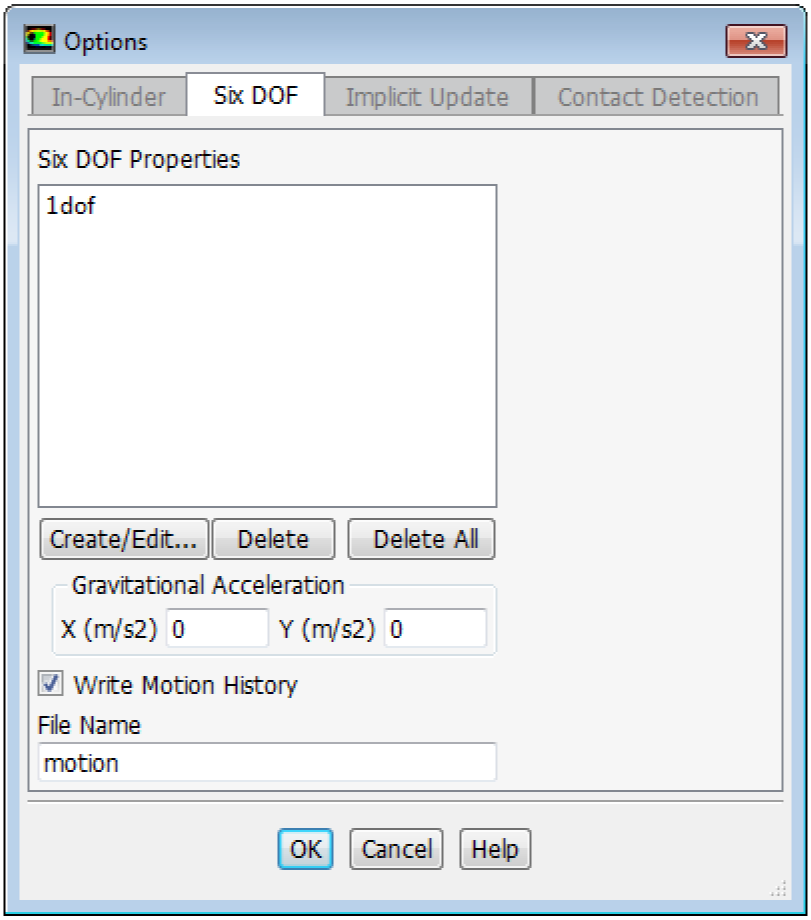
iii. Click **OK** to close the **Mesh Method Settings** dialog box.

2. Enable the six degrees of freedom (six DOF) solver and define a set of motion properties.

The six DOF solver computes external forces and moments (such as hydrodynamic forces) on objects that undergo rigid body motion. These forces are computed by numerical integration of pressure and shear stress over the object's surfaces. Additional load forces can be added, such as those produced by a spring.

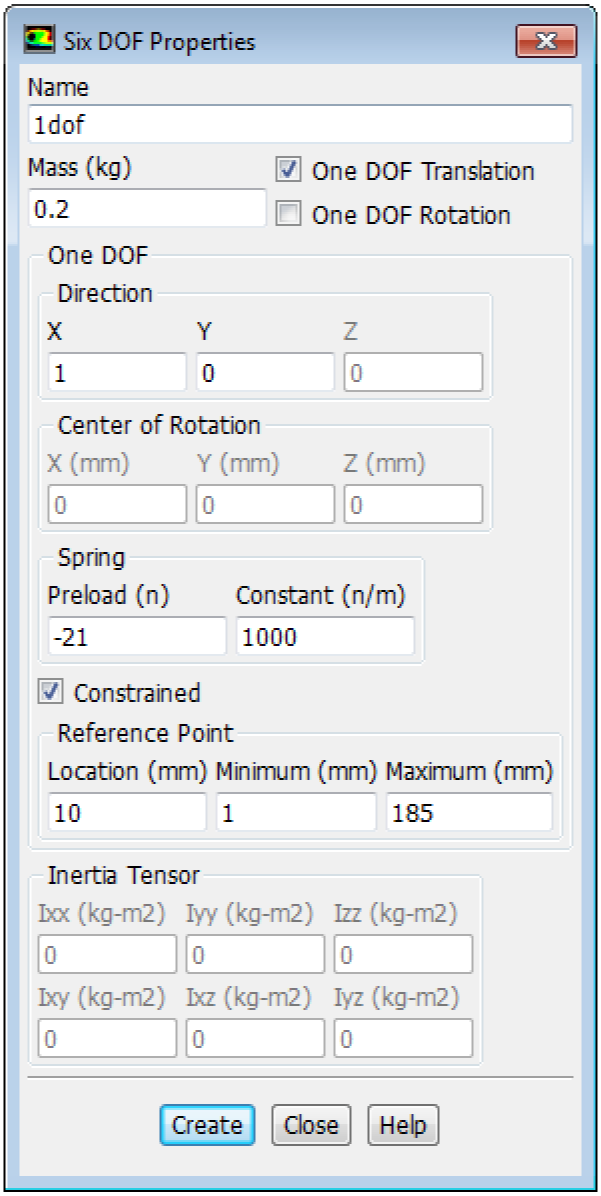
a. Enable **Six DOF** in the **Options** group box of the **Dynamic Mesh** task page.

b. Click the **Settings...** button in the **Options** group box to open the **Options** dialog box.



i. Click the **Create/Edit...** button in the **Six DOF** tab of the **Options** dialog box, in order to open the **Six DOF Properties** group box.

Here you will define a set of properties for the motion of the valve ball using the ANSYS Fluent graphical user interface. Note that you could instead define it through a user-defined function; this is not as easy to do, but may be necessary for more complex motion.



ii. Enter `1dof` for **Name**.

iii. Enter `0.2` for **Mass**.

This represents the total mass of the valve ball.

iv. Enable the **One DOF Translation** option to ensure the valve ball only translates in a single direction without rotation.

v. Enter `1` for **X** and `0` for **Y** in the **Direction** group box.

vi. Enter `-21` for **Preload** and `1000` for **Constant** in the **Spring** group box, in order to define a Hooke's law spring that exerts loads on the valve ball.

The ***Preload*** *field allows you to specify that the spring is deformed (that is, extended or compressed) with the current position of the valve ball. A negative value is used in this case in accordance with the defined* ***Direction****, that is, to specify that the initial force of the spring acts to move the valve ball toward the valve seat.*

vii. Enable the **Constrained** option, so that you can prevent the valve ball from contacting the valve seat.

The valve cannot be allowed to completely close: because dynamic mesh problems require that at least one layer of cells remains in order to maintain the topology, a small gap must be maintained between the valve ball and the valve seat.

viii. Enter the following values in the **Reference Point** group box.

Property	Value
Location	`10 mm`
Minimum	`1 mm`
Maximum	`185 mm`

The **Location** *assigns a coordinate value to a point located on the dynamic object (the valve ball) in its initial position. The value of the* **Location** *is arbitrary, and allows you to define the range using values that are most convenient for your problem. In this case, the surface of the valve ball is initially 10 mm away from the valve seat; the* **Minimum** *is defined relative to the* **Location** *and the values you entered here allow the valve ball to move only 9 mm in the direction opposite to the* **Direction**.

ix. Click **Create** to save your settings.

x. Close the **Six DOF Properties** dialog box.

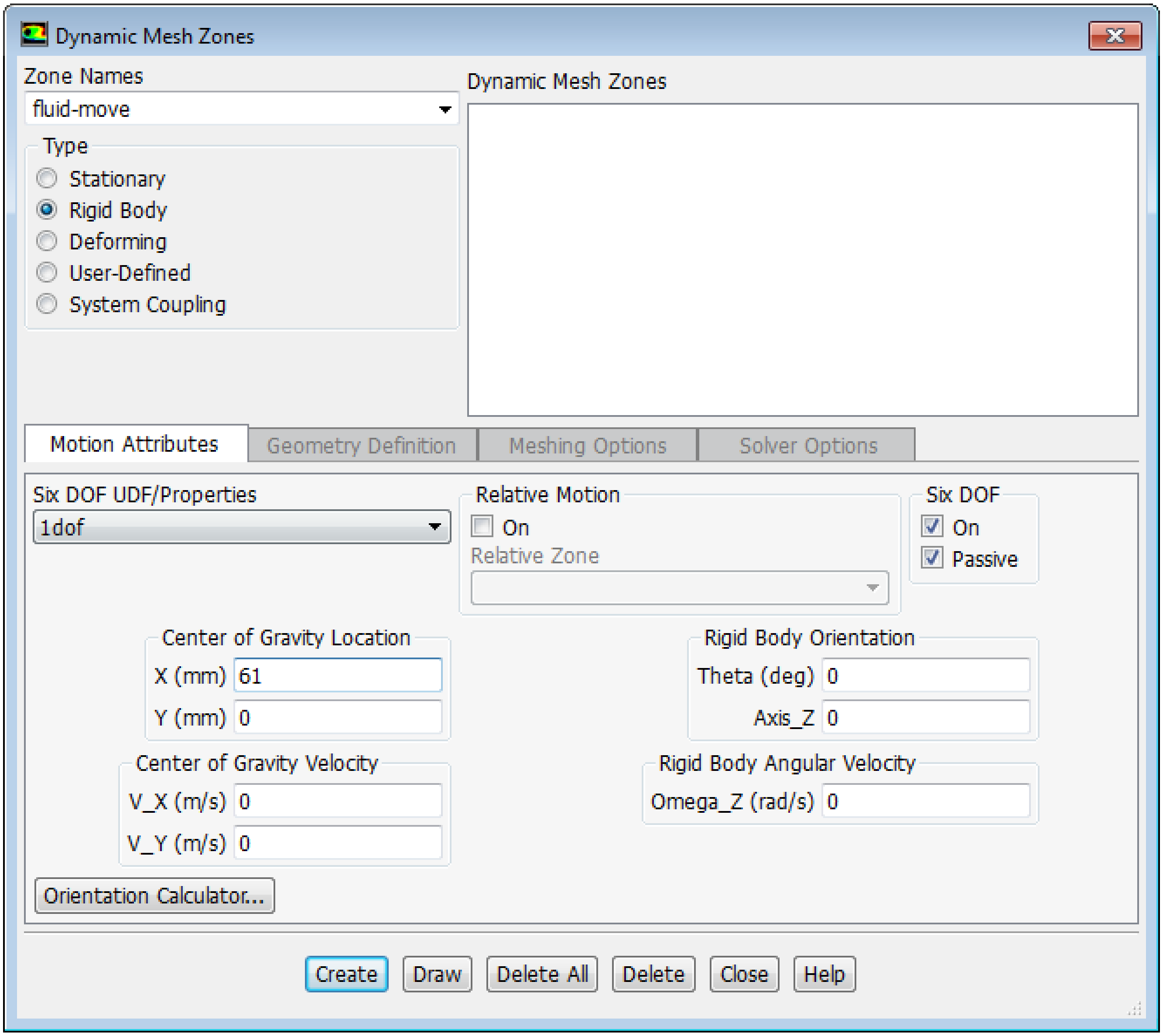
xi. Enable the **Write Motion History** option in the **Options** dialog box, and enter `motion` for the **File Name**.

This ensures that a file is created during the simulation that lists the location and orientation of the dynamic object at every time step, which could be used for postprocessing.

xii. Click **OK** to close the **Options** dialog box.

3. Specify the dynamic mesh zone motion for the fluid cell zone in which the valve ball moves (**fluid-move**).

   a. Click **Create/Edit...** in the **Dynamic Mesh** task page, to open the **Dynamic Mesh Zones** dialog box.



b. Select **fluid-move** from the **Zone Names** drop-down list.

c. Retain the default selection of **Rigid Body** from the **Type** list.

d. Ensure that the **On** option is enabled in the **Six DOF** group box of the **Motion Attributes** tab.

e. Enable the **Passive** option in the **Six DOF** group box.

   The only reason you are having this cell zone move is to preserve the mesh that surrounds the moving valve ball. Enabling the **Passive** *option ensures that the forces on this zone do not affect the motion of the valve ball.*

f. Ensure that **1dof** is selected from the **Six DOF UDF/Properties** drop-down list, to apply the motion properties you defined previously.

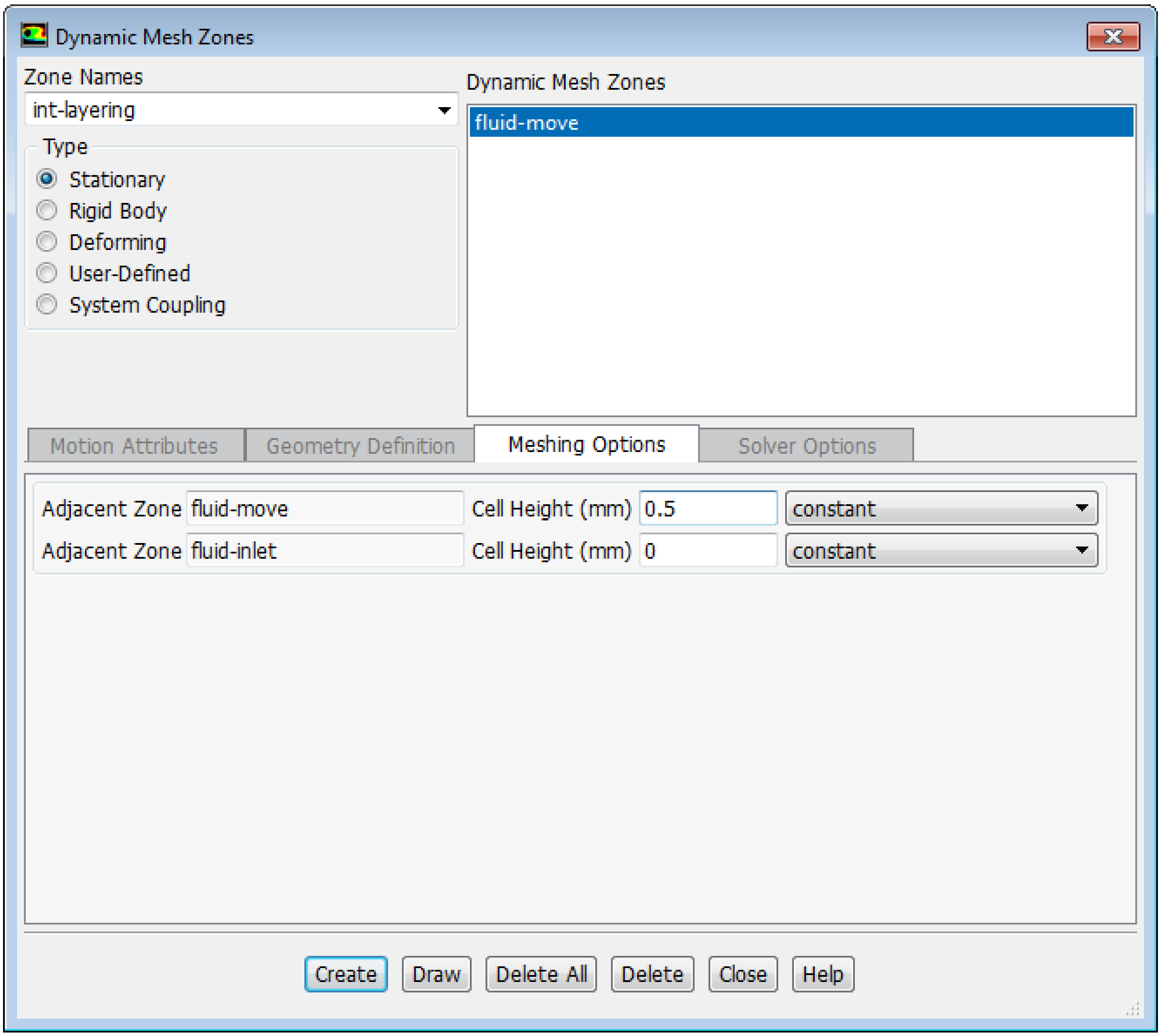
g. Enter `61` mm for **X** and `0` mm for **Y** in the **Center of Gravity Location**.

These coordinates correspond to the center of the valve ball in the initial position. These fields will be updated by ANSYS Fluent during the simulation, allowing you to track the location of this zone from within the **Dynamic Mesh Zones** *dialog box.*

h. Retain the default settings of `0` for the fields in the **Center of Gravity Velocity**, **Rigid Body Orientation**, and **Rigid Body Angular Velocity** group boxes.

i. Click **Create**.

4. Specify the meshing options for the boundary zone named **int-layering** in the **Dynamic Mesh Zones** dialog box. This boundary zone acts as a layering interface between the stationary inlet cell zone and the cell zone in which the valve ball moves.



a. Select **int-layering** from the **Zone Names** drop-down list.

b. Select **Stationary** from the **Type** list.

c. Enter `0.5` mm for the **Cell Height** of the `fluid-move` **Adjacent Zone**.

d. Retain the default value of `0` mm for the **Cell Height** of the `fluid-inlet` **Adjacent Zone**.

e. Click **Create**.

5. Specify the meshing options for the boundary zone named **outlet** in the **Dynamic Mesh Zones** dialog box. This boundary zone acts as a stationary outlet for the cell zone in which the valve ball moves.

   a. Select **outlet** from the **Zone Names** drop-down list.

   b. Retain the previous selection of **Stationary** from the **Type** list.

   c. Enter `1.9` mm for the **Cell Height** of the `fluid-move` **Adjacent Zone**.

   d. Click **Create**.

6. Specify the meshing options for the boundary zone named **seat-valve** in the **Dynamic Mesh Zones** dialog box. This boundary zone acts as the stationary valve seat.

   a. Select **seat-valve** from the **Zone Names** drop-down list.

   b. Retain the previous selection of **Stationary** from the **Type** list.

   c. Enter `0.5` mm for the **Cell Height** of the `fluid-move` **Adjacent Zone**.

   d. Click **Create**.

7. Specify the motion of the boundary zone named **valve** in the **Dynamic Mesh Zones** dialog box. This boundary zone surrounds the valve ball that moves relative to the valve seat.

   a. Select **valve** from the **Zone Names** drop-down list.

   b. Select **Rigid Body** from the **Type** list.

   c. Click the **Motion Attributes** tab.

      i. Enable the **On** option in the **Six DOF** group box.

      ii. Ensure that the **Passive** option is disabled in the **Six DOF** group box.

      This ensures that the forces on this zone affect the motion of the valve ball.

      iii. Select **1dof** from the **Six DOF UDF/Properties** drop-down list to apply the motion properties you defined previously.

      iv. Enter `61` mm for **X** and `0` mm for **Y** in the **Center of Gravity Location**.

      Again, these coordinates correspond to the center of the valve ball in the initial position.

      v. Retain the default settings of `0` for the fields in the **Center of Gravity Velocity**, **Rigid Body Orientation**, and **Rigid Body Angular Velocity** group boxes.

   d. Click the **Meshing Options** tab and enter `0` mm for the **Cell Height** of the `fluid-move` **Adjacent Zone**.

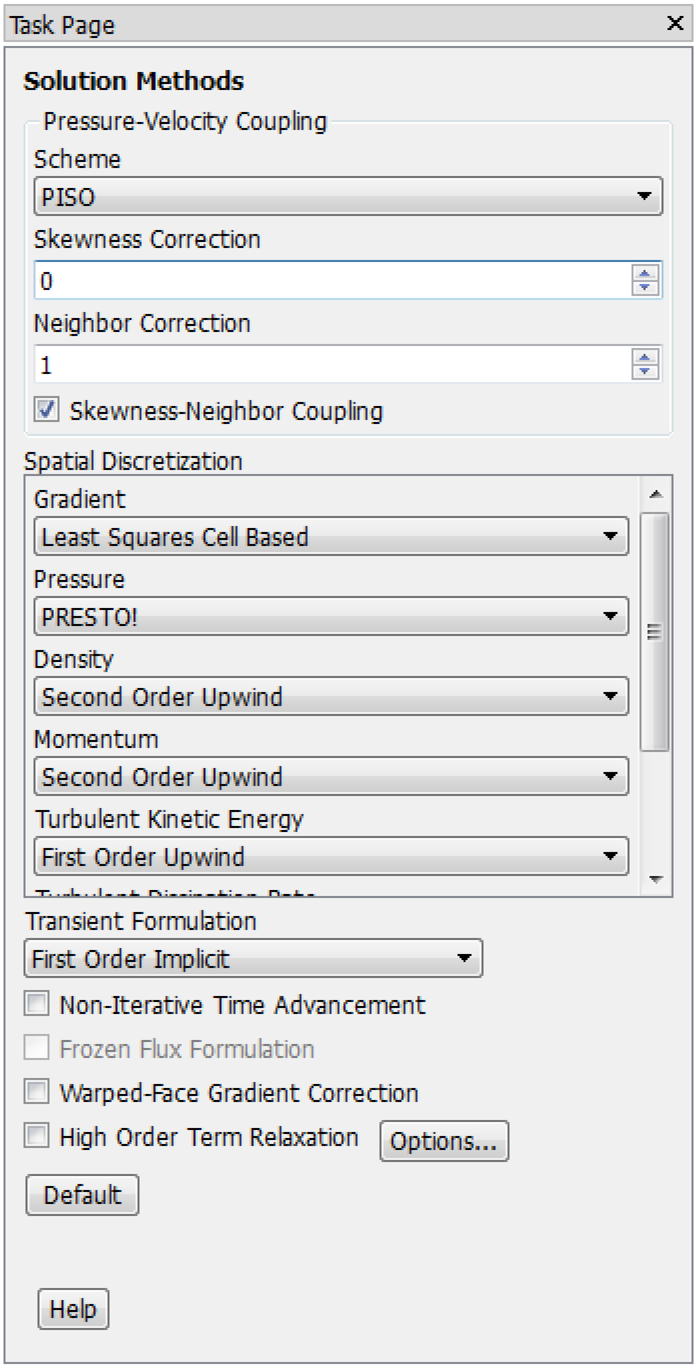
   e. Click **Create** and close the **Dynamic Mesh Zones** dialog box.

In many moving / deforming mesh (MDM) problems, you may want to preview the mesh motion before proceeding. In this problem, the mesh motion is affected by the pressure exerted by the fluid on the valve ball and acting against the inertia of the valve ball. Hence, for this problem, mesh motion in the absence of a flow field solution is meaningless, and you will not use this feature here.

12.4.10. Time-Dependent Solution

1. Set the solution parameters.

   Solving → **Solution** → **Methods...**



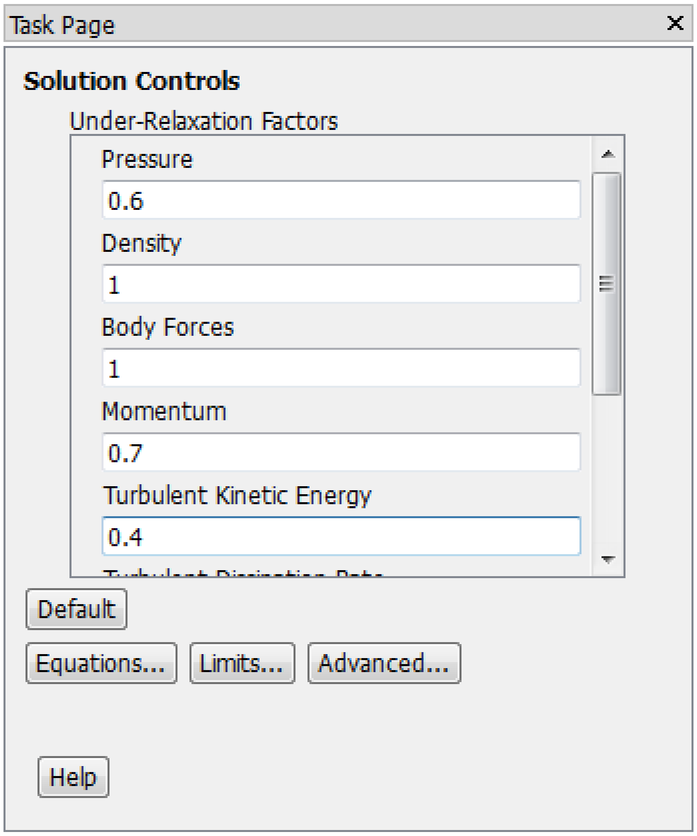
   a. Select **PISO** from the **Scheme** drop-down list in **Pressure-Velocity Coupling** group box.

b. Enter `0` for **Skewness Correction**.

c. Retain all of the other previously set schemes and defaults.

2. Set the relaxation factors.

Solving → **Controls** → **Controls...**



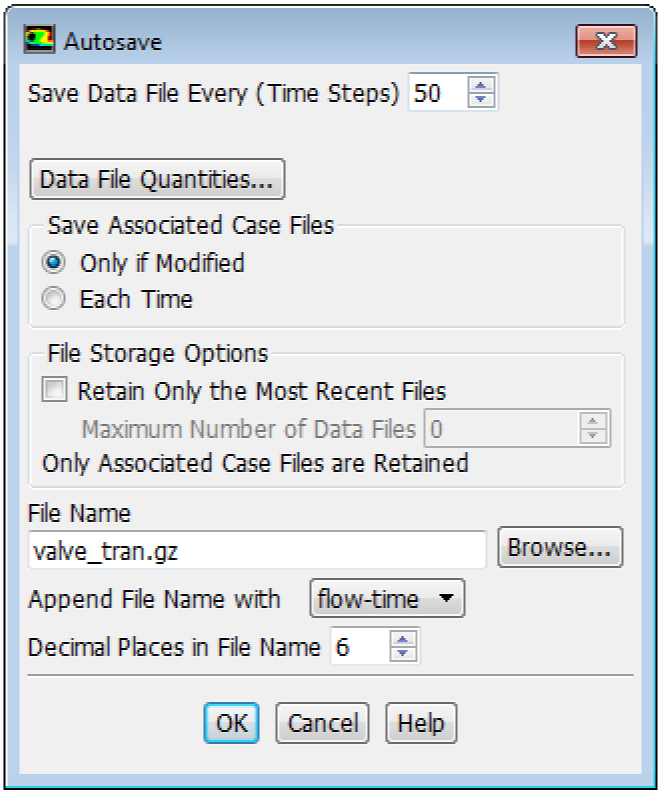
a. Enter `0.6` for **Pressure** in the **Under-Relaxation Factors** group box.

b. Enter `0.4` for **Turbulent Kinetic Energy**.

c. Scroll down and enter `0.4` for **Turbulent Dissipation Rate**.

3. Request that case and data files are automatically saved every 50 time steps.

Solving → **Activities** → **Autosave...**



a. Enter `50` for **Save Data File Every (Time Steps)**.

b. Enter `valve_tran.gz` in the **File Name** field.

c. Select **flow-time** from the **Append File Name with** drop-down list.

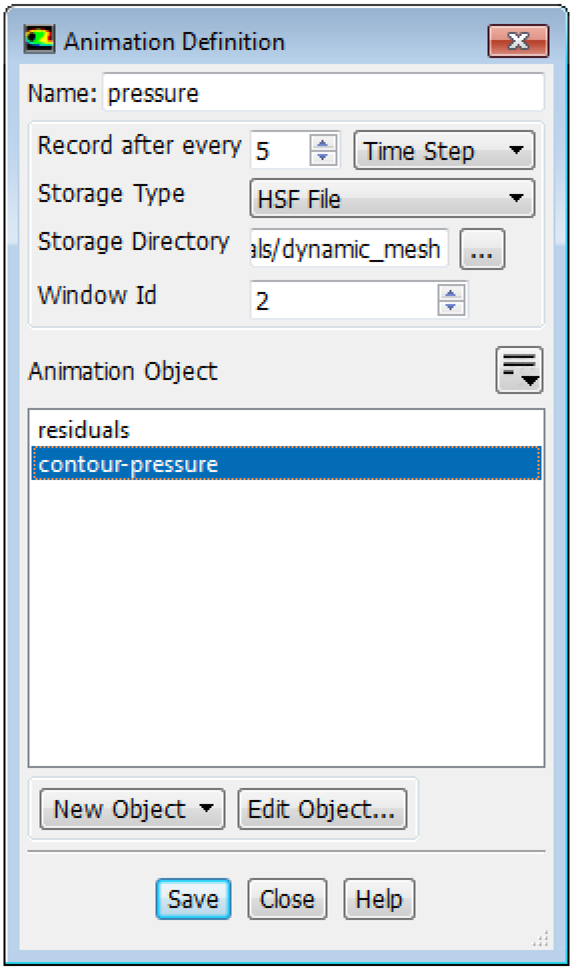
   When ANSYS Fluent saves a file, it will append the flow time value to the file name prefix (`valve_tran`). The gzipped standard extensions (`.cas.gz` and `.dat.gz`) will also be appended.

d. Click **OK** to close the **Autosave** dialog box.

4. Create an animation sequence for the static pressure contour plots for the valve.

   Use the solution animation feature to save pressure contour plots every five time steps. After the calculation is complete, you will use the solution animation playback feature to view the animated plots over time.

   Solving → **Activities** → **Create** → **Solution Animations...**



a. Enter `pressure` for **Name** in the **Animation Definition** dialog box.

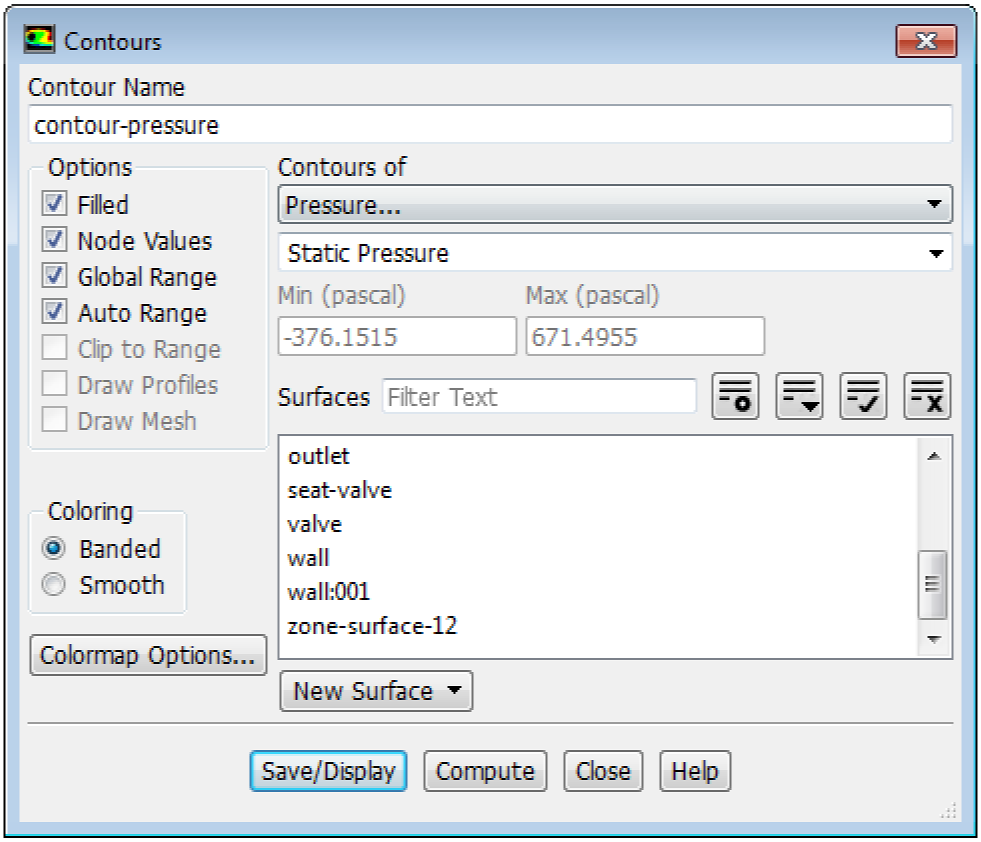
b. Set **Record after every** to 5 and select **Time Step** from the neighboring drop-down list.

   The default value of `1` *instructs ANSYS Fluent to update the animation sequence at every time step. For this case, this would generate a large number of files.*

c. Retain the default value of 2 for **Window Id**.

   This is where the animation frames will be displayed during the calculation.

d. Click the **New Object** drop-down list and select **Contours...** from the menu that opens.



i. Enter `contour-pressure` for **Contour Name** in the **Contours** dialog box.

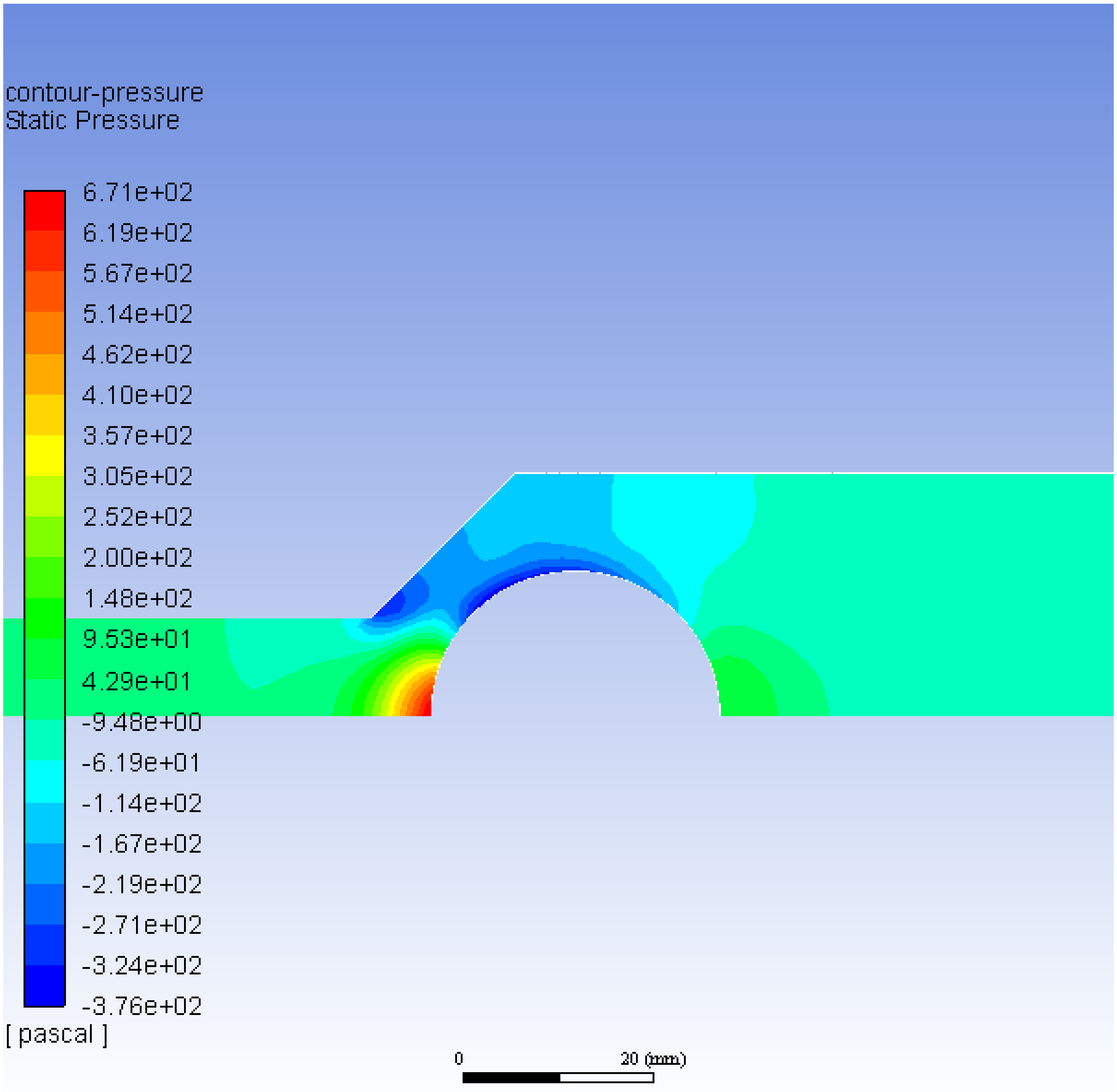
ii. Enable **Filled** in the **Options** group box.

iii. Retain the default selection of **Pressure...** and **Static Pressure** from the **Contours of** drop-down lists.

iv. Click **Save/Display** (Figure 12.3: Contours of Static Pressure at t=0 s (p. 527)).

v. Close the **Contours** dialog box.

Figure 12.3: Contours of Static Pressure at t=0 s

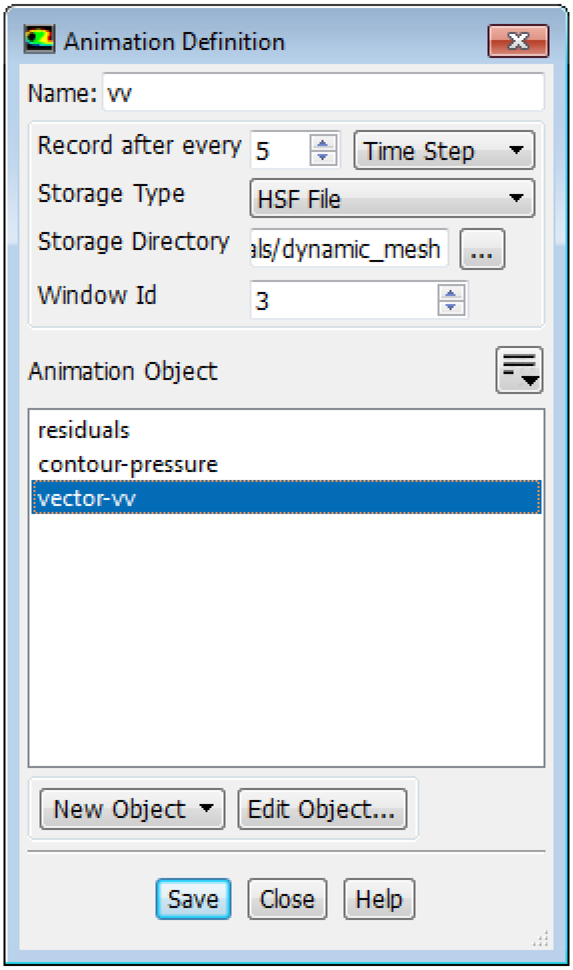


e. Select **contour-pressure** from the **Animation Object** selection list.

f. Click **Save** and close the **Animation Definition** dialog box.

5. Create an animation sequence for the velocity vectors plots for the valve.

Solving → **Activities** → **Create** → **Solution Animations...**

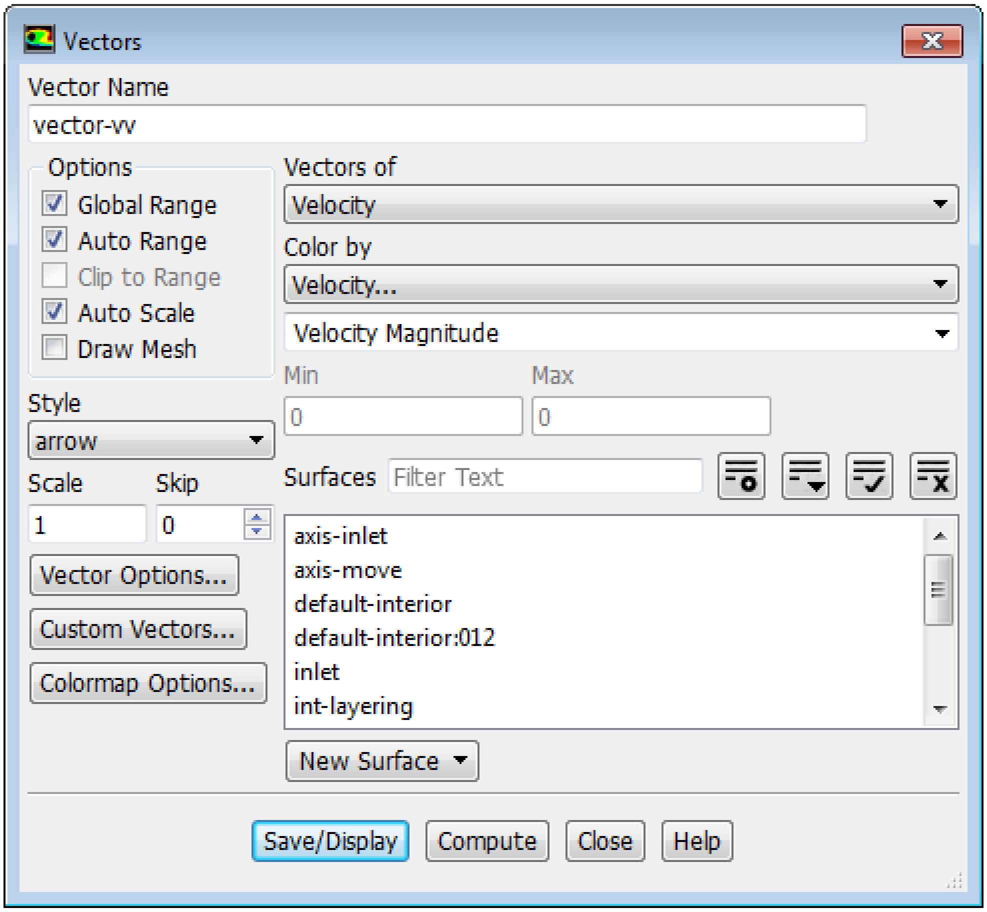


a. Enter vv for **Name**.

b. Set **Record after every** to 5 and select **Time Step** from the neighboring drop-down list.

c. Retain the default value of 3 for **Window Id**.

d. Click the **New Object** drop-down list and select **Vectors...** from the menu that opens.



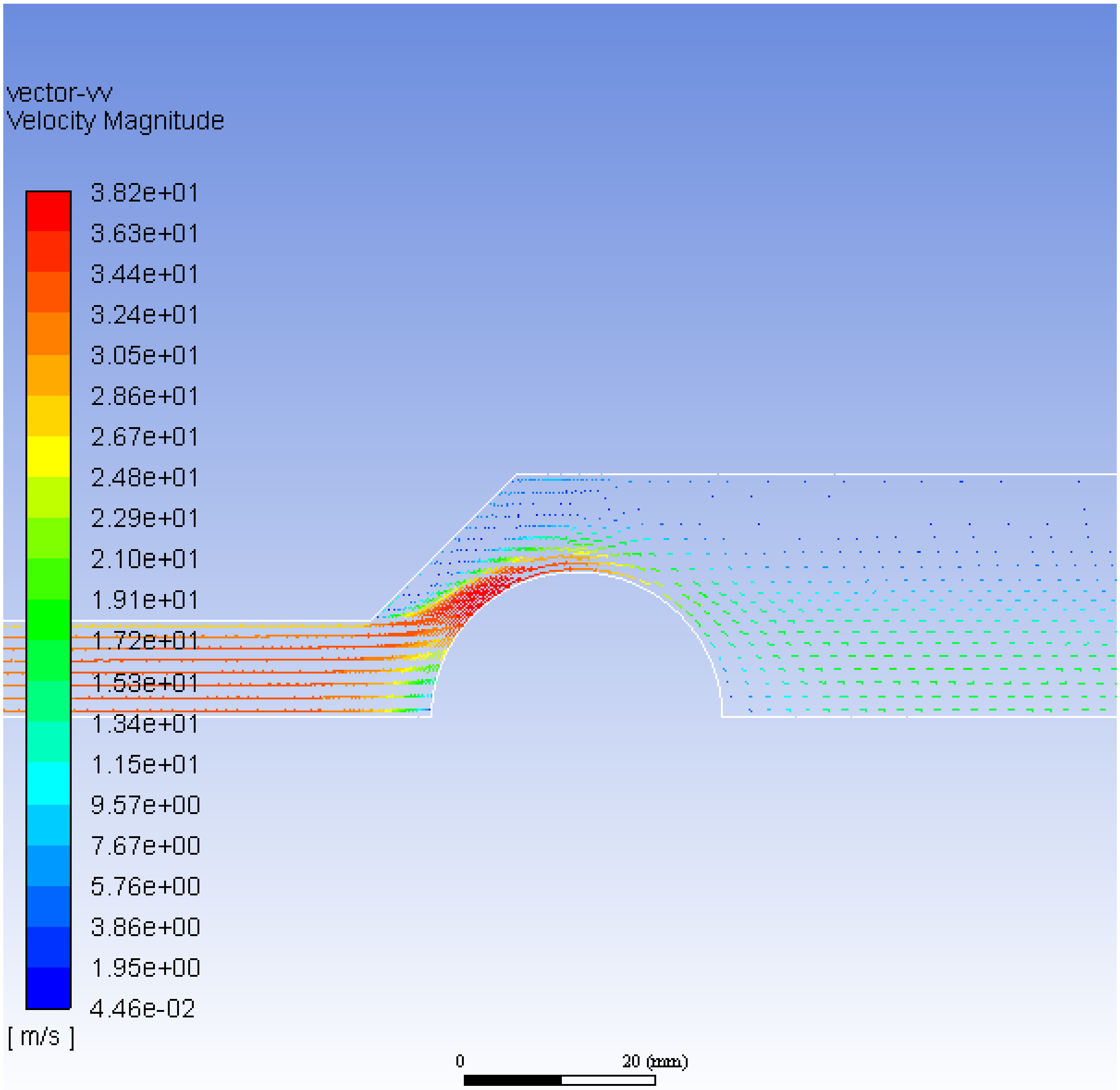
i. Enter `vector-vv` for **Name**.

ii. Retain all the other default settings.

iii. Click **Save/Display** (Figure 12.4: Vectors of Velocity at t=0 s (p. 530)).

iv. Close the **Vectors** dialog box.

Figure 12.4: Vectors of Velocity at t=0 s



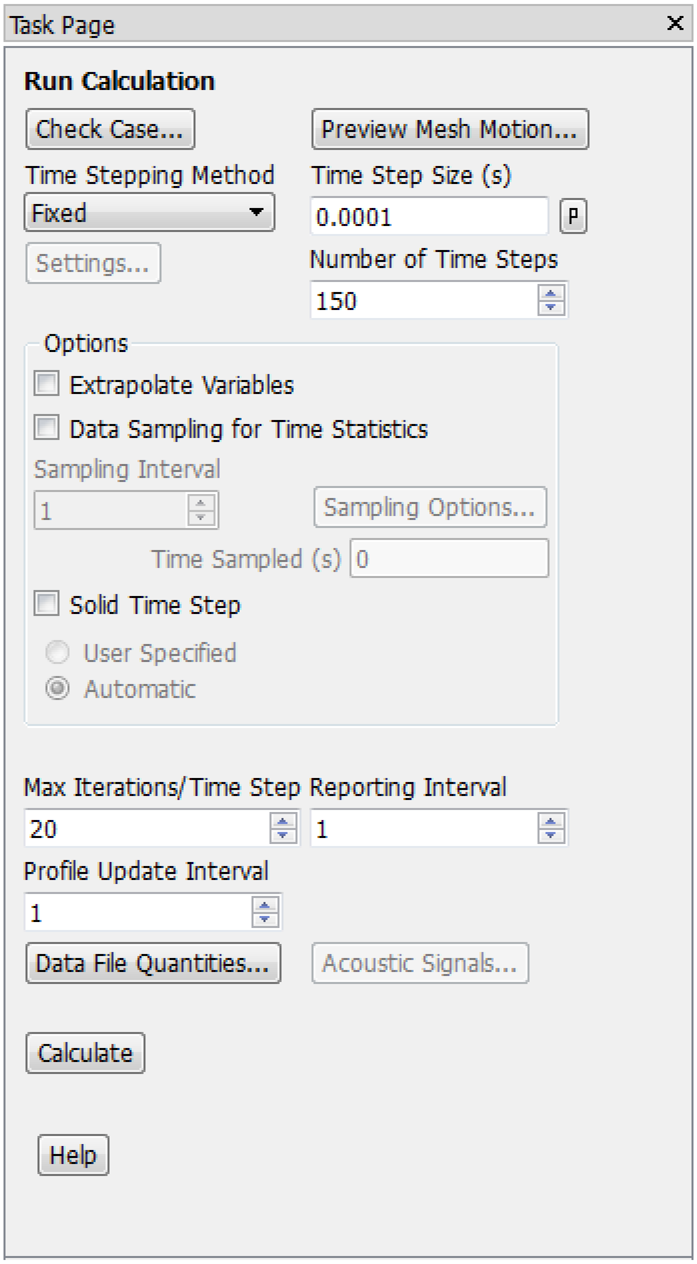
e. Select **vector-vv** from the **Animation Object** selection list.

f. Click **Save** and close the **Animation Definition** dialog box.

Note that the new animation definitions appear under the **Solution/Calculation Activities/Solution Animations** *tree branch. To edit an animation definition, right-click it and select* **Edit...** *from the menu that opens.*

6. Set the time step parameters for the calculation.

Solving → **Run Calculation** → **Advanced...**



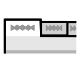
a. Enter `0.0001` s for **Time Step Size**.

b. Enter `150` for **Number of Time Steps**.

c. Retain `20` for **Max Iterations/Time Step**.

In the accurate solution of a real-life time-dependent CFD problem, it is important to make sure that the solution converges at every time step to within the desired accuracy. Here, the first few time steps will only come to a reasonably converged solution.

7. Save the initial case and data files for this transient problem (`valve_tran-0.000000.cas.gz` and `valve_tran-0.000000.dat.gz`).

 **File → Write → Case & Data...**

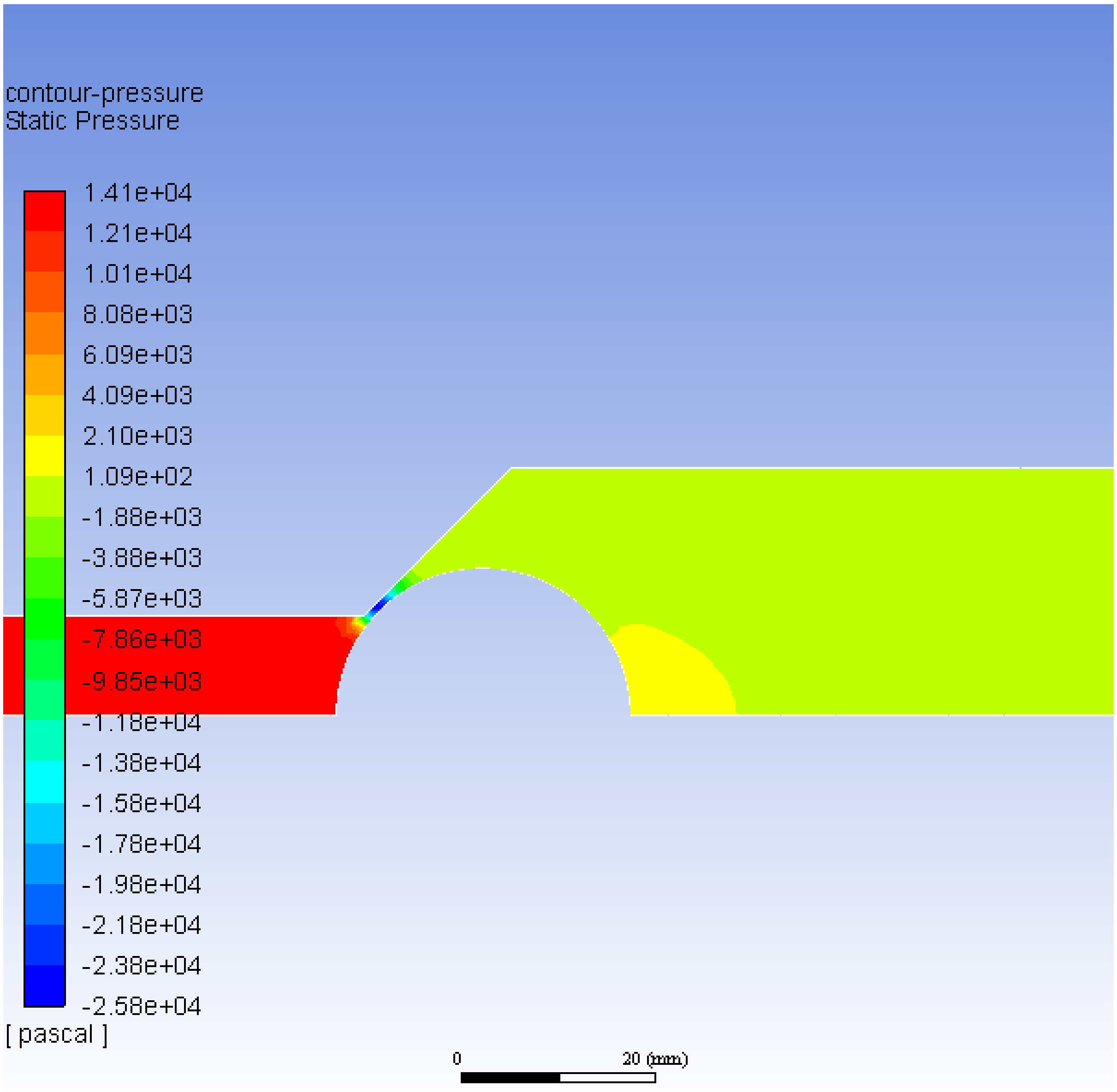
8. Calculate a solution by clicking **Calculate** in the **Run Calculation** task page.

12.4.11. Postprocessing

1. Inspect the solution at the final time step.

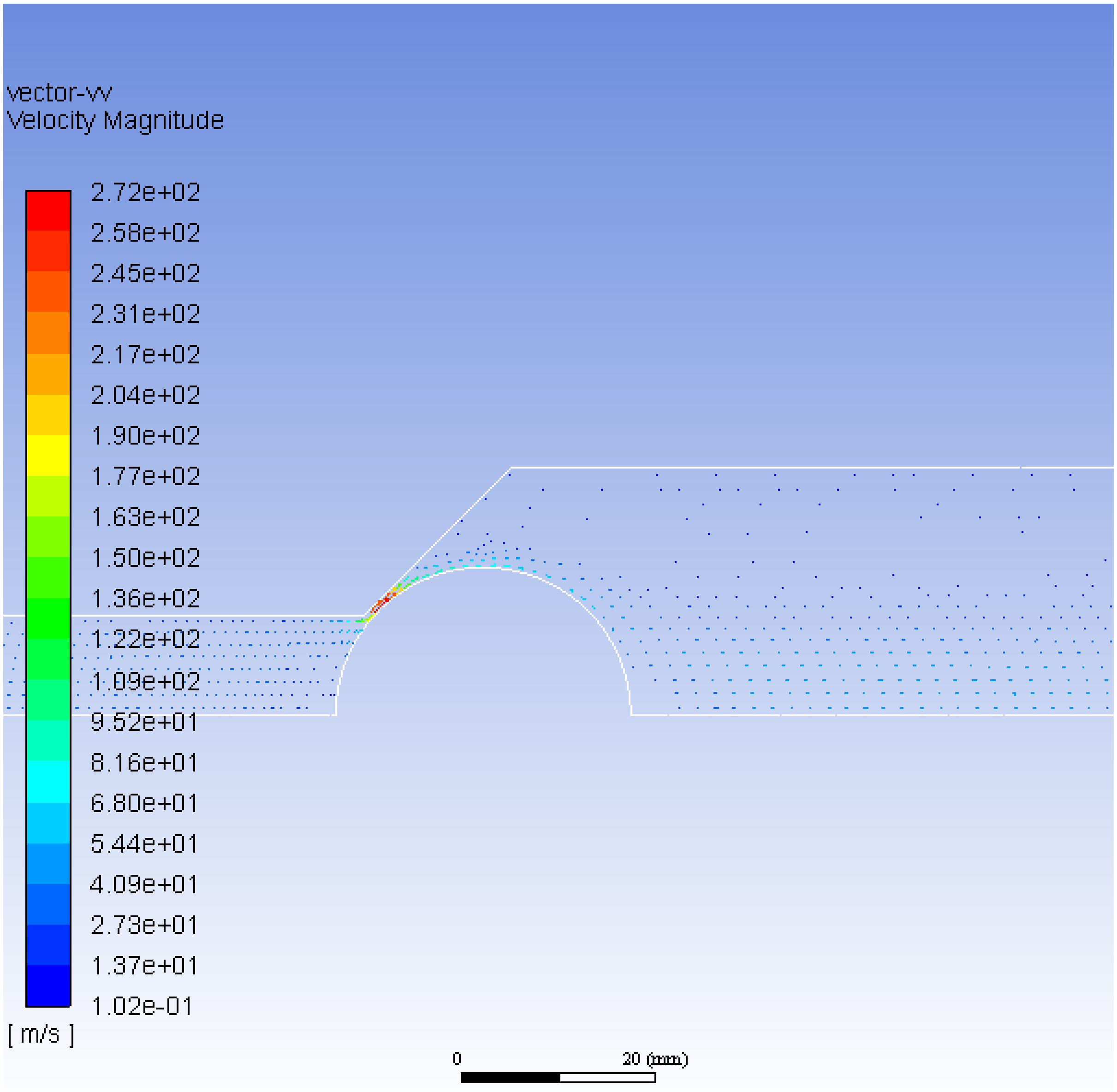
   a. Inspect the contours of static pressure in the valve, by clicking the **Contours of Static Pressure** tab in the graphics window (Figure 12.5: Contours of Static Pressure After 150 Time Steps (p. 532)).

   Figure 12.5: Contours of Static Pressure After 150 Time Steps



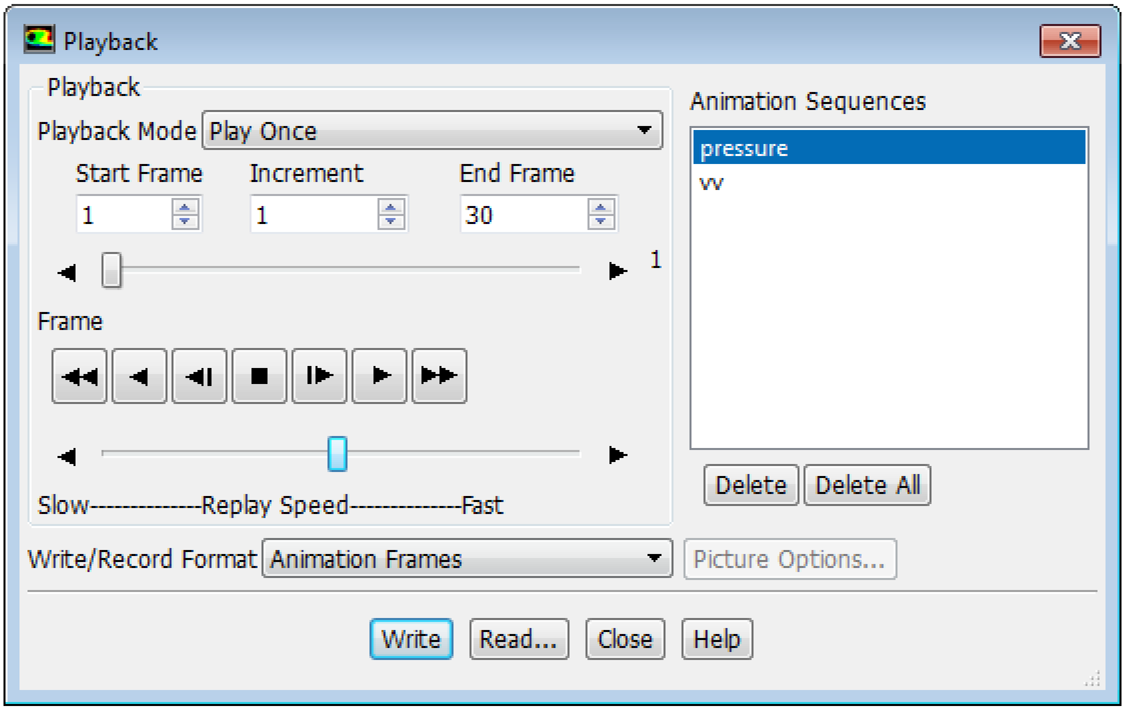
   b. Inspect the velocity vectors near the point where the valve ball meets the valve seat, by clicking the **Velocity Vectors Colored By Velocity Magnitude** tab in the graphics window (Figure 12.6: Velocity Vectors After 150 Time Steps (p. 533)).

Figure 12.6: Velocity Vectors After 150 Time Steps



2. Play the animation of the pressure contours.

   Postprocessing → **Animation** → **Solution Playback...**



a. Select **pressure** from the **Animation Sequences** list in the **Playback** dialog box.

If the **Animation Sequences** *list is empty, click* **Read...** *to select the* `pressure.cxa` *sequence file from your working folder.*

The playback control buttons will become active.

b. Set the slider bar above **Replay Speed** about halfway between **Slow** and **Fast**.

c. Retain the default settings in the rest of the dialog box and click the  button.

You may have to change the window to see the animation. Click the second tab in the graphics window, which corresponds to the **Window Id** *that you set for* `pressure` *in the* **Animation Definition** *dialog box.*

3. Play the animation of the velocity vectors.

a. Select **vv** from the **Animation Sequences** list in the **Playback** dialog box.

If the **Animation Sequences** *list does not contain* **vv**, *click* **Read...** *to select the* `vv.cxa` *sequence file from your working folder.*

b. Retain the settings in the rest of the dialog box and click the  button.

You may have to change the window to see the animation. Click the third tab in the graphics window, which corresponds to the **Window Id** *that you set for* `vv` *in the* **Animation Definition** *dialog box.*

For additional information on animating the solution, see Modeling Transient Compressible Flow (p. 267) and see the Fluent User's Guide.

c. Close the **Playback** dialog box.

4. You can also inspect the solution at different intermediate time steps.

   a. Read the corresponding case and data files (for example, `valve_tran-1-0.010000.cas.gz` and `valve_tran-1-0.010000.dat.gz`).

      File → **Read** → **Case & Data...**

   b. Display the desired contours and vectors.

12.5. Summary

In this tutorial, a check valve is used to demonstrate the dynamic layering capability within ANSYS Fluent, using one of the three dynamic mesh schemes available. You were also shown how to perform a one degree of freedom (one DOF) rigid body FSI by means of the six DOF solver.

12.6. Further Improvements

This tutorial guides you through the steps to generate an initial first-order solution. You may be able to increase the accuracy of the solution further by using an appropriate higher-order discretization scheme. For a more accurate solution, you can increase the number of layers across the valve seat area. This can be achieved either by using a finer mesh at the valve seat area and/or using a non-constant layer height instead of a constant layer height, as demonstrated in this tutorial.

Chapter 13: Modeling Species Transport and Gaseous Combustion

This tutorial is divided into the following sections:

13.1. Introduction

This tutorial examines the mixing of chemical species and the combustion of a gaseous fuel.

A cylindrical combustor burning methane (CH_4) in air is studied using the eddy-dissipation model in ANSYS Fluent.

This tutorial demonstrates how to do the following:

- Enable physical models, select material properties, and define boundary conditions for a turbulent flow with chemical species mixing and reaction.
- Initiate and solve the combustion simulation using the pressure-based solver.
- Examine the reacting flow results using graphics.
- Predict thermal and prompt NOx production.
- Use custom field functions to compute NO parts per million.

13.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

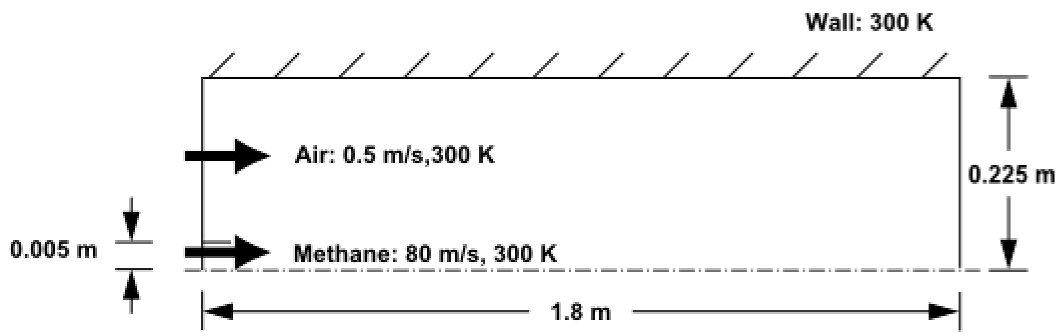
and that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

To learn more about chemical reaction modeling, see the Fluent User's Guide and the Fluent Theory Guide. Otherwise, no previous experience with chemical reaction or combustion modeling is assumed.

13.3. Problem Description

The cylindrical combustor considered in this tutorial is shown in Figure 13.1: Combustion of Methane Gas in a Turbulent Diffusion Flame Furnace (p. 538). The flame considered is a turbulent diffusion flame. A small nozzle in the center of the combustor introduces methane at 80 m/s. Ambient air enters the combustor coaxially at 0.5 m/s. The overall equivalence ratio is approximately 0.76 (approximately 28 $\%$ excess air). The high-speed methane jet initially expands with little interference from the outer wall, and entrains and mixes with the low-speed air. The Reynolds number based on the methane jet diameter is approximately 5.7×10^3.

Figure 13.1: Combustion of Methane Gas in a Turbulent Diffusion Flame Furnace



13.4. Background

In this tutorial, you will use the generalized eddy-dissipation model to analyze the methane-air combustion system. The combustion will be modeled using a global one-step reaction mechanism, assuming complete conversion of the fuel to CO_2 and H_2O. The reaction equation is

$$CH_4 + 2O_2 \rightarrow CO_2 + 2H_2O \tag{13.1}$$

This reaction will be defined in terms of stoichiometric coefficients, formation enthalpies, and parameters that control the reaction rate. The reaction rate will be determined assuming that turbulent mixing is the rate-limiting process, with the turbulence-chemistry interaction modeled using the eddy-dissipation model.

13.5. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

13.5.1. Preparation

To prepare for running this tutorial:

1. Set up a working folder on the computer you will be using.

2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

   > **Note**
   >
   > If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

3. Enter the name of this tutorial into the search bar.

4. Narrow the results by using the filter on the left side of the page.

   a. Click **ANSYS Fluent** under **Product**.

   b. Click **18.0** under **Version**.

5. Select this tutorial from the list.

6. Click the **species_transport_R180.zip** link to download the input files.

7. Unzip `species_transport_R180.zip` to your working folder.

   The file `gascomb.msh` can be found in the `species_transport` folder created after unzipping the file.

8. Use Fluent Launcher to start the **2D** version of ANSYS Fluent.

   Fluent Launcher displays your **Display Options** preferences from the previous session.

   For more information about Fluent Launcher, see starting ANSYS Fluent using the Fluent Launcher in the Fluent Getting Started Guide.

9. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.

10. Enable **Double-Precision**.

11. Ensure **Serial** is selected under **Processing Options**.

13.5.2. Mesh

1. Read the mesh file `gascomb.msh`.

   File → **Read** → **Mesh...**

   After reading the mesh file, ANSYS Fluent will report that 1615 quadrilateral fluid cells have been read, along with a number of boundary faces with different zone identifiers.

13.5.3. General Settings

1. Check the mesh.

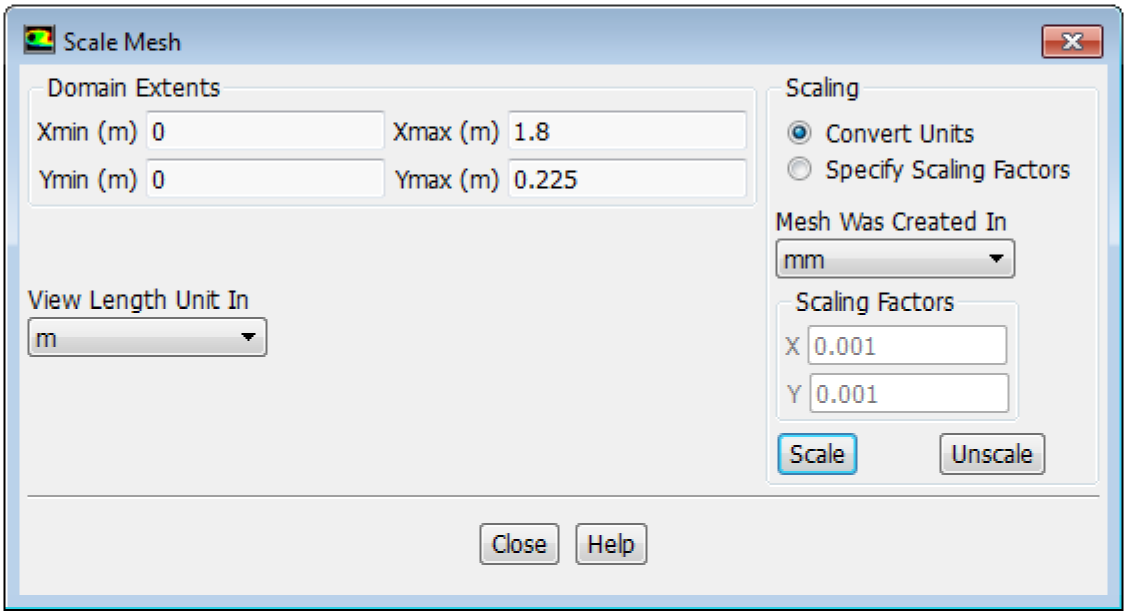
Setting Up Domain → Mesh → Check

ANSYS Fluent will perform various checks on the mesh and will report the progress in the console. Ensure that the reported minimum volume reported is a positive number.

2. Scale the mesh.

Setting Up Domain → Mesh → Scale...

Since this mesh was created in units of millimeters, you will need to scale the mesh into meters.



a. Select **mm** from the **Mesh Was Created In** drop-down list in the **Scaling** group box.

b. Click **Scale**.

c. Ensure that **m** is selected from the **View Length Unit In** drop-down list.

d. Ensure that **Xmax** and **Ymax** are reset to 1.8 m and 0.225 m respectively.

   The default SI units will be used in this tutorial, hence there is no need to change any units in this problem.

e. Close the **Scale Mesh** dialog box.

3. Check the mesh.

Setting Up Domain → Mesh → Check

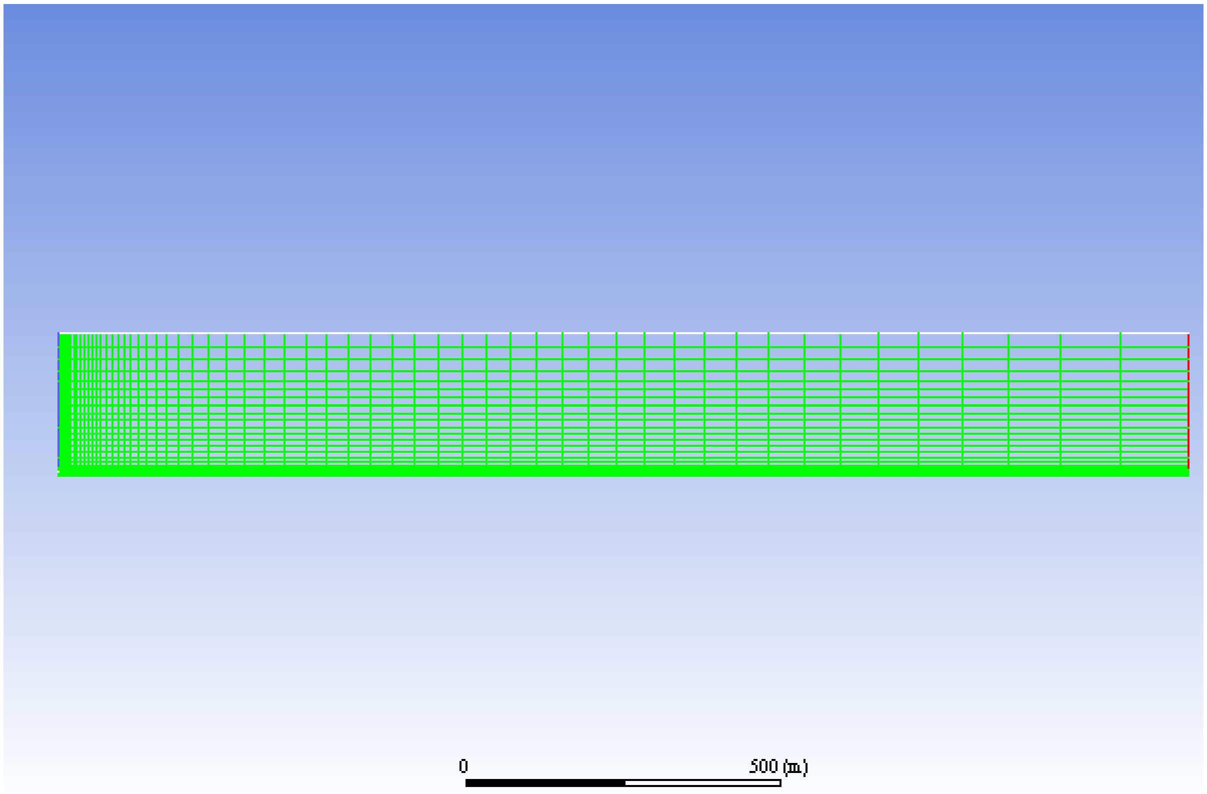
Note

You should check the mesh after you manipulate it (scale, convert to polyhedra, merge, separate, fuse, add zones, or smooth and swap). This will ensure that the quality of the mesh has not been compromised.

---

4. Examine the mesh with the default settings.

Figure 13.2: The Quadrilateral Mesh for the Combustor Model



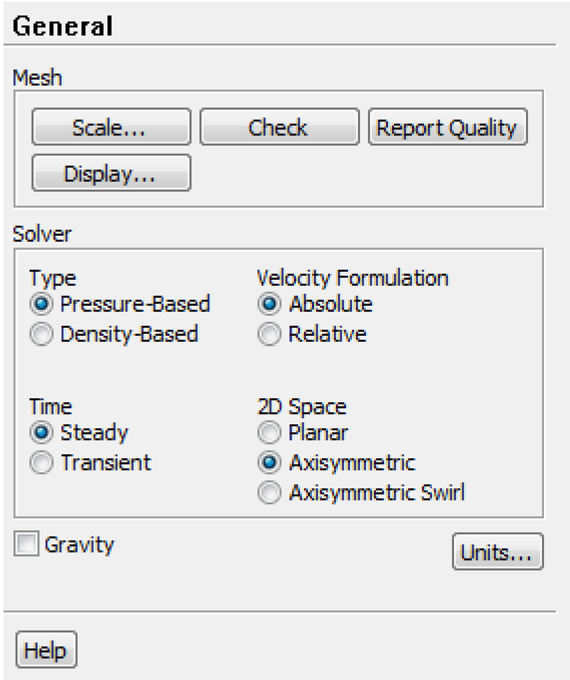
Extra

You can use the right mouse button to probe for mesh information in the graphics window. If you click the right mouse button on any node in the mesh, information will be displayed in the ANSYS Fluent console about the associated zone, including the name of the zone. This feature is especially useful when you have several zones of the same type and you want to distinguish between them quickly.

---

5. Select **Axisymmetric** in the **2D Space** list.

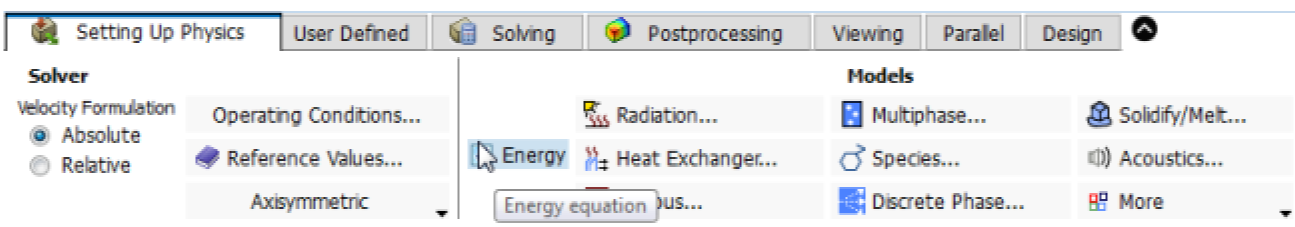
Setup → **General**



13.5.4. Models

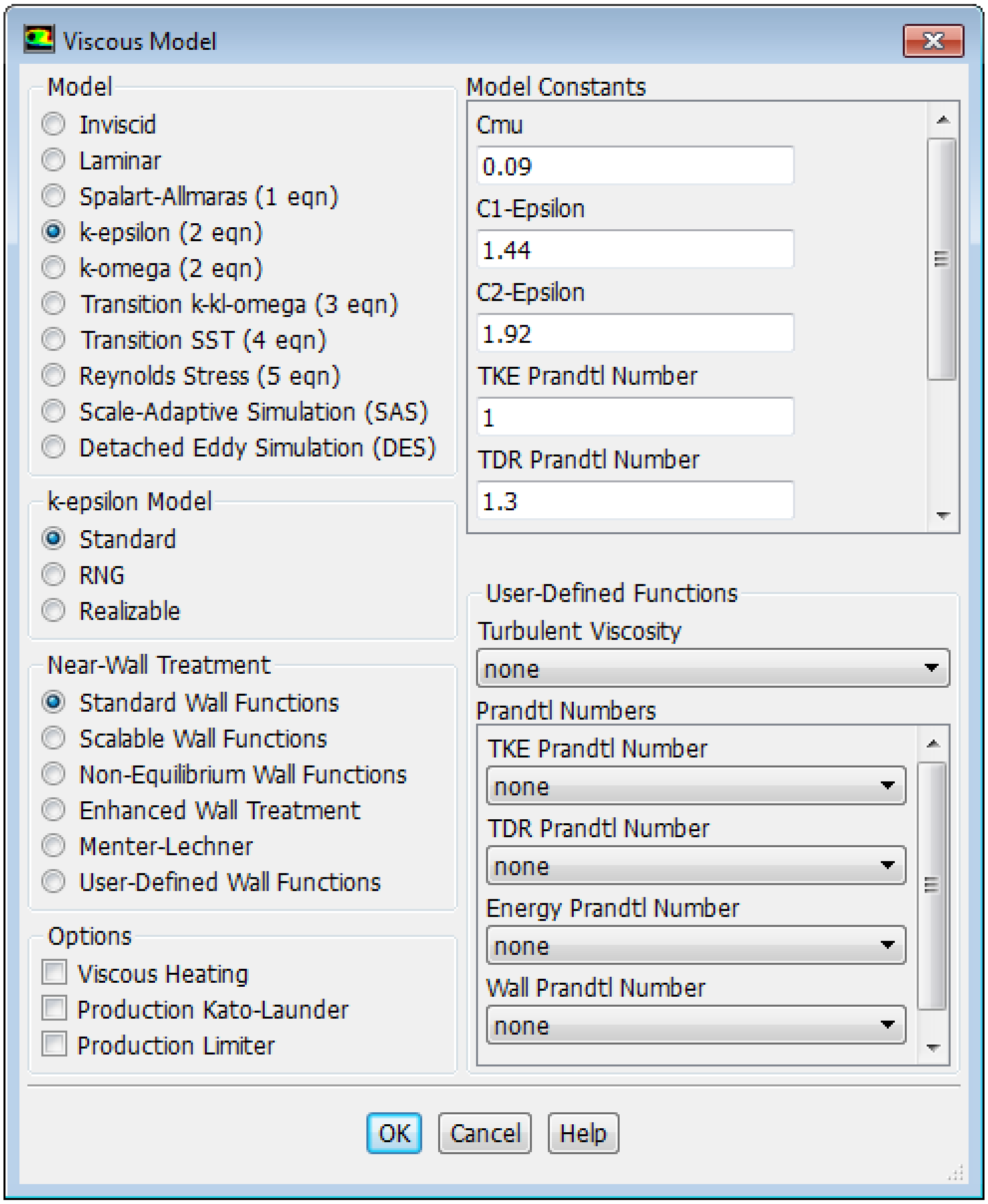
1. Enable heat transfer by enabling the energy equation.

   Setting Up Physics → **Models** → **Energy**



2. Select the standard k- ε turbulence model.

   Setting Up Physics → **Models** → **Viscous...**



a. Select **k-epsilon (2 eqn)** in the **Model** list.

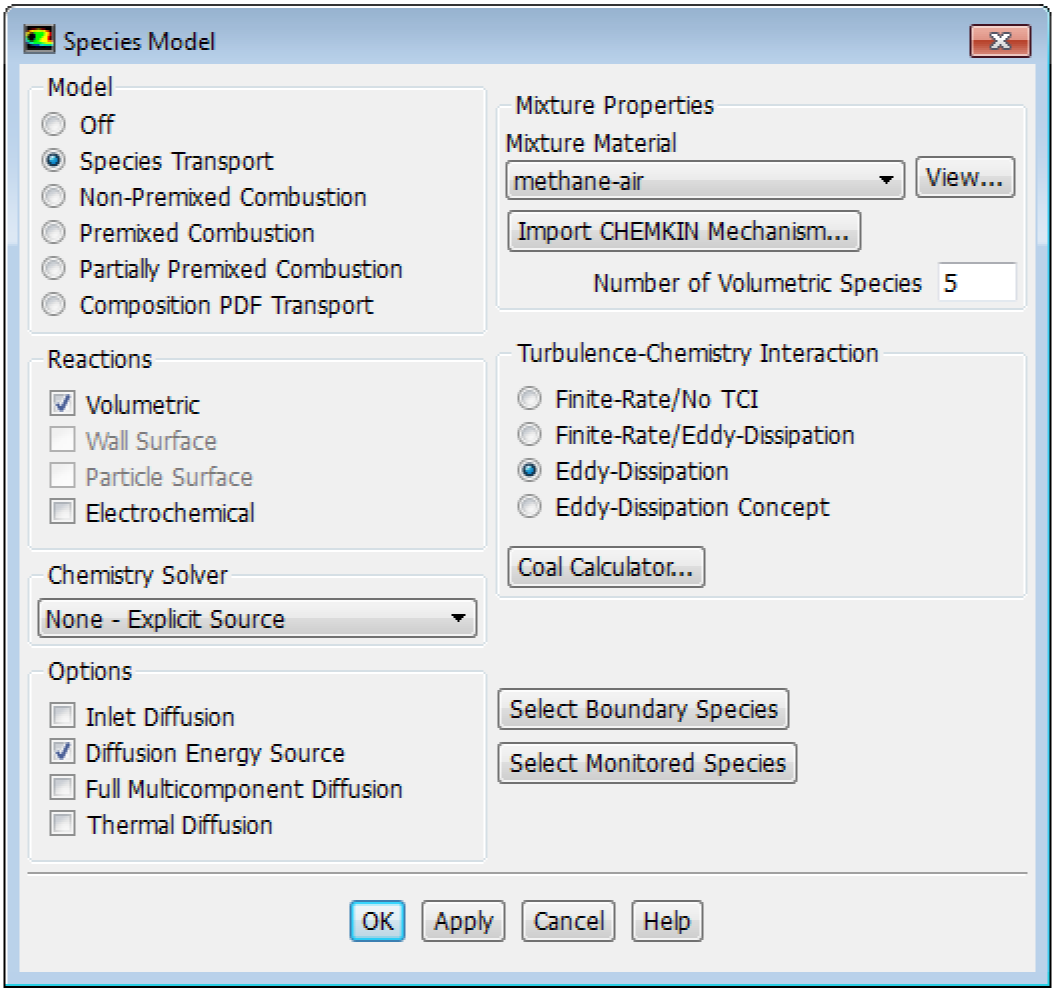
   *The **Viscous Model** dialog box will expand to provide further options for the **k-epsilon** model.*

b. Retain the default settings for the **k-epsilon** model.

c. Click **OK** to close the **Viscous Model** dialog box.

3. Enable chemical species transport and reaction.

   Setting Up Physics → Models → Species...



a. Select **Species Transport** in the **Model** list.

*The **Species Model** dialog box will expand to provide further options for the **Species Transport** model.*

b. Enable **Volumetric** in the **Reactions** group box.

c. Select **methane-air** from the **Mixture Material** drop-down list.

*Scroll down the list to find **methane-air**.*

Note

The **Mixture Material** list contains the set of chemical mixtures that exist in the ANSYS Fluent database. You can select one of the predefined mixtures to access a complete description of the reacting system. The chemical species in the system and their physical and thermodynamic properties are defined by your selection of the mixture material. You can alter the mixture material selection or modify the mixture material properties using the **Create/Edit Materials** dialog box (see Materials (p. 545)).

---

d. Select **Eddy-Dissipation** in the **Turbulence-Chemistry Interaction** group box.

The eddy-dissipation model computes the rate of reaction under the assumption that chemical kinetics are fast compared to the rate at which reactants are mixed by turbulent fluctuations (eddies).

e. Click **OK** to close the **Species Model** dialog box.

Prior to listing the properties that are required for the models you have enabled, ANSYS Fluent will display a warning about the symmetry zone in the console. You may have to scroll up to see this warning.

```
Warning: It appears that symmetry zone 5 should actually be an axis
          (it has faces with zero area projections).
          Unless you change the zone type from symmetry to axis,
          you may not be able to continue the solution without
          encountering floating point errors.
```

In the axisymmetric model, the boundary conditions should be such that the centerline is an axis type instead of a symmetry type. You will change the symmetry zone to an axis boundary in *Boundary Conditions (p. 548)*.

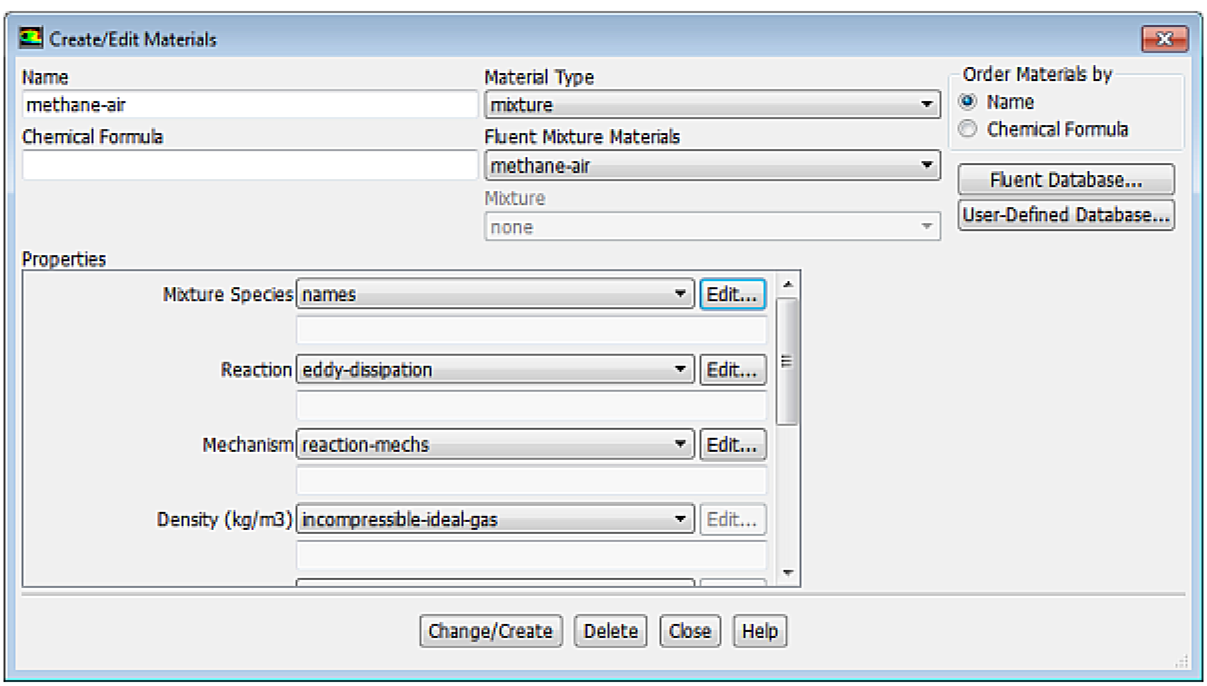
13.5.5. Materials

In this step, you will examine the default settings for the mixture material. This tutorial uses mixture properties copied from the Fluent Database. In general, you can modify these or create your own mixture properties for your specific problem as necessary.

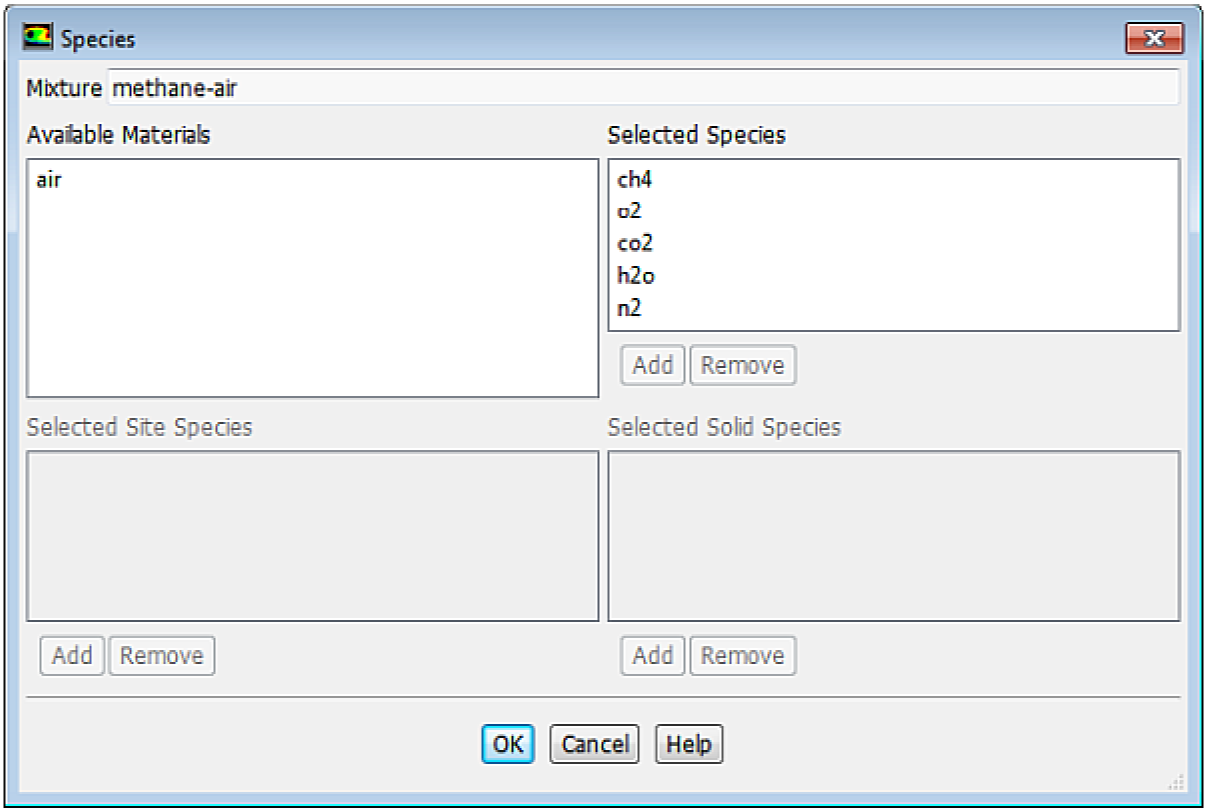
1. Confirm the properties for the mixture materials.

Setup → **Materials** → **Mixture** → **methane-air** **Edit...**

*The **Create/Edit Materials** dialog box will display the mixture material (**methane-air**) that was selected in the **Species Model** dialog box. The properties for this mixture material have been copied from the **Fluent Database...** and will be modified in the following steps.*



a. Click the **Edit...** button to the right of the **Mixture Species** drop-down list to open the **Species** dialog box.



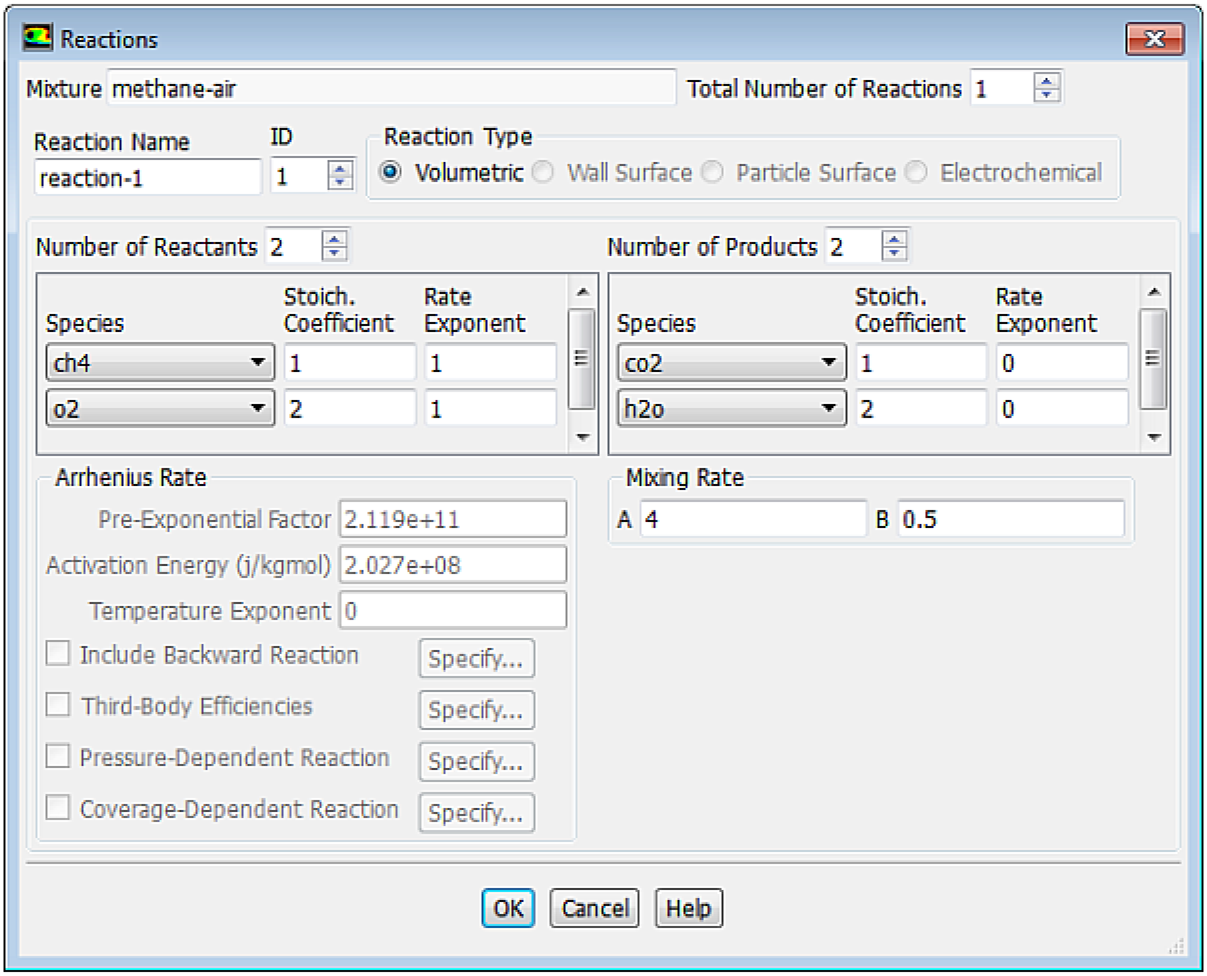
*You can add or remove species from the mixture material as necessary using the **Species** dialog box.*

i. Retain the default selections from the **Selected Species** selection list.

   The species that make up the methane-air mixture are predefined and require no modification.

ii. Click **OK** to close the **Species** dialog box.

b. Click the **Edit...** button to the right of the **Reaction** drop-down list to open the **Reactions** dialog box.



*The eddy-dissipation reaction model ignores chemical kinetics (the Arrhenius rate) and uses only the parameters in the **Mixing Rate** group box in the **Reactions** dialog box. The **Arrhenius Rate** group box will therefore be inactive. The values for **Rate Exponent** and **Arrhenius Rate** parameters are included in the database and are employed when the alternate finite-rate/eddy-dissipation model is used.*

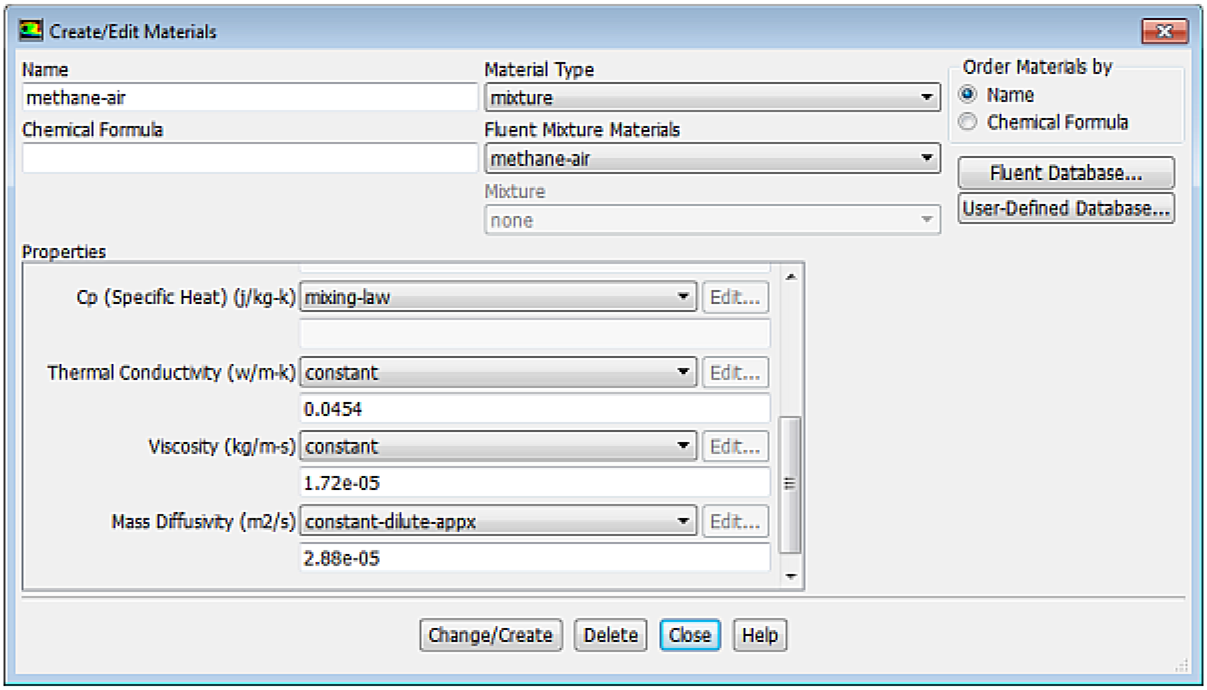
i. Retain the default values in the **Mixing Rate** group box.

ii. Click **OK** to close the **Reactions** dialog box.

c. Retain the selection of **incompressible-ideal-gas** from the **Density** drop-down list.

d. Retain the selection of **mixing-law** from the **Cp (Specific Heat)** drop-down list.

e. Retain the default values for **Thermal Conductivity**, **Viscosity**, and **Mass Diffusivity**.



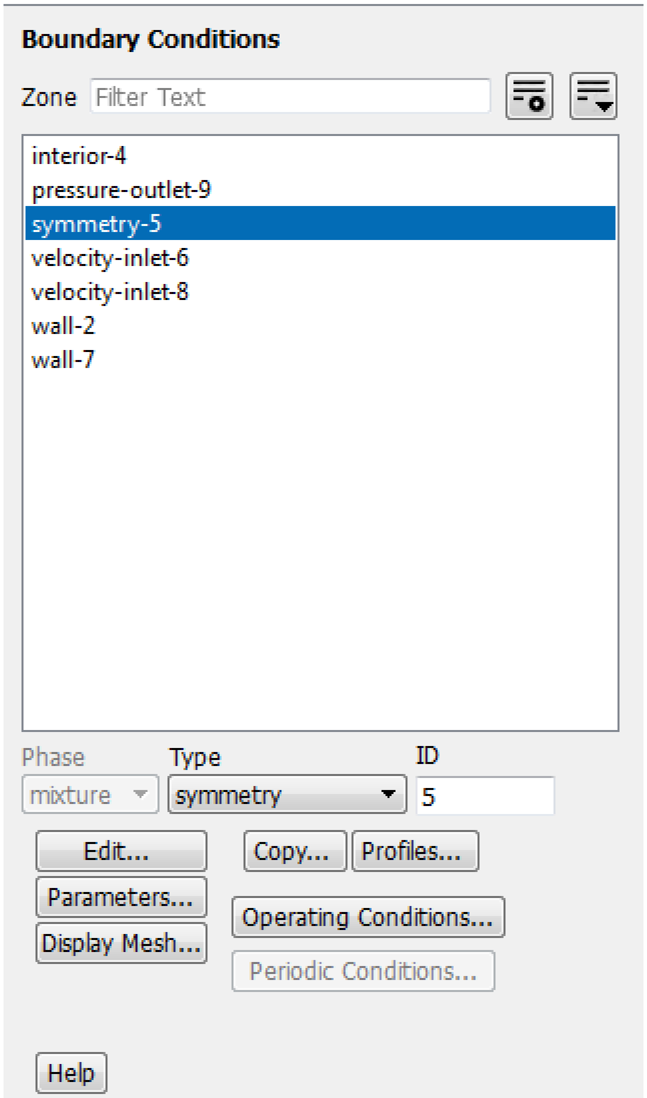
f. Click **Change/Create** to accept the material property settings.

g. Close the **Create/Edit Materials** dialog box.

The calculation will be performed assuming that all properties except density and specific heat are constant. The use of constant transport properties (viscosity, thermal conductivity, and mass diffusivity coefficients) is acceptable because the flow is fully turbulent. The molecular transport properties will play a minor role compared to turbulent transport.

13.5.6. Boundary Conditions

Setup → **Boundary Conditions**



1. Convert the symmetry zone to the axis type.

   Setup → **Boundary Conditions** → **symmetry-5**

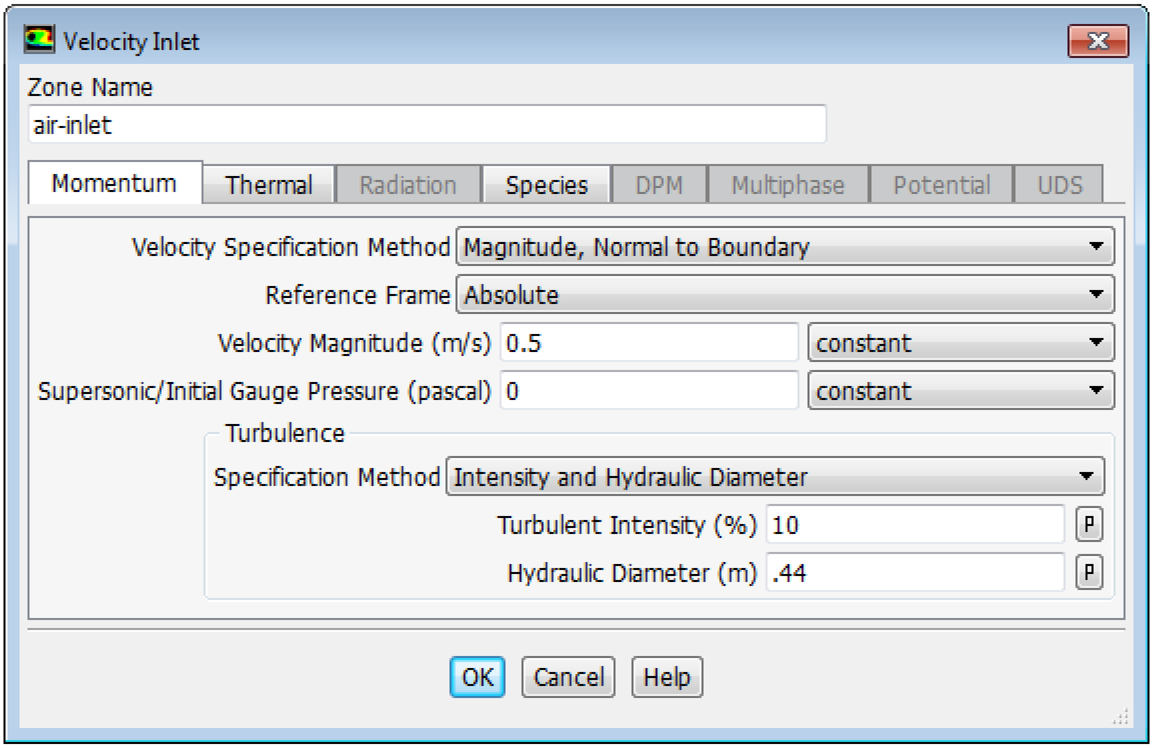
   The symmetry zone must be converted to an axis to prevent numerical difficulties where the radius reduces to zero.

   a. Select **axis** from the **Type** drop-down list.

2. Set the boundary conditions for the air inlet (**velocity-inlet-8**).

   Setup → **Boundary Conditions** → **velocity-inlet-8** **Edit...**

   *To determine the zone for the air inlet, display the mesh without the fluid zone to see the boundaries. Use the right mouse button to probe the air inlet. ANSYS Fluent will report the zone name (**velocity-inlet-8**) in the console.*



a. Enter `air-inlet` for **Zone Name**.

   This name is more descriptive for the zone than ***velocity-inlet-8***.

b. Enter `0.5` m/s for **Velocity Magnitude**.

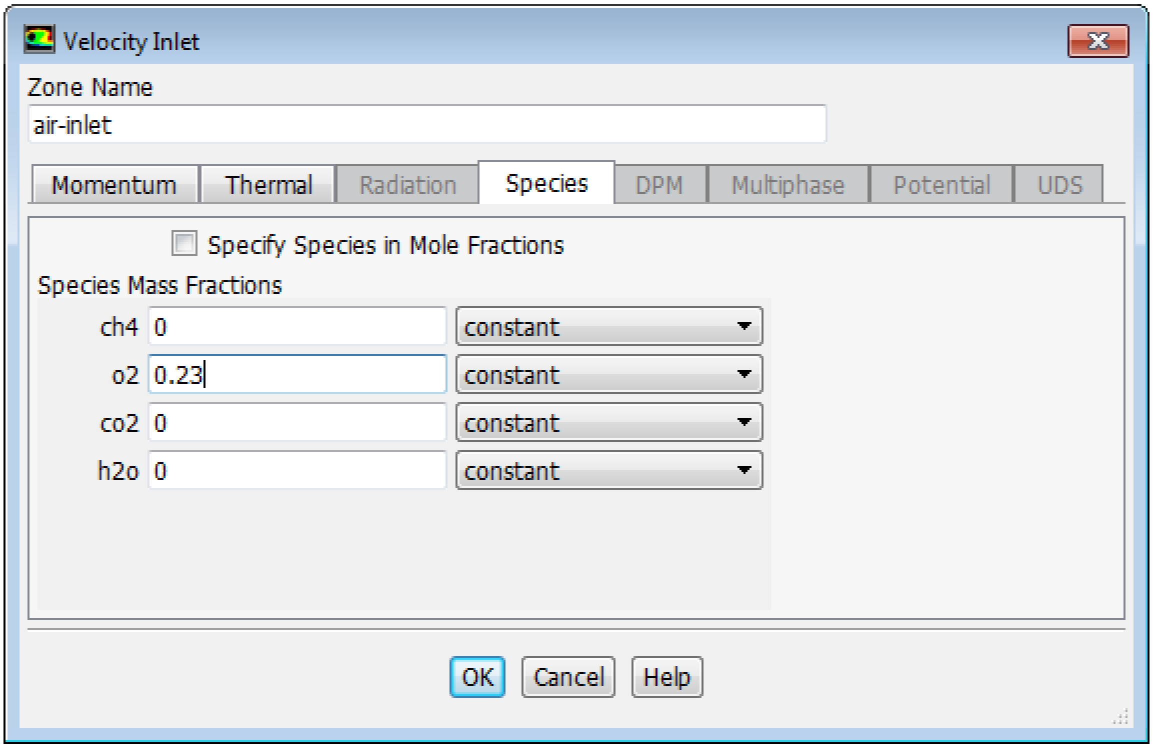
c. Select **Intensity and Hydraulic Diameter** from the **Specification Method** drop-down list in the **Turbulence** group box.

d. Enter `10` $\%$ for **Turbulent Intensity**.

e. Enter `0.44` m for **Hydraulic Diameter**.

f. Click the **Thermal** tab and retain the default value of `300` K for **Temperature**.

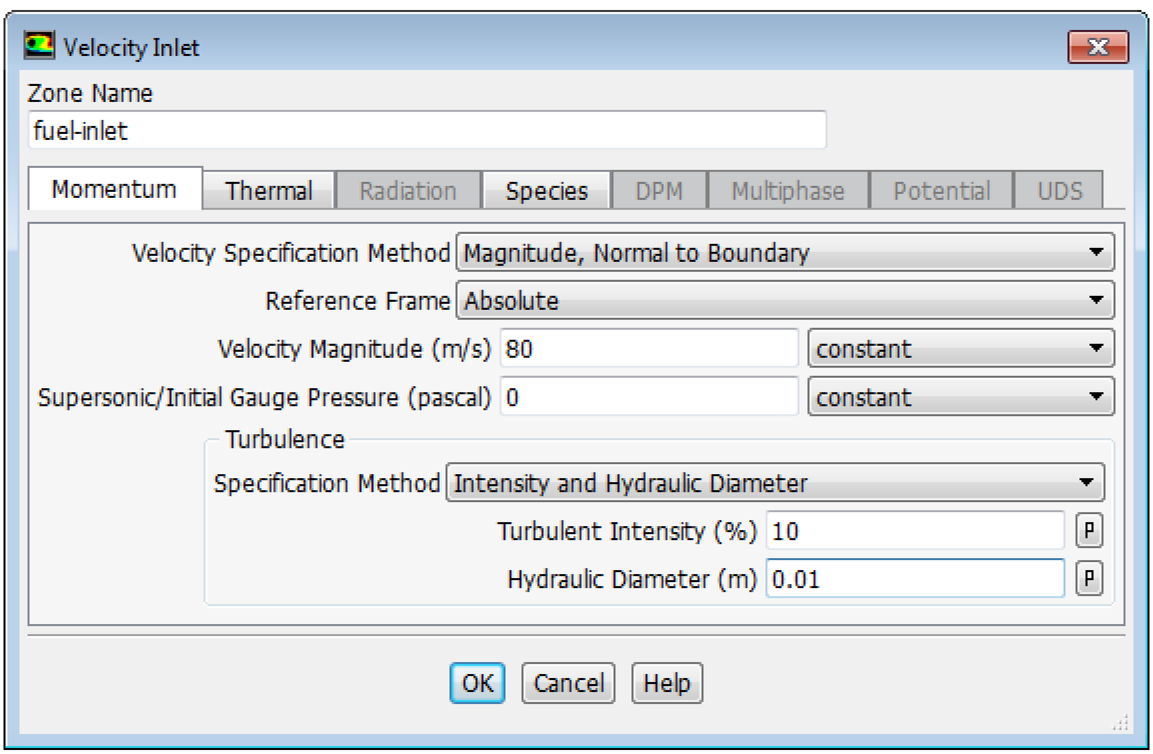
g. Click the **Species** tab and enter `0.23` for **o2** in the **Species Mass Fractions** group box.



h. Click **OK** to close the **Velocity Inlet** dialog box.

3. Set the boundary conditions for the fuel inlet (**velocity-inlet-6**).

Setup → **Boundary Conditions** → **velocity-inlet-6** **Edit...**



a. Enter `fuel-inlet` for **Zone Name**.

*This name is more descriptive for the zone than **velocity-inlet-6**.*

b. Enter 80 m/s for the **Velocity Magnitude**.

c. Select **Intensity and Hydraulic Diameter** from the **Specification Method** drop-down list in the **Turbulence** group box.

d. Enter 10 $\%$ for **Turbulent Intensity**.

e. Enter 0.01 m for **Hydraulic Diameter**.

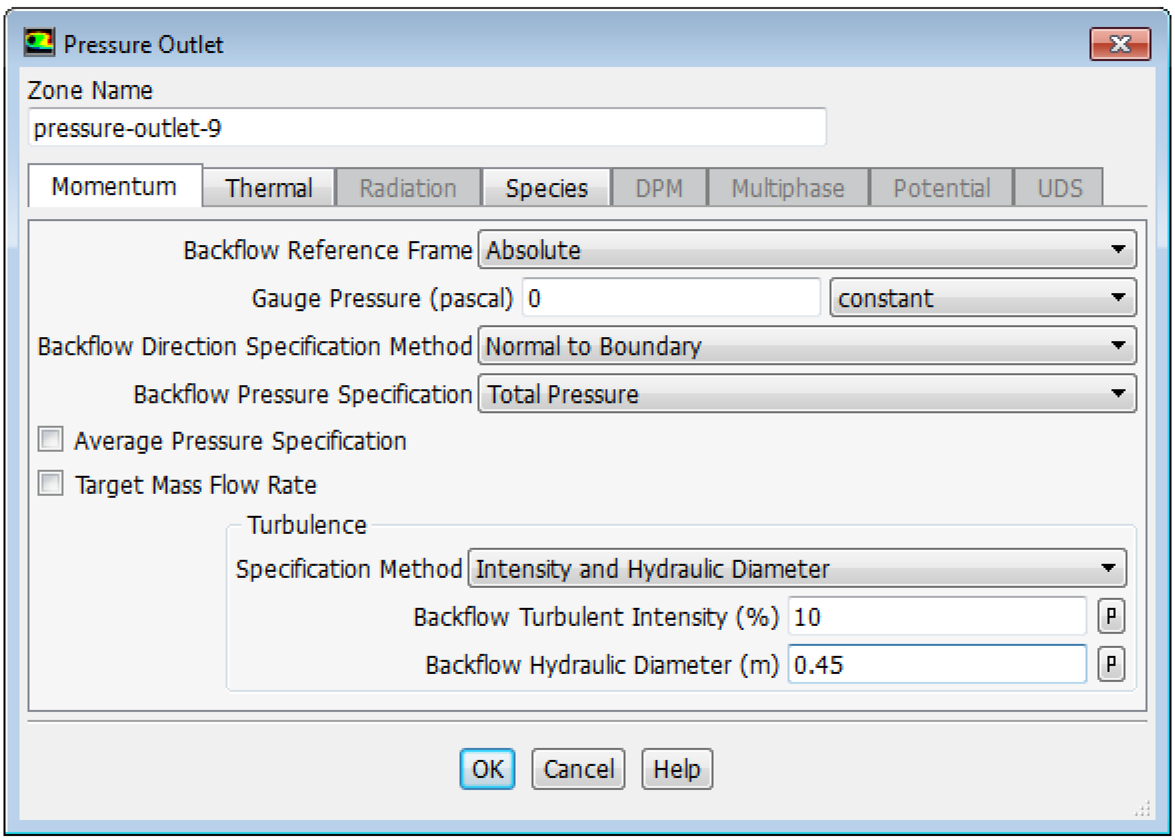
f. Click the **Thermal** tab and retain the default value of 300 K for **Temperature**.

g. Click the **Species** tab and enter 1 for **ch4** in the **Species Mass Fractions** group box.

h. Click **OK** to close the **Velocity Inlet** dialog box.

4. Set the boundary conditions for the exit boundary (**pressure-outlet-9**).

Setup → **Boundary Conditions** → **pressure-outlet-9** **Edit...**



a. Retain the default value of 0 Pa for **Gauge Pressure**.

b. Select **Intensity and Hydraulic Diameter** from the **Specification Method** drop-down list in the **Turbulence** group box.

c. Enter 10 $\%$ for **Backflow Turbulent Intensity**.

d. Enter 0.45 m for **Backflow Hydraulic Diameter**.

e. Click the **Thermal** tab and retain the default value of 300 K for **Backflow Total Temperature**.

f. Click the **Species** tab and enter 0.23 for **o2** in the **Species Mass Fractions** group box.

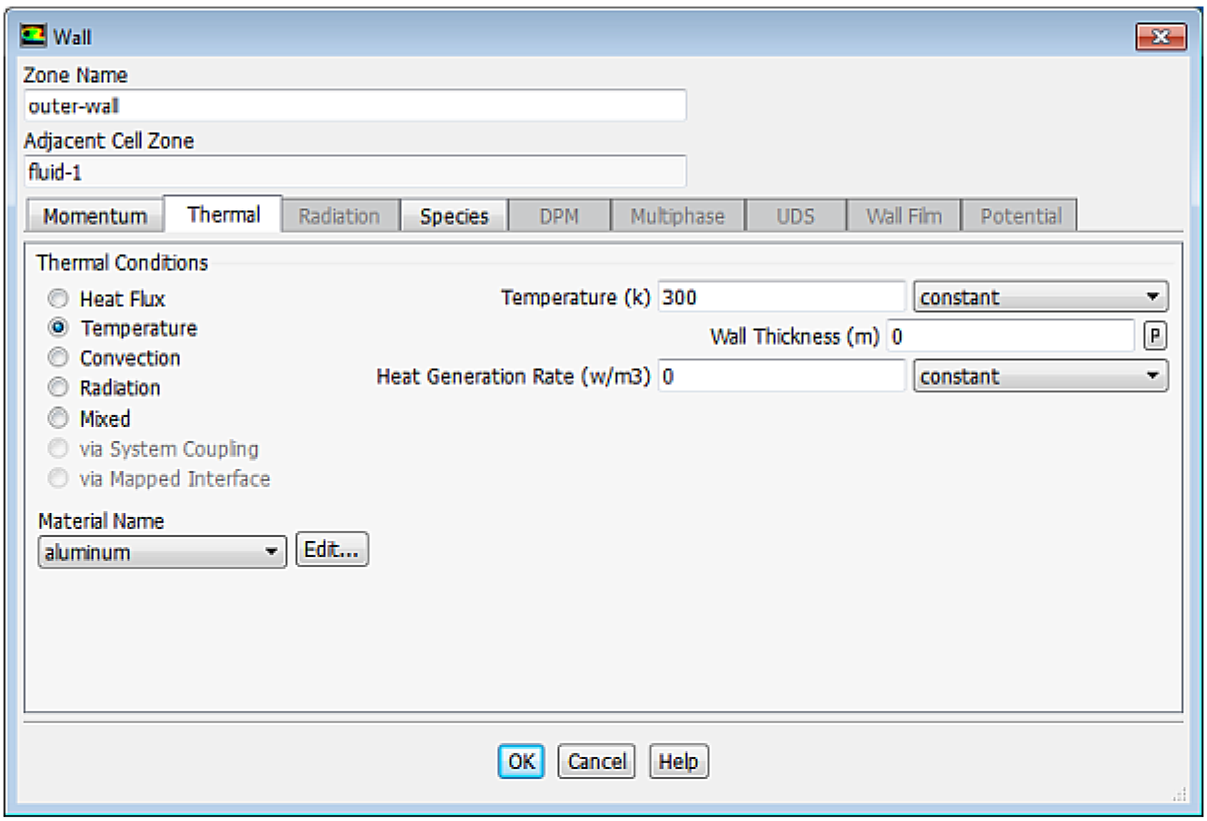
g. Click **OK** to close the **Pressure Outlet** dialog box.

*The **Backflow** values in the **Pressure Outlet** dialog box are utilized only when backflow occurs at the pressure outlet. Always assign reasonable values because backflow may occur during intermediate iterations and could affect the solution stability.*

5. Set the boundary conditions for the outer wall (**wall-7**).

Setup → **Boundary Conditions** → **wall-7** **Edit...**

Use the mouse-probe method described for the air inlet to determine the zone corresponding to the outer wall.



a. Enter outer-wall for **Zone Name**.

*This name is more descriptive for the zone than **wall-7**.*

b. Click the **Thermal** tab.

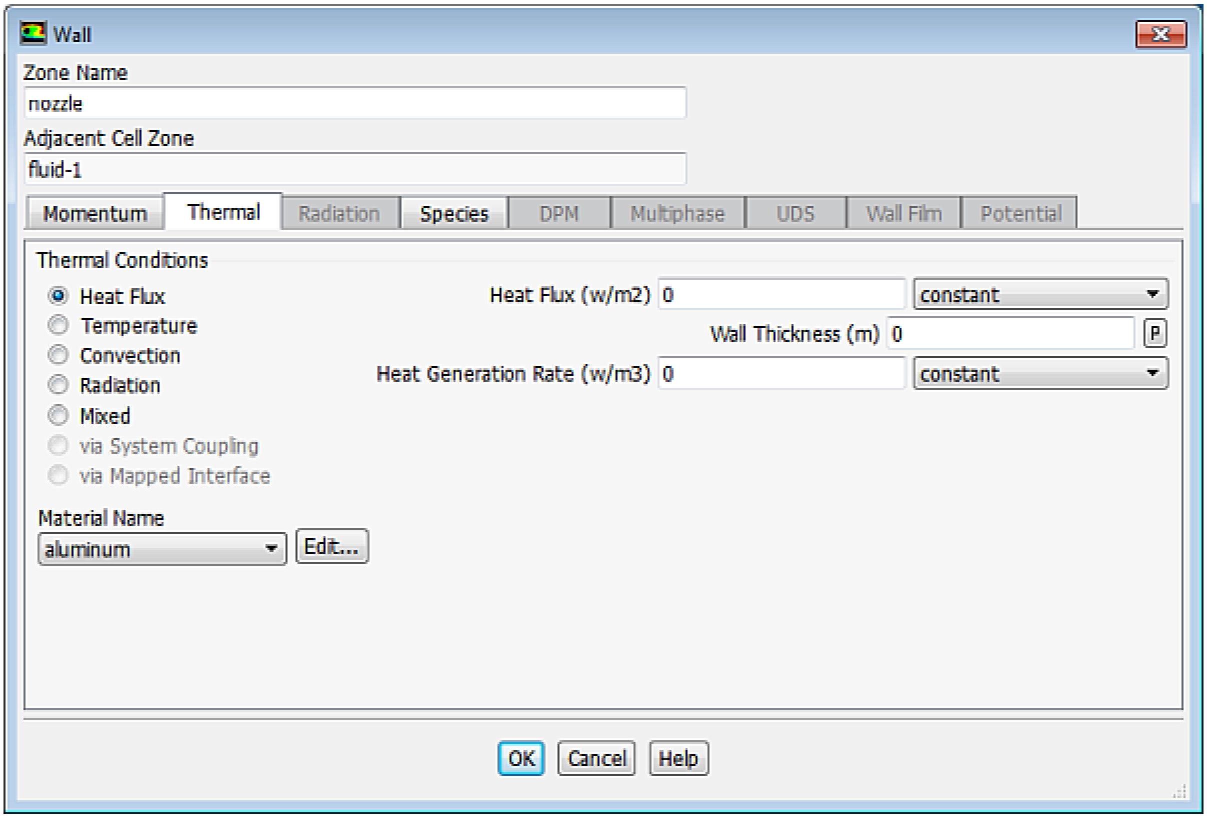
i. Select **Temperature** in the **Thermal Conditions** list.

ii. Retain the default value of 300 K for **Temperature**.

c. Click **OK** to close the **Wall** dialog box.

6. Set the boundary conditions for the fuel inlet nozzle (**wall-2**).

Setup → **Boundary Conditions** → **wall-2** **Edit...**



a. Enter `nozzle` for **Zone Name**.

*This name is more descriptive for the zone than **wall-2**.*

b. Click the **Thermal** tab.

i. Retain the default selection of **Heat Flux** in the **Thermal Conditions** list.

ii. Retain the default value of `0` W/m^2 for **Heat Flux**, so that the wall is adiabatic.

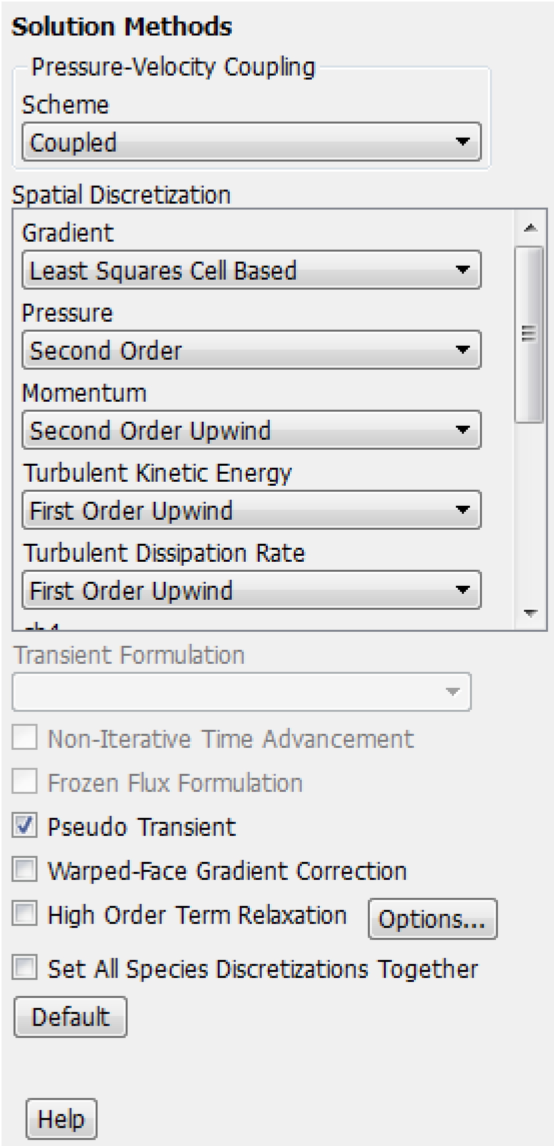
c. Click **OK** to close the **Wall** dialog box.

13.5.7. Initial Reaction Solution

You will first calculate a solution for the basic reacting flow neglecting pollutant formation. In a later step, you will perform an additional analysis to simulate NOx.

1. Select the Coupled Pseudo Transient solution method.

Solving → **Solution** → **Methods...**



a. Select **Coupled** from the **Scheme** drop-down list in the **Pressure-Velocity Coupling** group box.

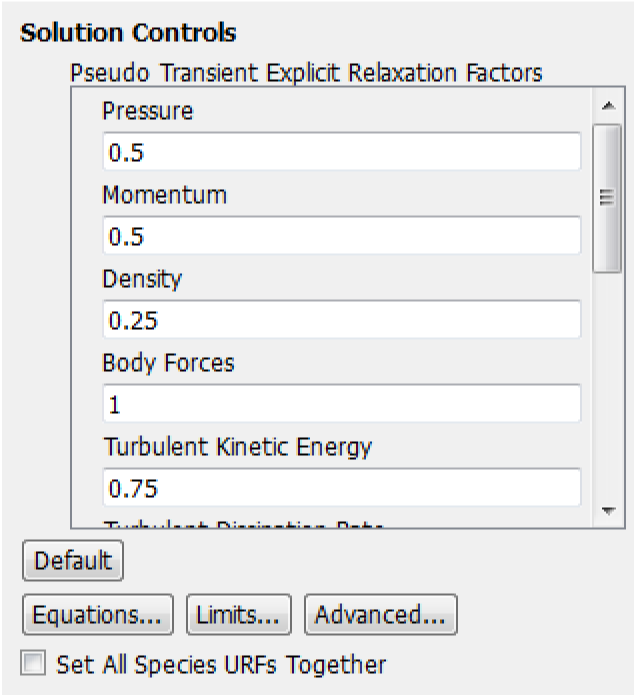
b. Retain the default selections in the **Spatial Discretization** group box.

c. Enable **Pseudo Transient**.

The Pseudo Transient option enables the pseudo transient algorithm in the coupled pressure-based solver. This algorithm effectively adds an unsteady term to the solution equations in order to improve stability and convergence behavior. Use of this option is recommended for general fluid flow problems.

2. Modify the solution controls.

Solving → **Controls** → **Controls...**

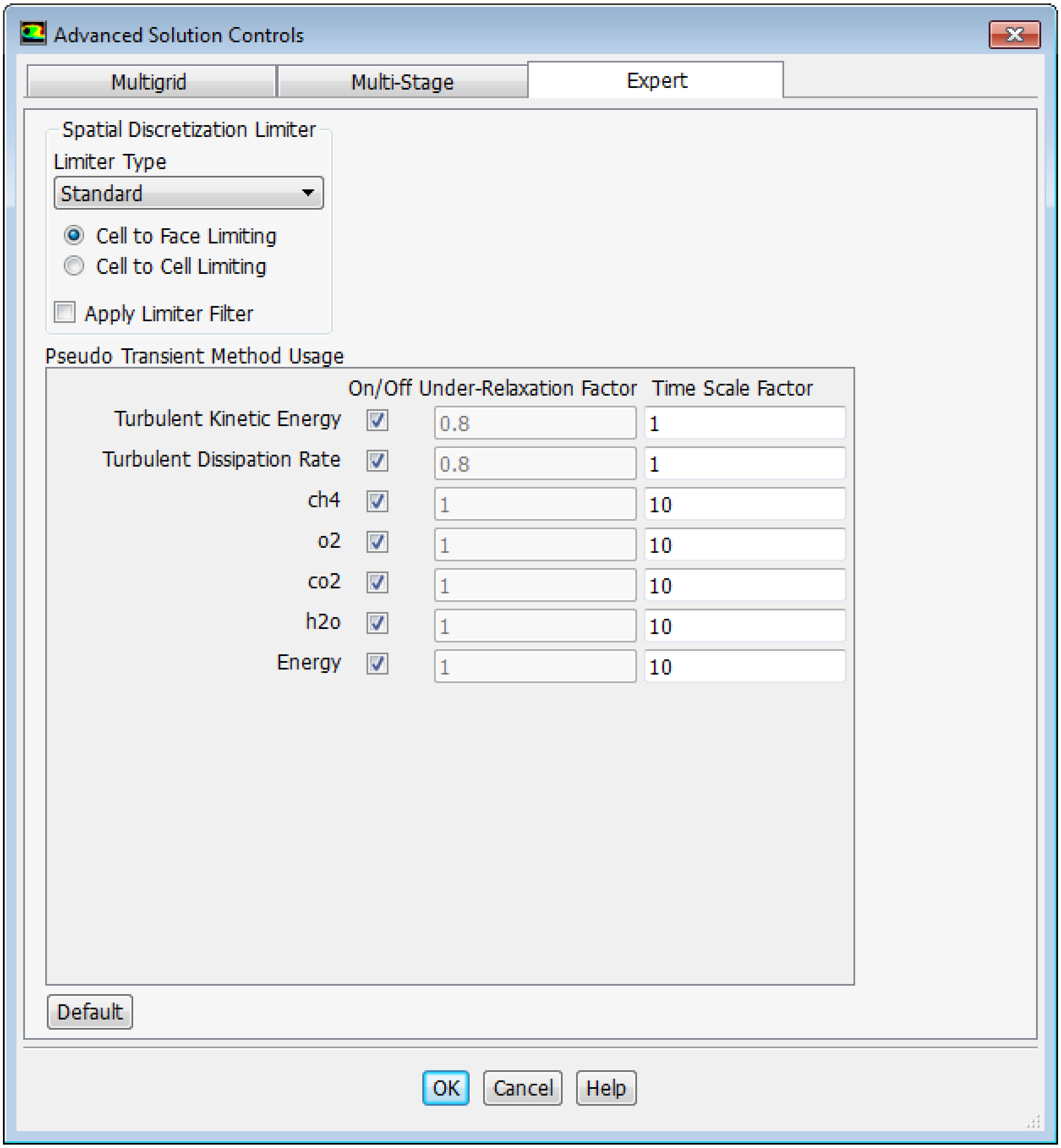


a. Enter `0.25` under **Density** in the **Pseudo Transient Explicit Relaxation Factors** group box.

   The default explicit relaxation parameters in ANSYS Fluent are appropriate for a wide range of general fluid flow problems. However, in some cases it may be necessary to reduce the relaxation factors to stabilize the solution. Some experimentation is typically necessary to establish the optimal values. For this tutorial, it is sufficient to reduce the density explicit relaxation factor to `0.25` for stability.

b. Click **Advanced...** to open the **Advanced Solution Controls** dialog box and select the **Expert** tab.

   The Expert tab in the Advanced Solution Controls dialog box allows you to individually specify the solution method and Pseudo Transient Time Scale Factors for each equation, except for the flow equations. When using the Pseudo Transient method for general reacting flow cases, increasing the species and energy time scales is recommended.



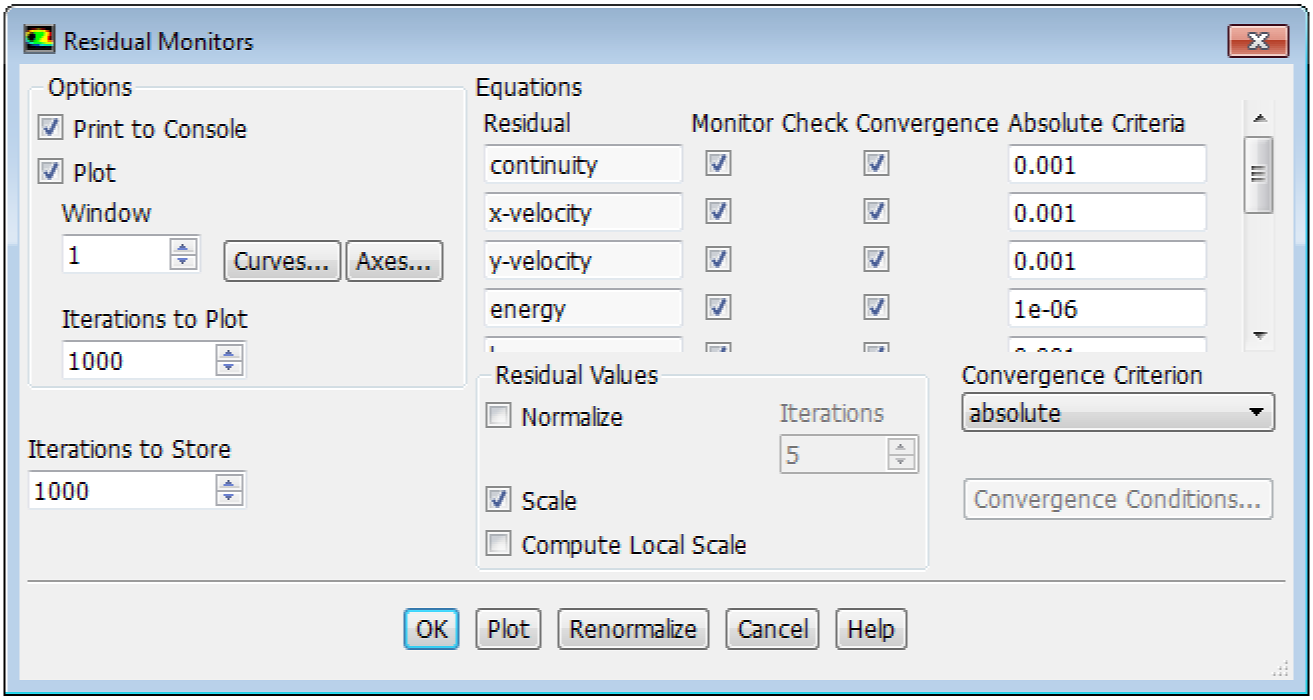
i. Enable the pseudo-transient method for **ch4**, **o2**, **co2**, **h2o**, and **Energy** in the **Expert** tab, by selecting each one under **On/Off**.

ii. Enter `10` for the **Time Scale Factor** for **ch4**, **o2**, **co2**, **h2o**, and **Energy**.

iii. Click **OK** to close the **Advanced Solution Controls** dialog box.

3. Ensure the plotting of residuals during the calculation.

Solving → **Reports** → **Residuals...**

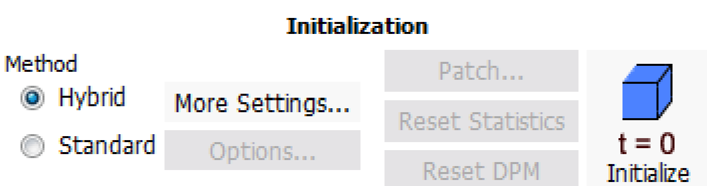


a. Ensure that **Plot** is enabled in the **Options** group box.

b. Click **OK** to close the **Residual Monitors** dialog box.

4. Initialize the field variables.

Solving → **Initialization**



a. Retain the default **Hybrid** initialization method and click **Initialize** to initialize the variables.

5. Save the case file (`gascomb1.cas.gz`).

File → **Write** → **Case...**

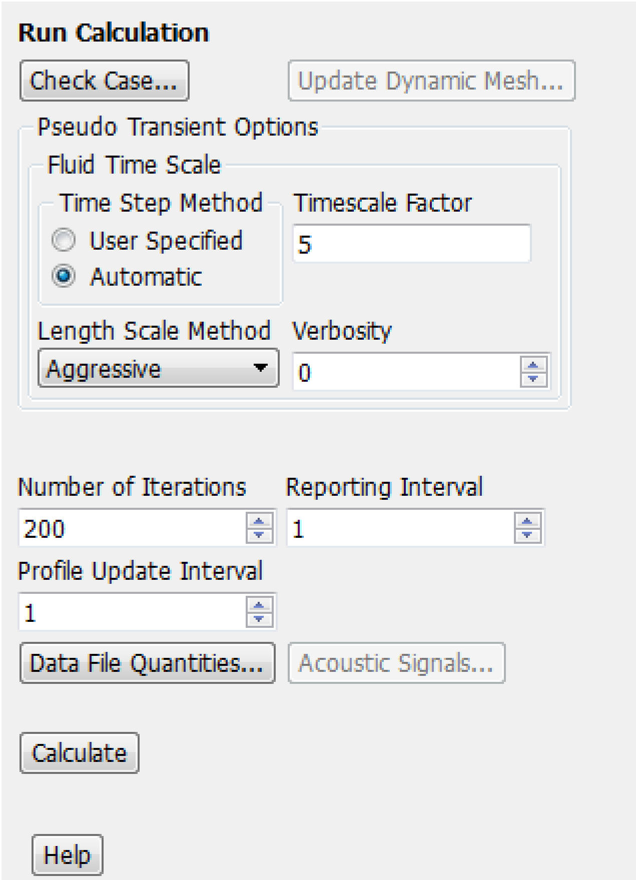
a. Enter `gascomb1.cas.gz` for **Case File**.

b. Ensure that **Write Binary Files** is enabled to produce a smaller, unformatted binary file.

c. Click **OK** to close the **Select File** dialog box.

6. Run the calculation by requesting 200 iterations.

Solving → **Run Calculation** → **Advanced...**



a. Select **Aggressive** from the **Length Scale Method** drop-down list.

   When using the Automatic Time Step Method ANSYS Fluent computes the Pseudo Transient time step based on characteristic length and velocity scales of the problem. The Conservative Length Scale Method uses the smaller of two computed length scales emphasizing solution stability. The Aggressive Length Scale Method uses the larger of the two which may provide faster convergence in some cases.

b. Enter 5 for the **Timescale Factor**.

   The Timescale Factor allows you to further manipulate the computed Time Step calculated by ANSYS Fluent. Larger time steps can lead to faster convergence. However, if the time step is too large it can lead to solution instability.

c. Enter `200` for **Number of Iterations**.

d. Click **Calculate**.

The solution will converge after approximately 160 iterations.

7. Save the case and data files (`gascomb1.cas.gz` and `gascomb1.dat.gz`).

File → Write → Case & Data...

Note

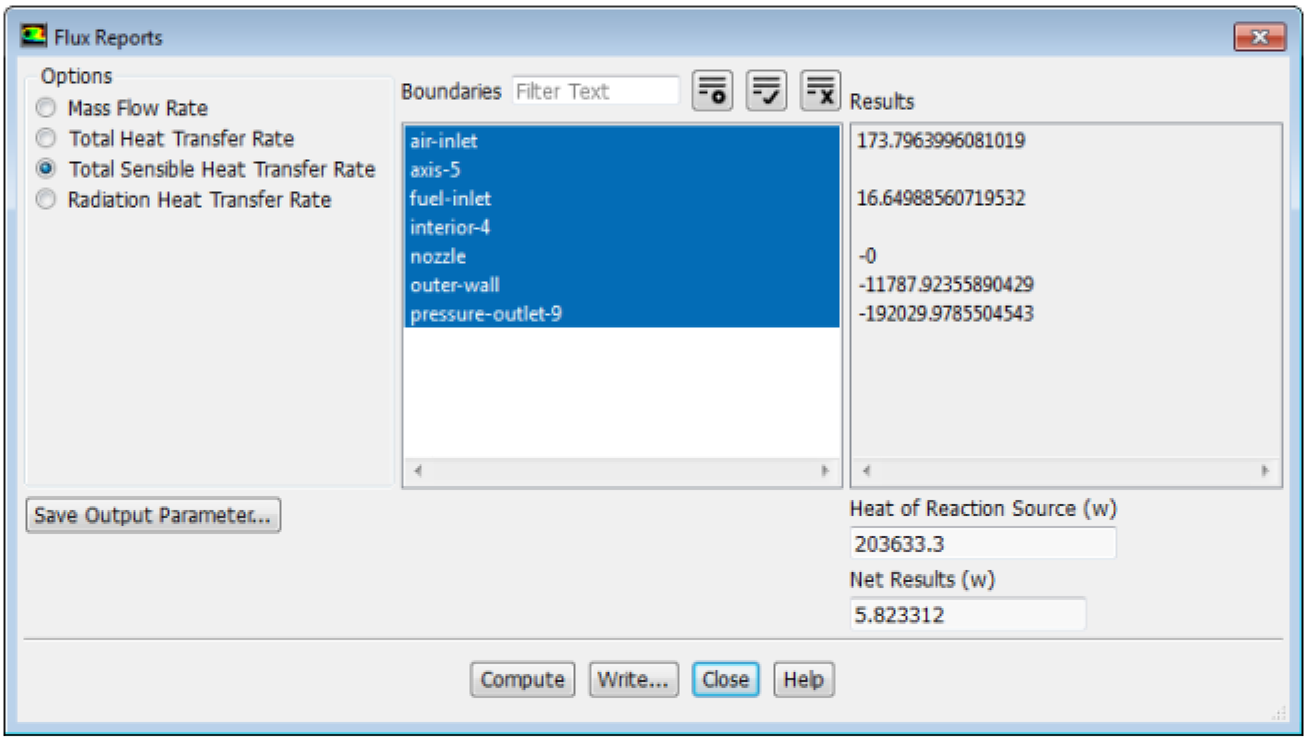
If you choose a file name that already exists in the current folder, ANSYS Fluent will ask you to confirm that the previous file is to be overwritten.

13.5.8. Postprocessing

Review the solution by examining graphical displays of the results and performing surface integrations at the combustor exit.

1. Report the total sensible heat flux.

   Postprocessing → Reports → Fluxes...



   a. Select **Total Sensible Heat Transfer Rate** in the **Options** list.

   b. Select all the boundaries from the **Boundaries** selection list (you can click the select-all button ( )).

   c. Click **Compute** and close the **Flux Reports** dialog box.

Note

The energy balance is good because the net result is small compared to the heat of reaction.

---

2. Display filled contours of temperature (Figure 13.3: Contours of Temperature (p. 561)).

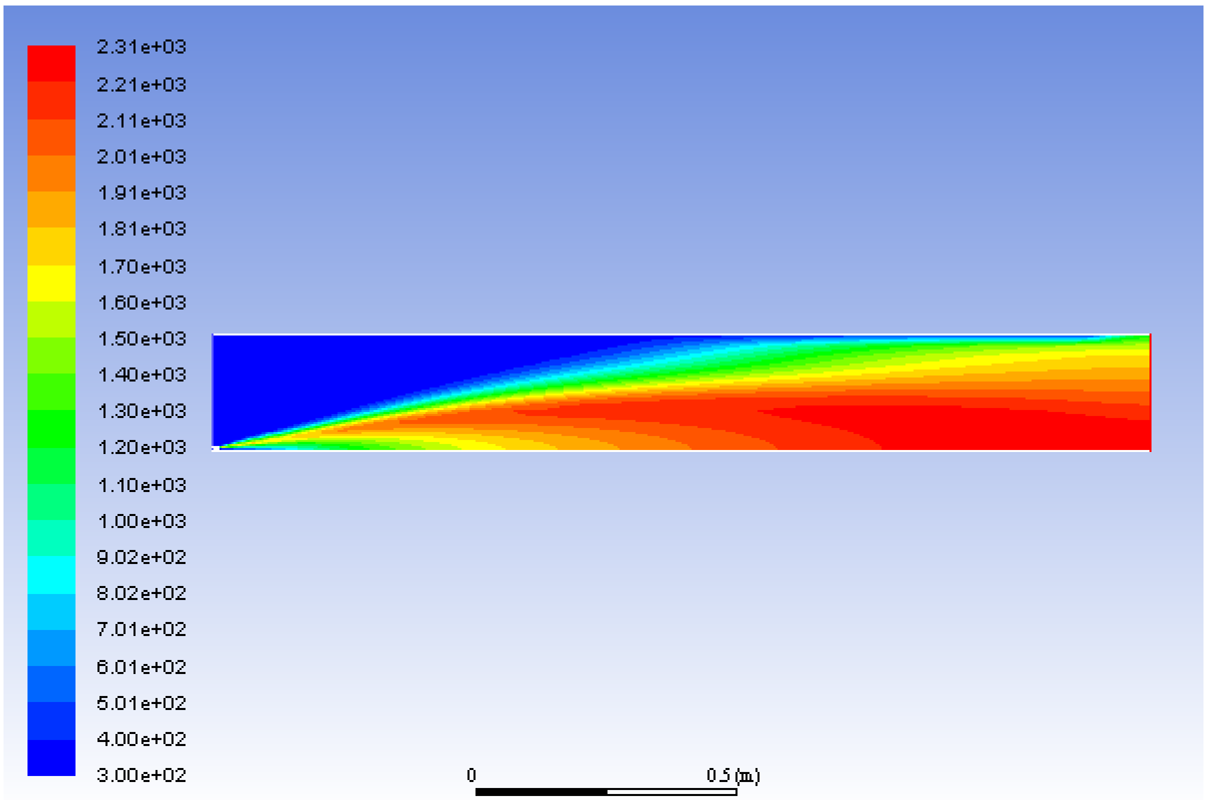
   Postprocessing → **Graphics** → **Contours** → **Edit...**

   a. Ensure that **Filled** is enabled in the **Options** group box.

   b. Select **Temperature...** and **Static Temperature** in the **Contours of** drop-down lists.

   c. Click **Display**.

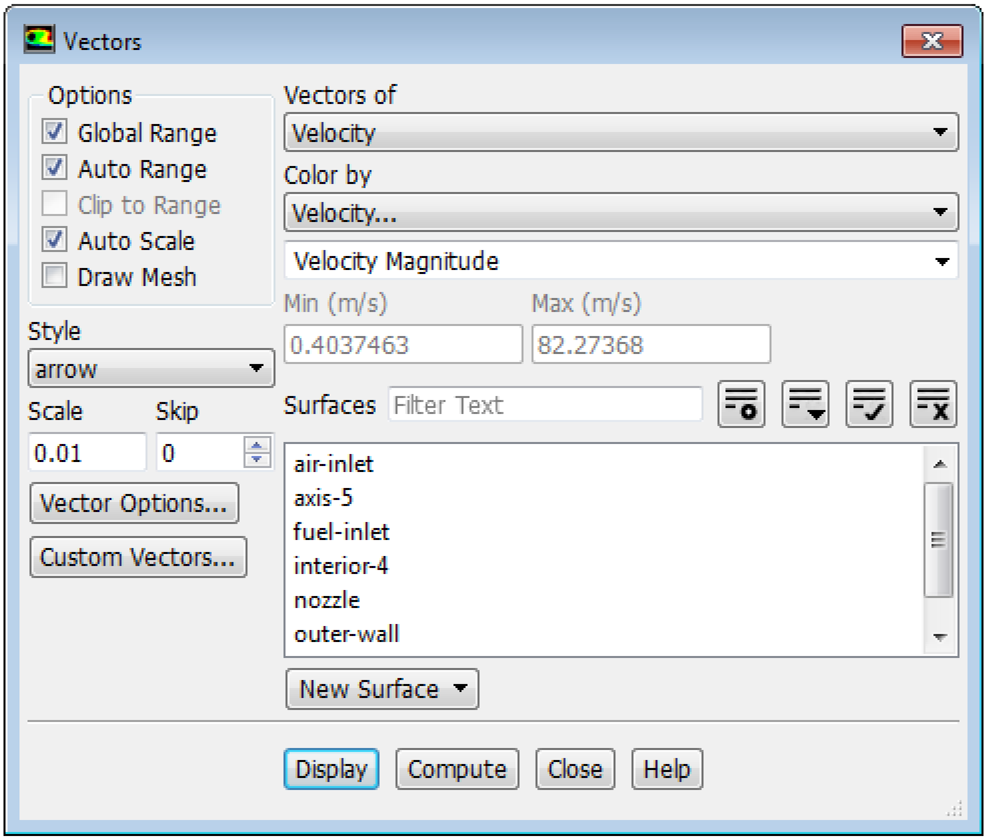
   Figure 13.3: Contours of Temperature



   The peak temperature is approximately 2310 K.

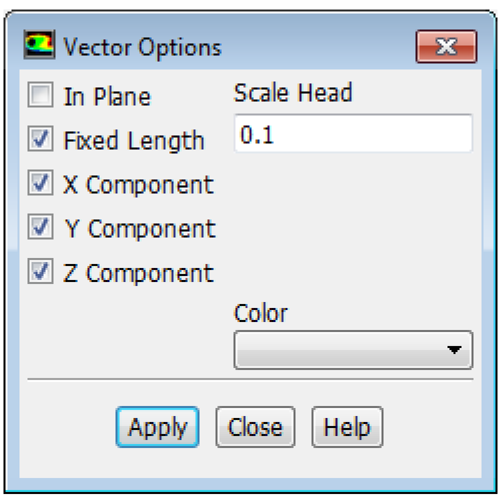
3. Display velocity vectors (Figure 13.4: Velocity Vectors (p. 563)).

   Postprocessing → **Graphics** → **Vectors** → **Edit...**



a. Enter `0.01` for **Scale**.

b. Click the **Vector Options...** button to open the **Vector Options** dialog box.



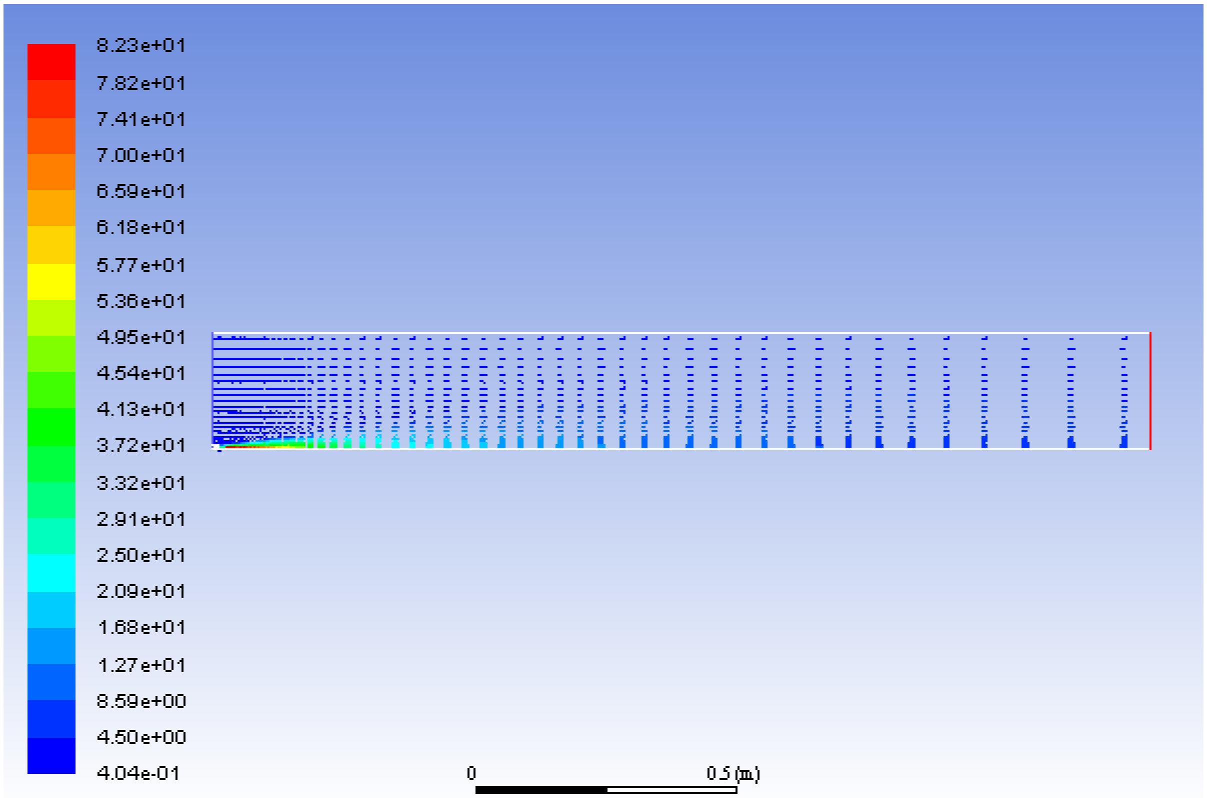
i. Enable **Fixed Length**.

The fixed length option is useful when the vector magnitude varies dramatically. With fixed length vectors, the velocity magnitude is described only by color instead of by both vector length and color.

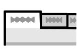
ii. Click **Apply** and close the **Vector Options** dialog box.

c. Click **Display** and close the **Vectors** dialog box.

Figure 13.4: Velocity Vectors



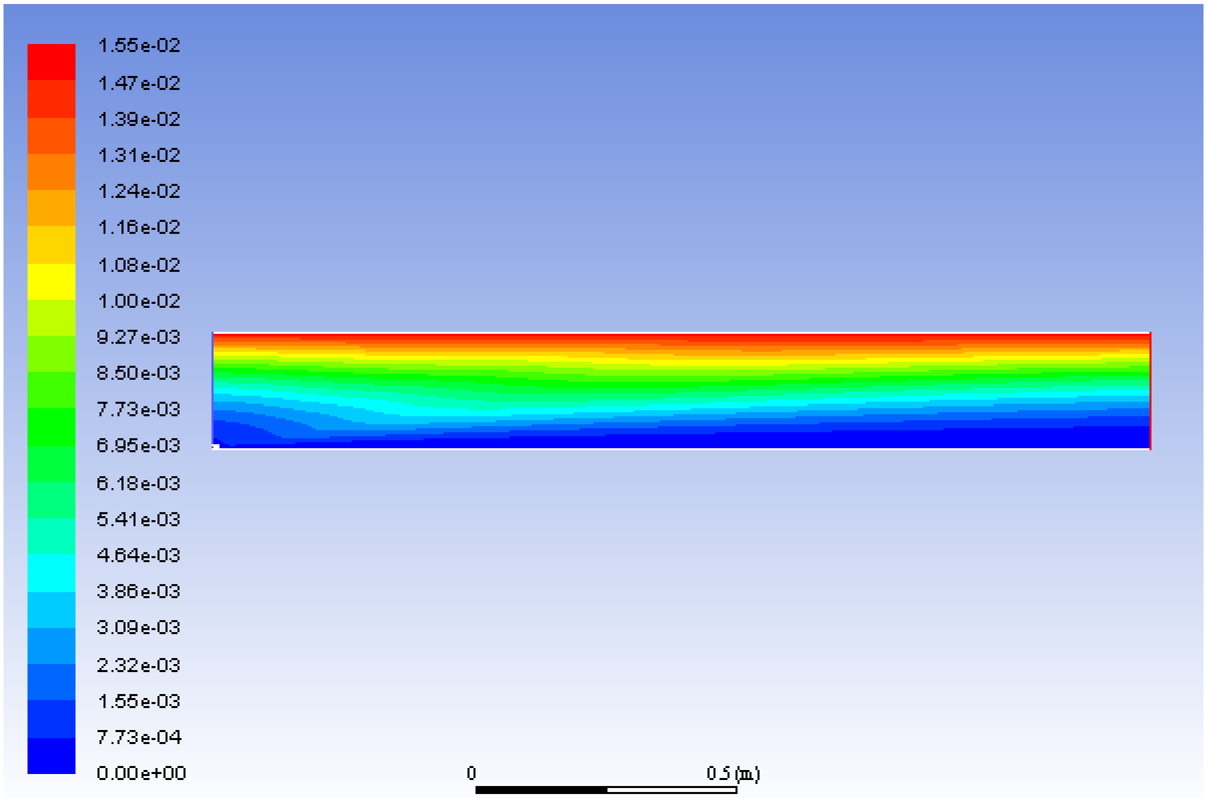
4. Display filled contours of stream function (Figure 13.5: Contours of Stream Function (p. 564)).

 **Postprocessing** → **Graphics** → **Contours** → **Edit...**

a. Select **Velocity...** and **Stream Function** from the **Contours of** drop-down lists.

b. Click **Display**.

Figure 13.5: Contours of Stream Function



The entrainment of air into the high-velocity methane jet is clearly visible in the streamline display.

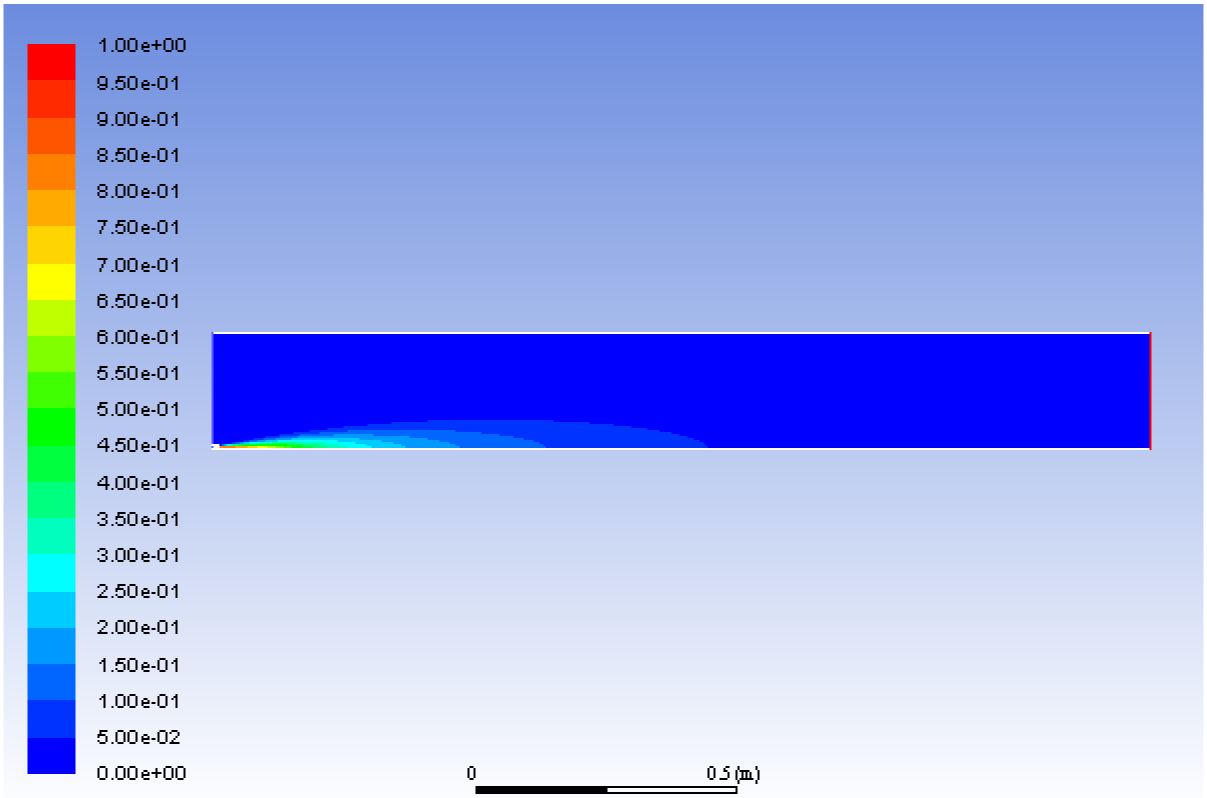
5. Display filled contours of mass fraction for CH_4 (Figure 13.6: Contours of CH4 Mass Fraction (p. 565)).

 **Postprocessing** → **Graphics** → **Contours** → **Edit...**

   a. Select **Species...** and **Mass fraction of ch4** from the **Contours of** drop-down lists.

   b. Click **Display**.

Figure 13.6: Contours of CH4 Mass Fraction



6. In a similar manner, display the contours of mass fraction for the remaining species O_2, CO_2, and H_2O (Figure 13.7: Contours of O2 Mass Fraction (p. 566), Figure 13.8: Contours of CO2 Mass Fraction (p. 566), and Figure 13.9: Contours of H2O Mass Fraction (p. 567)) Close the **Contours** dialog box when all of the species have been displayed.

Figure 13.7: Contours of O2 Mass Fraction

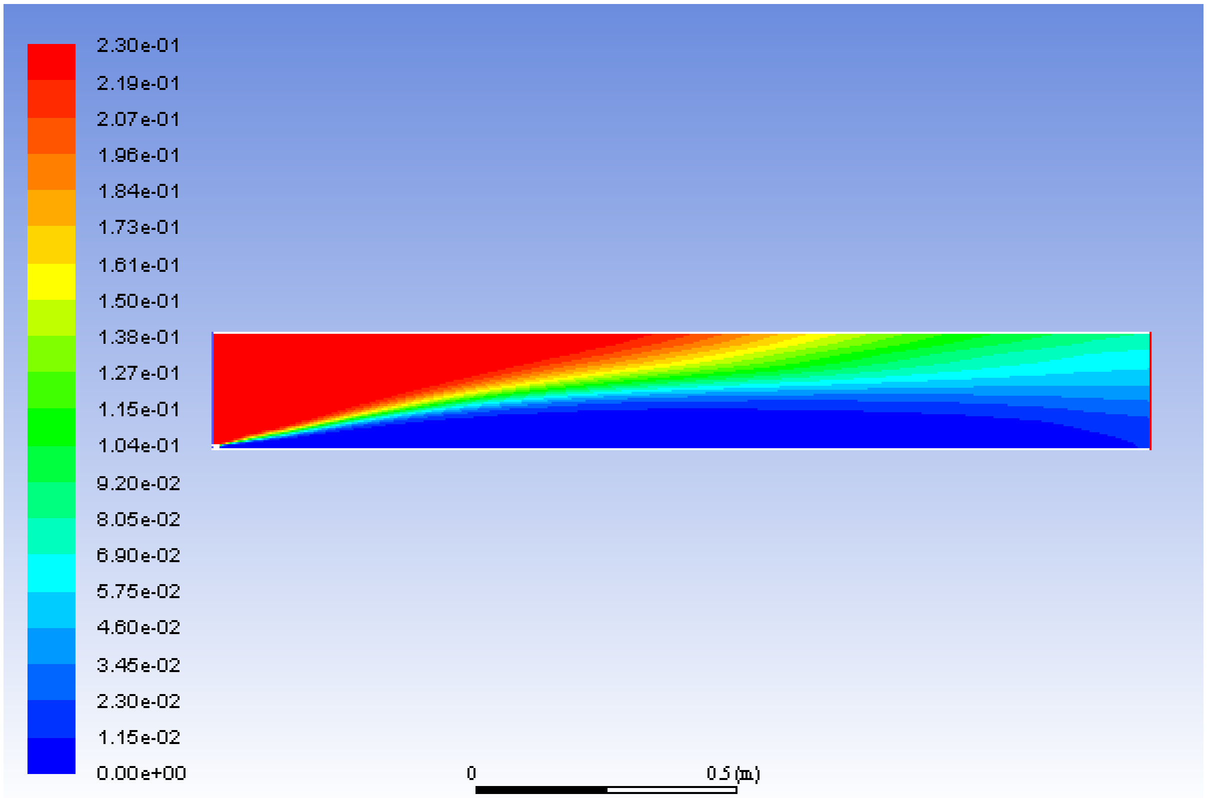


Figure 13.8: Contours of CO2 Mass Fraction

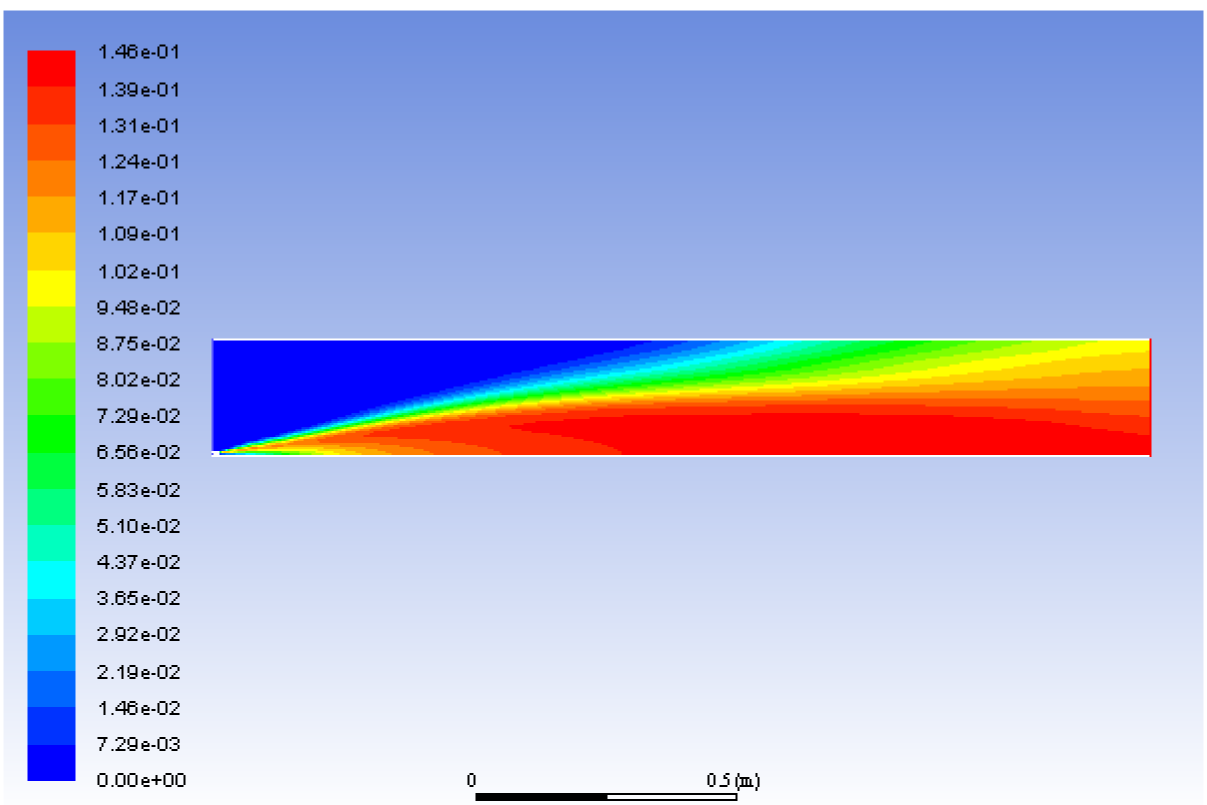
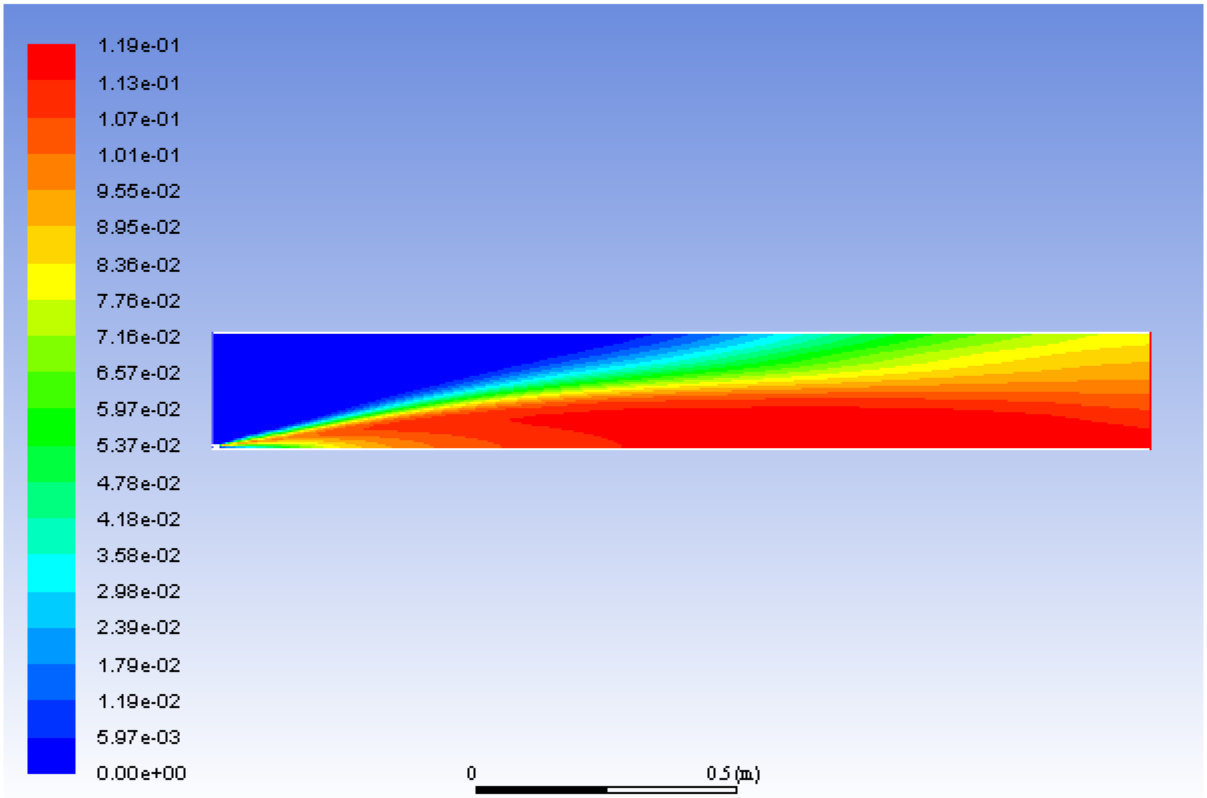
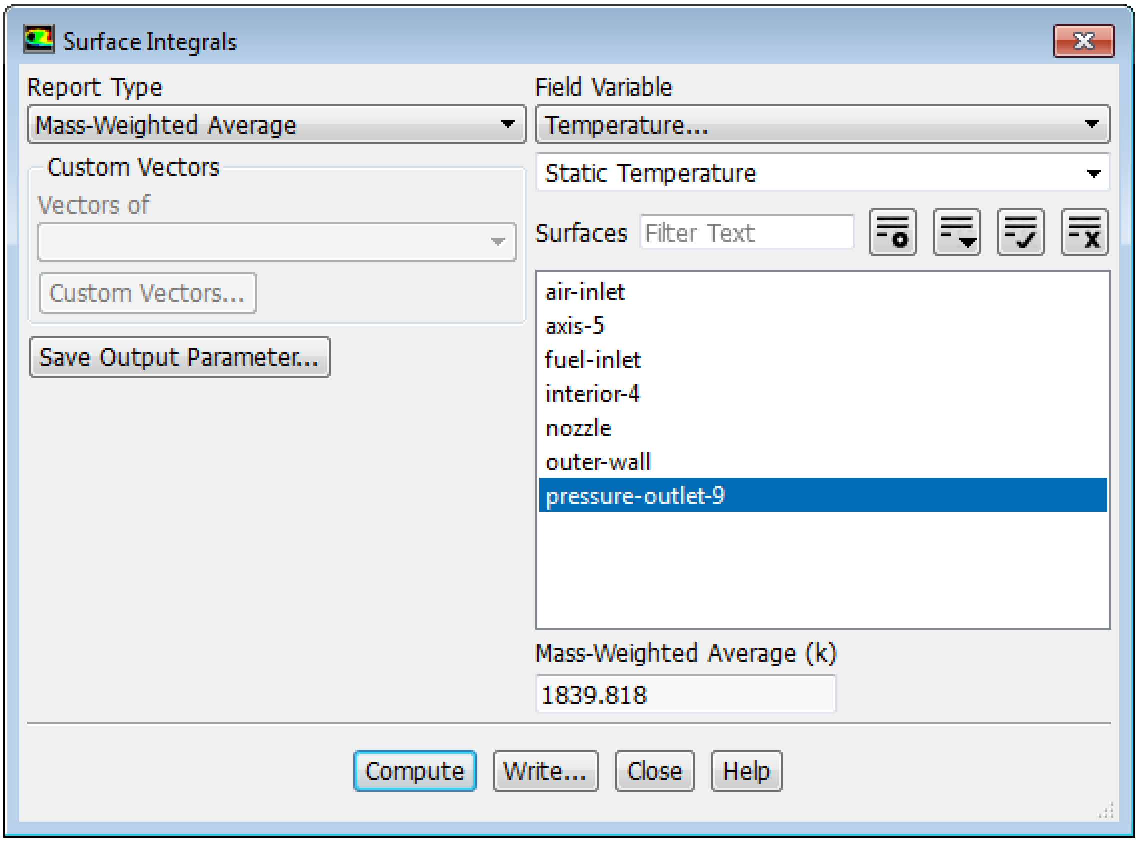


Figure 13.9: Contours of H2O Mass Fraction



7. Determine the average exit temperature.

 **Postprocessing → Reports → Surface Integrals...**



a. Select **Mass-Weighted Average** from the **Report Type** drop-down list.

b. Select **Temperature...** and **Static Temperature** from the **Field Variable** drop-down lists.

The mass-averaged temperature will be computed as:

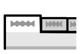
$$\overline{T}=\frac{\int T\rho\vec{v}\cdot d\vec{A}}{\int \rho\vec{v}\cdot d\vec{A}} \tag{13.2}$$

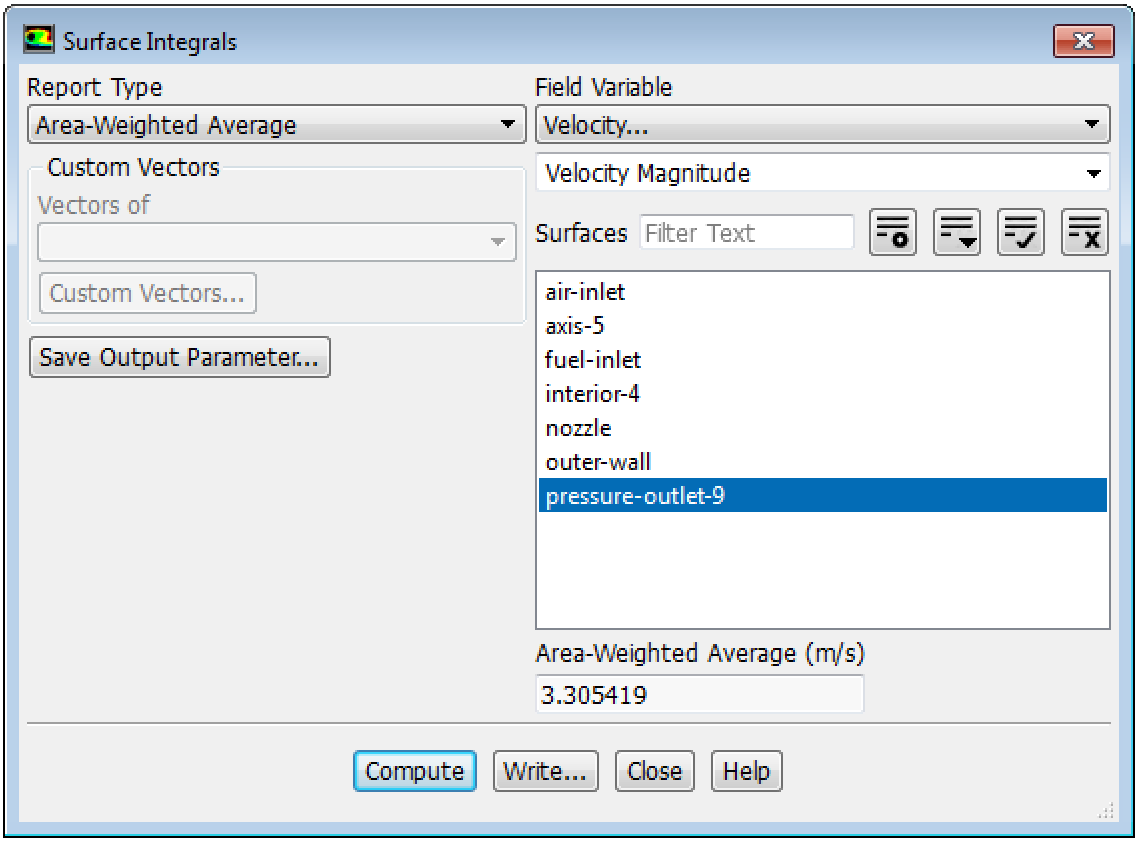
c. Select **pressure-outlet-9** from the **Surfaces** selection list, so that the integration is performed over this surface.

d. Click **Compute**.

*The **Mass-Weighted Average** field will show that the exit temperature is approximately 1840* K.

8. Determine the average exit velocity.

 **Postprocessing** → **Reports** → **Surface Integrals...**



a. Select **Area-Weighted Average** from the **Report Type** drop-down list.

b. Select **Velocity...** and **Velocity Magnitude** from the **Field Variable** drop-down lists.

The area-weighted velocity-magnitude average will be computed as:

$$\overline{v}=\frac{1}{A}\int v dA \tag{13.3}$$

c. Click **Compute**.

*The **Area-Weighted Average** field will show that the exit velocity is approximately 3.30* m/s.

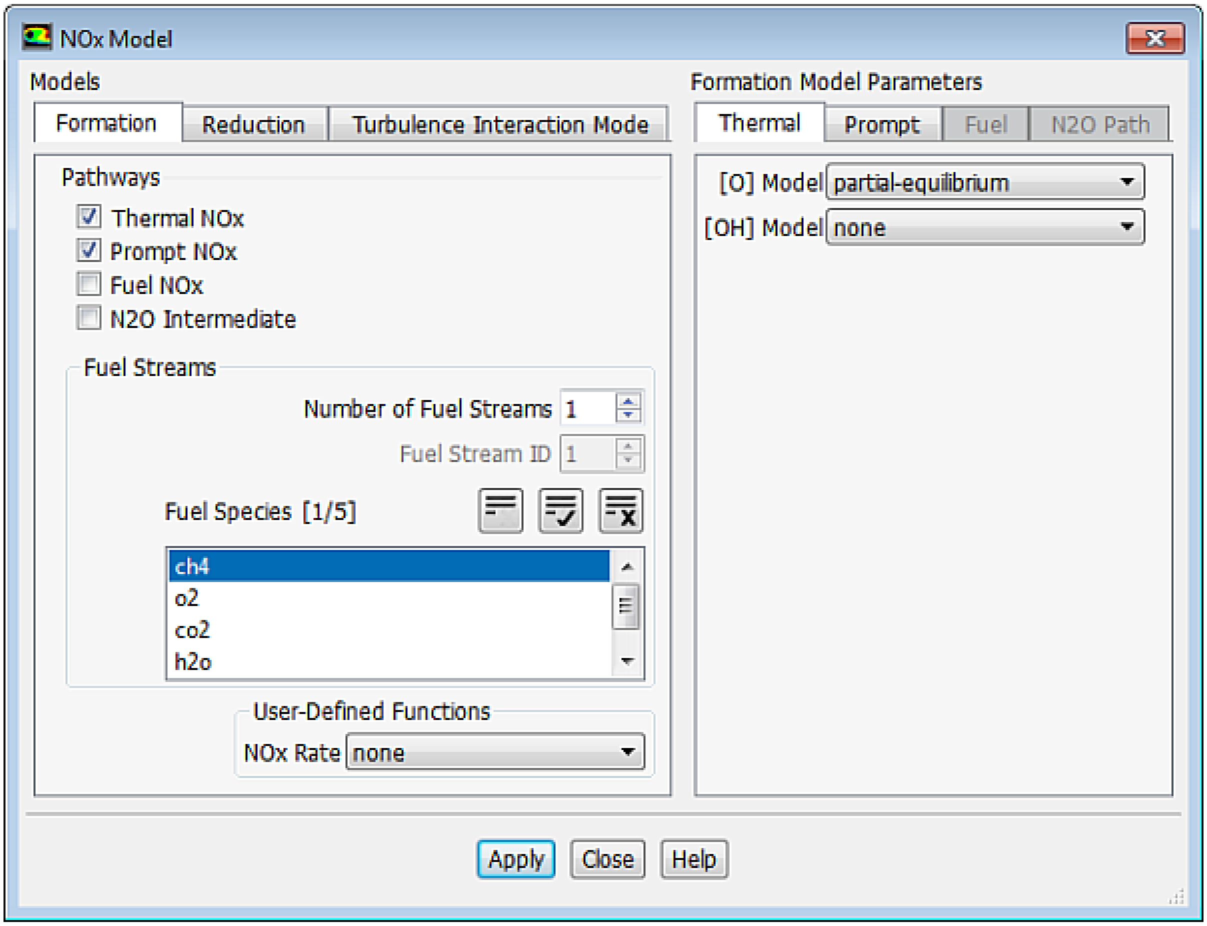
d. Close the **Surface Integrals** dialog box.

13.5.9. NOx Prediction

In this section you will extend the ANSYS Fluent model to include the prediction of NOx. You will first calculate the formation of both thermal and prompt NOx, then calculate each separately to determine the contribution of each mechanism.

1. Enable the NOx model.

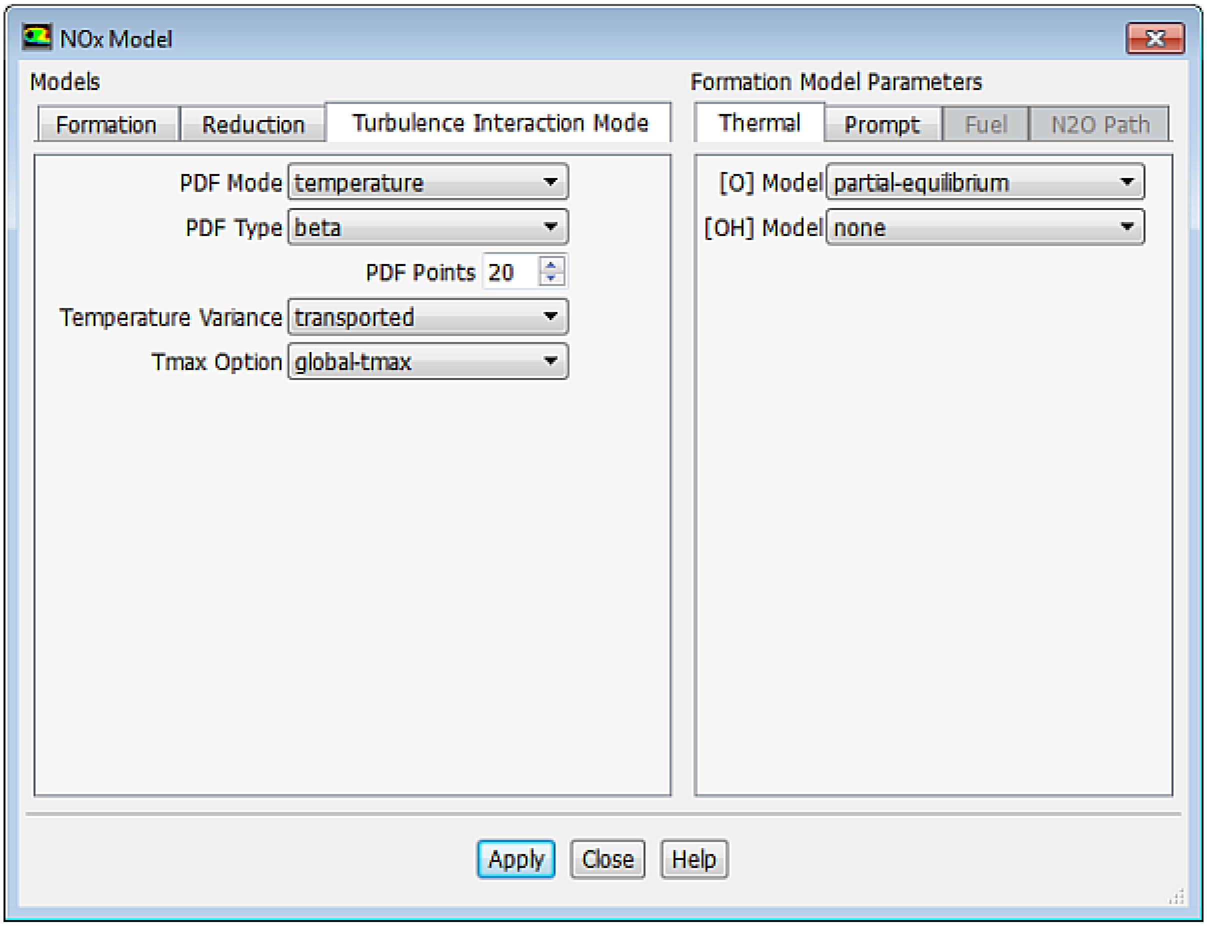
Setup → **Models** → **Species** → **NOx** **Edit...**



a. Enable **Thermal NOx** and **Prompt NOx** in the **Pathways** group box.

b. Select **ch4** from the **Fuel Species** selection list.

c. Click the **Turbulence Interaction Mode** tab.



i. Select **temperature** from the **PDF Mode** drop-down list.

   This will enable the turbulence-chemistry interaction. If turbulence interaction is not enabled, you will be computing NOx formation without considering the important influence of turbulent fluctuations on the time-averaged reaction rates.

ii. Retain the default selection of **beta** from the **PDF Type** drop-down list and enter `20` for **PDF Points**.

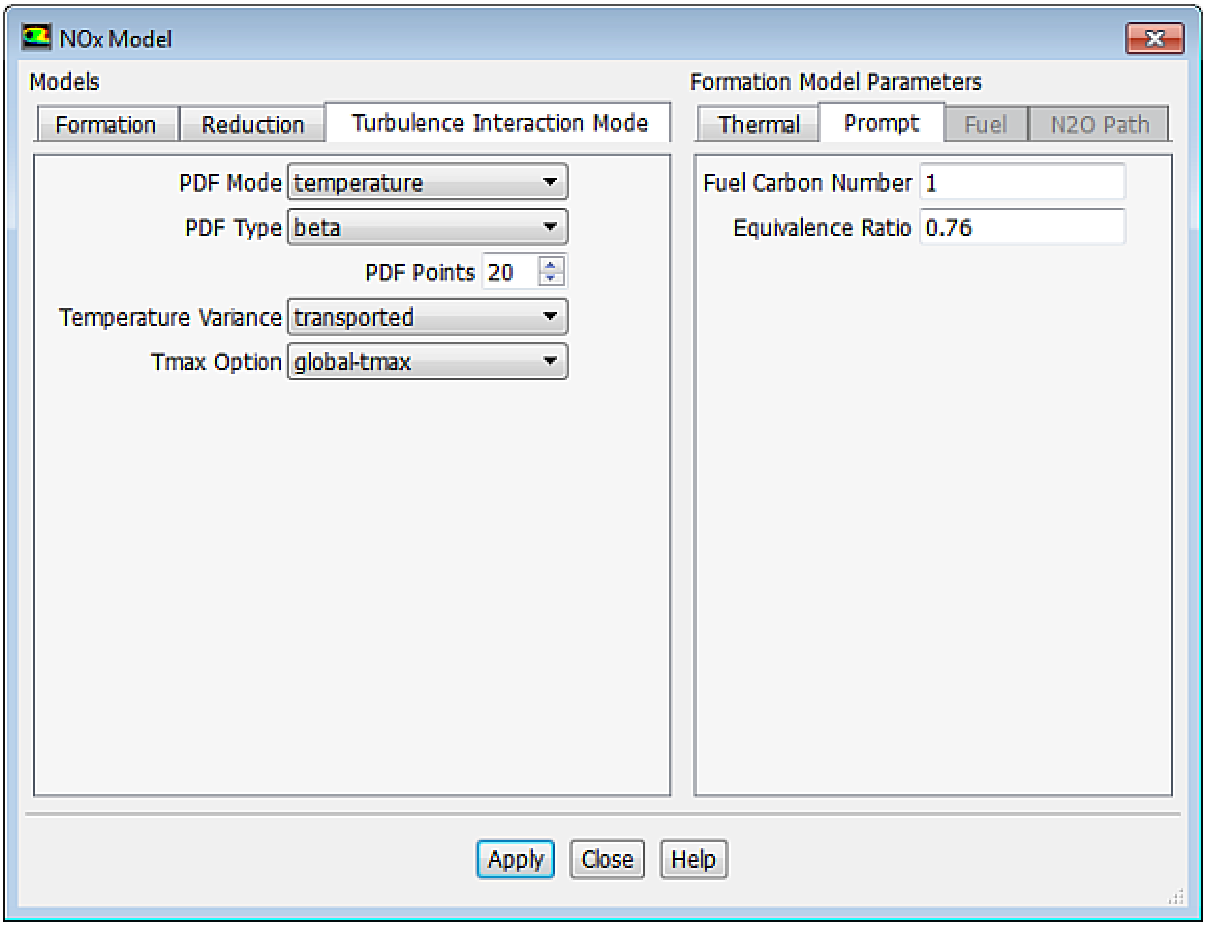
   The value for ***PDF Points*** *is increased from* `10` *to* `20` *to obtain a more accurate NOx prediction.*

iii. Select **transported** from the **Temperature Variance** drop-down list.

d. Select **partial-equilibrium** from the **[O] Model** drop-down list in the **Formation Model Parameters** group box in the **Thermal** tab.

   The partial-equilibrium model is used to predict the O radical concentration required for thermal NOx prediction.

e. Click the **Prompt** tab.



i. Retain the default value of `1` for **Fuel Carbon Number**.

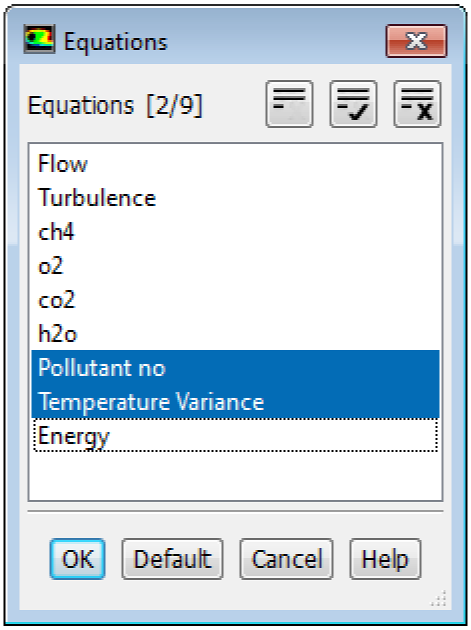
ii. Enter `0.76` for **Equivalence Ratio**.

*All of the parameters in the **Prompt** tab are used in the calculation of prompt NOx formation. The **Fuel Carbon Number** is the number of carbon atoms per molecule of fuel. The **Equivalence Ratio** defines the fuel-air ratio (relative to stoichiometric conditions).*

f. Click **Apply** to accept these changes and close the **NOx Model** dialog box.

2. Enable the calculation of NO species only and temperature variance.

Solving → **Controls** → **Equations...**



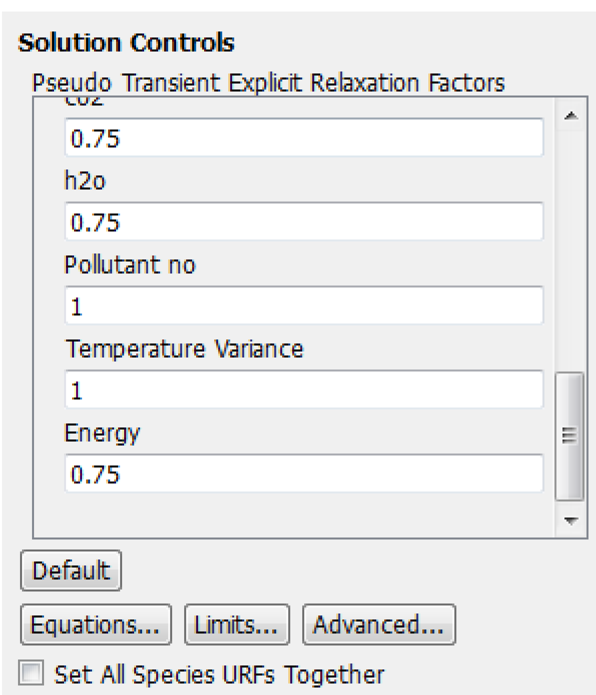
a. Deselect all variables except **Pollutant no** and **Temperature Variance** from the **Equations** selection list.

b. Click **OK** to close the **Equations** dialog box.

You will predict NOx formation in a "postprocessing" mode, with the flow field, temperature, and hydrocarbon combustion species concentrations fixed. Hence, only the NO equation will be computed. Prediction of NO in this mode is justified on the grounds that the NO concentrations are very low and have negligible impact on the hydrocarbon combustion prediction.

3. Modify the solution controls for **Pollutant no** and **Temperature Variance**.

Solving → **Controls** → **Controls...**



a. Click **Advanced...** to open the **Advanced Solution Controls** dialog box.

   i. In the **Expert** tab, enable the pseudo-transient method for **Pollutant no** and **Temperature Variance**, by selecting them under **On/Off**.

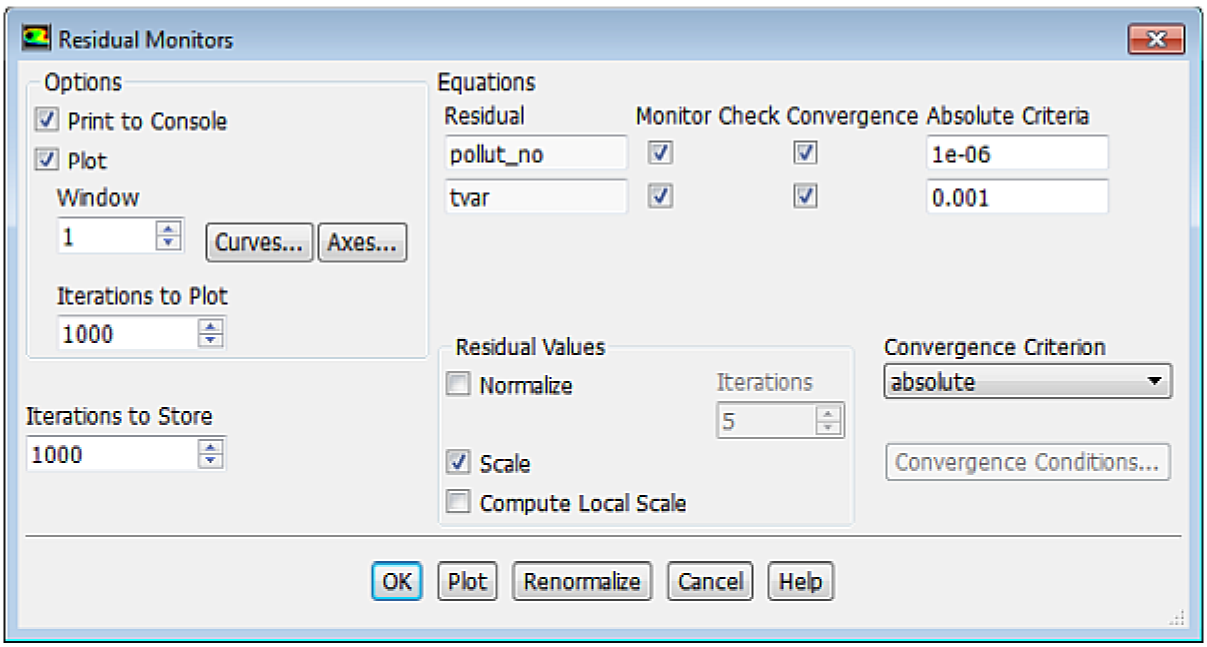
   ii. Enter `10` for **Time Scale Factor** for **Pollutant no** and **Temperature Variance**.

   iii. Click **OK** to close the **Advanced Solution Controls** dialog box.

b. Enter `1` for **Pollutant no** and **Temperature Variance** in the **Pseudo Transient Explicit Relaxation Factors** group box.

4. Confirm the convergence criterion for the NO species equation.

   Solving → Reports → Residuals...



   a. Ensure that the **Absolute Criteria** for **pollut_no** is set to `1e-06`.

   b. Click **OK** to close the **Residual Monitors** dialog box.

5. Request 25 more iterations.

   Solution → Run Calculation

   The solution will converge in approximately 10 iterations.

6. Save the new case and data files (`gascomb2.cas.gz` and `gascomb2.dat.gz`).

   File → Write → Case & Data...

7. Review the solution by creating and displaying a contour definition for NO mass fraction (Figure 13.10: Contours of NO Mass Fraction — Prompt and Thermal NOx Formation (p. 575)).

   Postprocessing → Graphics → Contours → New...

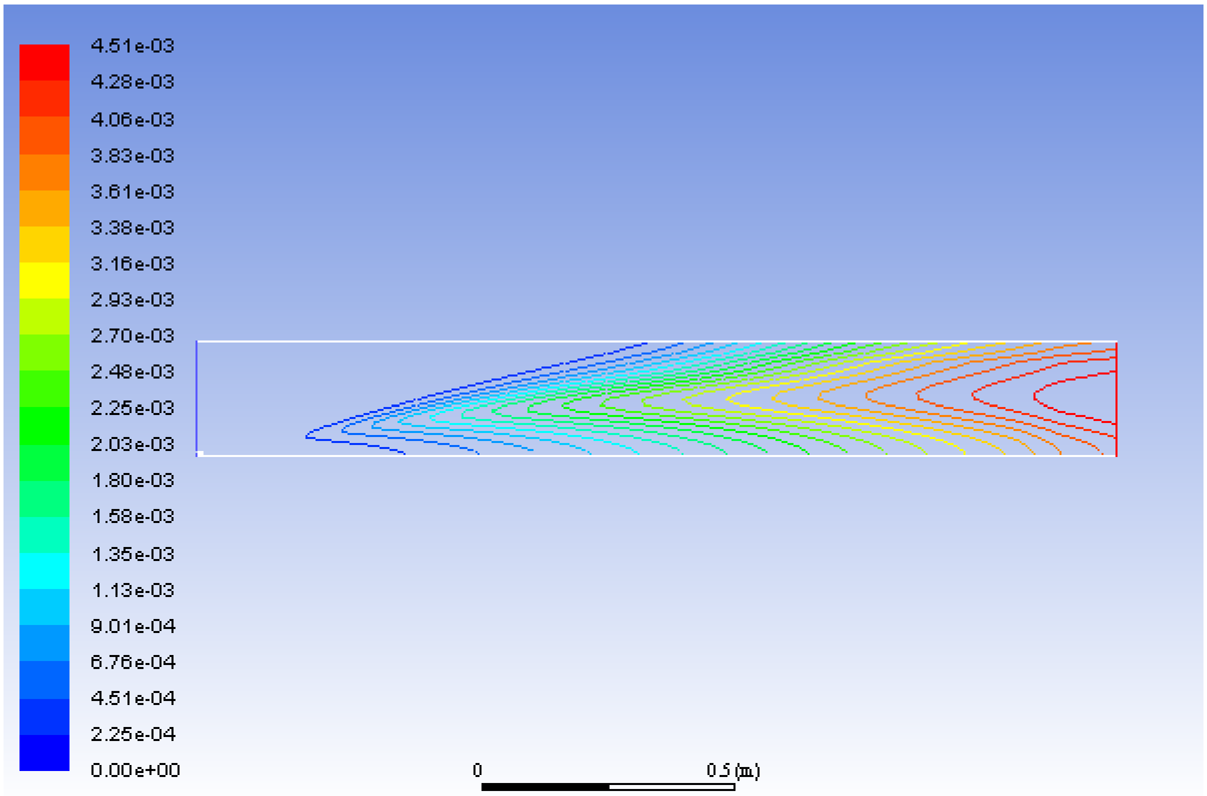
a. Enter `contour-no-mass-fraction` for **Contour Name**.

b. Disable **Filled** in the **Options** group box.

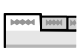
c. Select **NOx...** and **Mass fraction of Pollutant no** from the **Contours of** drop-down lists.

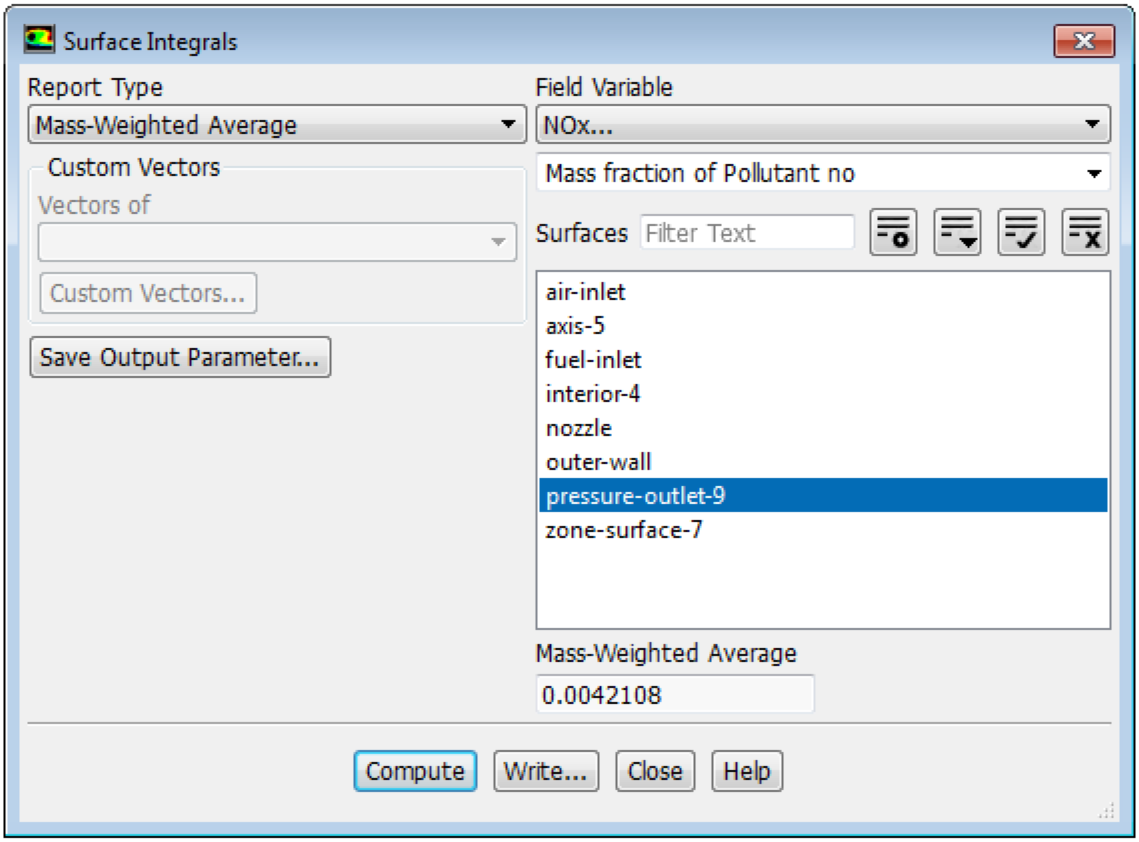
d. Click **Save/Display** and close the **Contours** dialog box.

Figure 13.10: Contours of NO Mass Fraction — Prompt and Thermal NOx Formation



8. Calculate the average exit NO mass fraction.

 **Postprocessing** → **Reports** → **Surface Integrals...**



a. Select **Mass-Weighted Average** from the **Report Type** drop-down list.

b. Select **NOx...** and **Mass fraction of Pollutant no** from the **Field Variable** drop-down lists.

c. Ensure that **pressure-outlet-9** is selected from the **Surfaces** selection list.

d. Click **Compute**.

*The **Mass-Weighted Average** field will show that the exit NO mass fraction is approximately 0.00421.*

e. Close the **Surface Integrals** dialog box.

9. Disable the prompt NOx mechanism in preparation for solving for thermal NOx only.

Setup → **Models** → **Species** → **NOx** **Edit...**

a. In the **Formation** tab, disable **Prompt NOx**.

b. Click **Apply** and close the **NOx Model** dialog box.

10. Request 25 iterations.

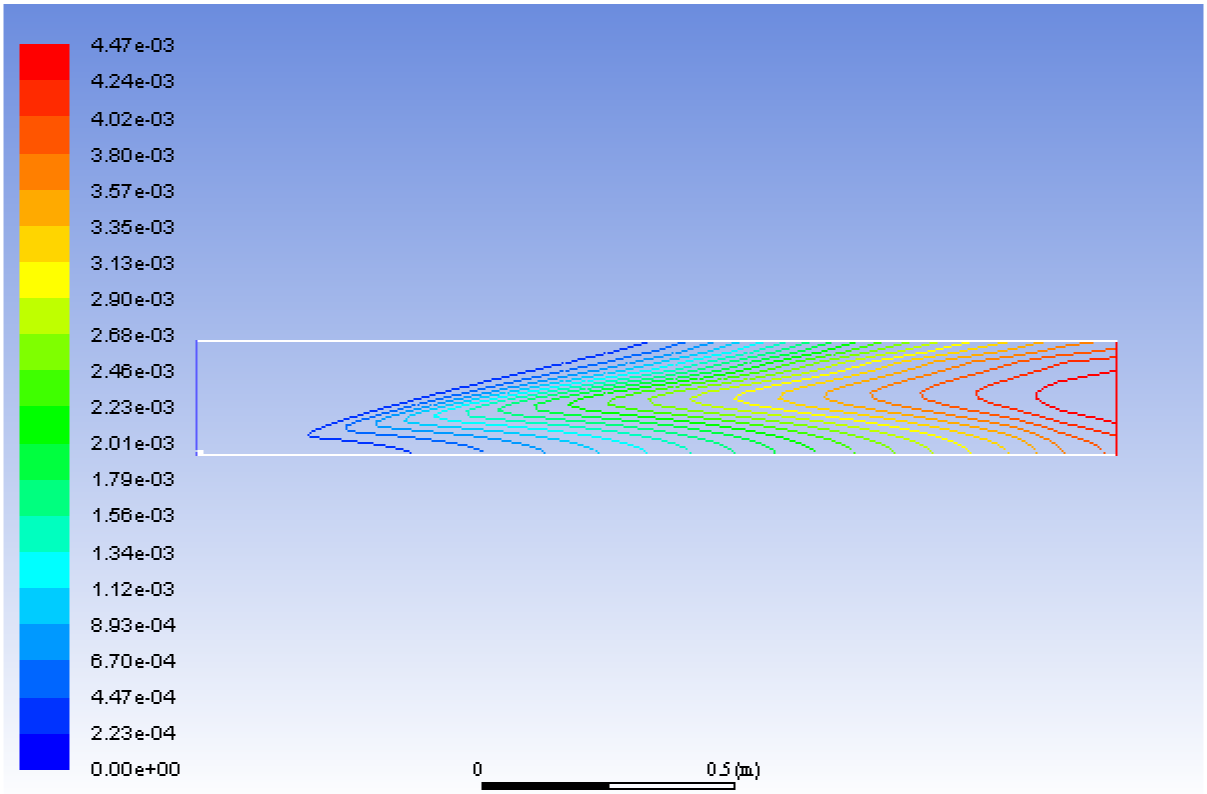
Solution → **Run Calculation**

The solution will converge in approximately 6 iterations.

11. Review the thermal NOx solution by displaying the **contour-no-mass-fraction** contour definition for NO mass fraction (under the **Results/Graphics/Contours** tree branch) you created earlier (Figure 13.11: Contours of NO Mass Fraction—Thermal NOx Formation (p. 577)).

    Results → **Graphics** → **Contours** → **contour-no-mass-fraction** **Display**

    Figure 13.11: Contours of NO Mass Fraction—Thermal NOx Formation

    Note that the concentration of NO is slightly lower without the prompt NOx mechanism.

12. Compute the average exit NO mass fraction with only thermal NOx formation.

    Postprocessing → **Reports** → **Surface Integrals...**

    Tip

    Follow the same procedure you used earlier for the calculation with both thermal and prompt NOx formation.

    ---

    *The **Mass-Weighted Average** field will show that the exit NO mass fraction with only thermal NOx formation (without prompt NOx formation) is approximately 0.004174.*

13. Solve for prompt NOx production only.

    Setup → **Models** → **Species** → **NOx** **Edit...**

    a. Disable **Thermal NOx** in the **Pathways** group box.

b. Enable **Prompt NOx**.

c. Click **Apply** and close the **NOx Model** dialog box.

14. Request 25 iterations.

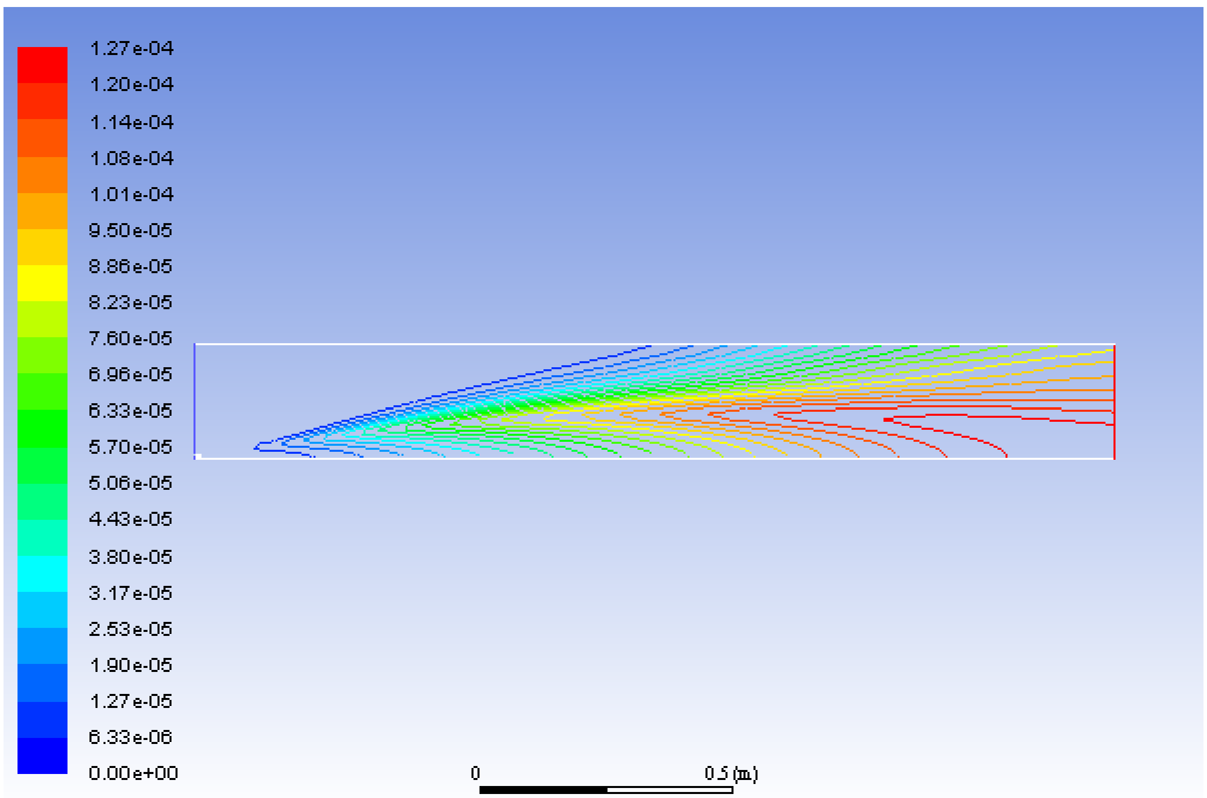
Solution → **Run Calculation**

The solution will converge in approximately 13 iterations.

15. Review the prompt NOx solution by displaying the **contour-no-mass-fraction** contour definition for NO mass fraction (under the **Results/Graphics/Contours** tree branch) (Figure 13.12: Contours of NO Mass Fraction—Prompt NOx Formation (p. 578)).

Results → **Graphics** → **Contours** → **contour-no-mass-fraction** **Display**

Figure 13.12: Contours of NO Mass Fraction—Prompt NOx Formation



The prompt NOx mechanism is most significant in fuel-rich flames. In this case the flame is lean and prompt NO production is low.

16. Compute the average exit NO mass fraction only with prompt NOx formation.

Postprocessing → Reports → Surface Integrals...

Tip

Follow the same procedure you used earlier for the calculation with both thermal and prompt NOx formation.

*The **Mass-Weighted Average** field will show that the exit NO mass fraction with only prompt NOx formation is approximately 9.975e-05.*

Note

The individual thermal and prompt NO mass fractions do not add up to the levels predicted with the two models combined. This is because reversible reactions are involved. NO produced in one reaction can be destroyed in another reaction.

17. Use a custom field function to compute NO parts per million (ppm).

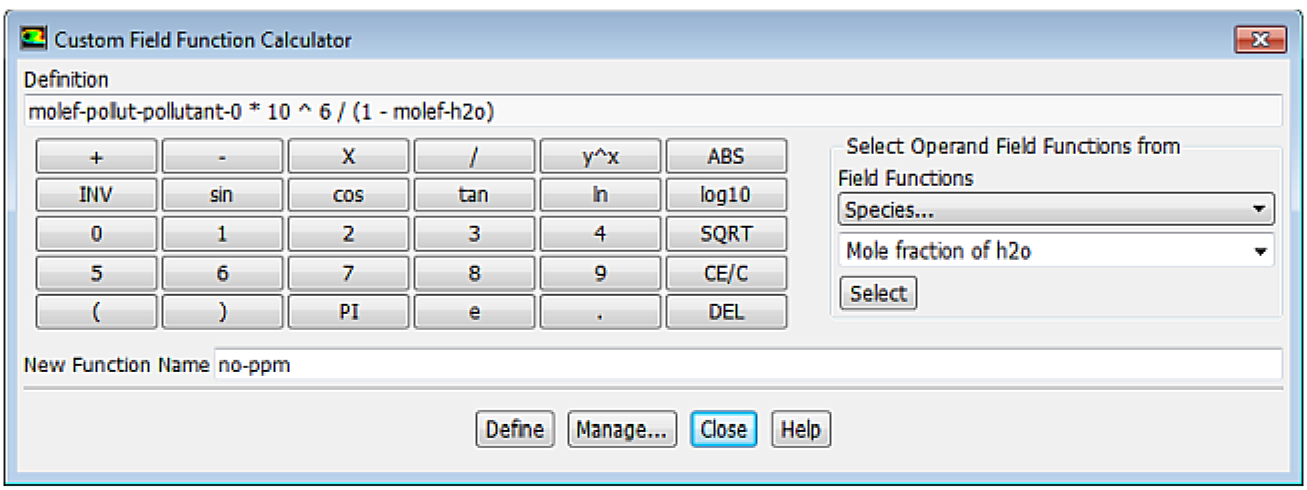
The NOppm will be computed from the following equation:

$$NOppm = \frac{NO mole fraction \times 10^6}{1 - H_2O mole fraction} \tag{13.4}$$

Note

This is the dry ppm. Therefore, the value is normalized by removing the water mole fraction in the denominator.

User Defined → Field Functions → Custom...



a. Select **NOx...** and **Mole fraction of Pollutant no** from the **Field Functions** drop-down lists, and click the **Select** button to enter **molef-pollut-pollutant-0** in the **Definition** field.

b. Click the appropriate calculator buttons to enter

```
*10^6/(1-
```

in the **Definition** field, as shown in the previous dialog box.

Tip

If you make a mistake, click the **DEL** button on the calculator pad to delete the last item you added to the function definition.

For more explicit instructions on using the **Custom Field Function** calculator buttons, see Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121).

c. Select **Species...** and **Mole fraction of h2o** from the **Field Functions** drop-down lists, and click the **Select** button to enter **molef-h2o** in the **Definition** field.

d. Click the **)** button to complete the field function.

e. Enter `no-ppm` for **New Function Name**.

f. Click **Define** to add the new field function to the variable list and close the **Custom Field Function Calculator** dialog box.

18. Display contours of NO ppm (Figure 13.13: Contours of NO ppm — Prompt NOx Formation (p. 581)).

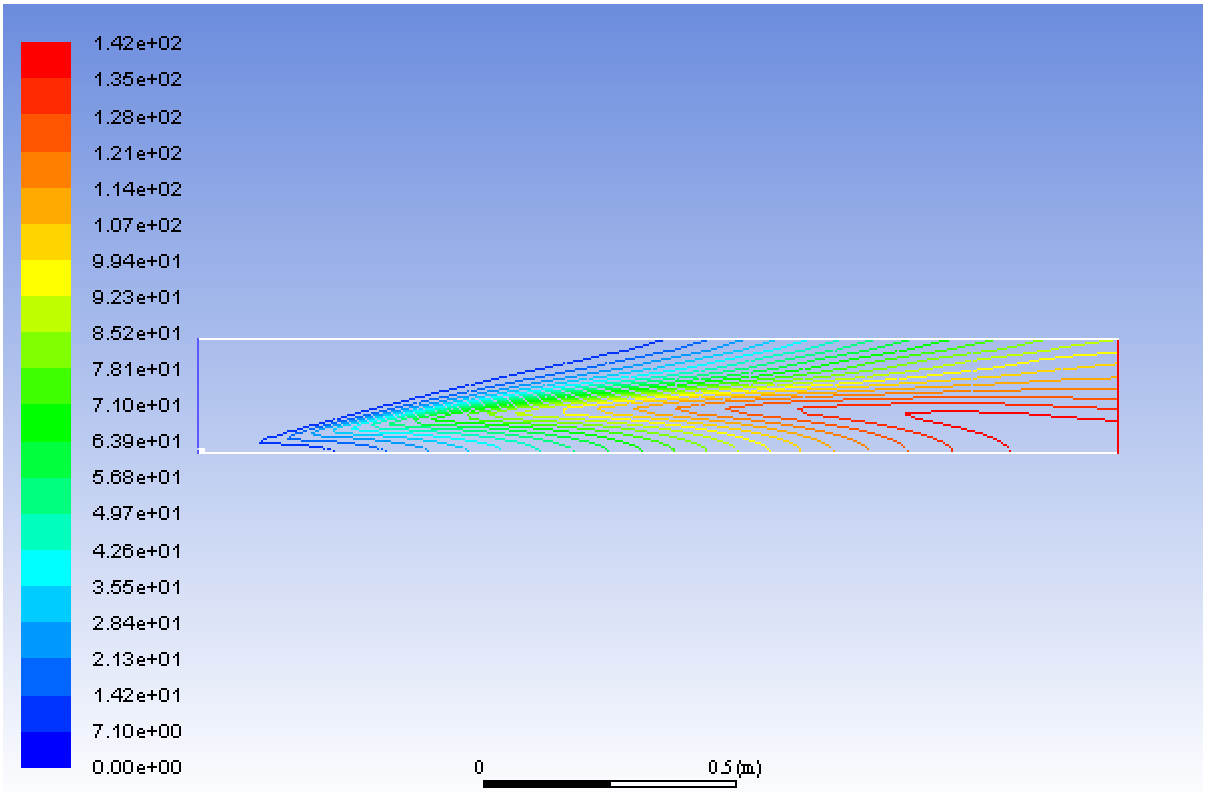
 **Postprocessing** → **Graphics** → **Contours** → **Edit...**

a. Select **Custom Field Functions...** and **no-ppm** from the **Contours of** drop-down lists.

*Scroll up the list to find **Custom Field Functions...**.*

b. Click **Display** and close the **Contours** dialog box.

Figure 13.13: Contours of NO ppm — Prompt NOx Formation



The contours closely resemble the mass fraction contours (Figure 13.12: Contours of NO Mass Fraction—Prompt NOx Formation (p. 578)), as expected.

13.6. Summary

In this tutorial you used ANSYS Fluent to model the transport, mixing, and reaction of chemical species. The reaction system was defined by using a mixture-material entry in the ANSYS Fluent database. The procedures used here for simulation of hydrocarbon combustion can be applied to other reacting flow systems.

The NOx production in this case was dominated by the thermal NO mechanism. This mechanism is very sensitive to temperature. Every effort should be made to ensure that the temperature solution is not overpredicted, since this will lead to unrealistically high predicted levels of NO.

13.7. Further Improvements

Further improvements can be expected by including the effects of intermediate species and radiation, both of which will result in lower predicted combustion temperatures.

The single-step reaction process used in this tutorial cannot account for the moderating effects of intermediate reaction products, such as CO and H_2. Multiple-step reactions can be used to address these species. If a multi-step Magnussen model is used, considerably more computational effort is required to solve for the additional species. Where applicable, the nonpremixed combustion model can be used to account for intermediate species at a reduced computational cost.

For more details on the nonpremixed combustion model, see the Fluent User's Guide.

Radiation heat transfer tends to make the temperature distribution more uniform, thereby lowering the peak temperature. In addition, radiation heat transfer to the wall can be very significant (especially here, with the wall temperature set at 300 K). The large influence of radiation can be anticipated by computing the Boltzmann number for the flow:

$$Bo=\frac{(\rho U C_p)_{inlet}}{\sigma T_{AF}^3} \sim \frac{\text{convection}}{\text{radiation}}$$

where σ is the Boltzmann constant (5.729×10^{-8} $W/m^2 - K^4$) and T_{AF} is the adiabatic flame temperature. For a quick estimate, assume $\rho=1$ kg/m^3, $U=0.5$ m/s, and $C_p=1000$ $J/kg-K$ (the majority of the inflow is air). Assume $T_{AF}=2000$ K. The resulting Boltzmann number is Bo = 1.09, which shows that radiation is of approximately equal importance to convection for this problem.

For details on radiation modeling, see the Fluent User's Guide.

This tutorial guides you through the steps to reach an initial set of solutions. You may be able to obtain a more accurate solution by using an appropriate higher-order discretization scheme and by adapting the mesh. Mesh adaption can also ensure that the solution is independent of the mesh. These steps are demonstrated in Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121).

Chapter 14: Using the Non-Premixed Combustion Model

This tutorial is divided into the following sections:

14.1. Introduction

The goal of this tutorial is to accurately model the combustion processes in a 300 KW BERL combustor. The reaction can be modeled using either the species transport model or the non-premixed combustion model. In this tutorial you will set up and solve a natural gas combustion problem using the non-premixed combustion model for the reaction chemistry.

This tutorial demonstrates how to do the following:

- Define inputs for modeling non-premixed combustion chemistry.
- Prepare the PDF table in ANSYS Fluent.
- Solve a natural gas combustion simulation problem.
- Use the Discrete Ordinates (DO) radiation model for combustion applications.
- Use the k- ε turbulence model.

The non-premixed combustion model uses a modeling approach that solves transport equations for one or two conserved scalars (mixture fractions). Multiple chemical species, including radicals and intermediate species, may be included in the problem definition. Their concentrations will be derived from the predicted mixture fraction distribution.

Property data for the species are accessed through a chemical database, and turbulence-chemistry interaction is modeled using a β-function for the PDF. For details on the non-premixed combustion modeling approach, the Fluent User's Guide.

14.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)

- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

and that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

14.3. Problem Description

The flow considered is an unstaged natural gas flame in a 300 kW swirl-stabilized burner. The furnace is vertically-fired and of octagonal cross-section with a conical furnace hood and a cylindrical exhaust duct. The furnace walls are capable of being refractory-lined or water-cooled. The burner features 24 radial fuel ports and a bluff centerbody. Air is introduced through an annular inlet and movable swirl blocks are used to impart swirl. The combustor dimensions are described in Figure 14.1: Problem Description (p. 584), and Figure 14.2: Close-Up of the Burner (p. 585) shows a close-up of the burner assuming 2D axisymmetry. The boundary condition profiles, velocity inlet boundary conditions of the gas, and temperature boundary conditions are based on experimental data [1].

Figure 14.1: Problem Description

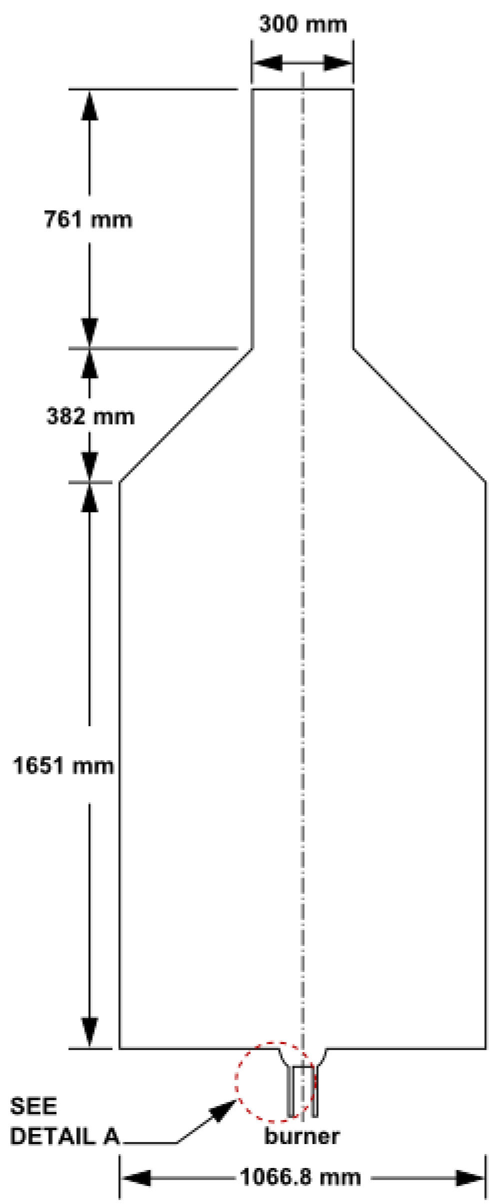
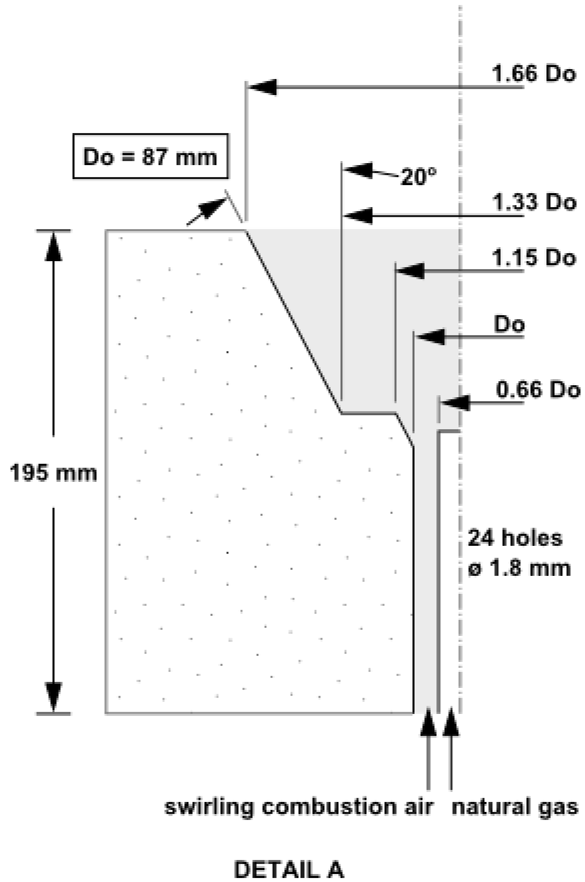


Figure 14.2: Close-Up of the Burner



14.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

14.4.1. Preparation

To prepare for running this tutorial:

1. Set up a working folder on the computer you will be using.

2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

   Note

   If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

3. Enter the name of this tutorial into the search bar.

4. Narrow the results by using the filter on the left side of the page.

   a. Click **ANSYS Fluent** under **Product**.

   b. Click **18.0** under **Version**.

5. Select this tutorial from the list.

6. Click the **non_premix_combustion_R180.zip** link to download the input files.

7. Unzip `non_premix_combustion_R180.zip` to your working folder.

   The files, `berl.msh` and `berl.prof`, can be found in the `non_premix_combustion` folder, which will be created after unzipping the file.

   The mesh file, `berl.msh`, is a quadrilateral mesh describing the system geometry shown in Figure 14.1: Problem Description (p. 584) and Figure 14.2: Close-Up of the Burner (p. 585).

8. Use Fluent Launcher to start the **2D** version of ANSYS Fluent.

   Fluent Launcher displays your **Display Options** preferences from the previous session.

9. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.

10. Enable **Double-Precision**.

    For more information about Fluent Launcher, see starting ANSYS Fluent using the Fluent Launcher in the Fluent Getting Started Guide.

11. Ensure that the **Serial** processing option is selected.

14.4.2. Reading and Checking the Mesh

1. Read the mesh file `berl.msh`.

   File → Read → Mesh...

   The ANSYS Fluent console will report that the mesh contains 9784 quadrilateral cells. A warning will be generated informing you to consider making changes to the zone type, or to change the problem definition to axisymmetric. You will change the problem to axisymmetric swirl in Specifying Solver and Analysis Type (p. 590).

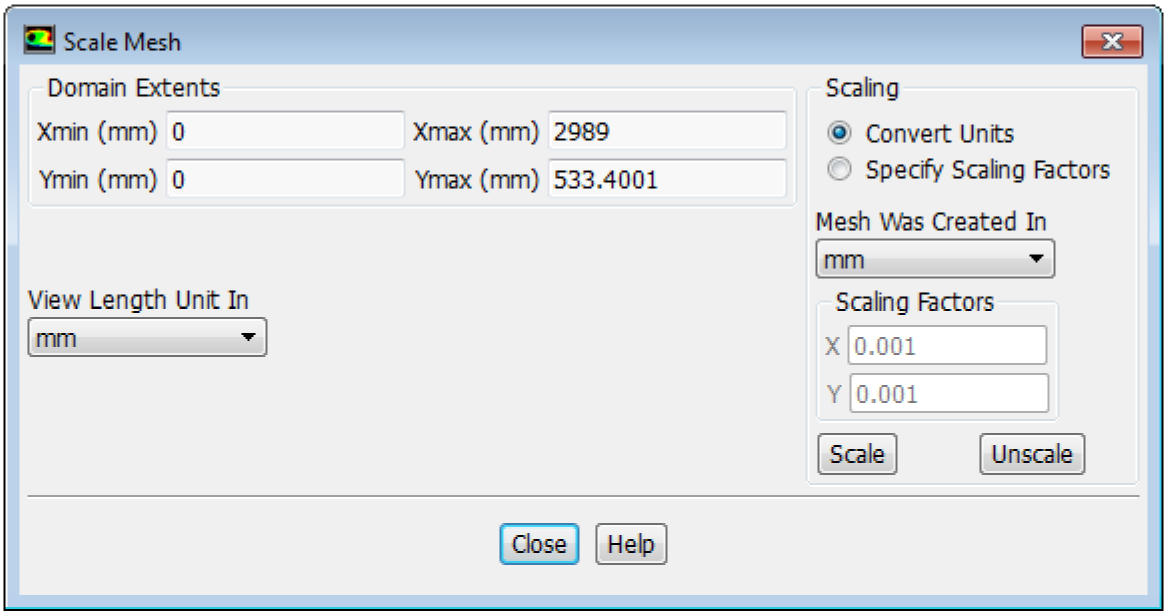
2. Check the mesh.

Setting Up Domain → Mesh → Check

ANSYS Fluent will perform various checks on the mesh and will report the progress in the console. Ensure that the reported minimum volume is a positive number.

3. Scale the mesh.

Setting Up Domain → Mesh → Scale...



a. Select **mm** from the **View Length Unit In** drop-down list.

All dimensions will now be shown in millimeters.

b. Select **mm** from the **Mesh Was Created In** drop-down list in the **Scaling** group box.

c. Click **Scale** and verify that the domain extents are as shown in the **Scale Mesh** dialog box.

d. Close the **Scale Mesh** dialog box.

4. Check the mesh.

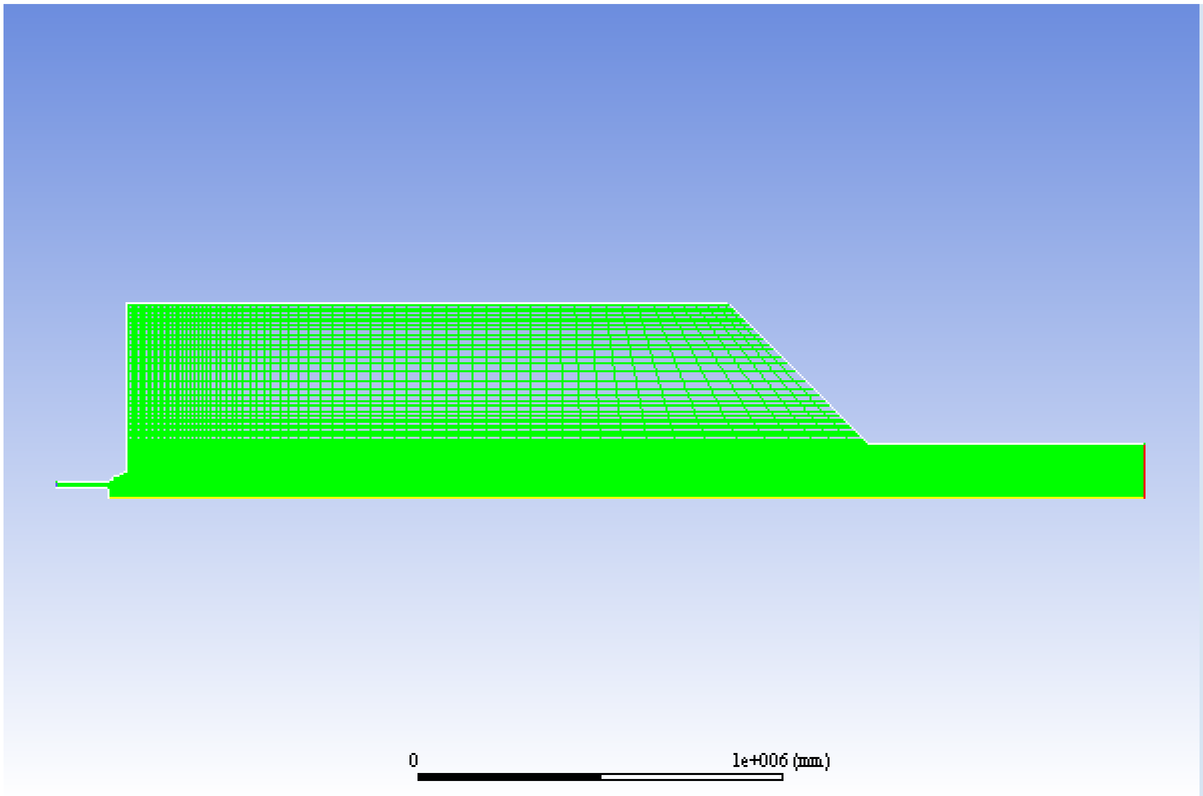
Setting Up Domain → Mesh → Check

Note

It is a good idea to check the mesh after you manipulate it (that is, scale, convert to polyhedra, merge, separate, fuse, add zones, or smooth and swap.) This will ensure that the quality of the mesh has not been compromised.

5. Examine the mesh (Figure 14.3: 2D BERL Combustor Mesh Display (p. 588)).

Figure 14.3: 2D BERL Combustor Mesh Display



Due to the mesh resolution and the size of the domain, you may find it more useful to display just the outline, or to zoom in on various portions of the mesh display.

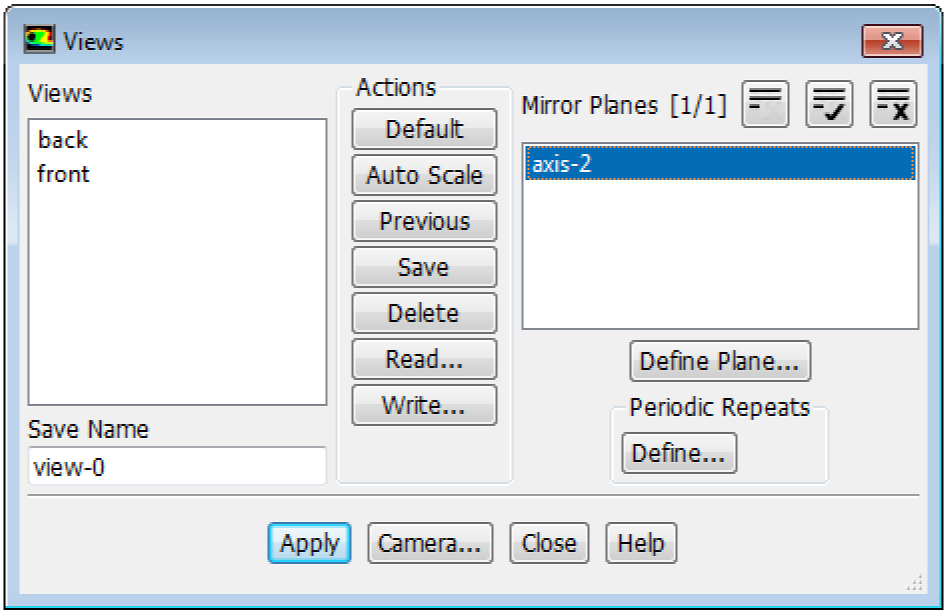
Extra

You can use the mouse zoom button (middle button, by default) to zoom in to the display and the mouse probe button (right button, by default) to find out the boundary zone labels. The zone labels will be displayed in the console.

---

6. Mirror the display about the symmetry plane.

   Results → **Graphics** **Views...**

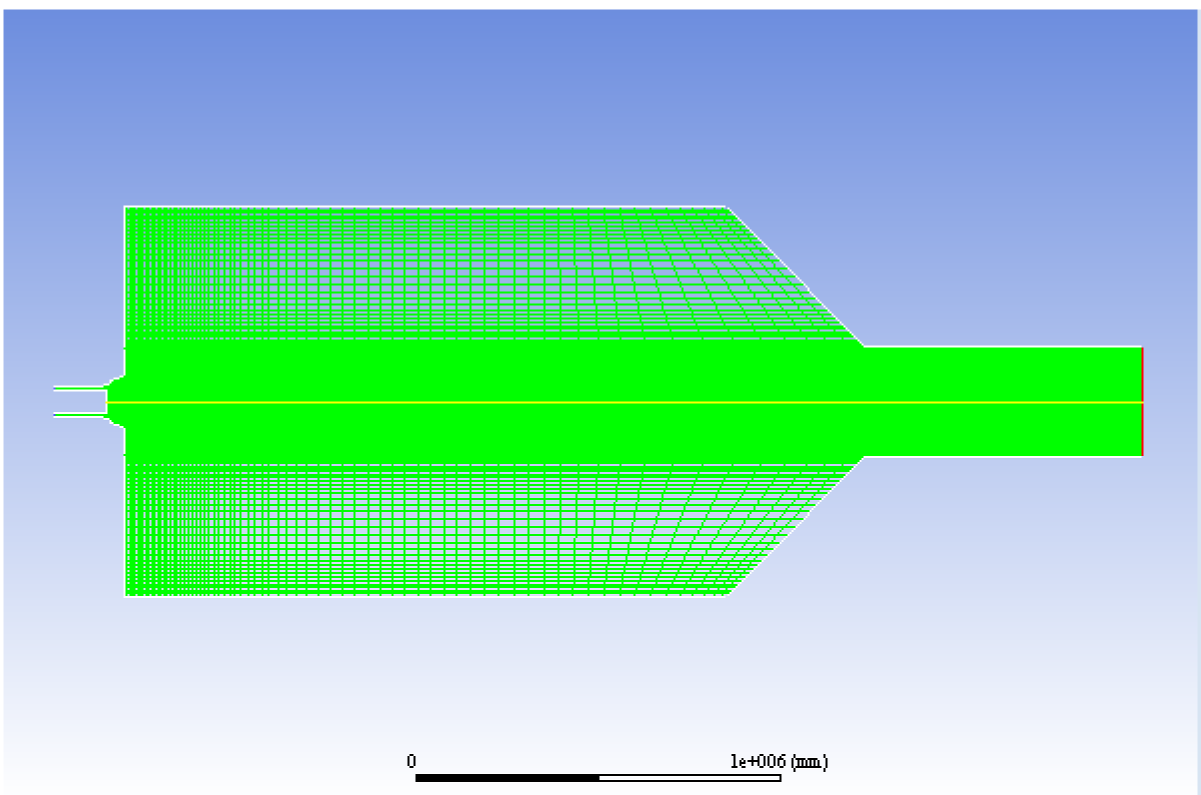


a. Select **axis-2** from the **Mirror Planes** selection list.

b. Click **Apply** and close the **Views** dialog box.

The full geometry is displayed, as shown in Figure 14.4: 2D BERL Combustor Mesh Display Including the Symmetry Plane (p. 589)

Figure 14.4: 2D BERL Combustor Mesh Display Including the Symmetry Plane



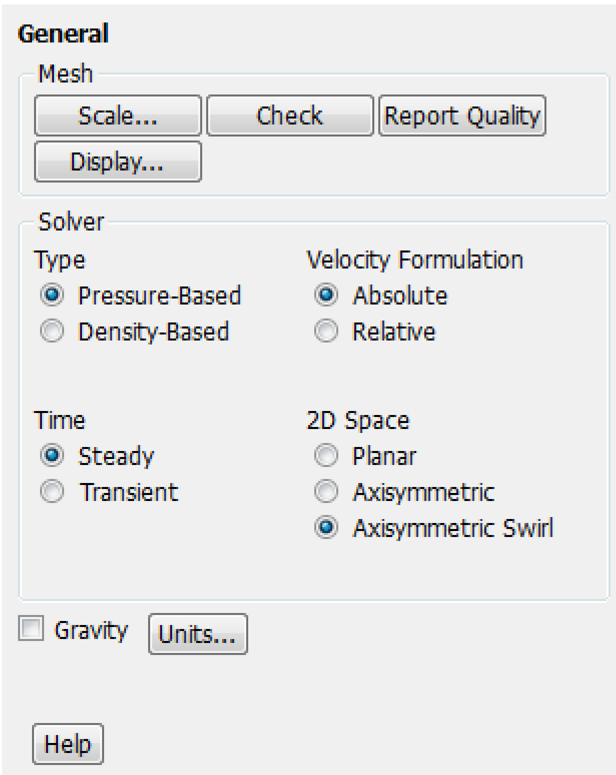
14.4.3. Specifying Solver and Analysis Type

1. Retain the default settings of pressure-based steady-state solver in the **Solver** group box.

   Setup → **General**

   The non-premixed combustion model is available only with the pressure-based solver.

2. Change the spatial definition to axisymmetric swirl by selecting **Axisymmetric Swirl** in the **2D Space** list.



14.4.4. Specifying the Models

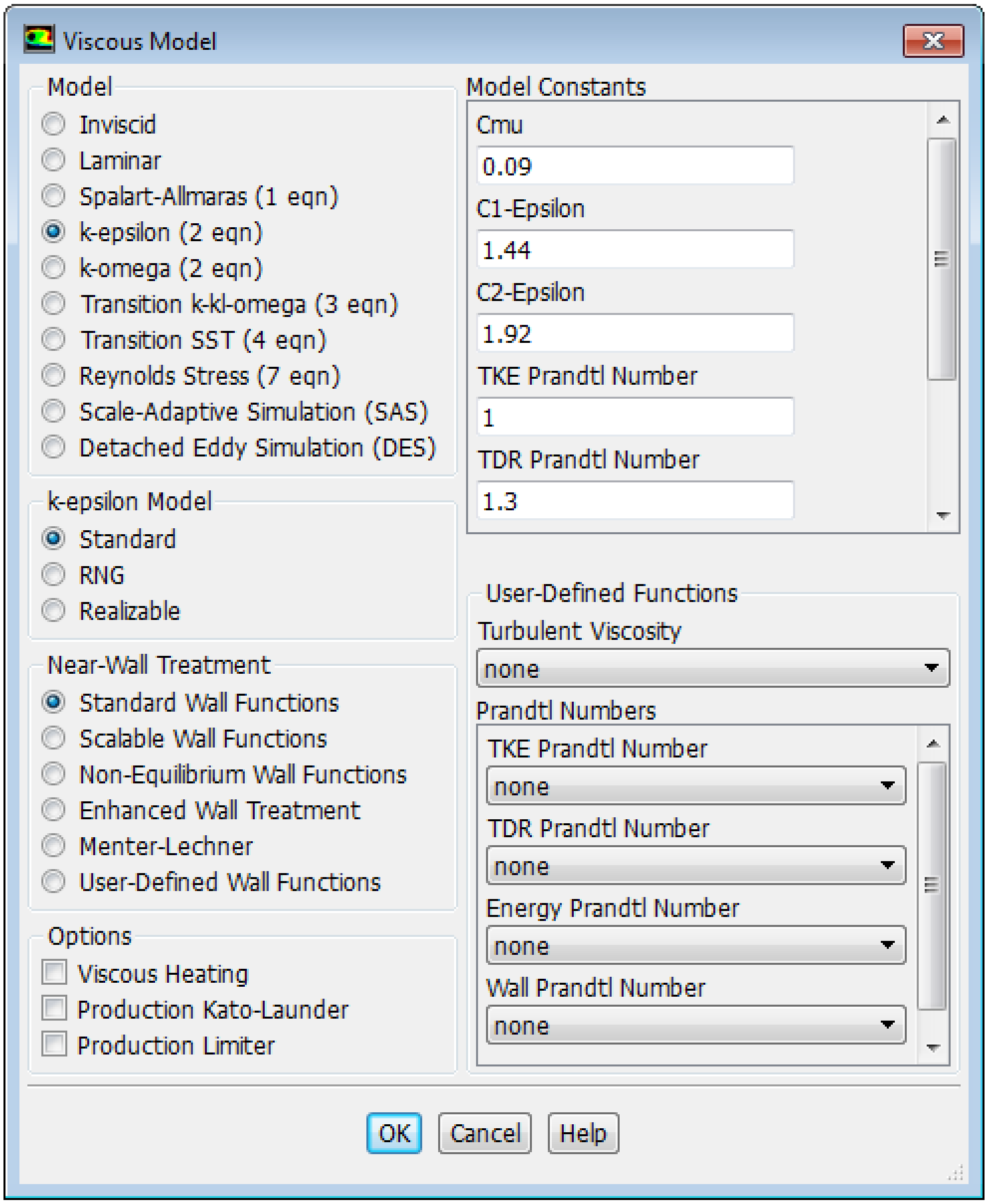
1. Enable the **Energy Equation**.

   Setting Up Physics → **Models** → **Energy**

   Since heat transfer occurs in the system considered here, you will have to solve the energy equation.

2. Select the standard k-epsilon turbulence model.

   Setting Up Physics → **Models** → **Viscous...**



a. Select **k-epsilon (2eqn)** in the **Model** list.

For axisymmetric swirling flow, the RNG k-epsilon model can also be used, but for this case you will retain the default ***Standard***, ***k-epsilon*** *model.*

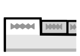
b. Retain the default **Standard Wall Functions** in the **Near-Wall Treatment** group box.

c. Click **OK** to accept all other default settings and close the **Viscous Model** dialog box.

3. Select the **Discrete Ordinates (DO)** radiation model.

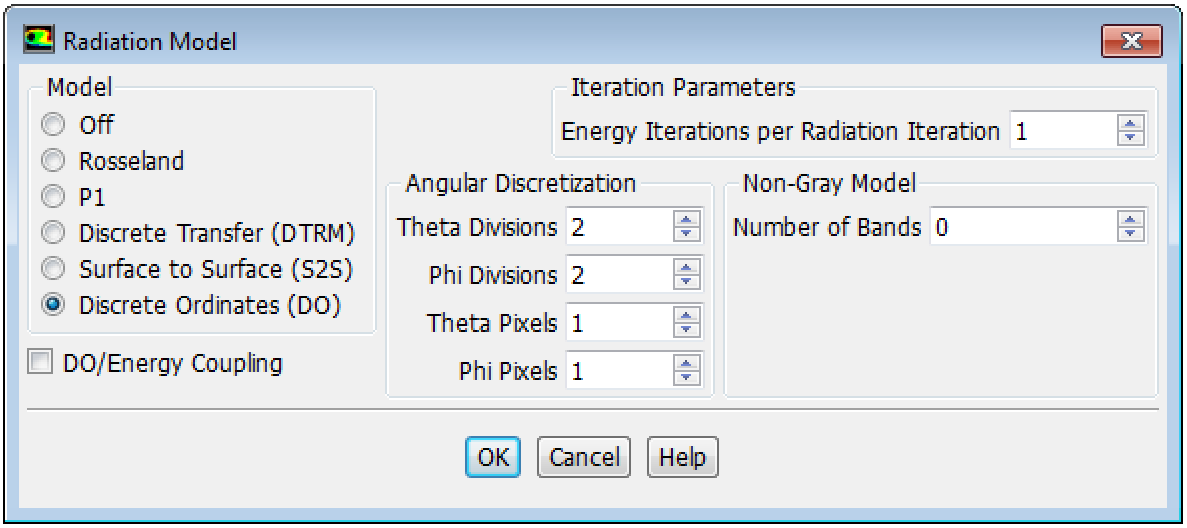
The DO radiation model provides a high degree of accuracy, but it can be CPU intensive. In cases where the computational expense of the DO model is deemed too great, the P1 model may provide an acceptable solution more quickly.

For details on the different radiation models available in ANSYS Fluent, see the Fluent User's Guide.

 **Setting Up Physics → Models → Radiation...**

a. Select **Discrete Ordinates (DO)** in the **Model** list.

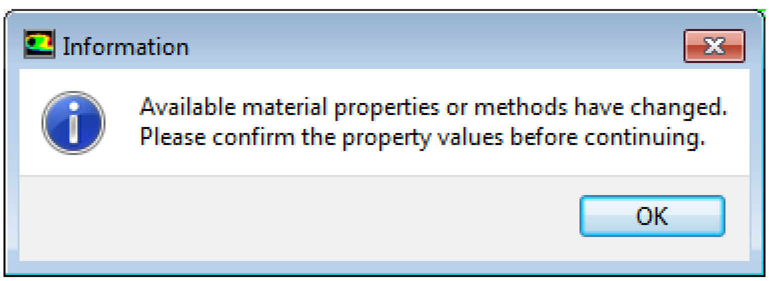
The dialog box will expand to show related inputs.



b. Enter 1 for **Energy Iterations per Radiation Iteration**.

c. Click **OK** to accept all other default settings and close the **Radiation Model** dialog box.

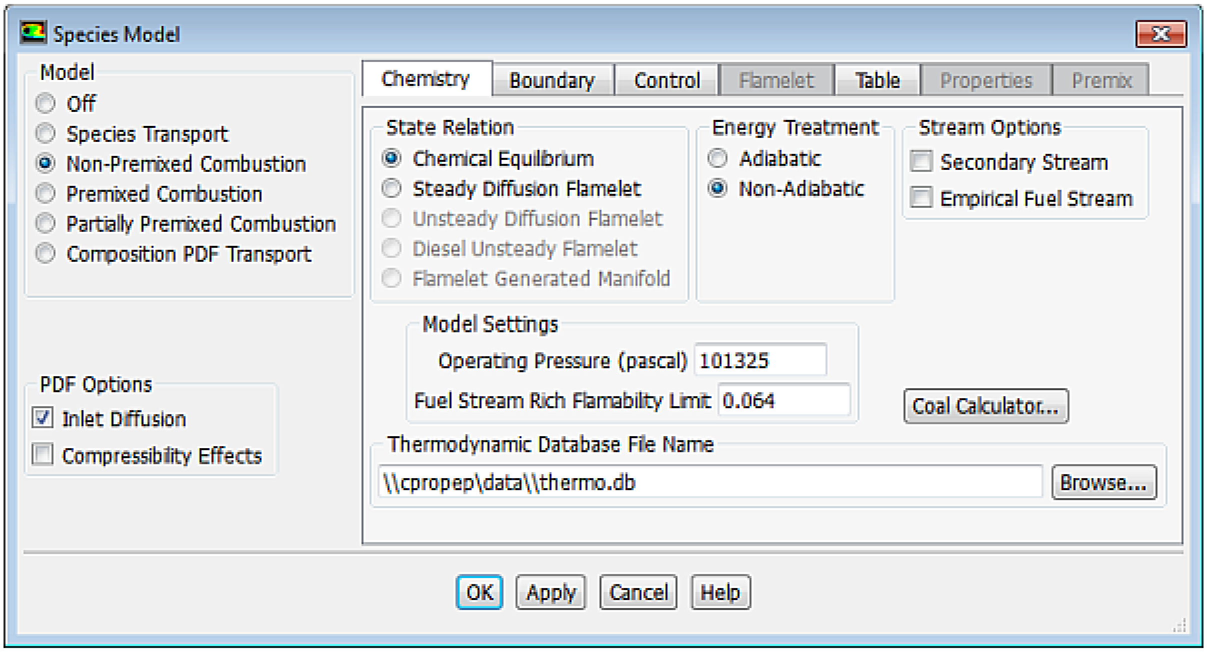
*The ANSYS Fluent console lists the properties that are required for the model you have enabled. An **Information** dialog box opens, reminding you to confirm the property values.*



d. Click **OK** to close the **Information** dialog box.

4. Select the **Non-Premixed Combustion** model.

Setting Up Physics → Models → Species...



a. Select **Non-Premixed Combustion** in the **Model** list.

The dialog box will expand to show the related inputs. You will use this dialog box to create the PDF table.

When you use the non-premixed combustion model, you need to create a PDF table. This table contains information on the thermo-chemistry and its interaction with turbulence. ANSYS Fluent interpolates the PDF during the solution of the non-premixed combustion model.

b. Enable **Inlet Diffusion** in the **PDF Options** group box.

*The **Inlet Diffusion** option enables the mixture fraction to diffuse out of the domain through inlets and outlets.*

c. Define chemistry models.

   i. Retain the default selection of the **Chemical Equilibrium** state relation and the **Non-Adiabatic** energy treatment.

   *In most non-premixed combustion simulations, the **Chemical Equilibrium** model is recommended. The **Steady Diffusion Flamelet** option can model local chemical non-equilibrium due to turbulent strain.*

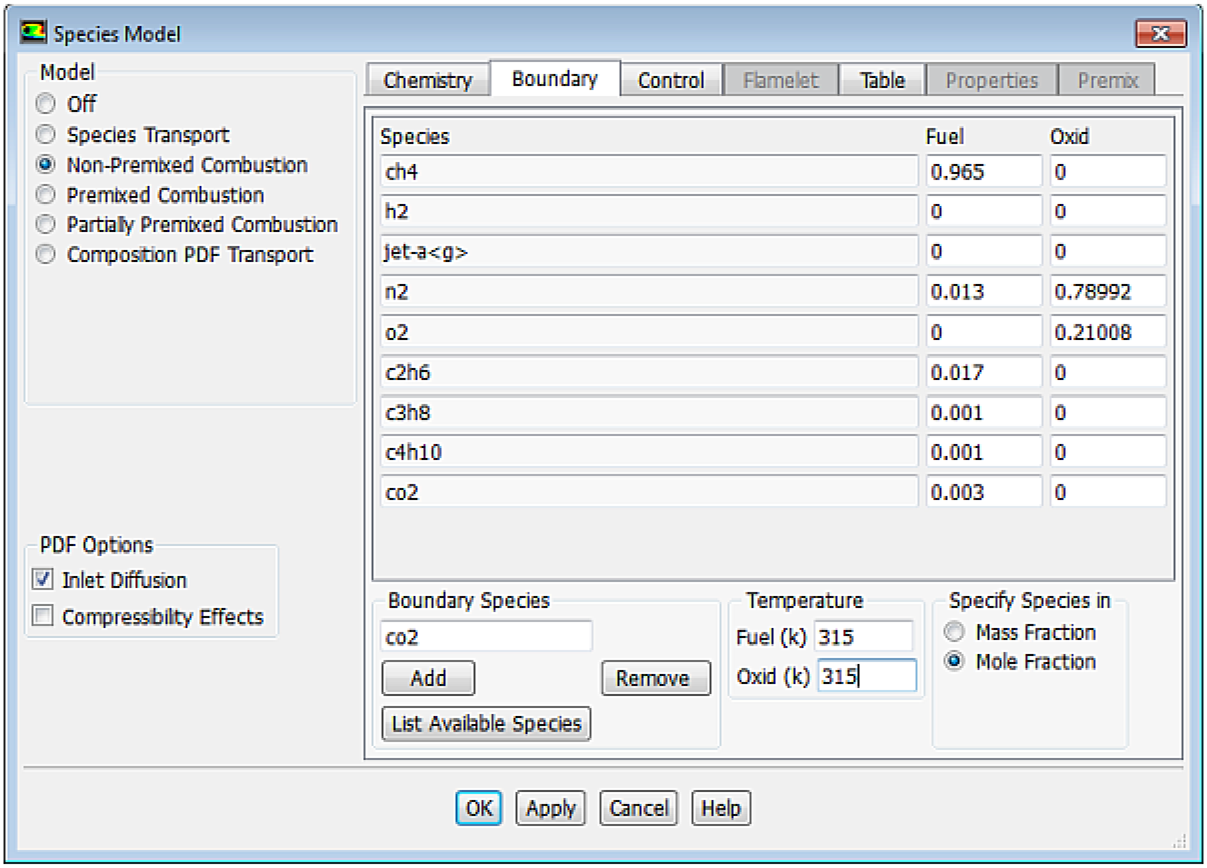
   ii. Retain the default value for **Operating Pressure**.

   iii. Enter `0.064` for **Fuel Stream Rich Flammability Limit**.

   *The **Fuel Stream Rich Flammability Limit** enables you to perform a "partial equilibrium" calculation, suspending equilibrium calculations when the mixture fraction exceeds the specified rich limit. This increases the efficiency of the PDF calculation, allowing you to bypass the complex equilibrium calculations in the fuel-rich region. This is also more physically realistic than the assumption of full equilibrium.*

For combustion cases, a value 10% – 50% larger than the stoichiometric mixture fraction can be used for the rich flammability limit of the fuel stream. In this case, the stoichiometric fraction is 0.058, therefore a value that is 10% greater is 0.064.

d. Click the **Boundary** tab to add and define the boundary species.



i. Enter `c2h6` in the **Boundary Species** text-entry field and click **Add**.

*The **c2h6** species appears at the bottom of the table.*

ii. Similarly, add `c3h8`, `c4h10`, and `co2`.

All the added species will appear in the table.

iii. Select **Mole Fraction** in the **Specify Species in** list.

iv. Retain the default values for **n2** and **o2** for **Oxid**.

The oxidizer (air) consists of 21% O_2 *and 79%* $2N_2$ *by volume.*

v. Specify the fuel composition by entering the following values for **Fuel**:

The fuel composition is entered in mole fractions of the species, c2h6, c3h8, c4h10, and co2.

Species	Mole Fraction
ch4	`0.965`
n2	`0.013`

Species	Mole Fraction
c2h6	0.017
c3h8	0.001
c4h10	0.001
co2	0.003

Tip

Scroll down to see all the species.

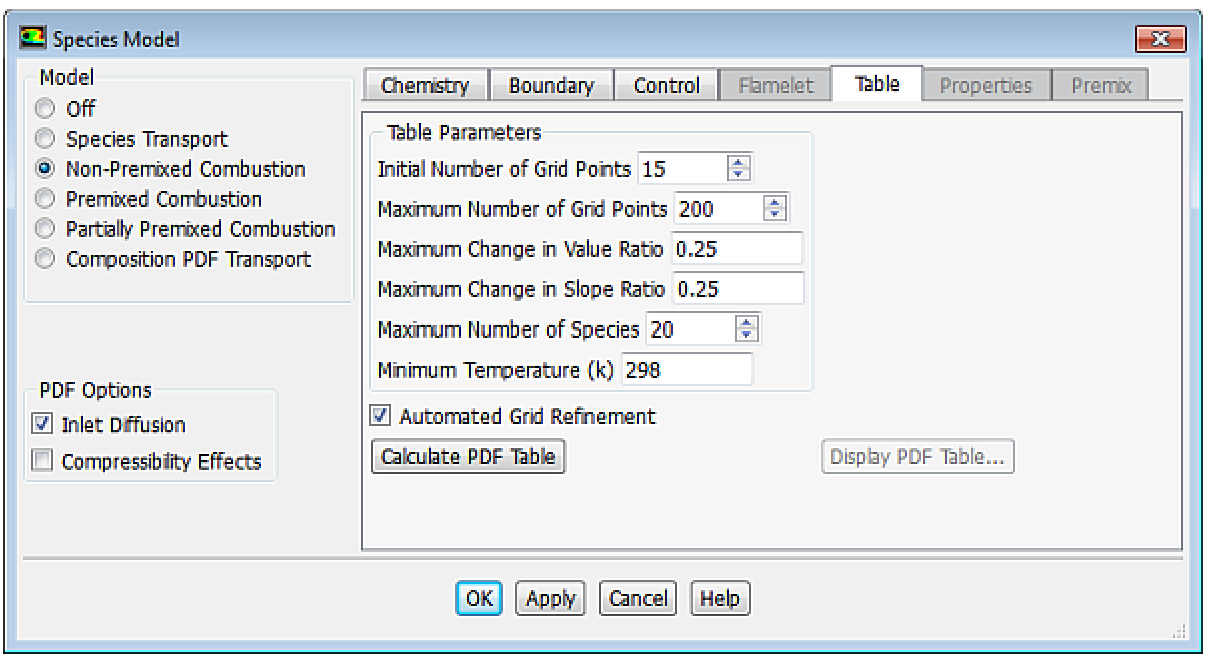
Note

All boundary species with a mass or mole fraction of zero will be ignored.

vi. Enter 315 K for **Fuel** and **Oxid** in the **Temperature** group box.

e. Click the **Control** tab and retain default species to be excluded from the equilibrium calculation.

f. Click the **Table** tab to specify the table parameters and calculate the PDF table.



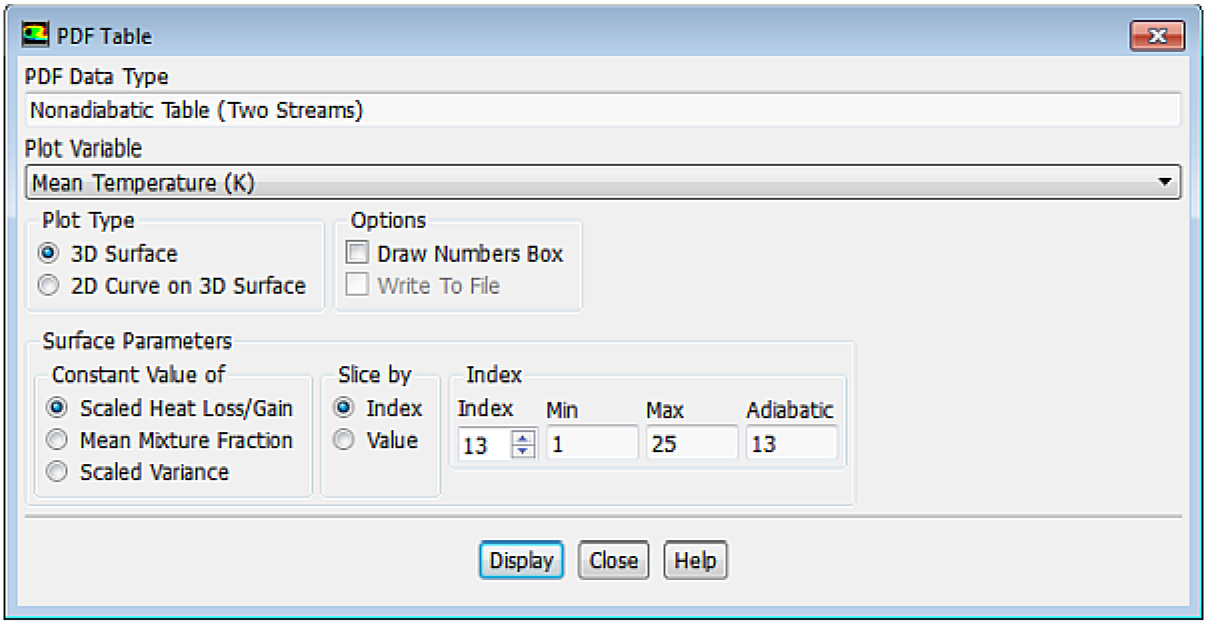
i. Ensure that **Automated Grid Refinement** is enabled.

ii. Retain the default values for all the parameters in the **Table Parameters** group box.

The maximum number of species determines the number of most preponderant species to consider after the equilibrium calculation is performed.

iii. Click **Calculate PDF Table** to compute the non-adiabatic PDF table.

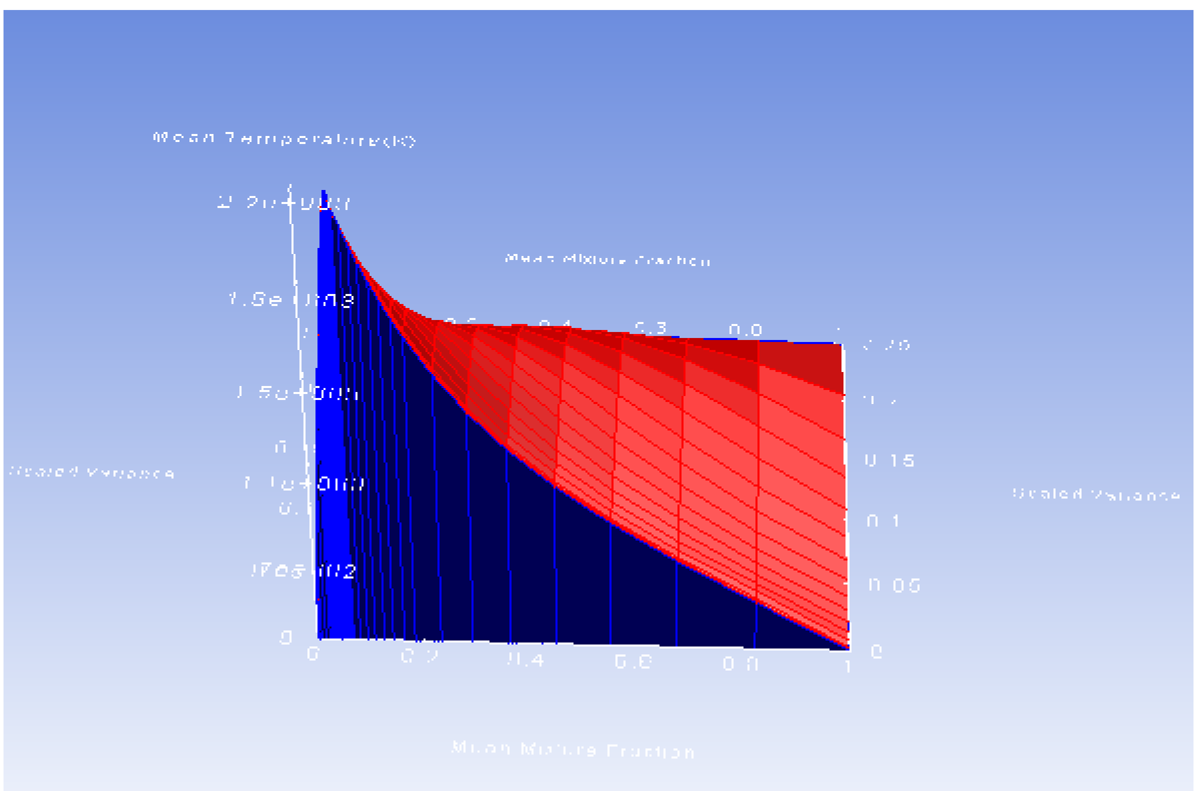
iv. Click the **Display PDF Table...** button.



A. In the **PDF Table** dialog box, retain the default parameters and click **Display** (Figure 14.5: Non-Adiabatic Temperature Look-Up Table on the Adiabatic Enthalpy Slice (p. 596)).

B. Close the **PDF Table** dialog box.

Figure 14.5: Non-Adiabatic Temperature Look-Up Table on the Adiabatic Enthalpy Slice



The 3D look-up tables are reviewed on a slice-by-slice basis. By default, the slice selected corresponds to the adiabatic enthalpy values. You can also select other slices of constant enthalpy for display.

The maximum and minimum values for mean temperature and the corresponding mean mixture fraction is also reported in the console. The maximum mean temperature is reported as 2246 K at a mean mixture fraction of 0.058.

g. Save the PDF output file (`berl.pdf`).

File → **Write** → **PDF...**

i. Retain `berl.pdf` for **PDF File** name.

ii. Click **OK** to write the file.

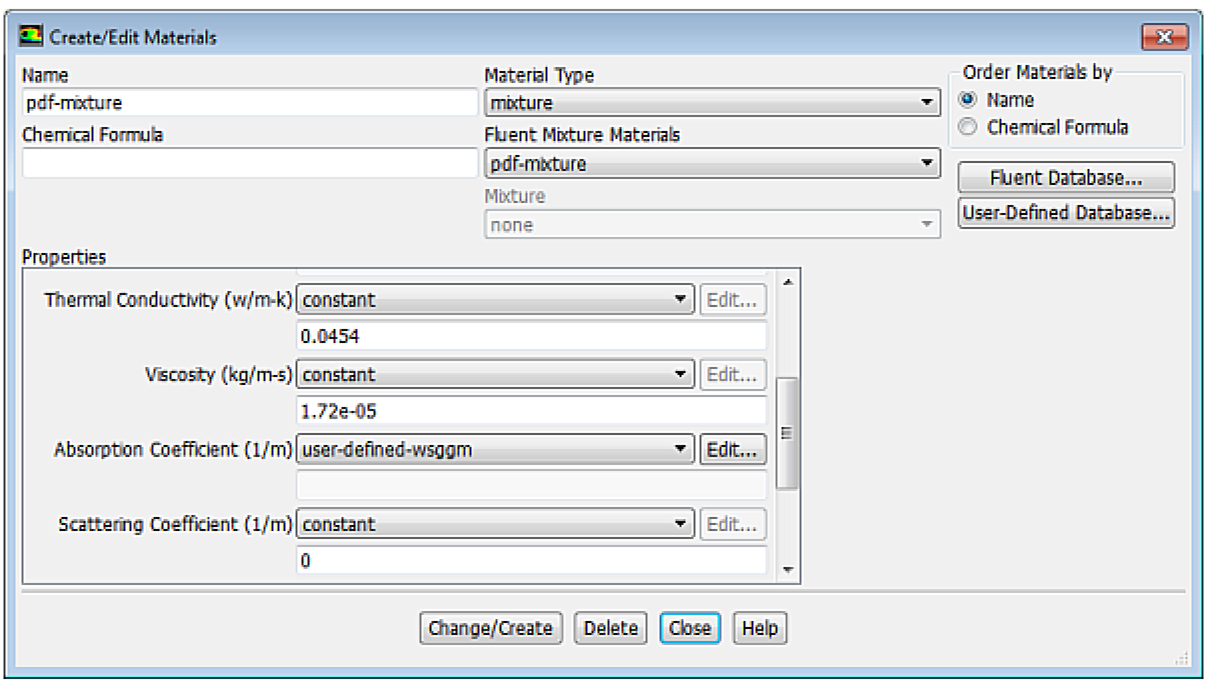
By default, the file will be saved as formatted (ASCII, or text). To save a binary (unformatted) file, enable the ***Write Binary Files*** *option in the* ***Select File*** *dialog box.*

h. Click **OK** to close the **Species Model** dialog box.

14.4.5. Defining Materials and Properties

1. Specify the continuous phase (**pdf-mixture**) material.

Setup → **Materials** → **pdf-mixture** → **Create/Edit...**



All thermodynamic data for the continuous phase, including density, specific heat, and formation enthalpies are extracted from the chemical database when the non-premixed combustion model is used. These properties are transferred to the ***pdf-mixture*** *material, for which only transport properties, such as viscosity and thermal conductivity, need to be defined. The* ***Density*** *and* ***Cp (Specific Heat)*** *laws cannot be altered: these properties are stored in the non-premixed combustion look-up tables.*

Under ***Properties****, you can click the* ***View...*** *button next to* ***Mixture Species*** *to view the species included in the* ***pdf-mixture*** *material. These are the species included during the system chemistry setup.*

a. Under **Properties**, from the **Absorption Coefficient** drop-down list, select **wsggm-domain-based**.

> **Tip**
>
> Scroll down to view the **Absorption Coefficient** option.

This specifies a composition-dependent absorption coefficient, using the weighted-sum-of-gray-gases model. WSGGM-domain-based is a variable coefficient that uses a length scale, based on the geometry of the model.

For more details, see radiation in combusting flows in the Fluent Theory Guide.

b. Click **Change/Create** and close the **Create/Edit Materials** dialog box.

ANSYS Fluent uses the gas law to compute the mixture density and a mass-weighted mixing law to compute the mixture C_p. When the non-premixed combustion model is used, do not alter the properties of the individual species. This will create an inconsistency with the PDF look-up table.

14.4.6. Specifying Boundary Conditions

1. Read the boundary conditions profile file.

   File → **Read** → **Profile...**

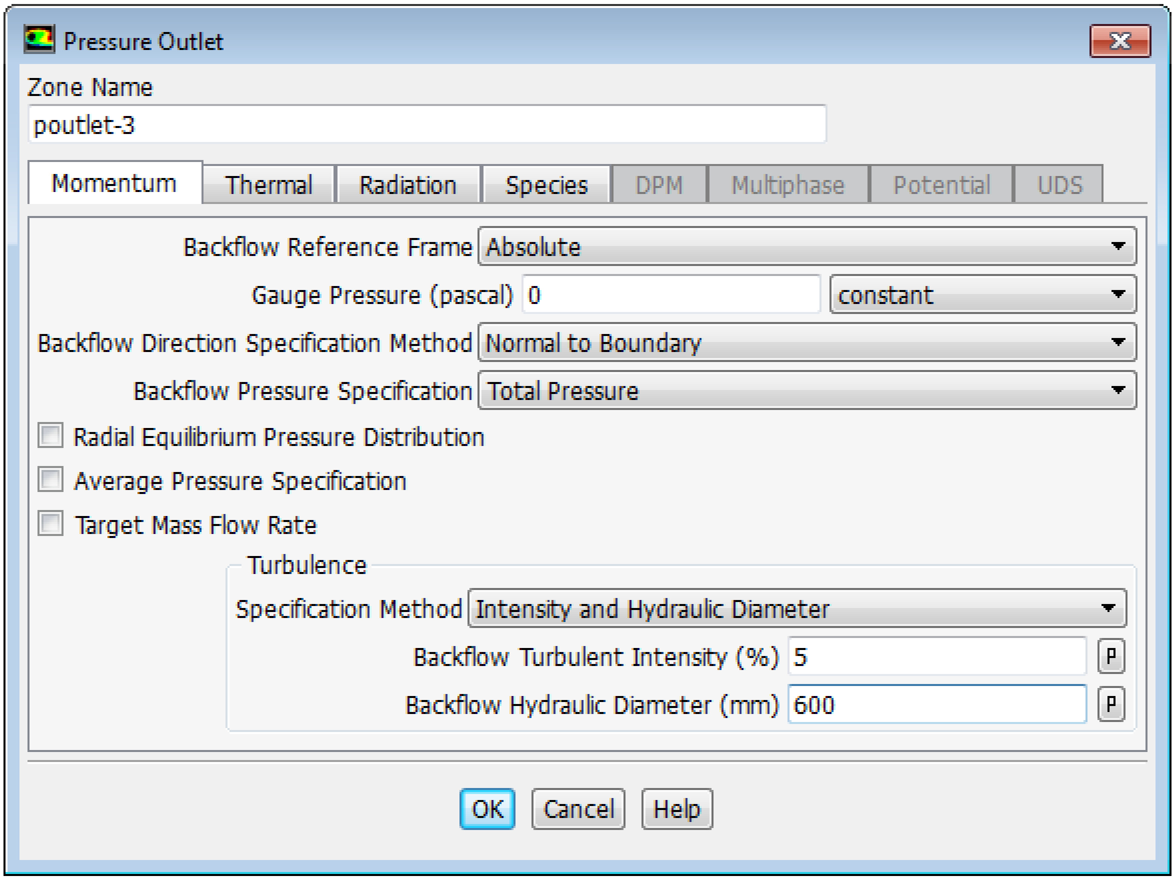
   a. Select **berl.prof** from the **Select File** dialog box.

   b. Click **OK**.

   *The CFD solution for reacting flows can be sensitive to the boundary conditions, in particular the incoming velocity field and the heat transfer through the walls. Here, you will use profiles to specify the velocity at **air-inlet-4**, and the wall temperature for **wall-9**. The latter approach of fixing the wall temperature to measurements is common in furnace simulations, to avoid modeling the wall convective and radiative heat transfer. The data used for the boundary conditions was obtained from experimental data [1].*

2. Set the boundary conditions for the pressure outlet (**poutlet-3**).

   Setup → **Boundary Conditions** → **poutlet-3** → **Edit...**



a. In the **Turbulence** group, from the **Specification Method** drop-down list, select **Intensity and Hydraulic Diameter**.

b. Retain 5% for **Backflow Turbulent Intensity**.

c. Enter `600` mm for **Backflow Hydraulic Diameter**.

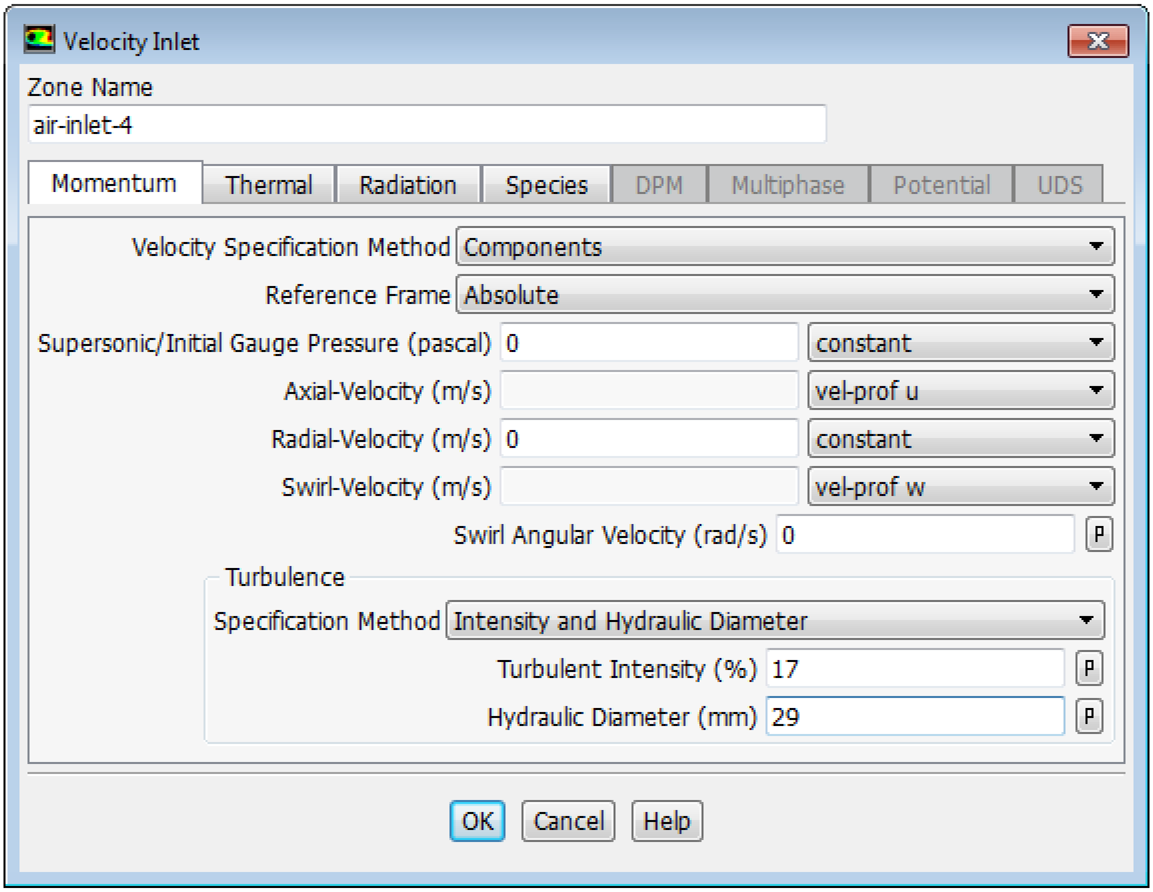
d. Click the **Thermal** tab and enter `1300` K for **Backflow Total Temperature**.

e. Click **OK** to close the **Pressure Outlet** dialog box.

The exit gauge pressure of zero defines the system pressure at the exit to be the operating pressure. The backflow conditions for scalars (temperature, mixture fraction, turbulence parameters) will be used only if flow is entrained into the domain through the exit. It is a good idea to use reasonable values in case flow reversal occurs at the exit at some point during the solution process.

3. Set the boundary conditions for the velocity inlet (**air-inlet-4**).

Setup → **Boundary Conditions** → **air-inlet-4** → **Edit...**



a. From the **Velocity Specification Method** drop-down list, select **Components**.

b. From the **Axial-Velocity** drop-down list, select **vel-prof u**.

c. From the **Swirl-Velocity** drop-down list, select **vel-prof w**.

d. In the **Turbulence** group, from the **Specification Method** drop-down list, select **Intensity and Hydraulic Diameter**.

e. Enter 17% for **Turbulent Intensity**.

f. Enter 29 mm for **Hydraulic Diameter**.

Turbulence parameters are defined based on intensity and length scale. The relatively large turbulence intensity of 17% may be typical for combustion air flows.

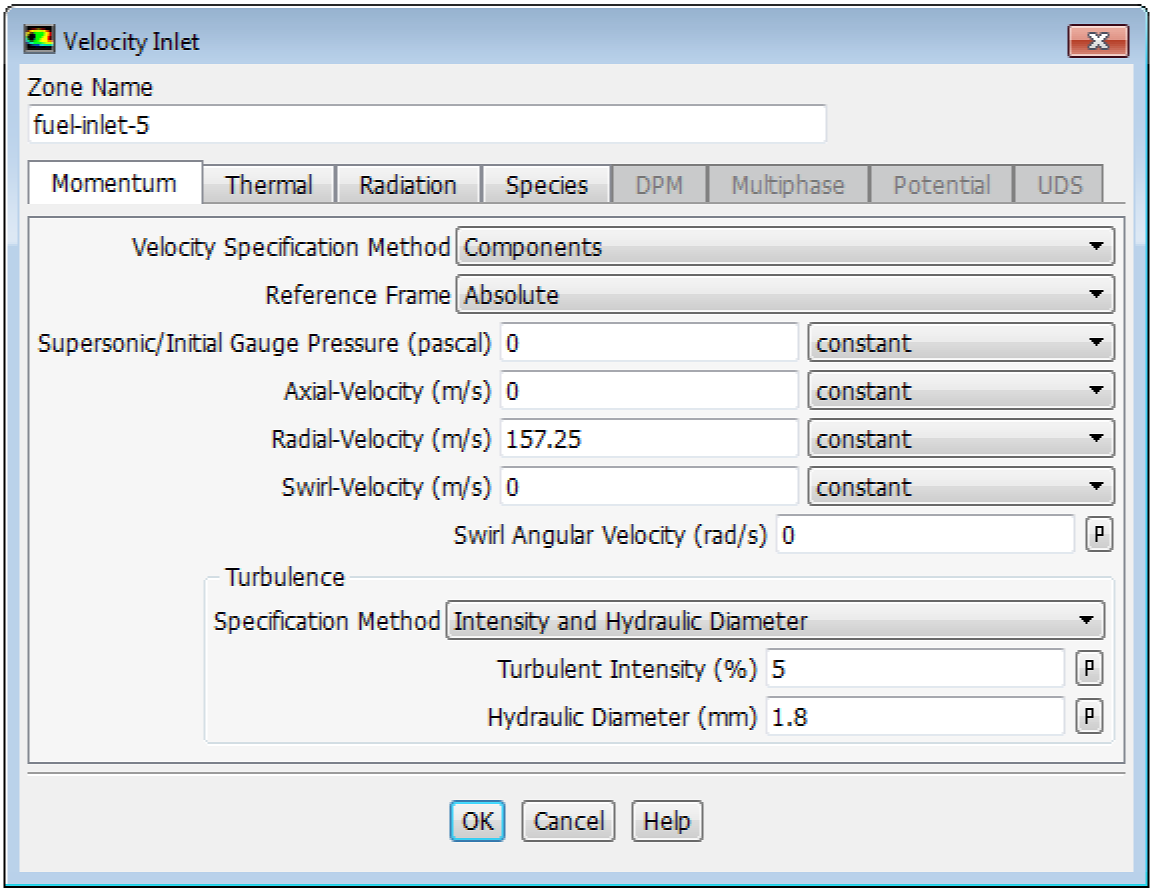
g. Click the **Thermal** tab and enter 312 K for **Temperature**.

h. Click the **Species** tab. For the non-premixed combustion calculation, you have to define the inlet **Mean Mixture Fraction** and **Mixture Fraction Variance**. In this case, the gas phase air inlet has a zero mixture fraction. Therefore, you can retain the zero default settings.

i. Click **OK** to close the **Velocity Inlet** dialog box.

4. Set the boundary conditions for the velocity inlet (**fuel-inlet-5**).

Setup → **Boundary Conditions** → **fuel-inlet-5** → **Edit...**



a. From the **Velocity Specification Method** drop-down list, select **Components**.

b. Enter `157.25` m/s for **Radial-Velocity**.

c. In the **Turbulence** group, from the **Specification Method** drop-down list, select **Intensity and Hydraulic Diameter**.

d. Retain 5% for **Turbulent Intensity**.

e. Enter `1.8` mm for **Hydraulic Diameter**.

   The hydraulic diameter has been set to twice the height of the 2D inlet stream.

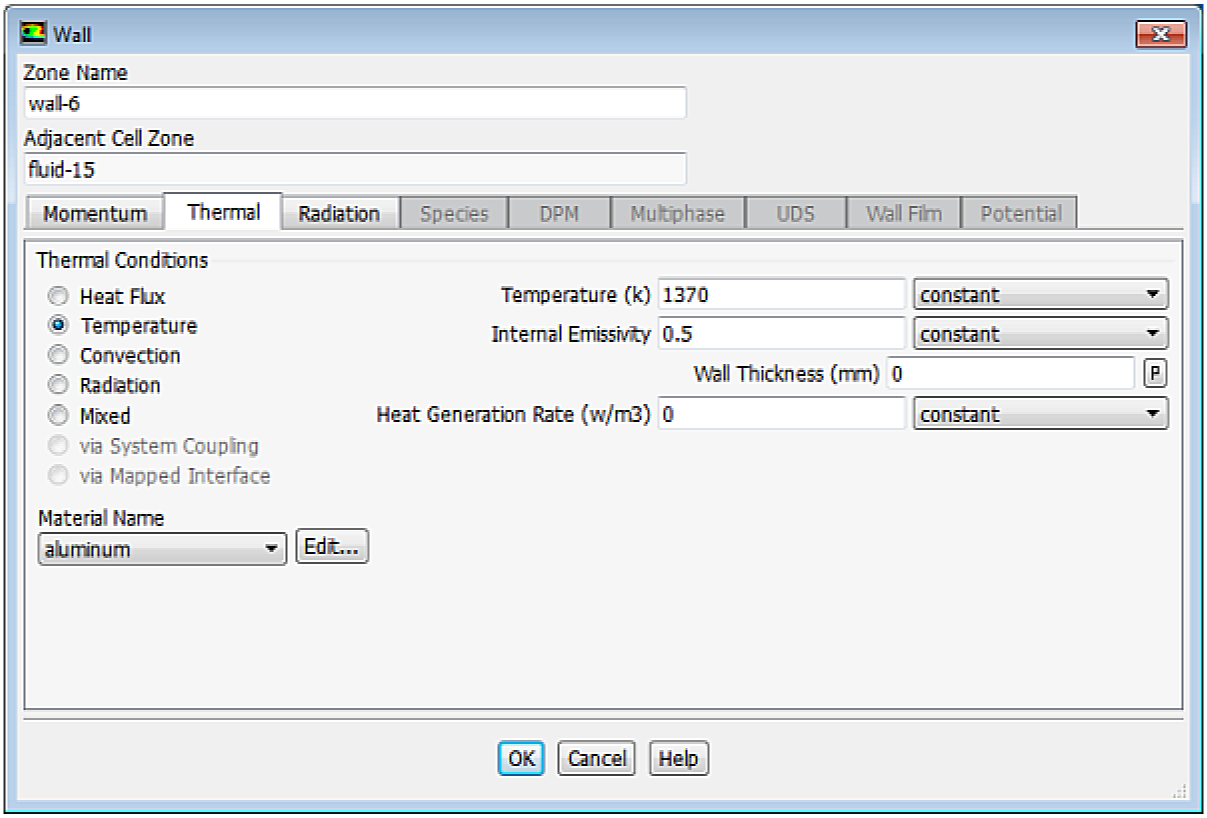
f. Click the **Thermal** tab and enter `308` K for **Temperature**.

g. Click the **Species** tab and enter `1` for **Mean Mixture Fraction** for the fuel inlet.

h. Click **OK** to close the **Velocity Inlet** dialog box.

5. Set the boundary conditions for **wall-6**.

   Setup → **Boundary Conditions** → **wall-6** → **Edit...**



a. Click the **Thermal** tab.

   i. In the **Thermal Conditions** list, select **Temperature**.

   ii. Enter `1370` K for **Temperature**.

   iii. Enter `0.5` for **Internal Emissivity**.

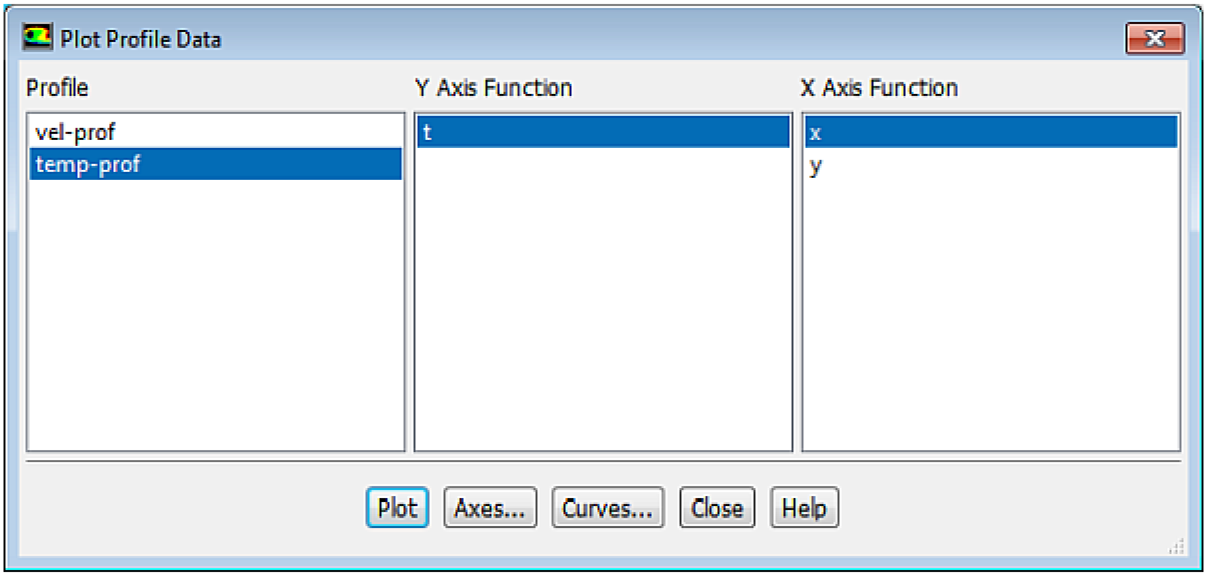
b. Click **OK** to close the **Wall** dialog box.

6. Similarly, set the boundary conditions for **wall-7** through **wall-13** using the following values:

Zone Name	Temperature	Internal Emissivity
wall-7	`312`	`0.5`
wall-8	`1305`	`0.5`
wall-9	**temp-prof t** (from the drop-down list)	`0.5`
wall-10	`1100`	`0.5`
wall-11	`1273`	`0.5`
wall-12	`1173`	`0.5`
wall-13	`1173`	`0.5`

7. Plot the profile of temperature for the wall furnace (**wall-9**).

Postprocessing → **Plots** → **Profile Data...**

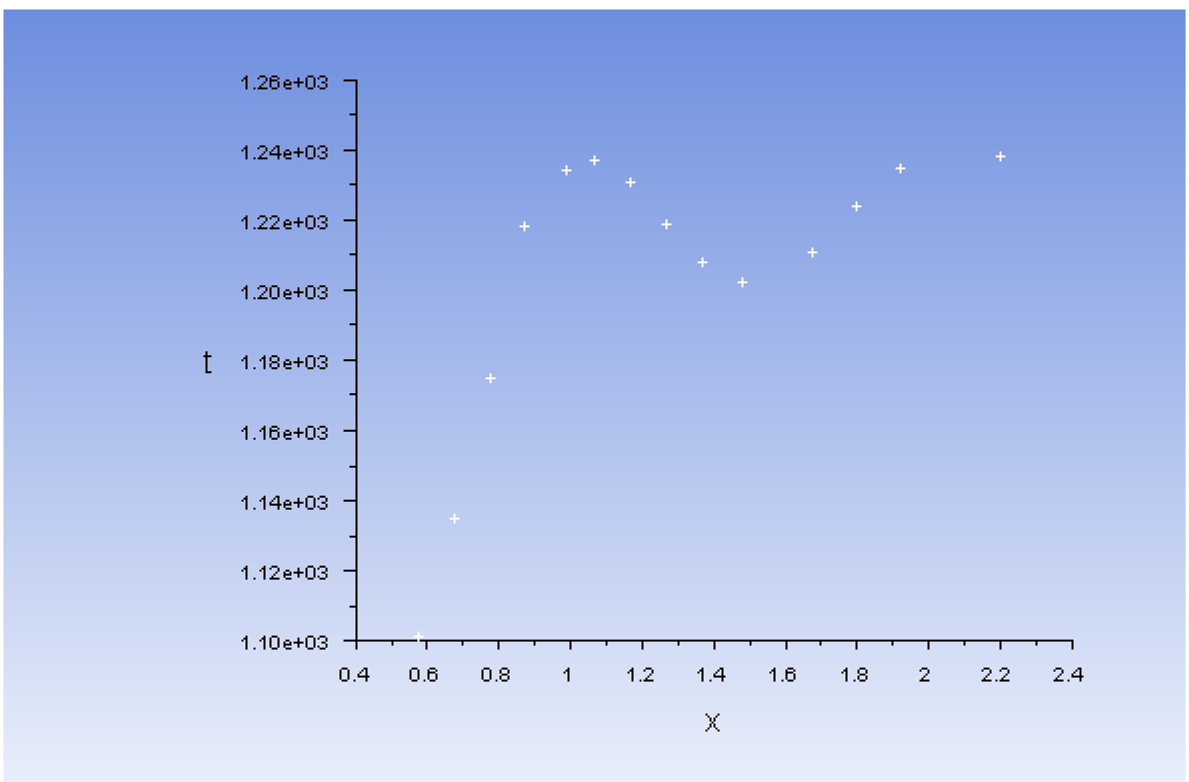


a. From the **Profile** selection list, select **temp-prof**.

b. Retain the selection of **t** and **x** from the **Y Axis Function** and **X Axis Function** selection lists, respectively.

c. Click **Plot** (Figure 14.6: Profile Plot of Temperature for wall-9 (p. 603)).

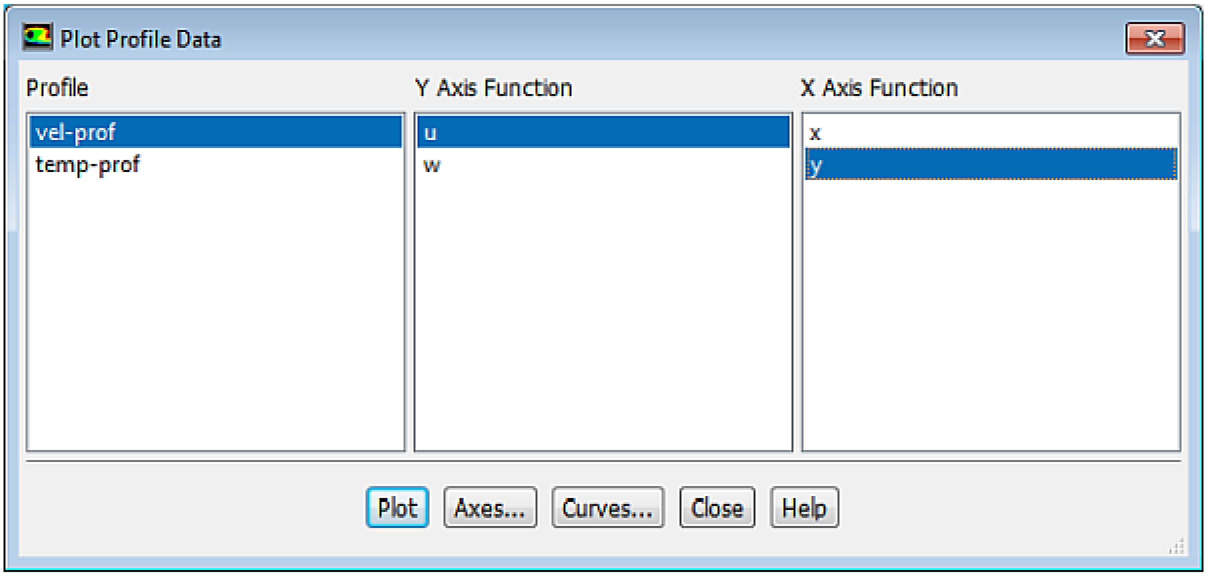
Figure 14.6: Profile Plot of Temperature for wall-9



8. Plot the profiles of velocity for the swirling air inlet (**air-inlet-4**).

   a. Plot the profile of axial-velocity for the swirling air inlet.

Postprocessing → Plots → Profile Data...



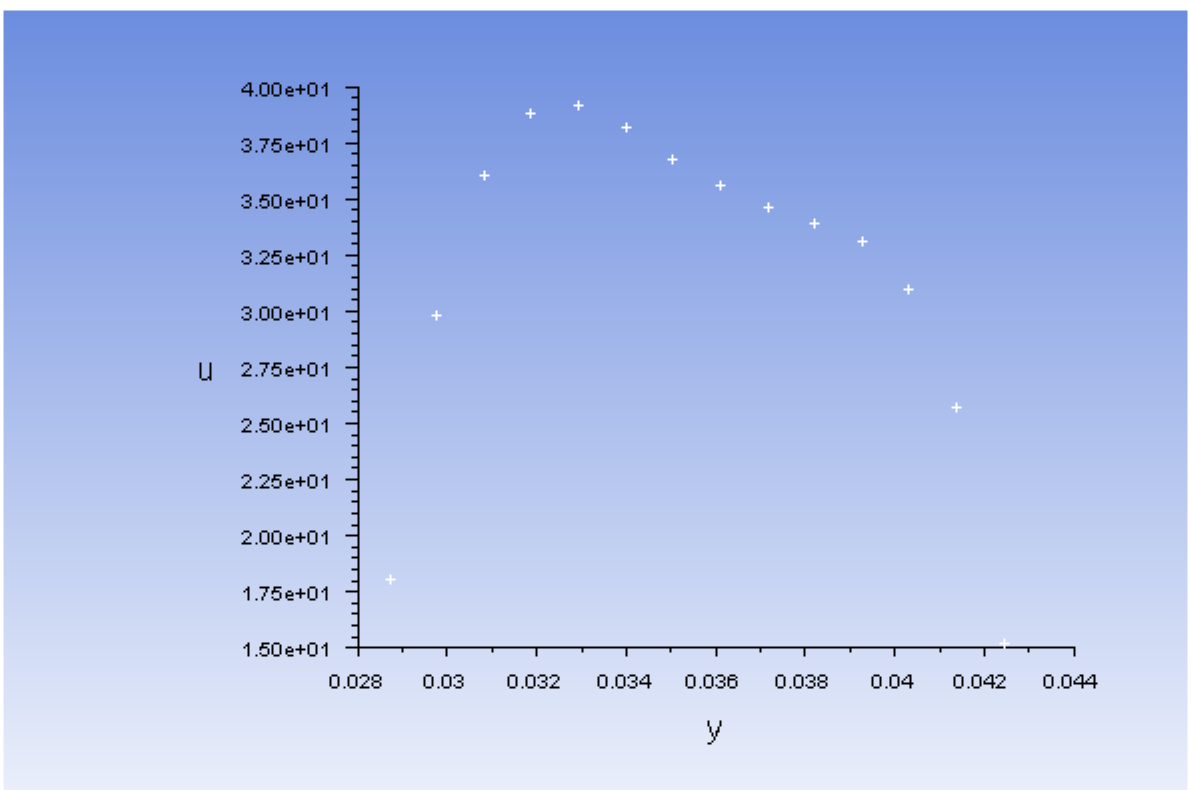
i. From the **Profile** selection list, select **vel-prof**.

ii. From the **Y Axis Function** selection list, retain the selection of **u**.

iii. From the **X Axis Function** selection list, select **y**.

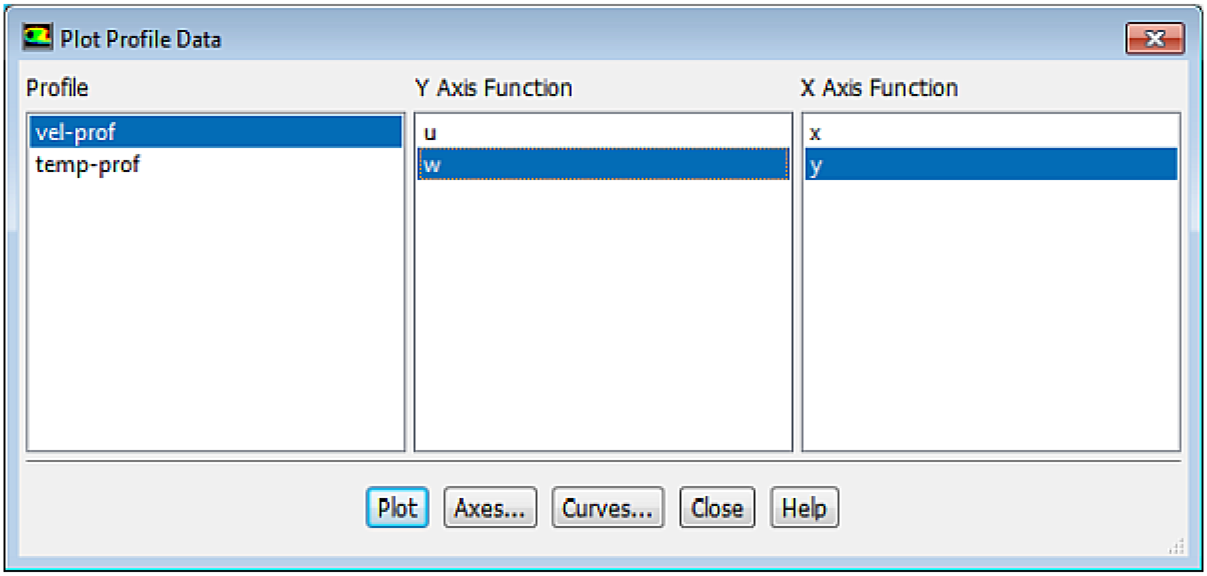
iv. Click **Plot** (Figure 14.7: Profile Plot of Axial-Velocity for the Swirling Air Inlet (air-inlet-4) (p. 604)).

Figure 14.7: Profile Plot of Axial-Velocity for the Swirling Air Inlet (air-inlet-4)



b. Plot the profile of swirl-velocity for swirling air inlet.

Postprocessing → **Plots** → **Profile Data...**



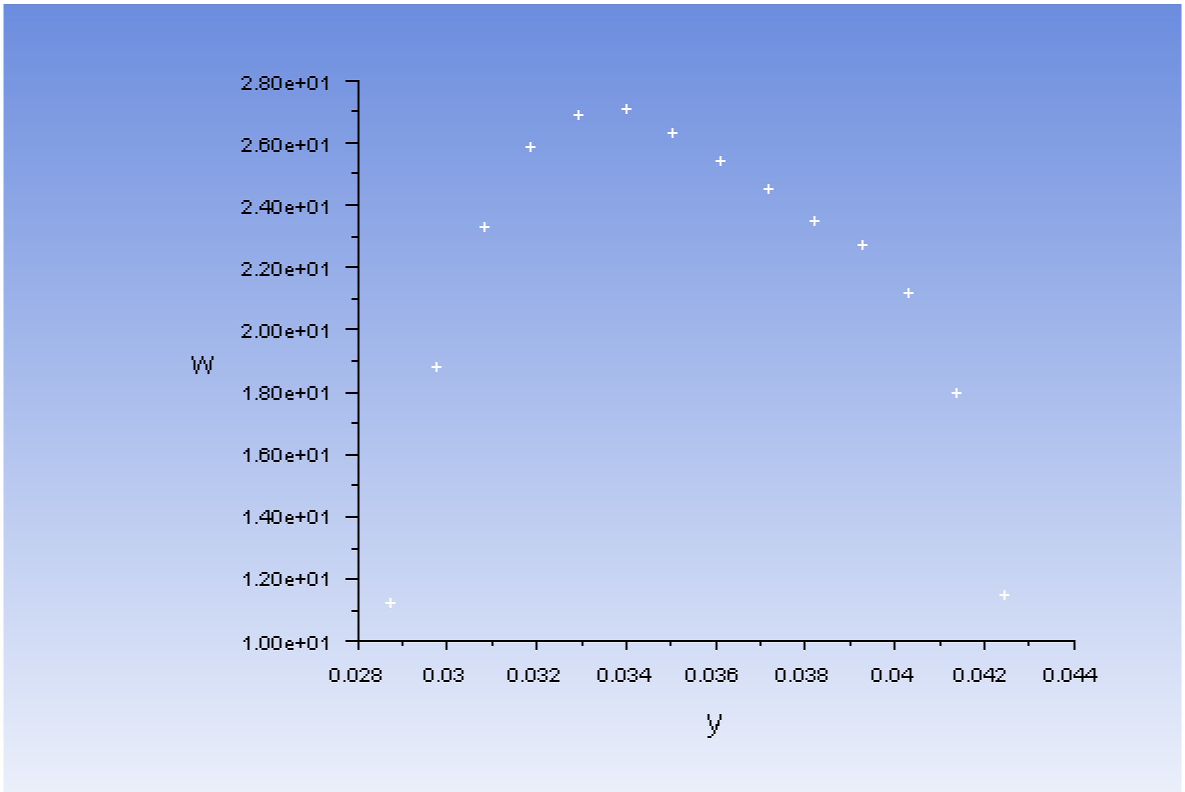
i. From the **Profile** selection list, retain the selection of **vel-prof**.

ii. From the **Y Axis Function** selection list, select **w**.

iii. From the **X Axis Function** selection list, retain the selection of **y**.

iv. Click **Plot** (Figure 14.8: Profile Plot of Swirl-Velocity for the Swirling Air Inlet (air-inlet-4) (p. 606)) and close the **Plot Profile Data** dialog box.

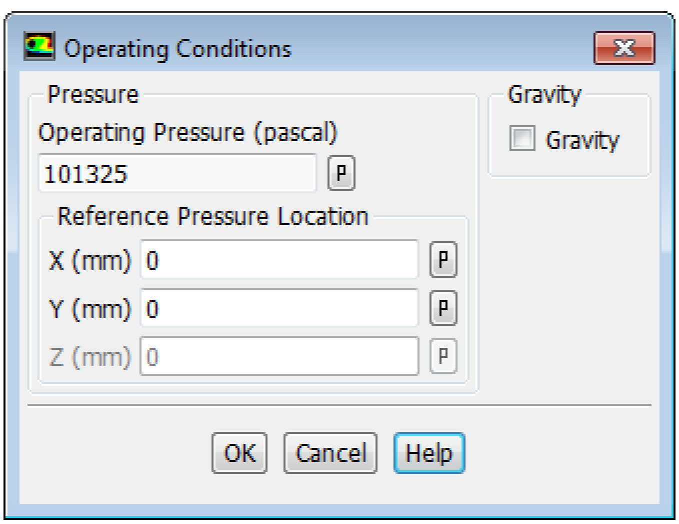
Figure 14.8: Profile Plot of Swirl-Velocity for the Swirling Air Inlet (air-inlet-4)



14.4.7. Specifying Operating Conditions

1. Retain the default operating conditions.

Setup → **Boundary Conditions** → **Operating Conditions...**

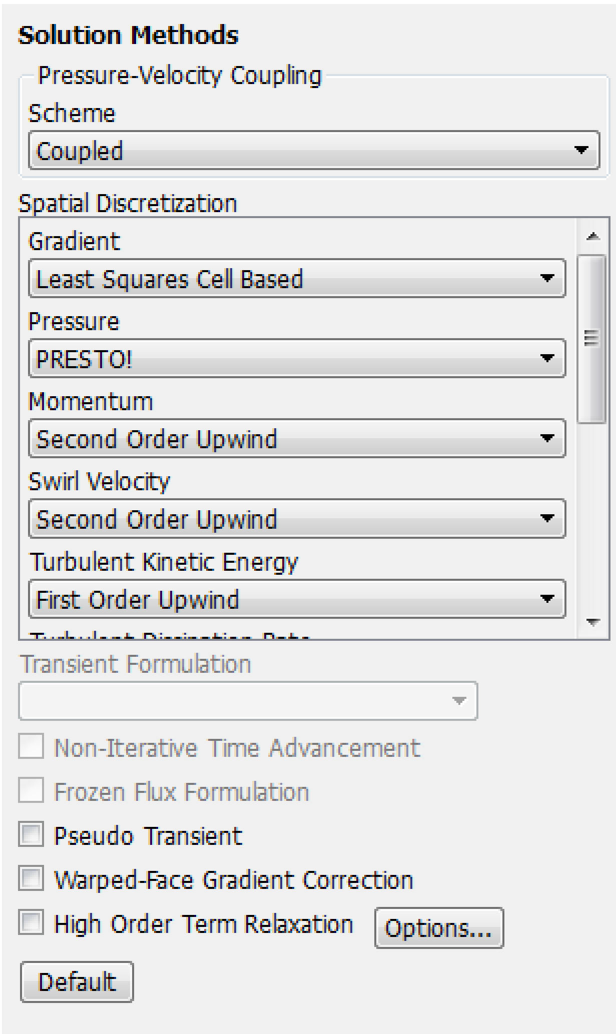


*The **Operating Pressure** was already set in the PDF table generation in Specifying the Models (p. 590).*

14.4.8. Obtaining Solution

1. Set the solution parameters.

   Solving → **Solution** → **Methods...**



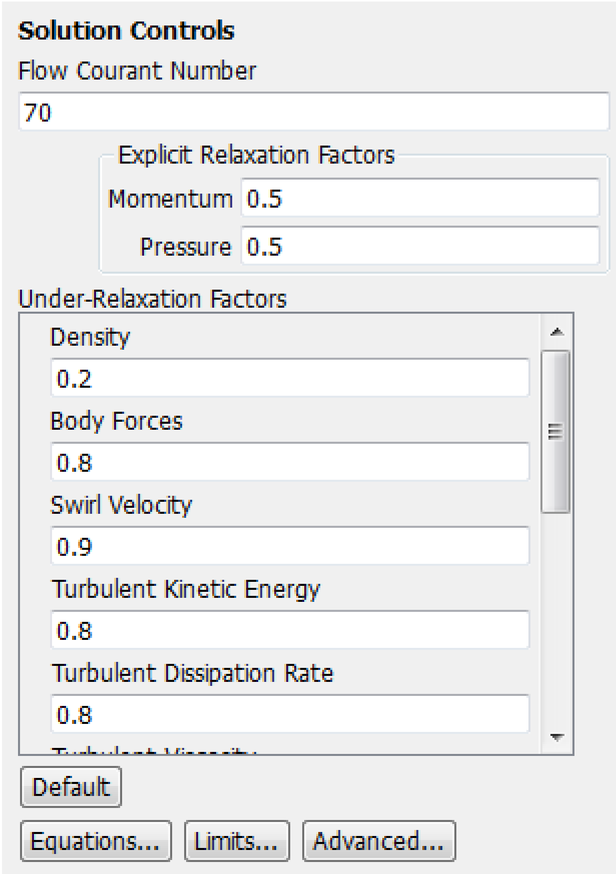
   a. In the **Pressure-Velocity Coupling** group, from the **Scheme** drop-down list, select **Coupled**.

   b. In the **Spatial Discretization** group, from the **Pressure** drop-down list, select **PRESTO!**.

   c. Retain the other default selections and settings.

2. Set the solution controls.

   Solving → **Controls** → **Controls...**



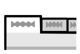
a. Enter 70 for **Flow Courant Number**.

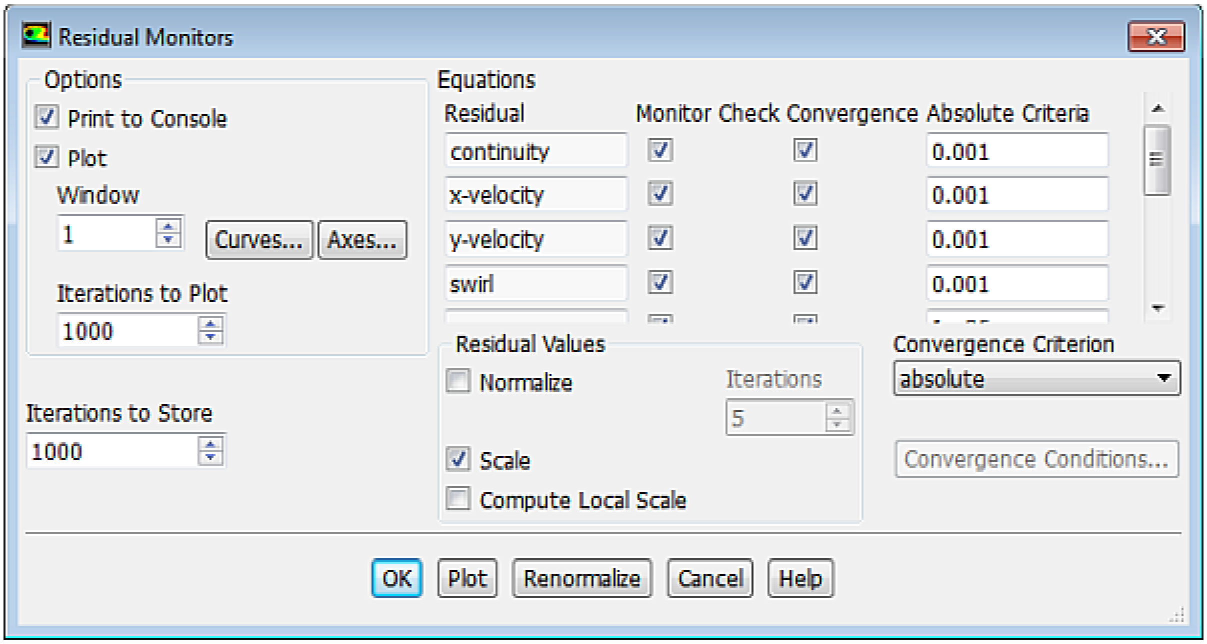
b. Set the following parameters in the **Under-Relaxation Factors** group box:

Under-Relaxation Factor	Value
Density	`0.2`
Body Forces	`0.8`

The default under-relaxation factors are considered to be too aggressive for reacting flow cases with high swirl velocity.

3. Enable the display of residuals during the solution process.

 **Solving** → **Reports** → **Residuals...**

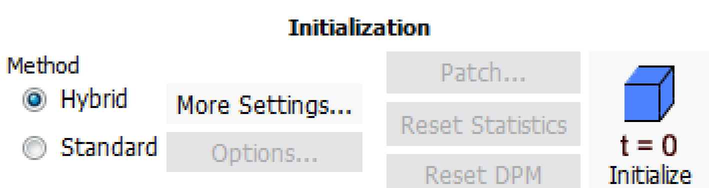


a. Ensure that the **Plot** is enabled in the **Options** group box.

b. Click **OK** to close the **Residual Monitors** dialog box.

4. Initialize the flow field.

Solving → **Initialization**



a. Retain the **Method** at the default of **Hybrid** in the **Initialization** group.

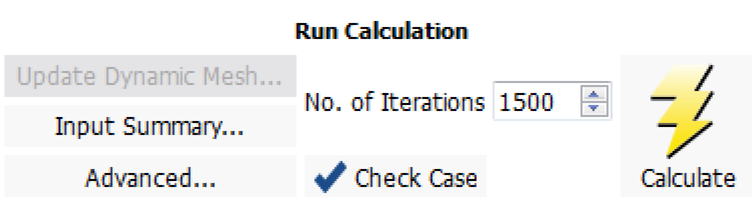
b. Click **Initialize**.

5. Save the case file (`berl-1.cas.gz`).

File → **Write** → **Case...**

6. Start the calculation by requesting 1500 iterations.

Solving → **Run Calculation** →



The solution will converge in approximately 1060 iterations.

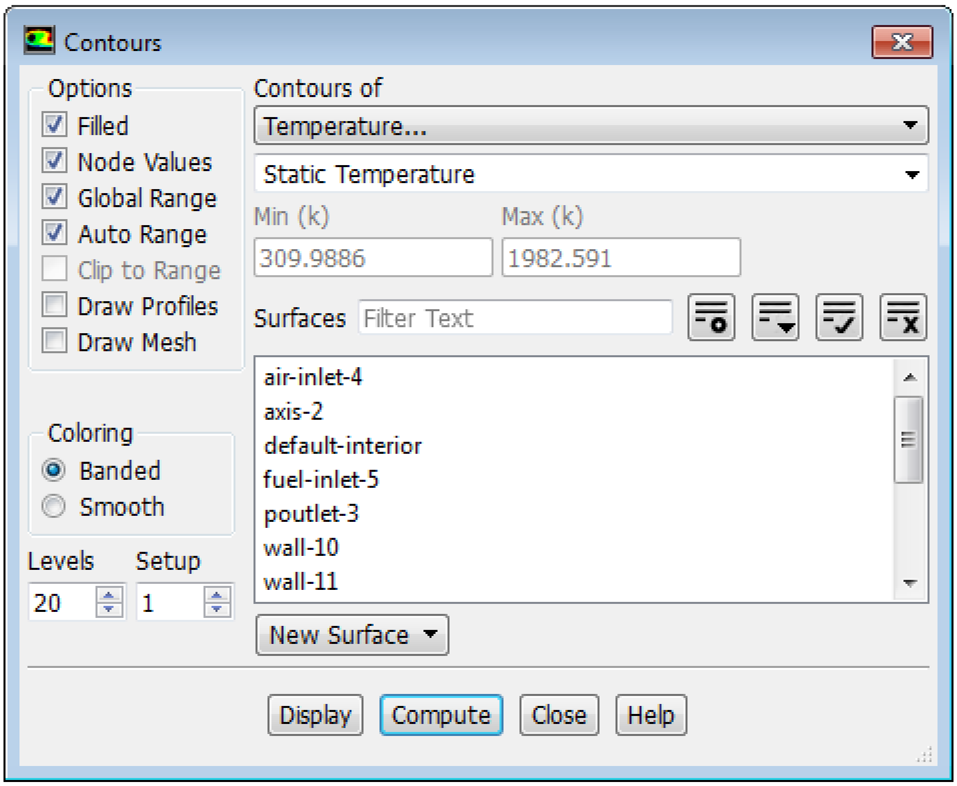
7. Save the converged solution (`berl-1.dat.gz`).

   File → **Write** → **Data...**

14.4.9. Postprocessing

1. Display the predicted temperature field (Figure 14.9: Temperature Contours (p. 611)).

   Postprocessing → **Graphics** → **Contours** → **Edit...**



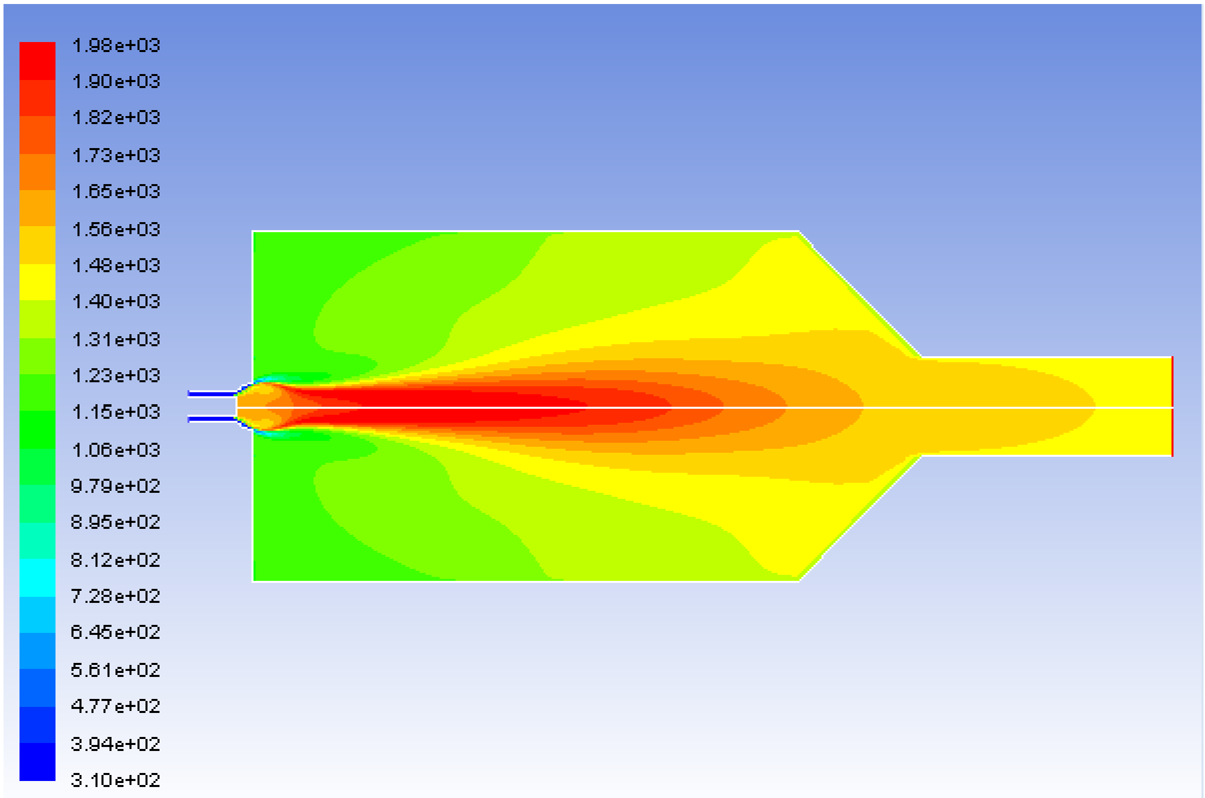
   a. Enable **Filled** in the **Options** group box.

   b. Select **Temperature...** and **Static Temperature** from the **Contours of** drop-down lists.

   c. Click **Display**.

   The peak temperature in the system is 1981 K.

Figure 14.9: Temperature Contours



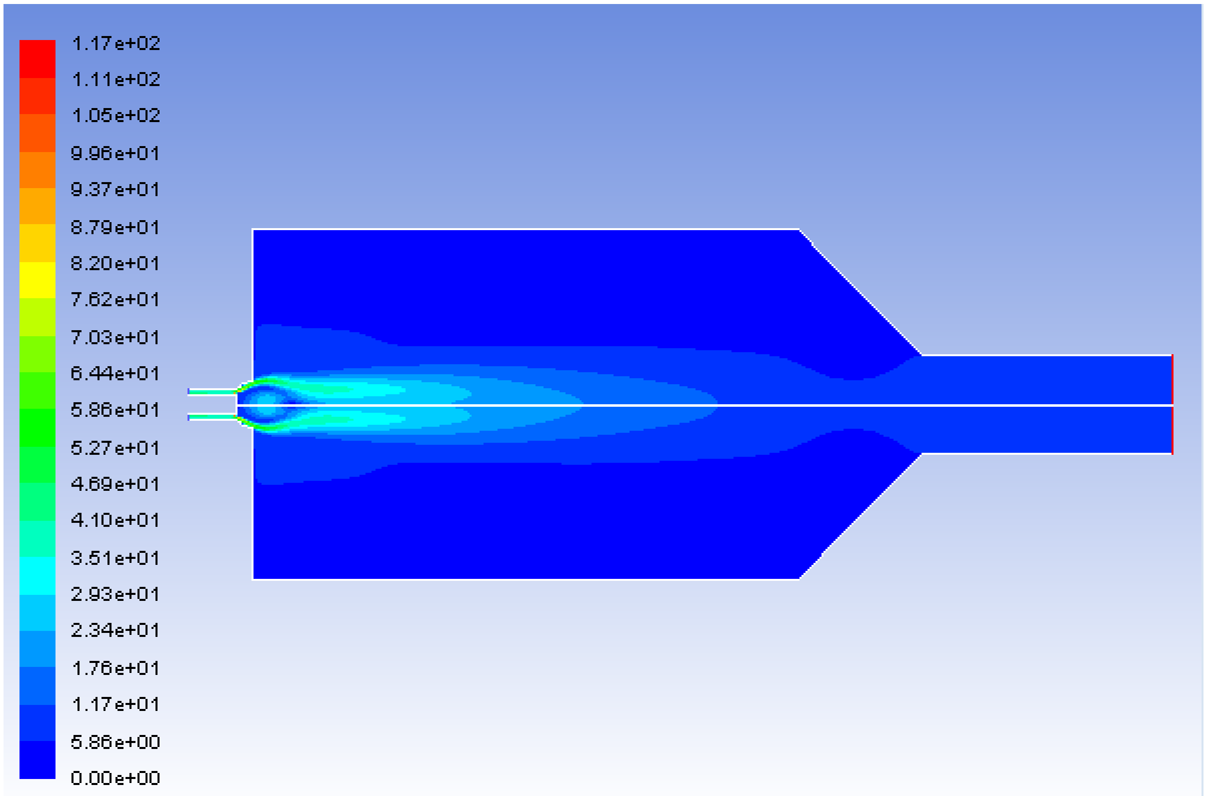
2. Display contours of velocity (Figure 14.10: Velocity Contours (p. 612)).

   Postprocessing → **Graphics** → **Contours** → **Edit...**

   a. Select **Velocity...** and **Velocity Magnitude** from the **Contours of** drop-down lists.

   b. Click **Display**.

Figure 14.10: Velocity Contours



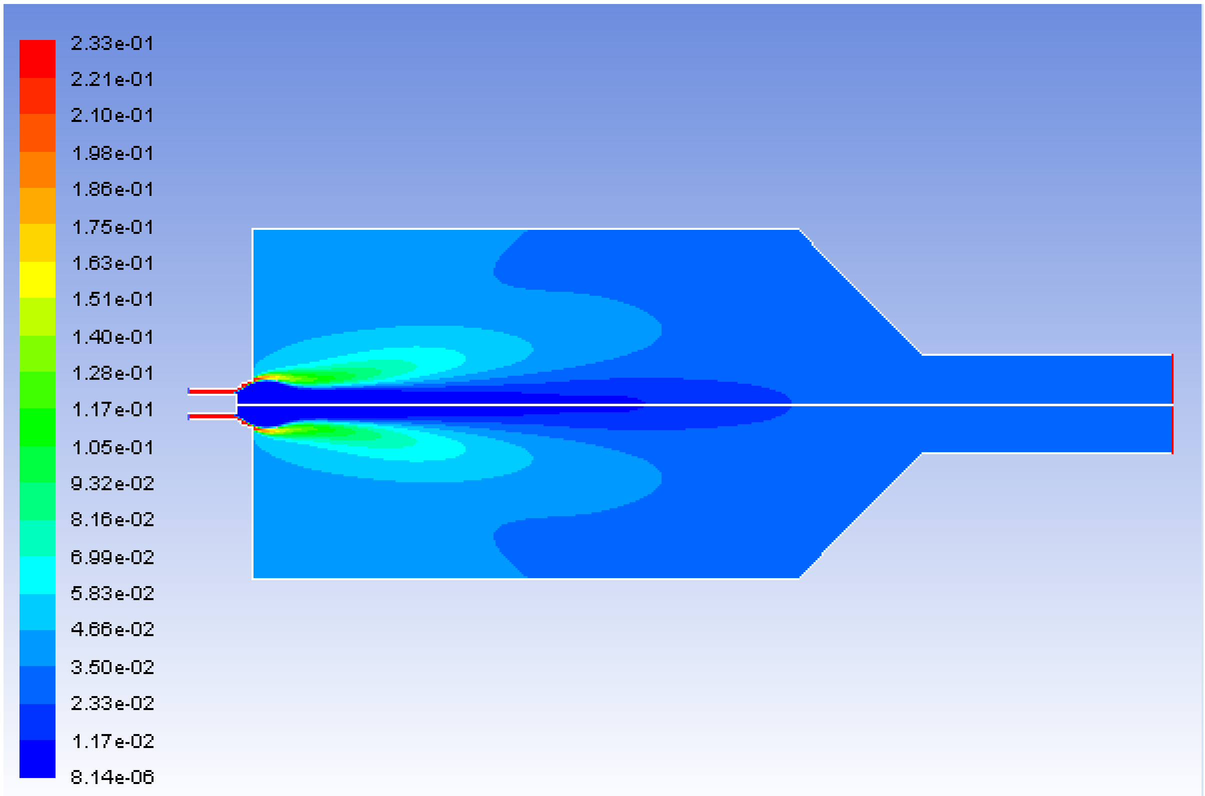
3. Display the contours of mass fraction of O_2 (Figure 14.11: Contours of Mass Fraction of o2 (p. 613)).

 **Postprocessing** → **Graphics** → **Contours** → **Edit...**

a. Select **Species...** and **Mass fraction of o2** from the **Contours of** drop-down lists.

b. Click **Display** and close the **Contours** dialog box.

Figure 14.11: Contours of Mass Fraction of o2

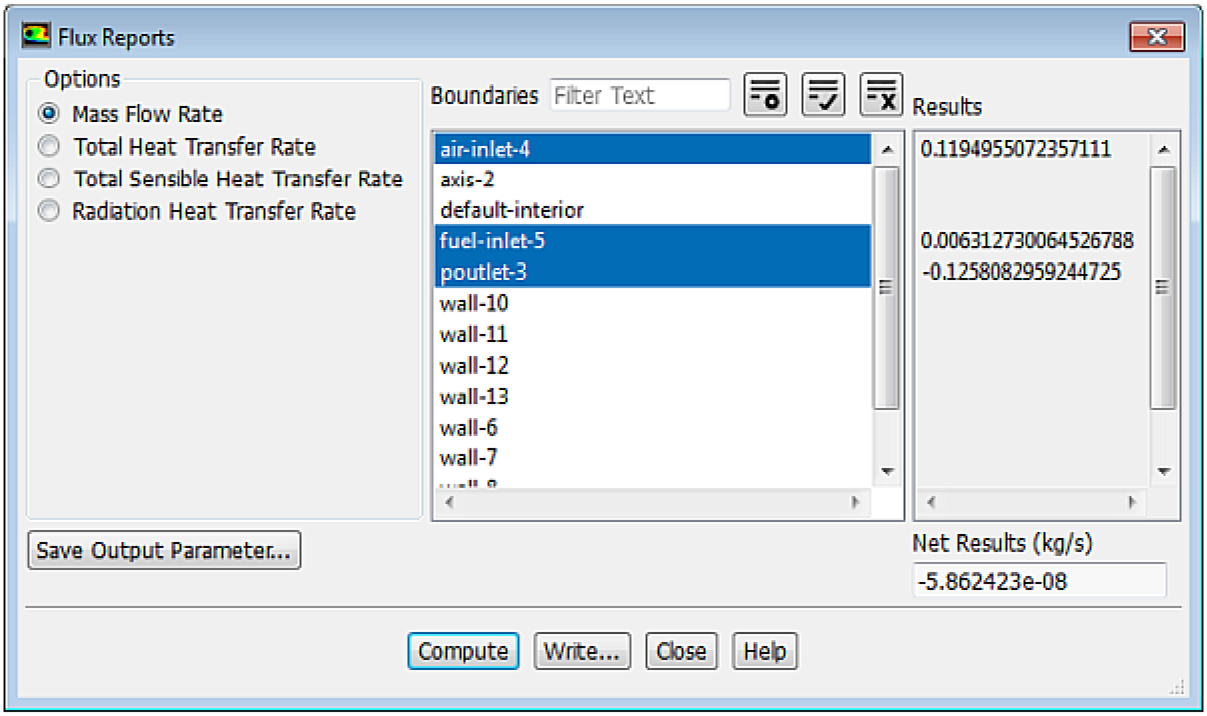


14.4.10. Energy Balances Reporting

ANSYS Fluent can report the overall energy balance and details of the heat and mass transfer.

1. Compute the gas phase mass fluxes through the domain boundaries.

   Postprocessing → **Reports** → **Fluxes...**



a. Retain the default selection of **Mass Flow Rate** in the **Options** group box.

b. Select **air-inlet-4**, **fuel-inlet-5**, and **poutlet-3** from the **Boundaries** selection list.

c. Click **Compute**.

The net mass imbalance should be a small fraction of the total flux through the system. If a significant imbalance occurs, you should decrease your residual tolerances by at least an order of magnitude and continue iterating.

2. Compute the fluxes of heat through the domain boundaries.

Postprocessing → **Reports** → **Fluxes...**

a. Select **Total Heat Transfer Rate** in the **Options** group box.

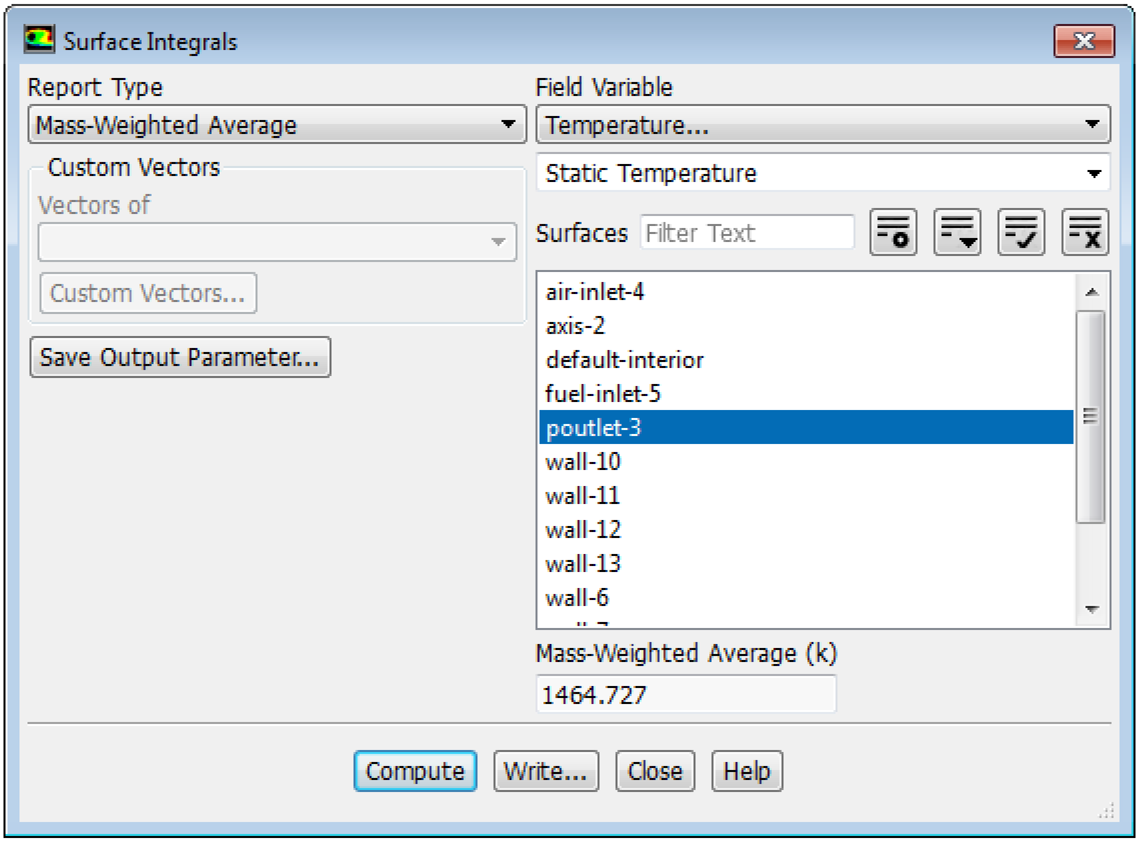
b. Select all the zones from the **Boundaries** selection list.

c. Click **Compute**, examine the resulting values, and close the **Flux Reports** dialog box.

The value will be displayed in the console. Positive flux reports indicate heat addition to the domain. Negative values indicate heat leaving the domain. Again, the net heat imbalance should be a small fraction (for example, 0.5% or less) of the total energy flux through the system. The reported value may change for different runs.

3. Compute the mass weighted average of the temperature at the pressure outlet.

Postprocessing → **Reports** → **Surface Integrals...**



a. Select **Mass-Weighted Average** from the **Report Type** drop-down list.

b. Select **Temperature...** and **Static Temperature** from the **Field Variable** drop-down lists.

c. Select **poutlet-3** from the **Surfaces** selection list.

d. Click **Compute**.

   A value of approximately 1464 K will be displayed in the console.

e. Close the **Surface Integrals** dialog box.

14.5. Summary

In this tutorial you learned how to use the non-premixed combustion model to represent the gas phase combustion chemistry. In this approach the fuel composition was defined and assumed to react according to the equilibrium system data. This equilibrium chemistry model can be applied to other turbulent, diffusion-reaction systems. You can also model gas combustion using the finite-rate chemistry model.

You also learned how to set up and solve a gas phase combustion problem using the Discrete Ordinates radiation model, and applying the appropriate absorption coefficient.

14.6. References

1. A. Sayre, N. Lallement, J. Dugu, and R. Weber "Scaling Characteristics of Aerodynamics and Low-NOx Properties of Industrial Natural Gas Burners", The SCALING 400 Study, Part IV: The 300 KW BERL Test Results, IFRF Doc No F40/y/11, International Flame Research Foundation, The Netherlands.

14.7. Further Improvements

This tutorial guides you through the steps to first generate an initial solution, and then to reach a more accurate second-order solution. You may be able to increase the accuracy of the solution even further by using an appropriate higher-order discretization scheme and by adapting the mesh. Mesh adaption can also ensure that your solution is independent of the mesh. These steps are demonstrated in Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121).

Chapter 15: Modeling Surface Chemistry

This tutorial is divided into the following sections:

15.1. Introduction

In chemically reacting laminar flows, such as those encountered in chemical vapor deposition (CVD) applications, accurate modeling of time-dependent hydrodynamics, heat and mass transfer, and chemical reactions (including wall surface reactions) is important.

In this tutorial, surface reactions are considered. Modeling the reactions taking place at gas-solid interfaces is complex and involves several elementary physicochemical processes like adsorption of gas-phase species on the surface, chemical reactions occurring on the surface, and desorption of gases from the surface back to the gas phase.

This tutorial demonstrates how to do the following:

- Create new materials and set the mixture properties.
- Model surface reactions involving site species.
- Enable physical models and define boundary conditions for a chemically reacting laminar flow involving wall surface reactions.
- Calculate the deposition solution using the pressure-based solver.
- Examine the flow results using graphics.

15.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

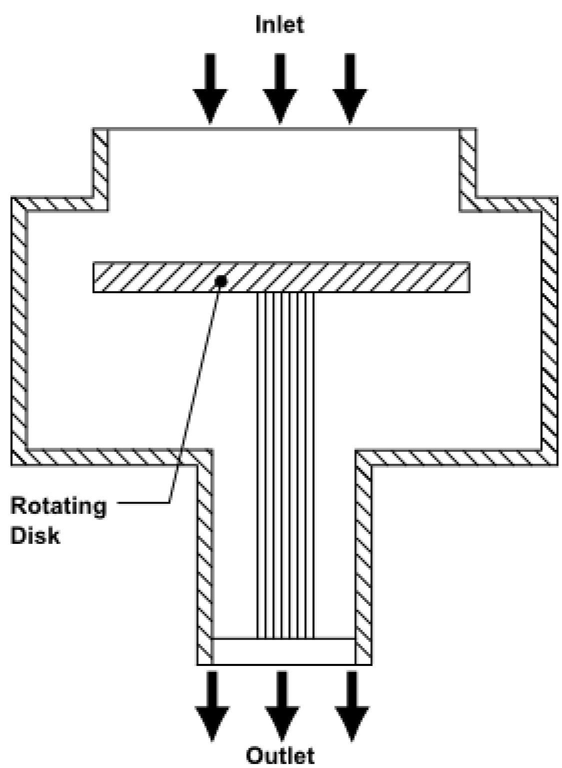
and that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

Before beginning with this tutorial, see the Fluent User's Guide for more information about species transport, chemically reacting flows, wall surface reaction modeling, and chemical vapor deposition. In particular, you should be familiar with the Arrhenius rate equation, as this equation is used for the surface reactions modeled in this tutorial.

15.3. Problem Description

A rotating disk CVD reactor for the growth of Gallium Arsenide (GaAs) shown in Figure 15.1: Schematic of the Reactor Configuration (p. 618) will be modeled.

Figure 15.1: Schematic of the Reactor Configuration



The process gases, Trimethyl Gallium ($Ga(CH_3)_3$) and Arsine (AsH_3) enter the reactor at 293 K through the inlet at the top. These gases flow over the hot, spinning disk depositing thin layers of gallium and arsenide on it in a uniform, repeatable manner. The disk rotation generates a radially pumping effect, which forces the gases to flow in a laminar manner down to the growth surface, outward across the disk, and finally to be discharged from the reactor.

The semiconductor materials Ga(s) and As(s) are deposited on the heated surface governed by the following surface reactions.

$$AsH_3 + Ga\_s \rightarrow Ga + As\_s + 1.5H_2 \tag{15.1}$$

$$Ga(CH_3)_3 + As\_s \rightarrow As + Ga\_s + 3CH_3 \tag{15.2}$$

The inlet gas is a mixture of Trimethyl Gallium, which has a mass fraction of 0.15, and Arsine, which has a mass fraction of 0.4, the remainder is hydrogen. The mixture velocity at the inlet is 0.02189 m/s. The disk rotates at 80 rad/sec. The top wall (wall-1) is heated to 473 K and the sidewalls (wall-2) of the reactor are maintained at 343 K. The susceptor (wall-4) is heated to a uniform temperature of 1023 K and the bottom wall (wall-6) is at 303 K. These CVD reactors are typically known as cold-wall reactors, where only the wafer surface is heated to higher temperatures, while the remaining reactor walls are maintained at low temperatures.

In this tutorial, simultaneous deposition of Ga and As is simulated and examined. The mixture properties and the mass diffusivity are determined based on kinetic theory. Detailed surface reactions with multiple sites and site species, and full multi-component/thermal diffusion effects are also included in the simulation.

The purpose of this tutorial is to demonstrate surface reaction capabilities in ANSYS Fluent. Convective heat transfer is considered to be the dominant mechanism compared to radiative heat transfer, thus radiation effects are ignored.

15.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

15.4.1. Preparation

To prepare for running this tutorial:

1. Set up a working folder on the computer you will be using.

2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

   Note

   If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

3. Enter the name of this tutorial into the search bar.

4. Narrow the results by using the filter on the left side of the page.

   a. Click **ANSYS Fluent** under **Product**.

   b. Click **18.0** under **Version**.

5. Select this tutorial from the list.

6. Click the **surface_chem_R180.zip** link to download the input files.

7. Unzip the `surface_chem_R180.zip` file you have downloaded to your working folder.

The file `surface.msh` can be found in the `surface_chem` folder created after unzipping the file.

8. Use Fluent Launcher to start the **3D** version of ANSYS Fluent.

   Fluent Launcher displays your **Display Options** preferences from the previous session.

   For more information about the Fluent Launcher, see starting ANSYS Fluent using the Fluent Launcher in the Fluent Getting Started Guide.

9. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.

10. Ensure that the **Serial** processing option is selected.

11. Enable **Double Precision**.

15.4.2. Reading and Checking the Mesh

1. Read in the mesh file `surface.msh`.

   File → Read → Mesh...

2. Check the mesh.

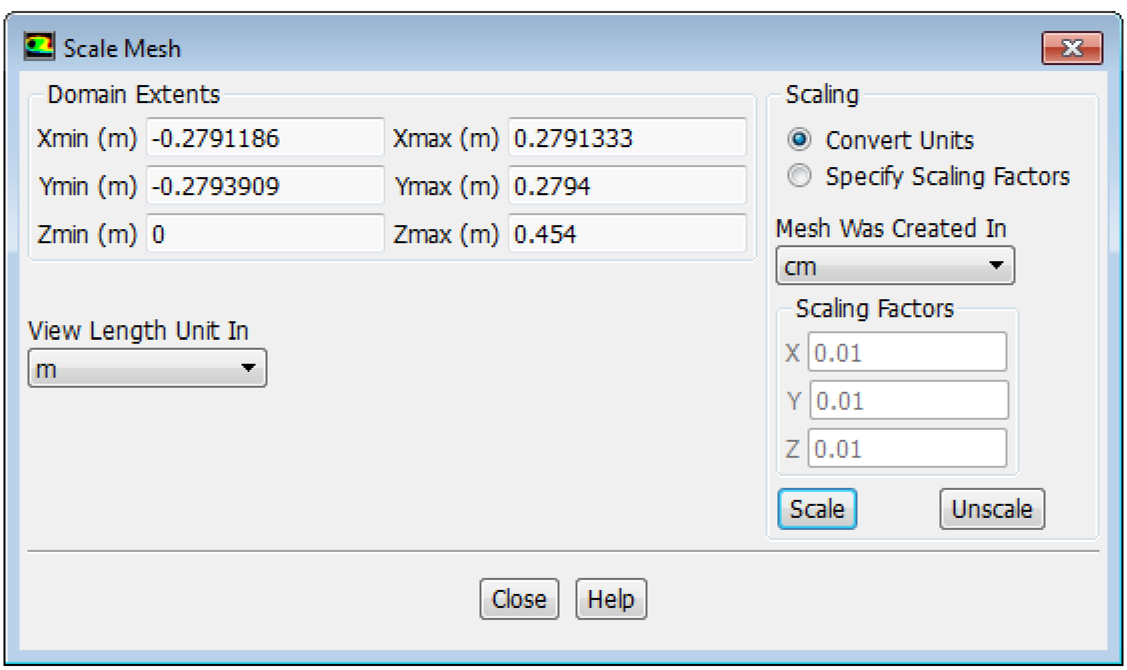
   Setting Up Domain → Mesh → Check

   ANSYS Fluent will perform various checks on the mesh and will report the progress in the console. Ensure that the reported minimum volume is a positive number.

3. Scale the mesh.

   Setting Up Domain → Mesh → Scale...

   Scale the mesh to meters as it was created in centimeters.



a. Select **cm** (centimeters) from the **Mesh Was Created In** drop-down list in the **Scaling** group box.

b. Click **Scale** and verify that the domain extents are as shown in the **Scale Mesh** dialog box.

The default SI units will be used in this tutorial, hence there is no need to change any units.

c. Close the **Scale Mesh** dialog box.

4. Check the mesh.

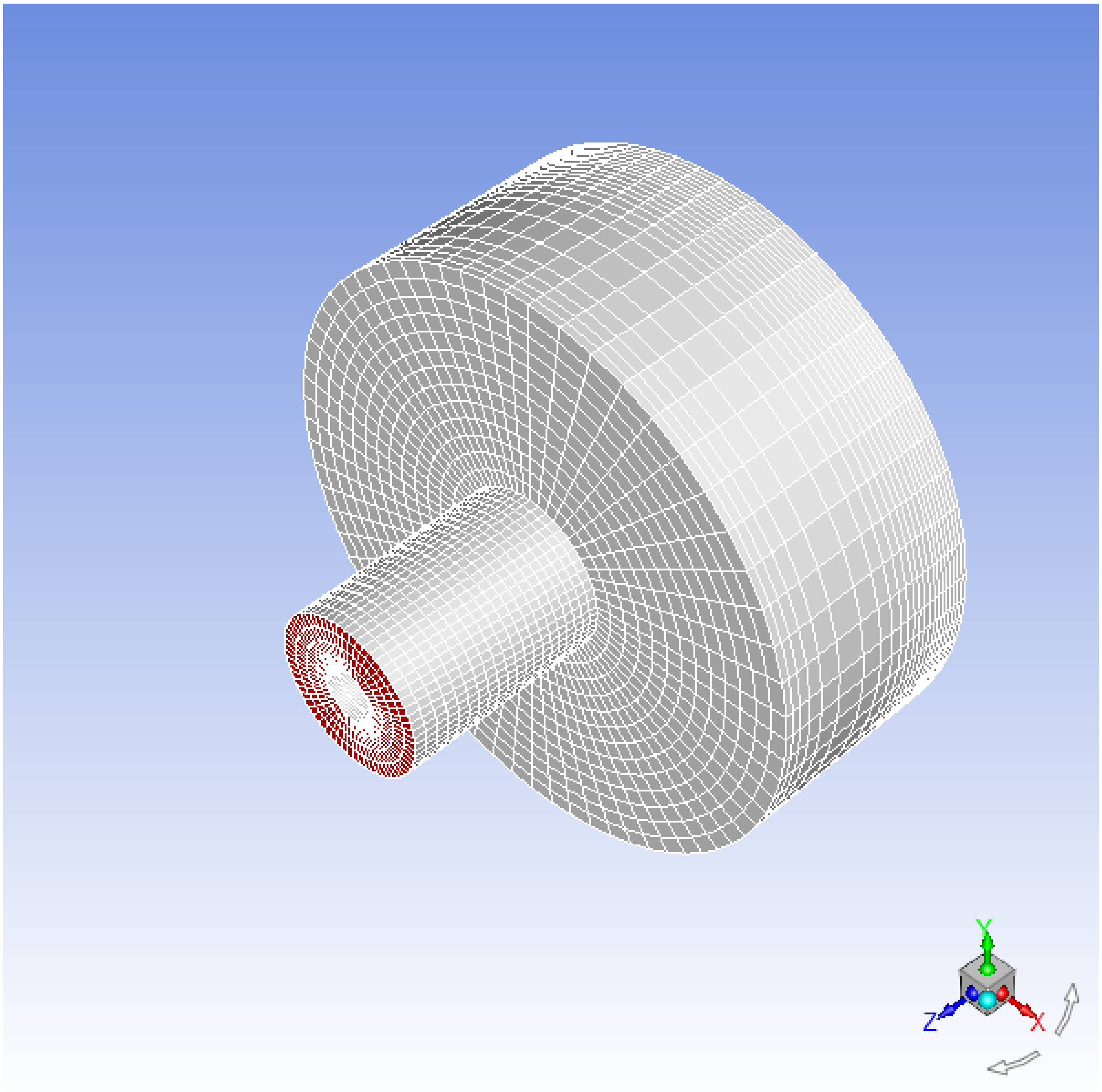
Setting Up Domain → Mesh → Check

Note

It is a good practice to check the mesh after manipulating it (scale, convert to polyhedra, merge, separate, fuse, add zones, or smooth and swap). This will ensure that the quality of the mesh has not been compromised.

5. Examine the mesh (Figure 15.2: Mesh Display (p. 622)).

Figure 15.2: Mesh Display



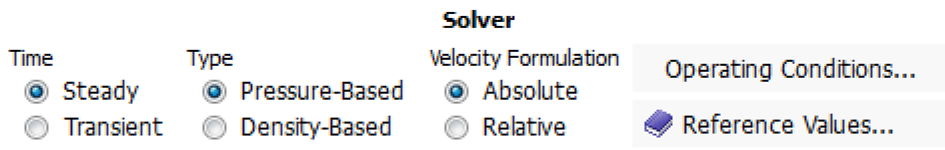
Extra

You can use the left mouse button to rotate the image and view it from different angles. Use the right mouse button to check which zone number corresponds to each boundary. If you click the right mouse button on one of the boundaries in the graphics window, its name and type will be printed in the ANSYS Fluent console. This feature is especially useful when you have several zones of the same type and you want to distinguish between them quickly. Use the middle mouse button to zoom the image.

15.4.3. Solver and Analysis Type

Retain the default solver settings of pressure-based steady-state solver in the **Setting Up Physics** tab (**Solver** group).

Setting Up Physics → Solver

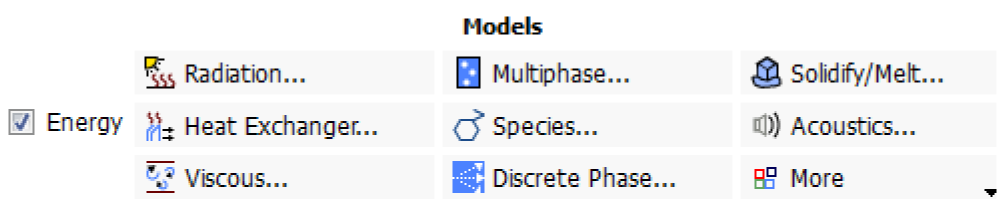


15.4.4. Specifying the Models

In this problem, the energy equation and the species conservation equations will be solved, along with the momentum and continuity equations.

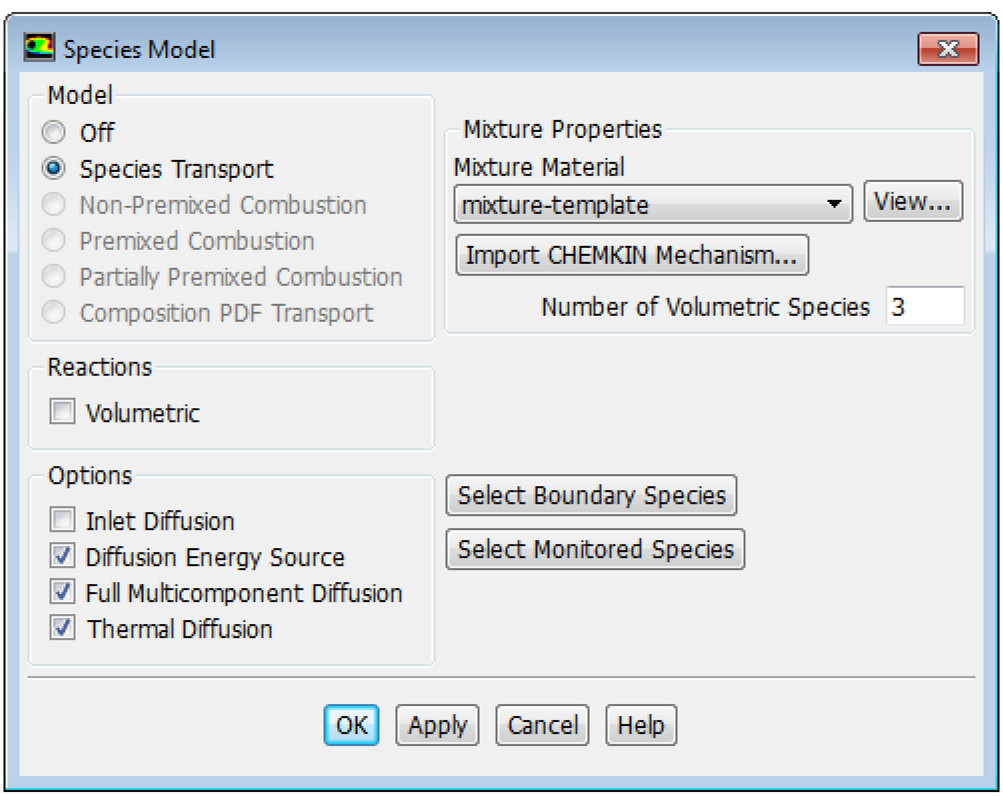
1. Enable heat transfer by turning on the energy equation.

   Setting Up Physics → Models → Energy



2. Enable chemical species transport.

   Setting Up Physics → Models → Species...



a. Select **Species Transport** in the **Model** list.

*The **Species Model** dialog box will expand to show relevant input options.*

b. Retain the selection of **mixture-template** from the **Mixture Material** drop-down list.

You will modify the mixture material later in this tutorial.

c. Retain the default setting for **Diffusion Energy Source**.

This includes the effect of enthalpy transport due to species diffusion in the energy equation, which contributes to the energy balance, especially for the case of Lewis numbers far from unity.

d. Enable **Full Multicomponent Diffusion** and **Thermal Diffusion**.

*The **Full Multicomponent Diffusion** activates Stefan-Maxwell's equations and computes the diffusive fluxes of all species in the mixture to all concentration gradients. The **Thermal Diffusion** effects cause heavy molecules to diffuse less rapidly, and light molecules to diffuse more rapidly, toward heated surfaces.*

e. Click **OK** to close the **Species Model** dialog box.

15.4.5. Defining Materials and Properties

In the following steps you will copy the gas-phase species (AsH_3 , $Ga(CH_3)_3$, CH_3 , and H_2) from the ANSYS Fluent database, specify the mixture materials, setup the reactions, and modify the material properties. You will also create the site species (Ga_s and As_s) and the solid species (Ga and As).

1. Copy arsenic-trihydride, hydrogen, methyl-radical, and trimethyl-gallium from the ANSYS Fluent material database to the list of fluid materials and modify their properties.

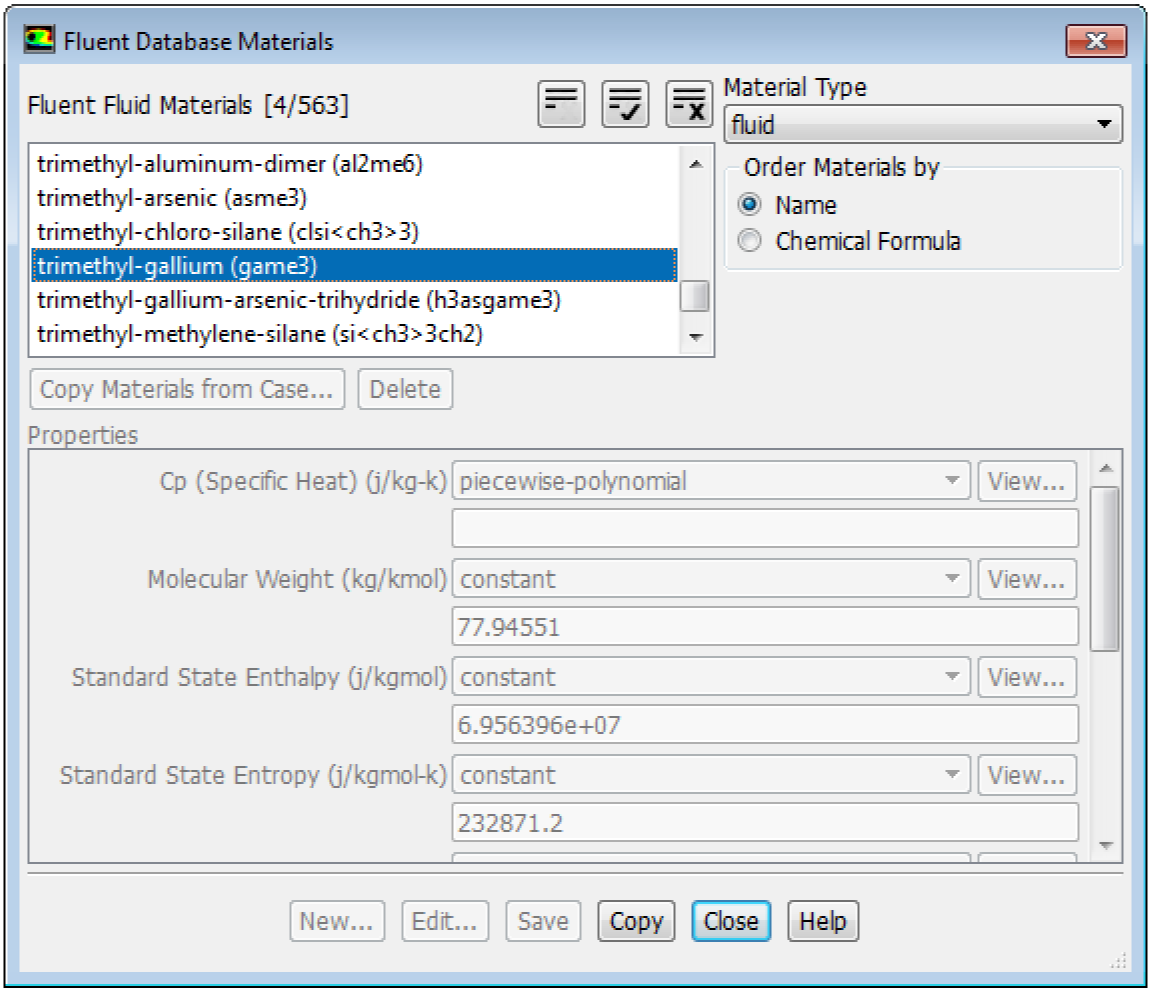
   Setup → **Materials** → **Fluid** → **air** **Edit...**

   a. Click **Fluent Database...** in the **Create/Edit Materials** dialog box to open the **Fluent Database Materials** dialog box.

   b. In the **Fluent Database Materials** dialog box, select **fluid** from the **Material Type** drop-down list.

   c. From the **Fluent Fluid Materials** selection list, select **arsenic-trihydride (ash3)**, **hydrogen (h2)**, **methyl-radical (ch3)**, and **trimethyl-gallium (game3)** by clicking each species once.

   Scroll down the **Fluent Fluid Materials** list to locate each species.



d. Click **Copy** to copy the selected species to your model.

e. Click **Close** to close the **Fluent Database Materials** dialog box.

   The **Create/Edit Materials** dialog box is updated to show the new materials, **arsenic-trihydride (ash3)**, **hydrogen (h2)**, **methyl-radical (ch3)**, and **trimethyl-gallium (game3)**, in the Fluent Fluid Materials drop-down list. The species are also listed under **Fluid** in the **Materials** task page and the **Setup/Materials/Fluid** tree branch.

2. Create the site species (Ga_s and As_s) and the solid species (Ga and As).

   a. In the **Create/Edit Materials** dialog box, select **air** from the **Fluent Fluid Materials** drop-down list.

   b. Enter ga_s for the **Name** text entry field.

   c. Enter ga_s for the **Chemical Formula** text entry field.

   d. Click **Change/Create** to create the new material.

   e. Click **No** in the **Question** dialog box when asked if you want to overwrite air.

      The new material **ga_s** is added to your model and listed under **Fluid** in the **Materials** task page and under the **Setup/Materials/Fluid** tree branch.

f. Create As_s, Ga, and As following the same procedure as for Ga_s and close the **Create/Edit Materials** dialog box.

Extra

To enter complex formulae such as Ga(CH$_3$*)*$_3$ *in the text entry box, use '<' and '>' instead of '(' and ')', respectively.*

3. Set the mixture species.

Setup → **Materials** → **Mixture** → **mixture-template** **Edit...**

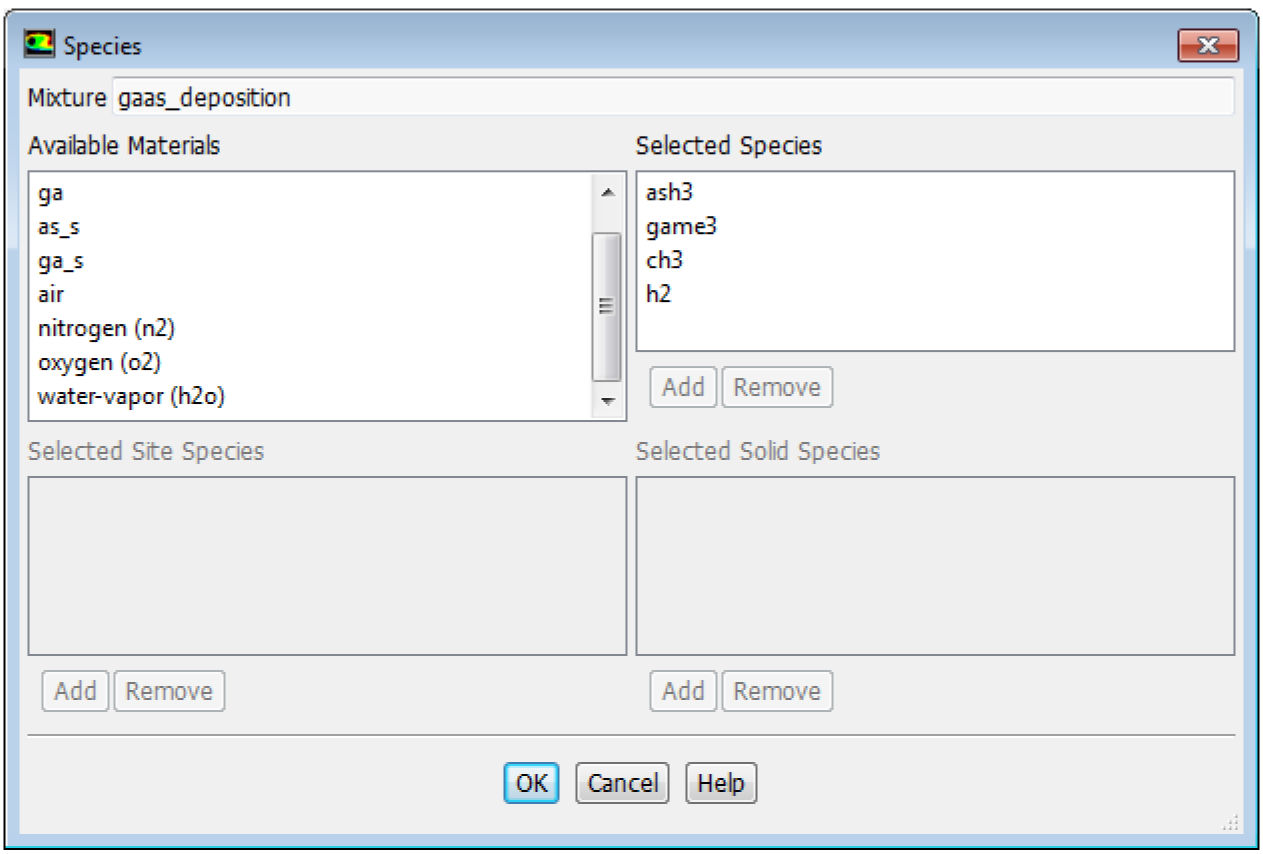
a. Enter `gaas_deposition` for **Name**.

b. Click **Change/Create**.

c. Click **Yes** in the **Question** dialog box to overwrite the mixture-template.

d. Set the **Selected Species**.

i. In **Properties** group box, click the **Edit...** button to the right of the **names** drop-down list for **Mixture Species** to open the **Species** dialog box.



ii. Set the **Selected Species** from the **Available Materials** selection list as shown in Table 15.1: Selected Species (p. 627) .

Table 15.1: Selected Species

Selected Species
ash3
game3
ch3
h2

Important

- Add **arsenic-trihydride (ash3)**, **trimethyl-gallium (game3)**, **methyl-radical (ch3)**, and **hydrogen (h2)** to the **Selected Species** list before removing **h2o**, **o2**, and **n2**.
- Ensure that **h2** is at the bottom in the **Selected Species** selection list as shown in Table 15.1: Selected Species (p. 627). ANSYS Fluent will interpret the last species in the list as the bulk species.

Note

To add/remove the species:

- To add a particular species to the list, select the required species from the **Available Materials** selection list and click **Add** in the corresponding species selection list (**Selected Species**, **Selected Site Species**, or **Selected Solid Species**). The species will be added to the end of the relevant list and removed from the **Available Materials** list.
- To remove an unwanted species from the selection list, select the species from the selection list (**Selected Species**, **Selected Site Species**, or **Selected Solid Species**) and click **Remove** in the corresponding selection list. The species will be removed from the list and added to the **Available Materials** list.

---

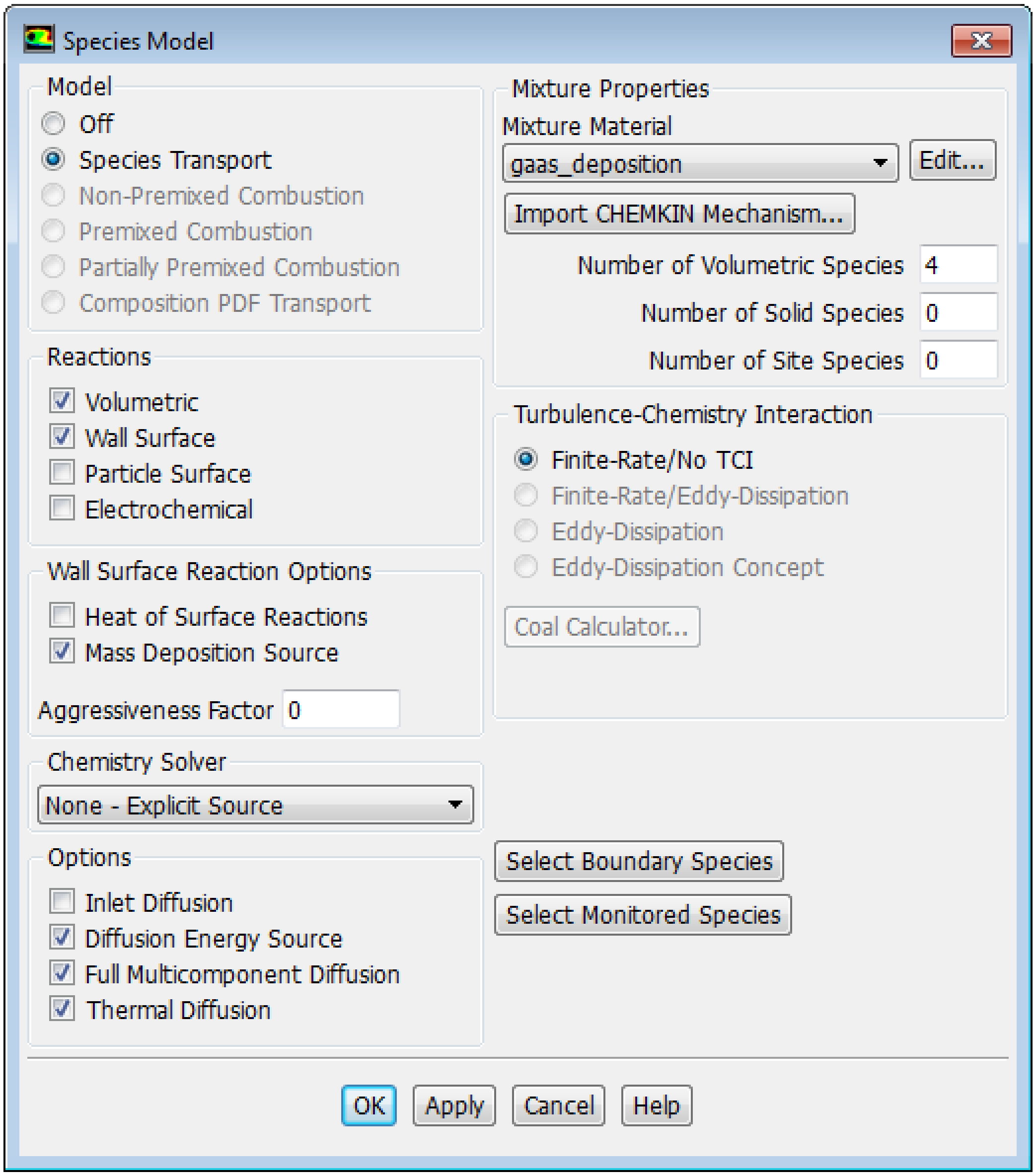
iii. Click **OK** to close the **Species** dialog box.

iv. Click **Change/Create** and close the **Creat/Edit Materials** dialog box.

4. Enable chemical species transport reaction.

Setting Up Physics → Models → Species...

Although you enable reactions, you still run a non-reacting flow to produce an initial solution. You will run a reacting flow in Simulating Reacting Flow (p. 644).



a. Enable **Volumetric** and **Wall Surface** in the **Reactions** group box.

b. Retain the selection of **gass_deposition** from the **Mixture Material** drop-down list.

c. Disable **Heat of Surface Reactions** and enable **Mass Deposition Source**.

d. Click **OK** to close the **Species Model** dialog box.

5. Set the site and solid species and the mixture reactions in a similar manner to the mixture species.

Setup → **Materials** → **Mixture** → **gaas_deposition** **Edit...**

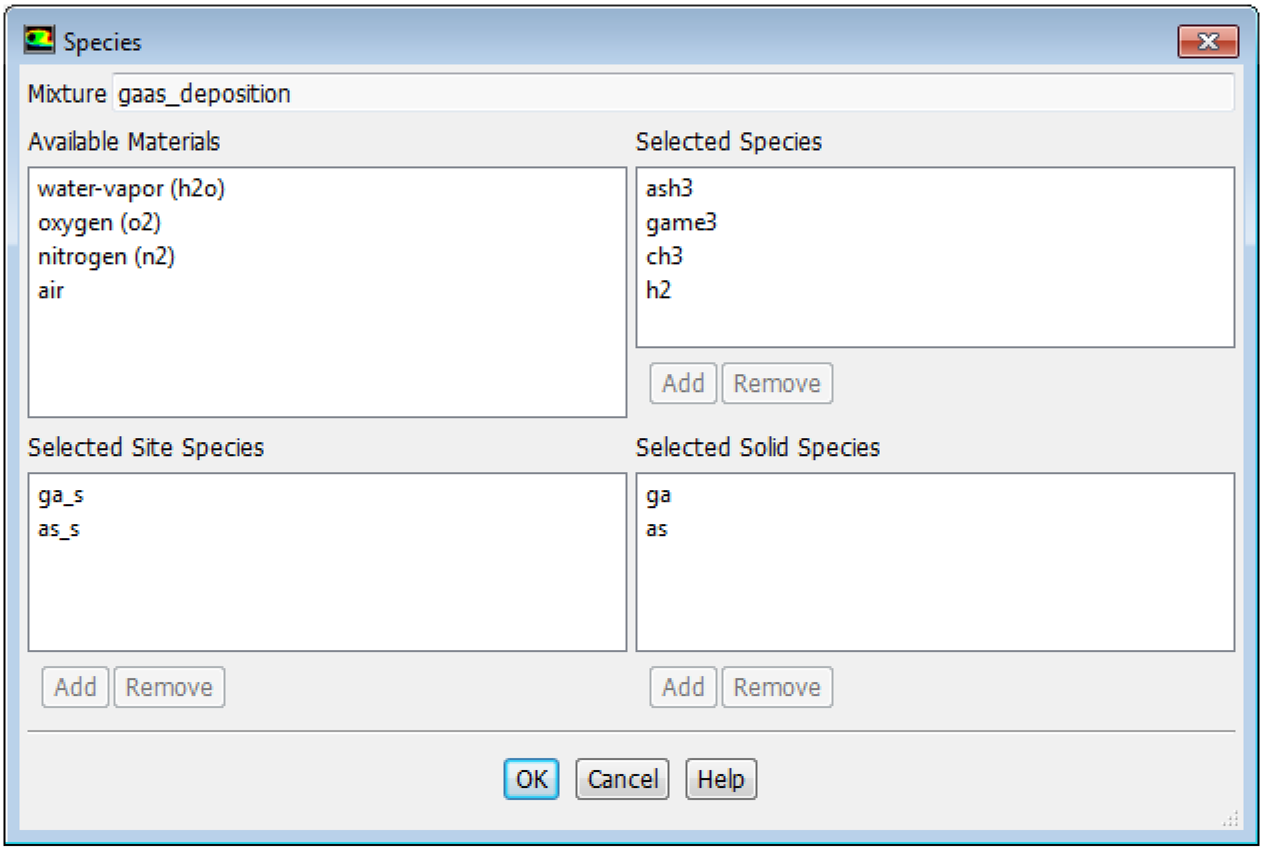
a. Click the **Edit...** button to the right of the **names** drop-down list for **Mixture Species** in the **Properties** group box.

Specify the **Selected Site Species** and the **Selected Solid Species** as shown in Table 15.2: Selected Site and Solid Species (p. 629).

Table 15.2: Selected Site and Solid Species

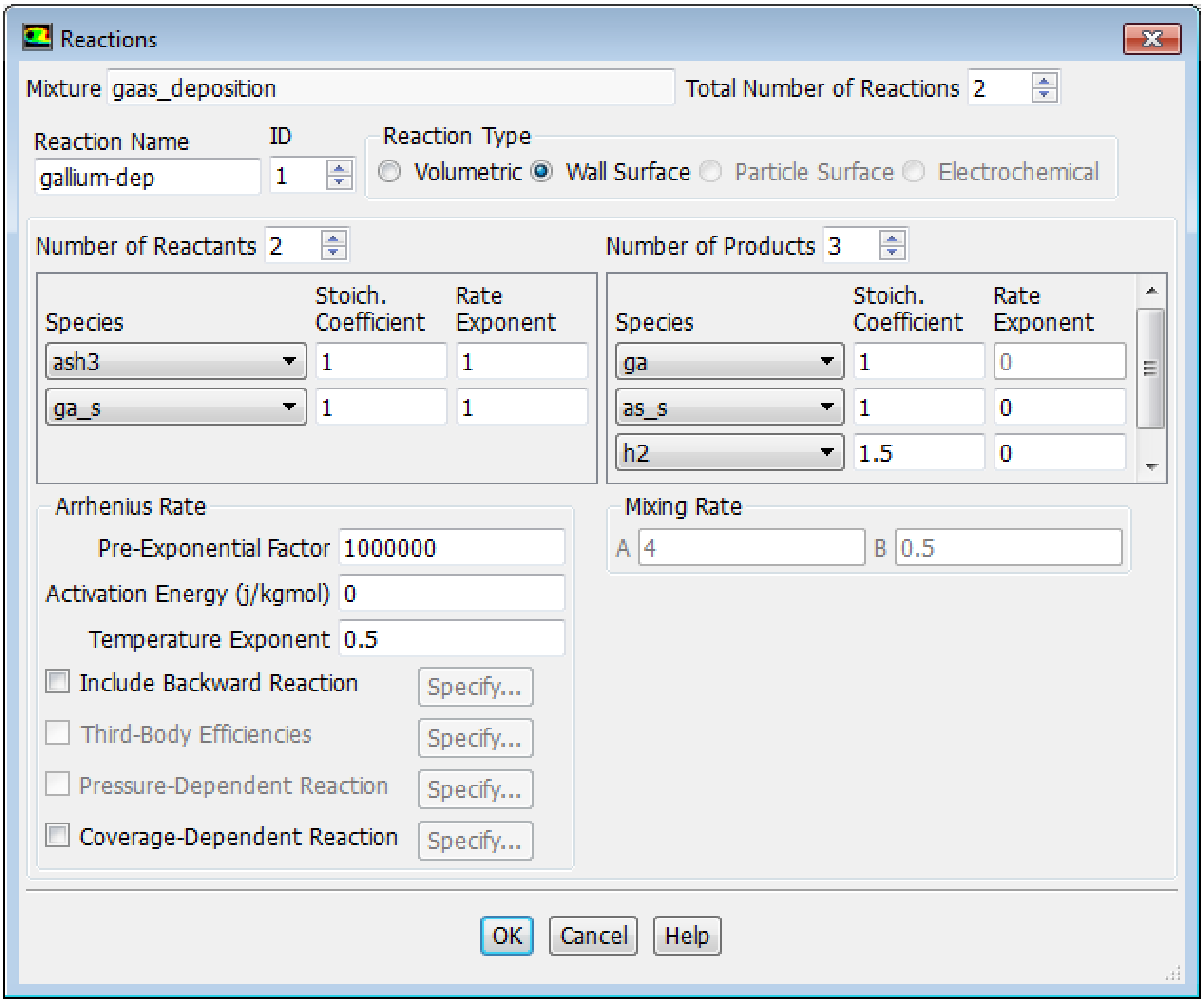
Selected Site Species	Selected Solid Species
ga_s	**ga**
as_s	**as**

Once you set the site and solid species, the **Species** dialog box should look like this:



b. Click **OK** to close the **Species** dialog box.

c. Click the **Edit...** button to the right of the **Reaction** drop-down list to open the **Reactions** dialog box.



d. Increase the **Total Number of Reactions** to 2, and define the following reactions using the parameters in Table 15.3: Reaction Parameters (p. 630) :

$$AsH_3 + Ga\_s \rightarrow Ga + As\_s + 1.5H_2 \tag{15.3}$$

$$Ga(CH_3)_3 + As\_s \rightarrow As + Ga\_s + 3CH_3 \tag{15.4}$$

Table 15.3: Reaction Parameters

Parameter	For Equation 15.3 (p. 630)	For Equation 15.4 (p. 630)
Reaction ID	1	2 [a]
Reaction Name	gallium-dep	arsenic-dep
Reaction Type	**Wall Surface**	**Wall Surface**
Number of Reactants	2	2
Species	**ash3**, **ga_s**	**game3**, **as_s**
Stoich. Coefficient	**ash3**= 1, **ga_s**= 1	**game3**= 1, **as_s**= 1
Rate Exponent	**ash3**= 1, **ga_s**= 1	**game3**= 1, **as_s**= 1
Arrhenius Rate	PEF= 1e+06, AE= 0, TE= 0.5 [b]	PEF= 1e+12, AE= 0, TE= 0.5

Parameter	For Equation 15.3 (p. 630)	For Equation 15.4 (p. 630)
Number of Products	3	3
Species	**ga**, **as_s**, **h2**	**as**, **ga_s**, **ch3**
Stoich. Coefficient	**ga**= 1, **as_s**= 1, **h2**= 1.5	**as**= 1, **ga_s**= 1, **ch3**= 3
Rate Exponent	**as_s**= 0, **h2**= 0	**ga_s**= 0, **ch3**= 0

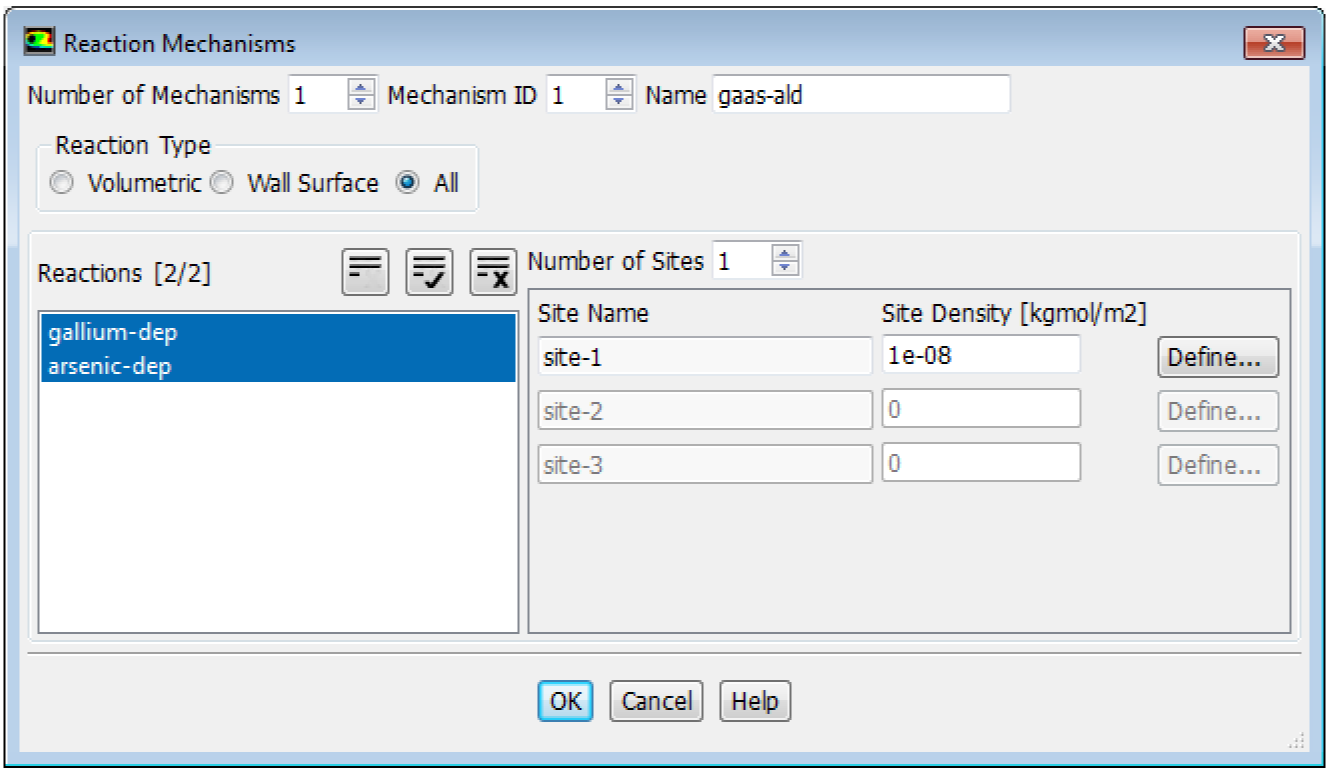
[a]Set the **ID** to 2 in order to set the parameters for the second reaction.

[b]Here, PEF = **Pre-Exponential Factor**, AE = **Activation Energy**, and TE = **Temperature Exponent**.

e. Click **OK** to save the data and close the **Reactions** dialog box.

f. Set the reaction mechanisms for the mixture.

   i. Click the **Edit...** button to the right of the **Mechanism** drop-down list to open the **Reaction Mechanisms** dialog box.



   ii. Retain **Number of Mechanisms** as **1**.

   iii. Enter `gaas-ald` for **Name**.

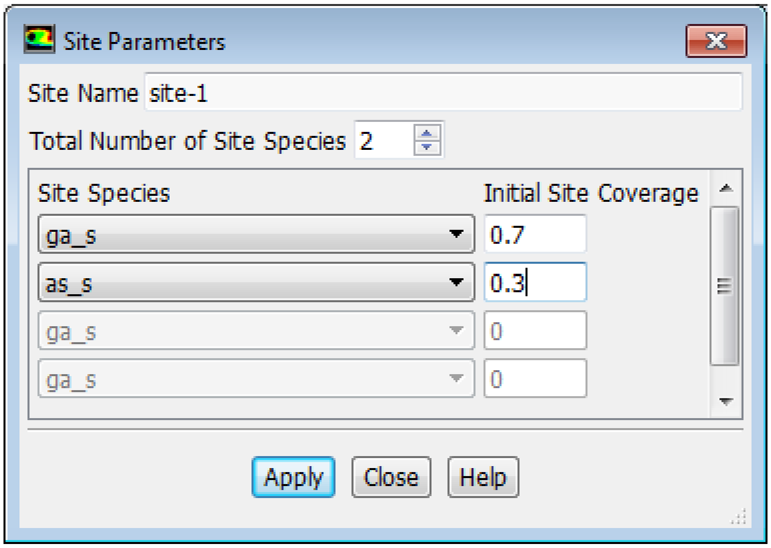
   iv. Select **Wall Surface** in the **Reaction Type** group box.

   v. Select **gallium-dep** and **arsenic-dep** from the **Reactions** selection list.

   vi. Set **Number of Sites** to 1.

   vii. Enter `1e-08` kgmol/m^2 for **Site Density** for **site-1**.

viii. Click the **Define...** button to the right of `site-1` to open the **Site Parameters** dialog box.



A. Set **Total Number of Site Species** to 2.

B. Select **ga_s** as the first site species and enter `0.7` for **Initial Site Coverage**.

C. Select **as_s** as the second site species and enter `0.3` for **Initial Site Coverage**.

D. Click **Apply** and close the **Site Parameters** dialog box.

ix. Click **OK** to close the **Reaction Mechanisms** dialog box.

g. Retain the default selection of **incompressible-ideal-gas** from the **Density** drop-down list.

h. Retain the default selection of **mixing-law** from the **Cp (Specific Heat)** drop-down list.

i. Select **mass-weighted-mixing-law** from the **Thermal Conductivity** drop-down list.

j. Select **mass-weighted-mixing-law** from the **Viscosity** drop-down list.

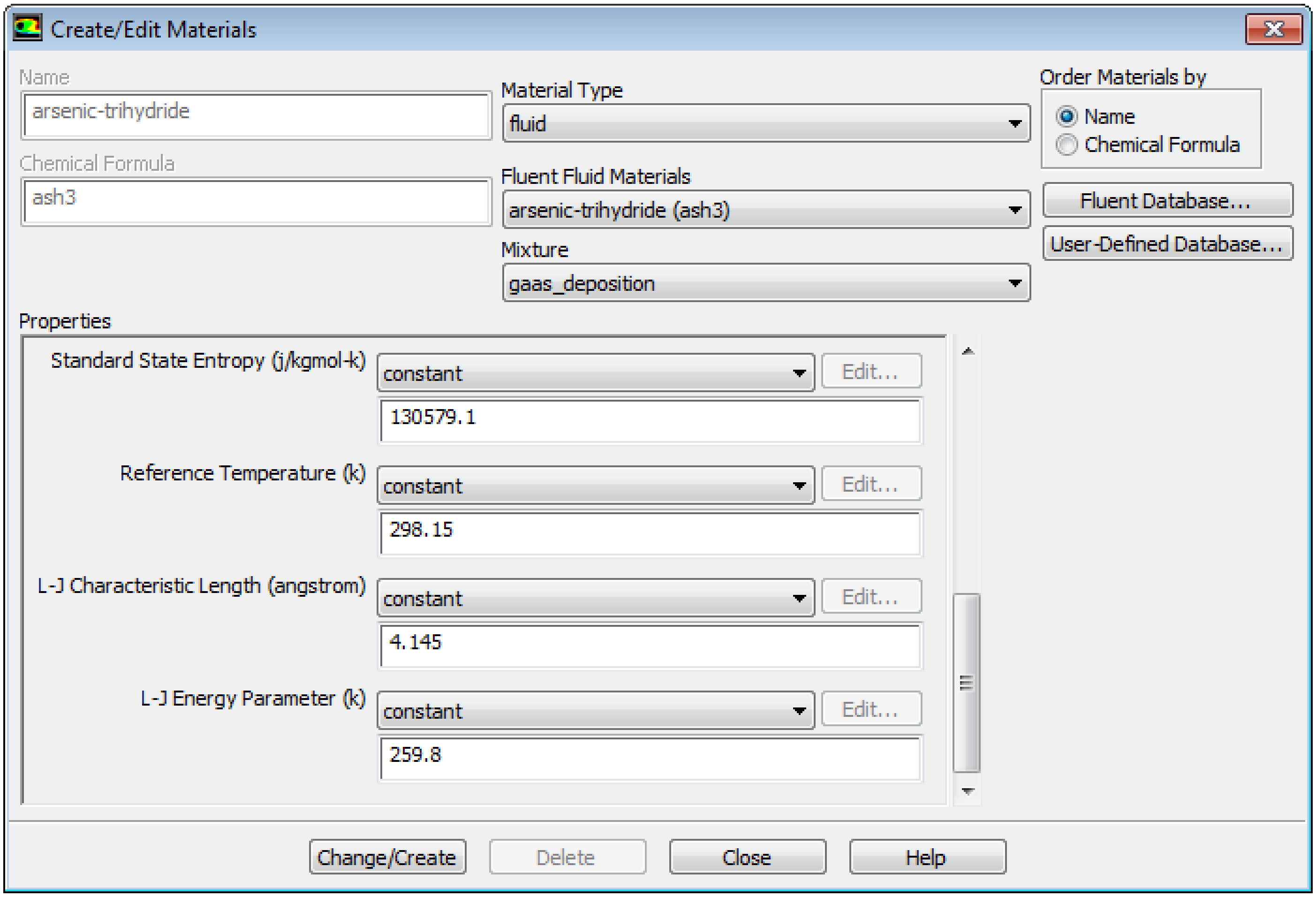
k. Retain the default selection of **kinetic-theory** from the **Mass Diffusivity** drop-down list.

l. Retain the default selection of **kinetic-theory** from the **Thermal Diffusion Coefficient** drop-down list.

m. Click **Change/Create** and close the **Create/Edit Materials** dialog box.

6. Specify the material properties for arsenic-trihydride, hydrogen, methyl-radical, trimethyl-gallium, site species (Ga_s and As_s), and solid species (Ga and As).

Setup → **Materials** → **Mixture** → **gaas_deposition** → **arsenic-trihydride** → **Edit...**



a. In the **Properties** group box, modify the arsenic-trihydride properties as shown in Table 15.4: Properties of Species (p. 633).

Important

Ensure **Mixture** is set to **gaas_deposition**

Tip

Scroll down in the **Properties** group box to see all the parameters.

Table 15.4: Properties of Species

Parameter	AsH_3	Ga(CH_3)_3	CH_3	H_2
Name	arsenic-tri-hydride	trimethyl-gallium	methyl-radic-al	hydrogen
Chemical Formula	ash3	game3	ch3	h2
Cp (Specific Heat)	**piecewise-polynomial**	**piecewise-poly-nomial**	**piecewise-poly-nomial**	**piecewise-poly-nomial**
Thermal Conductiv-ity	**kinetic-theory**	**kinetic-theory**	**kinetic-theory**	**kinetic-theory**

Parameter	AsH_3	Ga(CH_3)_3	CH_3	H_2
Viscosity	**kinetic-theory**	**kinetic-theory**	**kinetic-theory**	**kinetic-theory**
Molecular Weight	77.95	114.83	15	2.02
Standard State Enthalpy	0	0	2.044e+07	0
Standard State Entropy	130579.1	130579.1	257367.6	130579.1
Reference Temperature	298.15	298.15	298.15	298.15
L-J Characteristic Length	4.145	5.68	3.758	2.827
L-J Energy Parameter	259.8	398	148.6	59.7

b. When finished, click **Change/Create** to update your local copy of the species material.

Note

When you modify the properties of the material local copy, the original copy in Fluent material database stays intact.

c. In a similar way, modify the properties of **trimethyl-gallium (game3)**, **methyl-radical (ch3)**, and **hydrogen (h2)**.

Note

Make sure to click **Change/Create** each time you modify the properties for the material to apply the changes to the local copy.

d. Select **ga_s** from the **Fluent Fluid Materials** drop-down list.

e. Enter the parameter values for the ga_s species as shown in Table 15.5: Properties of Species (p. 634).

Table 15.5: Properties of Species

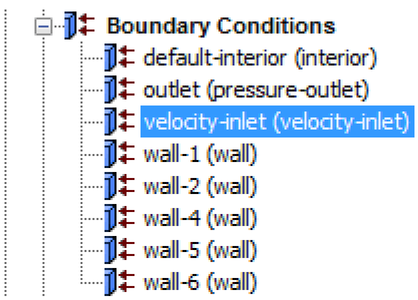
Parameter	Ga_s	As_s	Ga	As
Name	ga_s	as_s	ga	as
Chemical Formula	ga_s	as_s	ga	as
Cp (Specific Heat)	520.64	520.64	1006.43	1006.43
Thermal Conductivity	0.0158	0.0158	**kinetic-theory**	**kinetic-theory**
Viscosity	2.125e-05	2.125e-05	**kinetic-theory**	**kinetic-theory**
Molecular Weight	69.72	74.92	69.72	74.92

Parameter	Ga_s	As_s	Ga	As
Standard State Enthalpy	`-3117.71`	`-3117.71`	`0`	`0`
Standard State Entropy	`154719.3`	`154719.3`	`0`	`0`
Reference Temperature	`298.15`	`298.15`	`298.15`	`298.15`
L-J Characteristic Length	`0`	`0`	`0`	`0`
L-J Energy Parameter	`0`	`0`	`0`	`0`

f. Modify the material properties for As_s, Ga, and As as shown in Table 15.5: Properties of Species (p. 634).

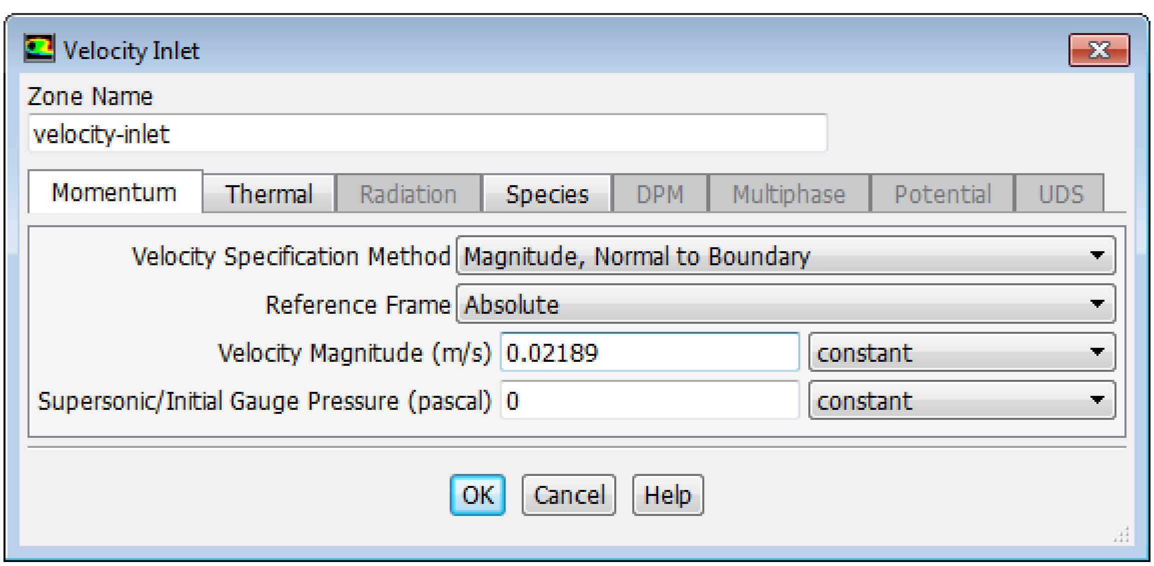
g. Close the **Create/Edit Materials** dialog box.

15.4.6. Specifying Boundary Conditions



1. Set the conditions for **velocity-inlet**.

Setup → **Boundary Conditions** → **velocity-inlet** **Edit...**



a. Retain the default selection of **Magnitude, Normal to Boundary** from the **Velocity Specification Method** drop-down list.

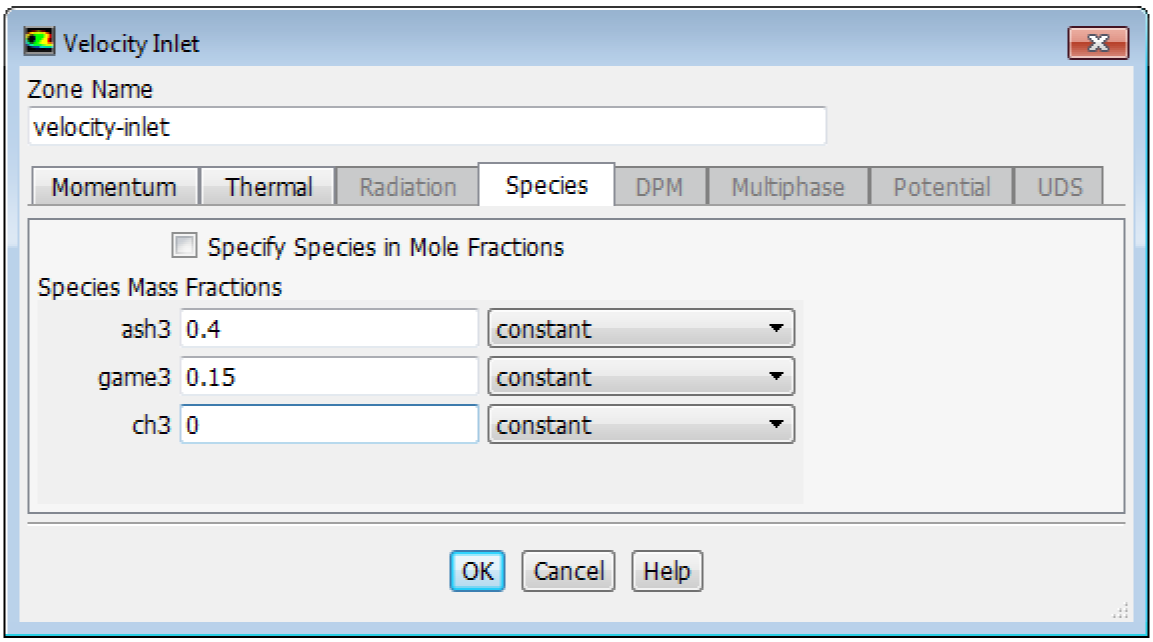
b. Retain the default selection of **Absolute** from the **Reference Frame** drop-down list.

c. Enter `0.02189` m/s for **Velocity Magnitude**.

d. Click the **Thermal** tab and enter 293 K for **Temperature**.

e. Under the **Species** tab, set the **Species Mass Fractions** for **ash3** to `0.4`, **game3** to `0.15`, and **ch3** to `0`.

The mass fraction of hydrogen is 0.45, but there is no need to specify this since it is the last species in the mixture.

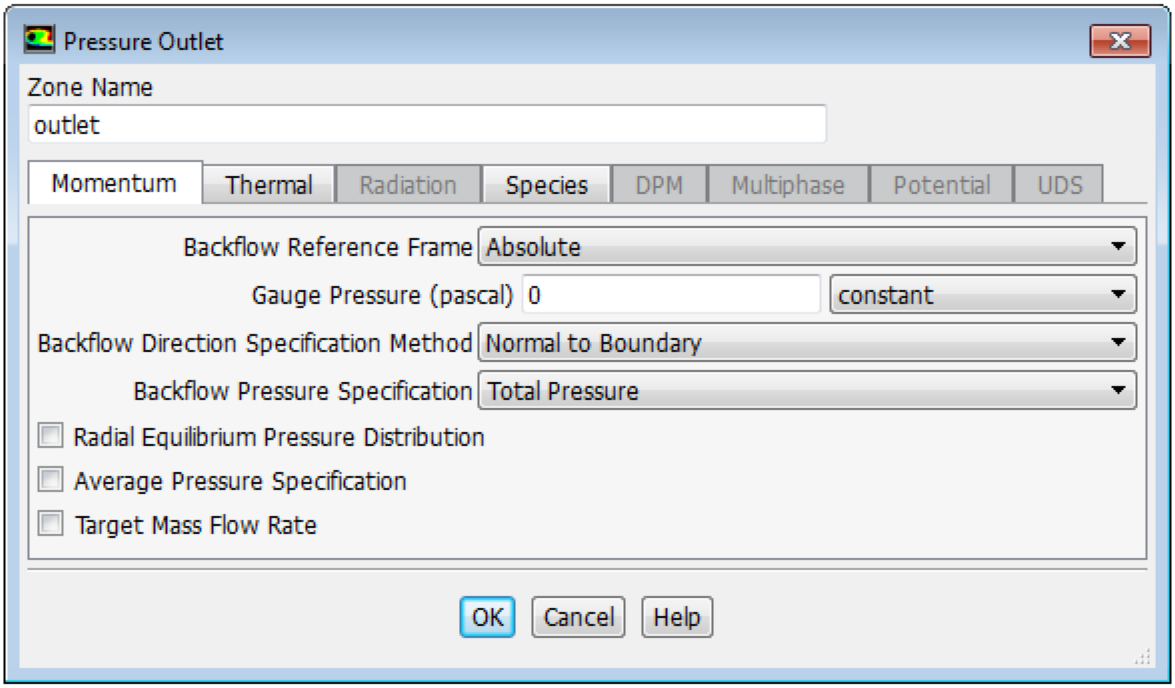


f. Click **OK** to close the **Velocity Inlet** dialog box.

2. Set the boundary conditions for **outlet**.

Setup → **Boundary Conditions** → **outlet** **Edit...**

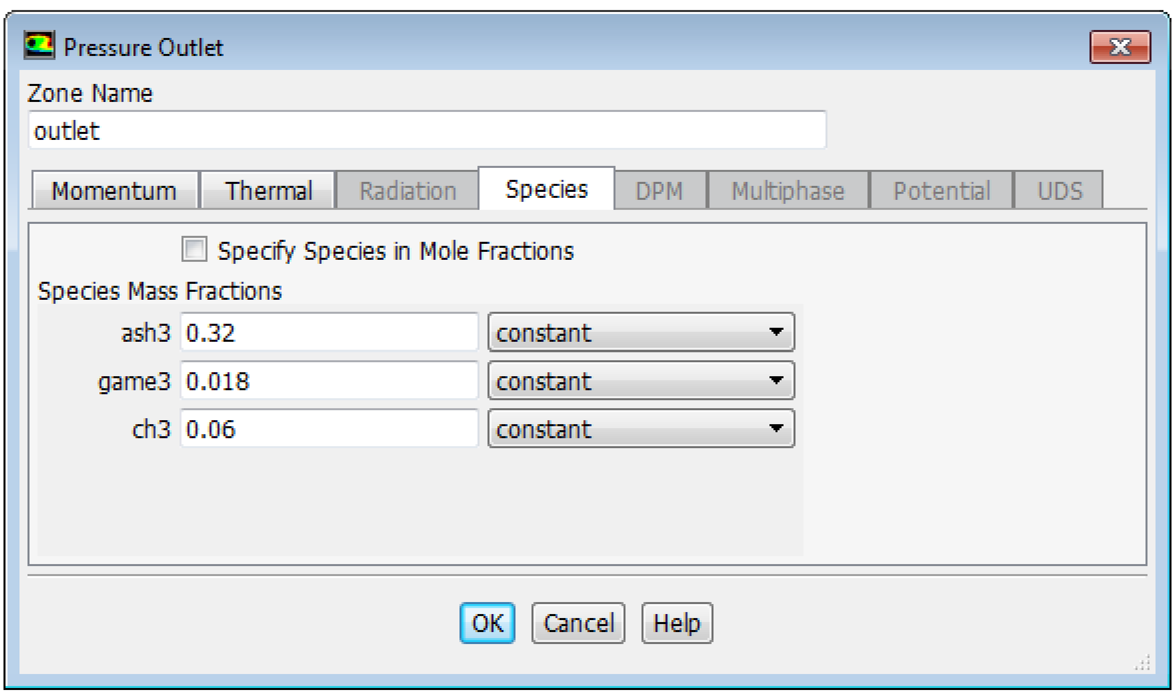
a. Retain the default settings under the **Momentum** tab.



b. Under the Thermal tab, enter `400` K for **Temperature**.

c. Under the **Species** tab, set the **Species Mass Fractions** for **ash3** to `0.32`, **game3** to `0.018`, and **ch3** to `0.06`.

Since a certain amount of backflow is expected in the flow regions around the rotating shaft, you should set the realistic backflow species mass fractions to minimize convergence difficulties.

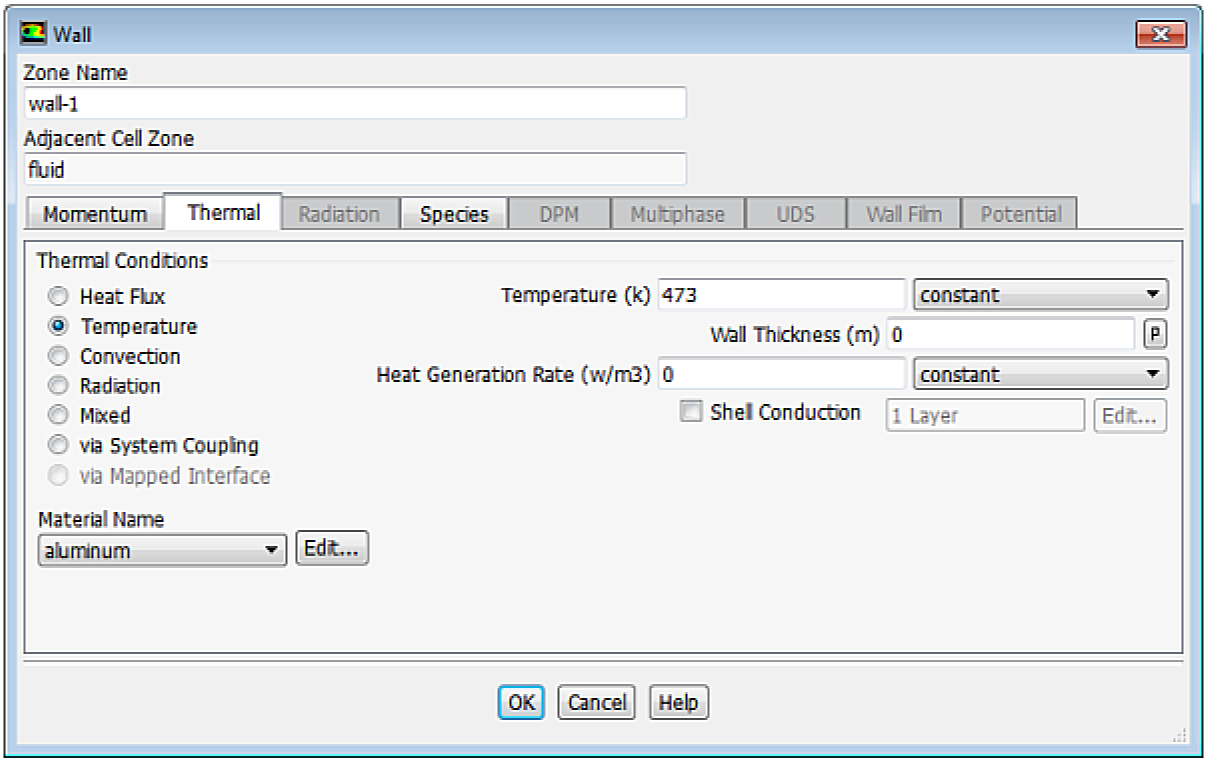


d. Click **OK** to accept the remaining default settings.

3. Set the boundary conditions for **wall-1**.

   Setup → **Boundary Conditions** → **wall-1** **Edit...**

   a. Click the **Thermal** tab.



      i. Select **Temperature** in the **Thermal Conditions** group box.

      ii. Enter 473 K for **Temperature**.

   b. Click **OK** to close the **Wall** dialog box.

4. Set the boundary conditions for **wall-2**.

   Setup → **Boundary Conditions** → **wall-2** **Edit...**

   a. Click the **Thermal** tab.

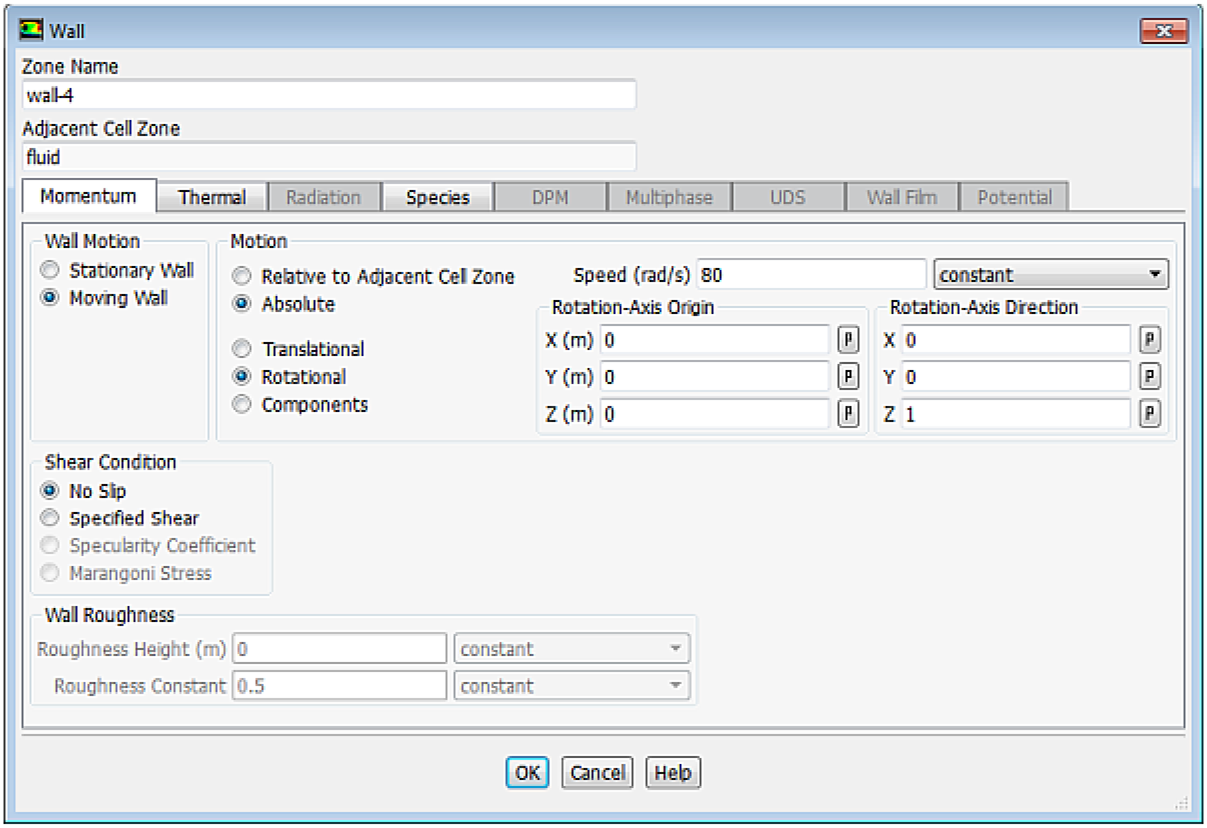
      i. Select **Temperature** in the **Thermal Conditions** group box.

      ii. Enter 343 K for **Temperature**.

   b. Click **OK** to close the **Wall** dialog box.

5. Set the boundary conditions for **wall-4**.

   Setup → **Boundary Conditions** → **wall-4** **Edit...**



a. Select **Moving Wall** in the **Wall Motion** group box.

   The **Wall** dialog box will expand to wall motion inputs and options.

b. Select **Absolute** and **Rotational** in the **Motion** group box.

c. Enter `80` rad/s for **Speed**.

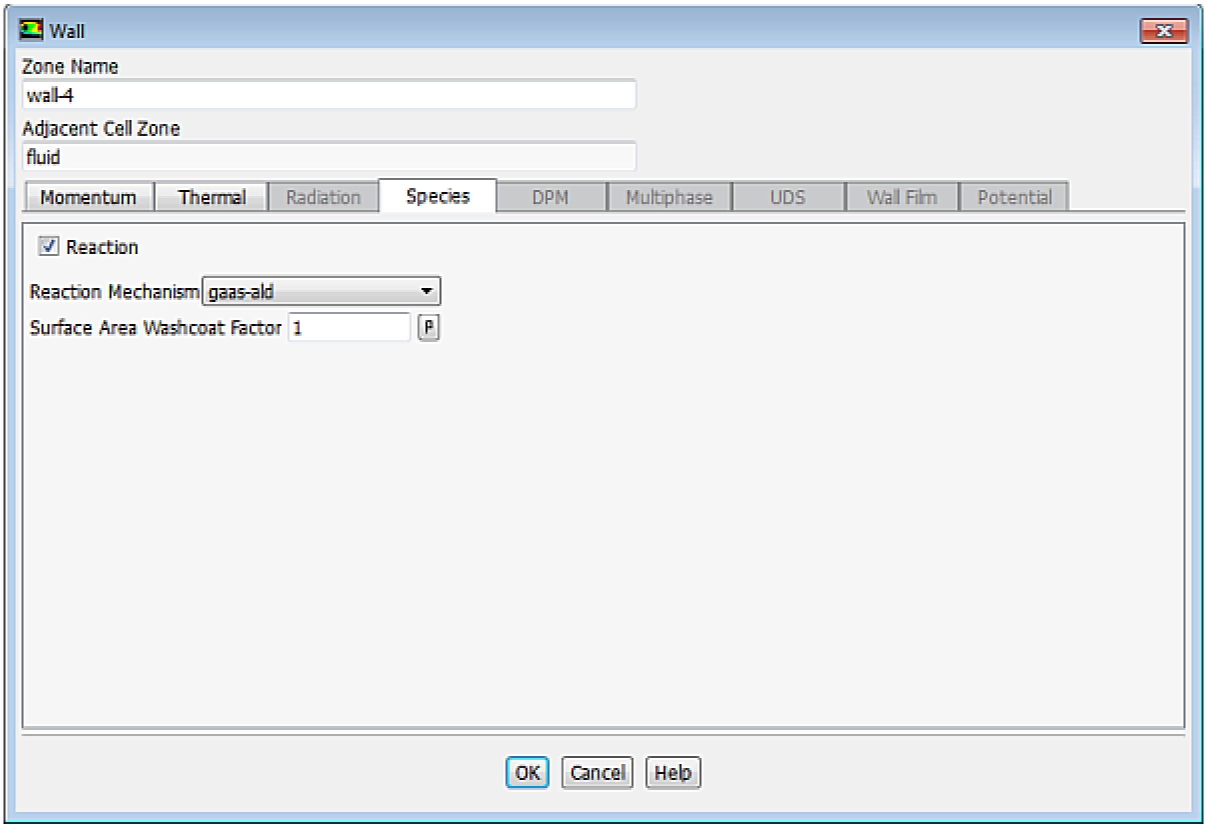
d. Retain the other default settings.

e. Click the **Thermal** tab.

   i. Select **Temperature** in the **Thermal Conditions** group box.

   ii. Enter `1023` K for **Temperature**.

f. Click the **Species** tab.



   i. Enable **Reaction**.

   ii. Retain the selection of **gaas-ald** from the **Reaction Mechanisms** drop-down list.

g. Click **OK** to close the **Wall** dialog box.

6. Set the boundary conditions for **wall-5**.

   Setup → **Boundary Conditions** → **wall-5** **Edit...**

   a. Select **Moving Wall** in the **Wall Motion** group box.

   b. Select **Absolute** and **Rotational** in the **Motion** group box.

   c. Enter `80` rad/s for **Speed**.

   d. Click the **Thermal** tab.

      i. Select **Temperature** in the **Thermal Conditions** group box.

      ii. Enter `720` K for **Temperature**.

   e. Click **OK** to close the **Wall** dialog box.

7. Set the boundary conditions for **wall-6**.

   Setup → **Boundary Conditions** → **wall-6** **Edit...**

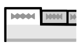
a. Click the **Thermal** tab.

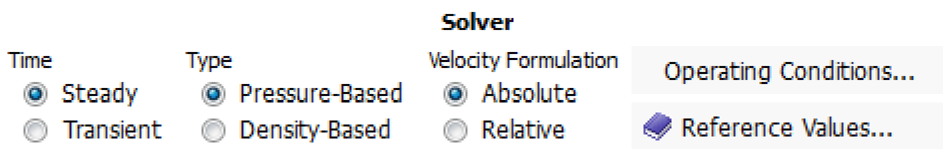
   i. Select **Temperature** in the **Thermal Conditions** group box.

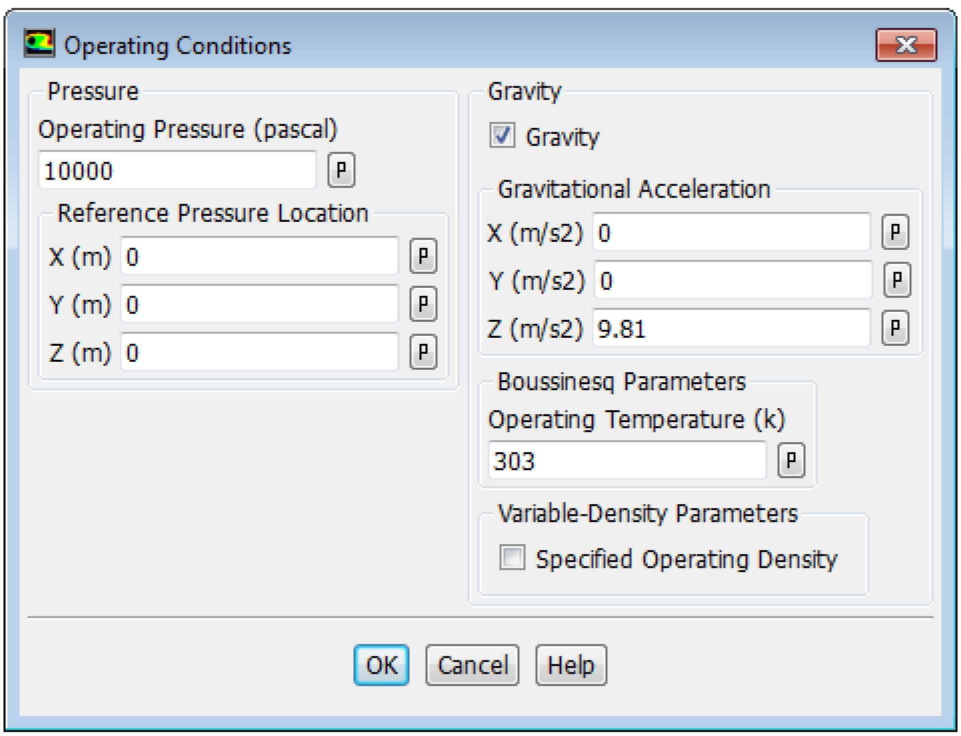
   ii. Enter `303` K for **Temperature**.

b. Click **OK** to close the **Wall** dialog box.

15.4.7. Setting the Operating Conditions

1.  **Setting Up Physics** → **Solver** → **Operating Conditions...**





a. Enter `10000` Pa for **Operating Pressure**.

b. Enable **Gravity**.

   The dialog box will expand to show related gravitational inputs.

c. Enter `9.81` m/s^2 for **Gravitational Acceleration** in the **Z** direction.

d. Enter `303` K for **Operating Temperature**.

e. Click **OK** to close the **Operating Conditions** dialog box.

*The **Operating Conditions** dialog box can be accessed from the **Cell Zone Conditions** task page.*

15.4.8. Simulating Non-Reacting Flow

1. Disable **Volumetric** for solving non-reacting flow.

   Setting Up Physics → Models → Species...

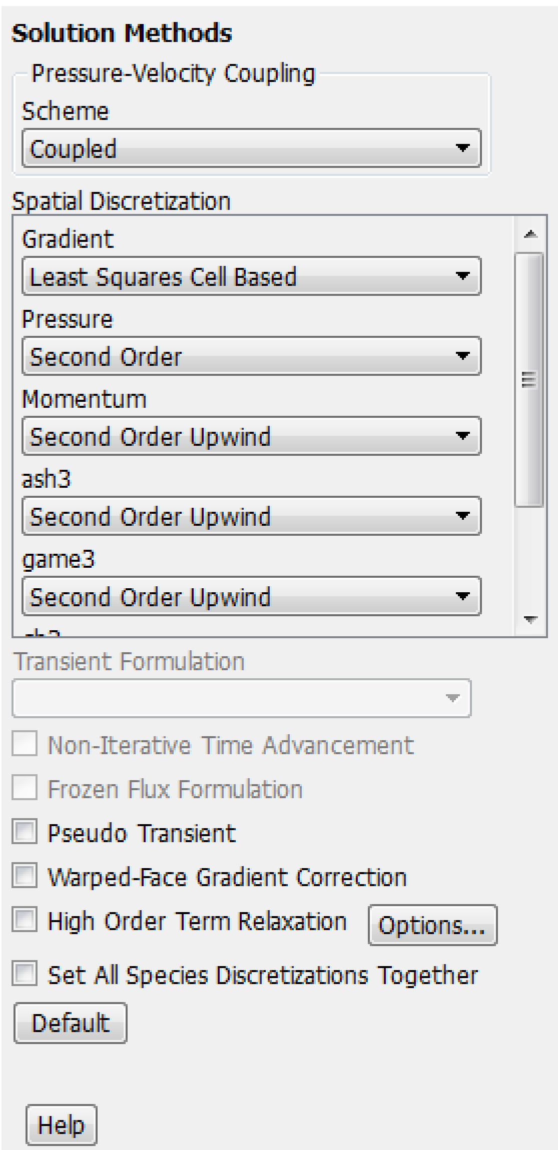
   a. Disable **Volumetric** in the **Reactions** group box.

   b. Click **OK** to close the **Species Model** dialog box.

   You will first run a non-reacting solution to establish the flow.

2. Select the **Coupled** solver method.

   Solution → Solution Methods

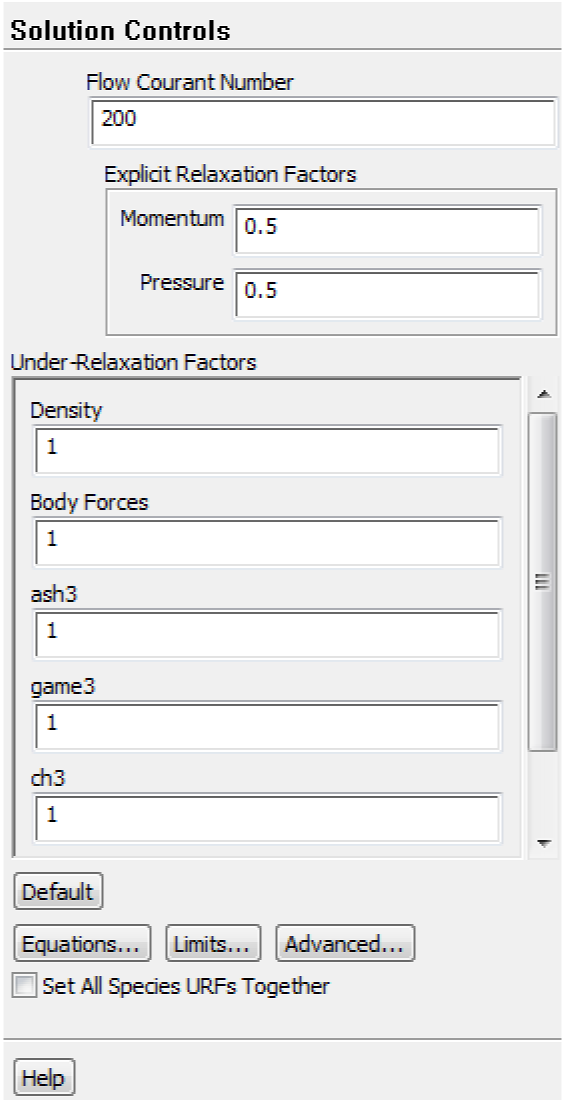


   a. Select **Coupled** from the **Scheme** drop-down list in the **Pressure-Velocity Coupling** group box.

   b. Retain the default selections in the **Spatial Discretization** group box.

3. Examine **Solution Controls** and retain the default settings.

   Solution → **Solution Controls**



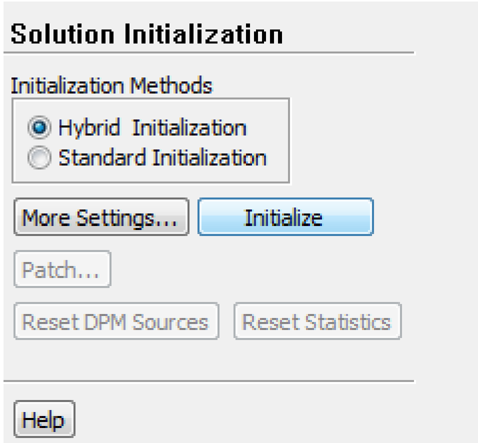
4. Enable residual plotting during the calculation.

   Solving → **Reports** → **Residuals...**

   a. Retain the default settings and close the **Residual Monitors** dialog box.

5. Initialize the flow field.

   Solution → **Solution Initialization**



a. Retain the default selection of **Hybrid Initialization** from the **Initialization Methods** group box.

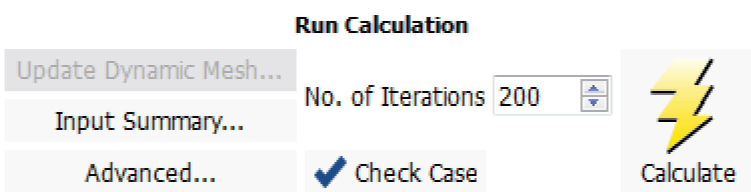
b. Click **Initialize.**

6. Save the case file (`surface-non-react.cas.gz`).

File → Write → Case...

7. Start the calculation by requesting 200 iterations.

Solving → Run Calculation



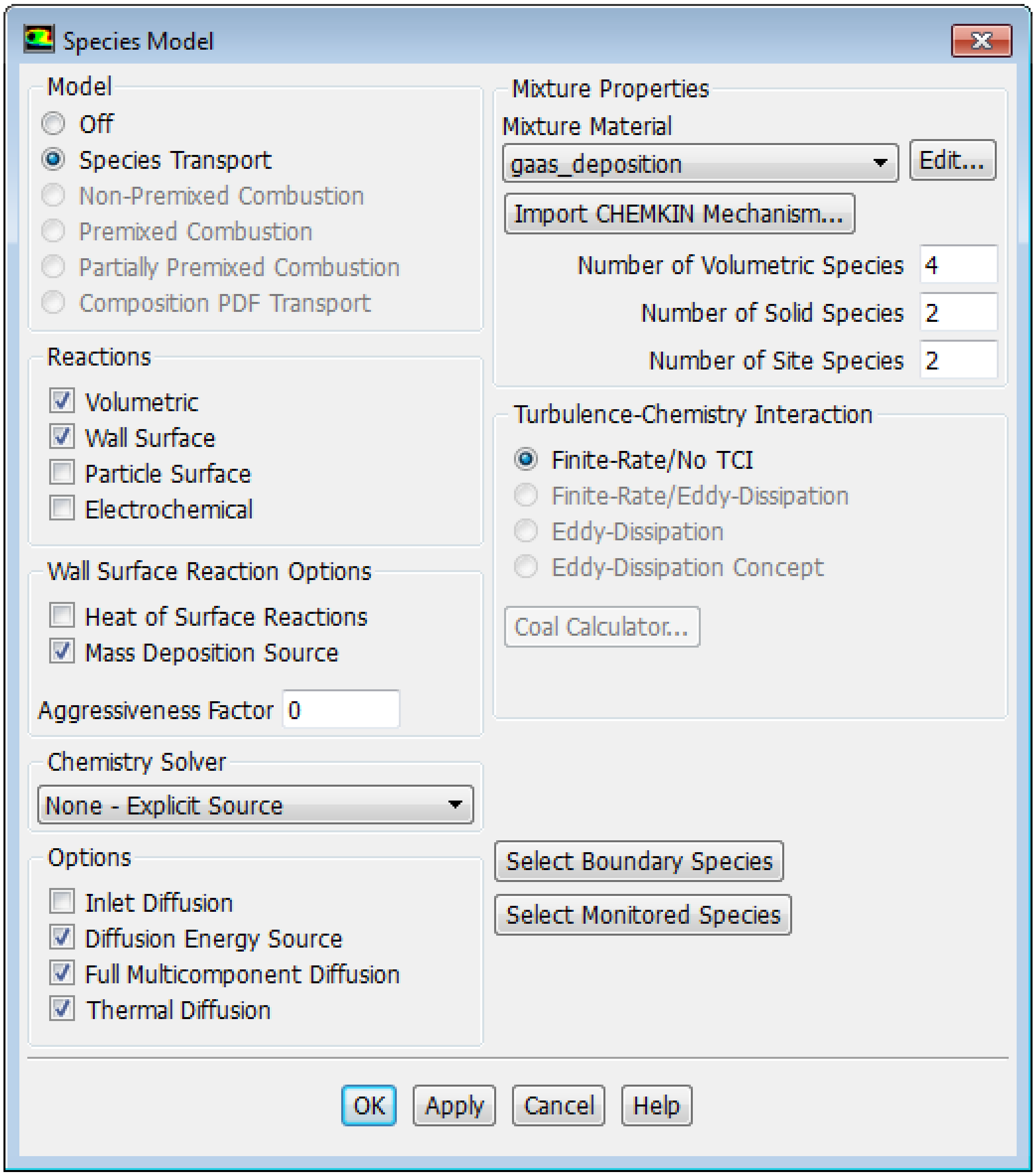
a. Enter `200` for **No. of Iterations** and click **Calculate**.

The solution will converge in approximately 50 iterations.

15.4.9. Simulating Reacting Flow

1. Enable **Volumetric** for the reacting flow solution.

Setting Up Physics → Models → Species...

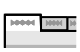


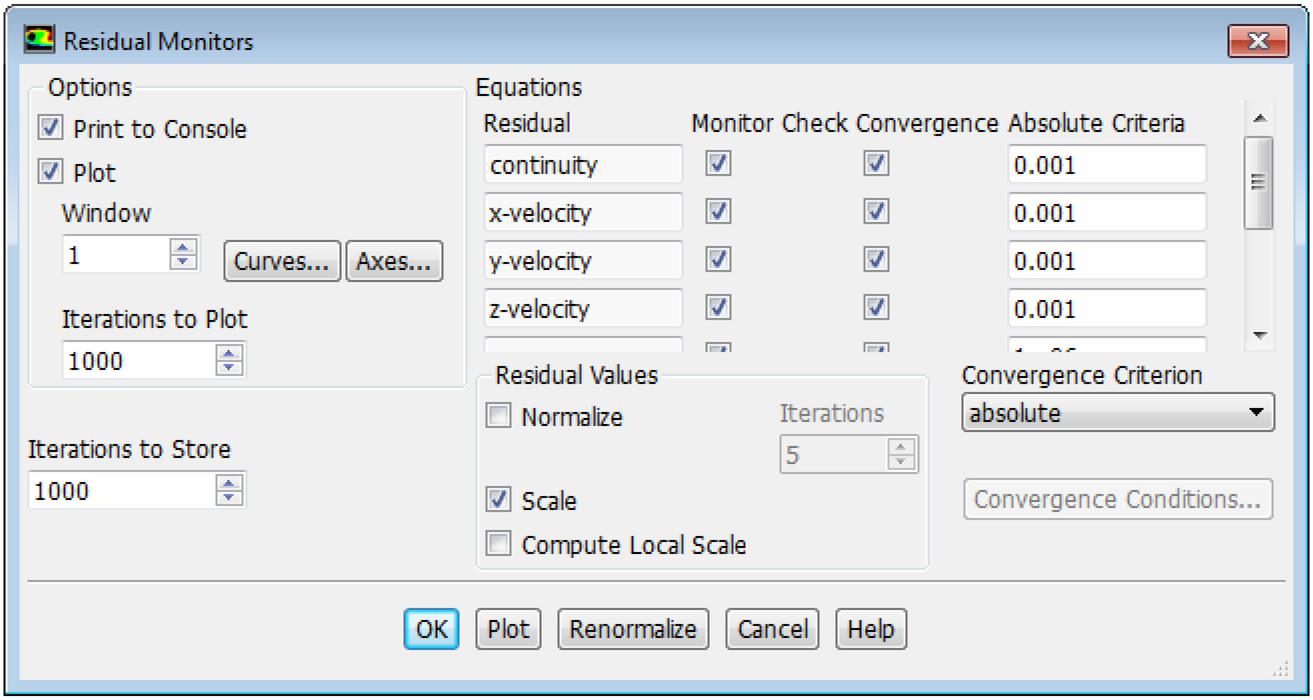
a. Enable **Volumetric** and **Wall Surface** in the **Reactions** group box.

b. Ensure that **Mass Deposition Source** is enabled in the **Wall Surface Reaction Options** group box.

c. Click **OK** to close the **Species Model** dialog box.

2. Retain the default convergence criteria for calculation.

 **Solving** → **Reports** → **Residuals...**



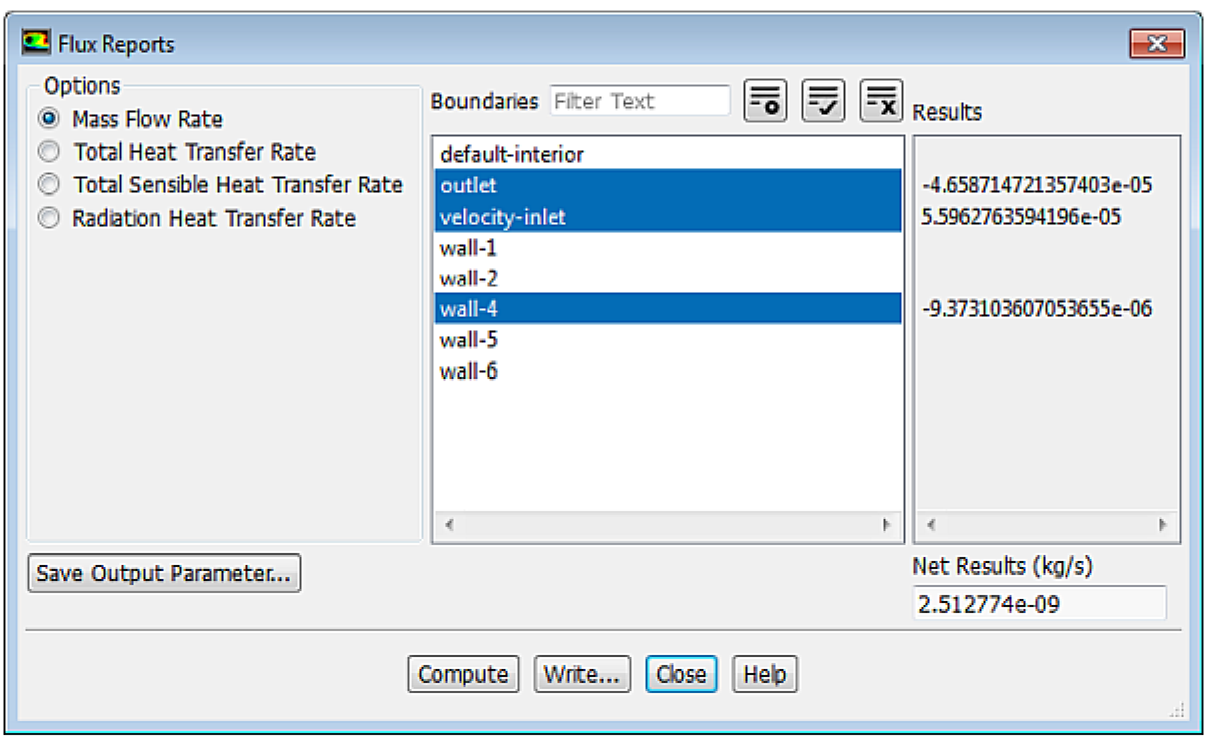
3. Request 200 more iterations.

Solving → **Run Calculation** → **Calculate**

The solution will converge in approximately 60 additional iterations.

4. Compute the mass fluxes.

Postprocessing → **Reports** → **Fluxes...**



a. Retain the default selection of **Mass Flow Rate** in the **Options** group box.

b. Select **outlet**, **velocity-inlet**, and **wall-4** from the **Boundaries** selection list.

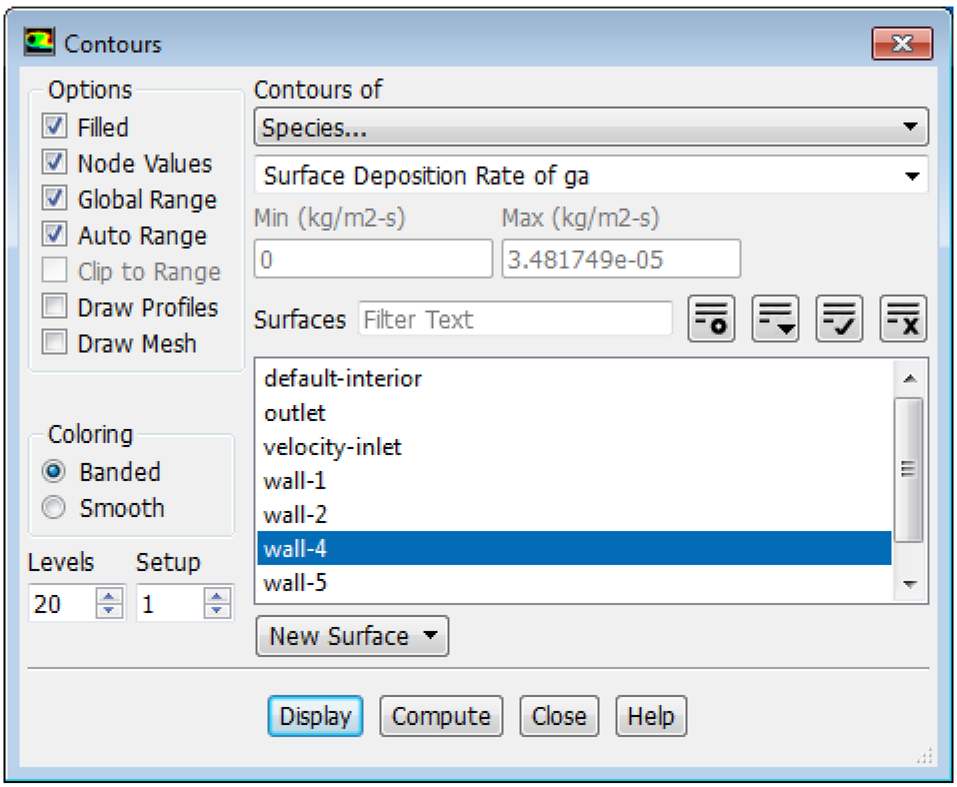
In order to properly assess the mass balance, you must account for the mass deposition on the spinning disk. Hence you select wall-4 in addition to the inlet and outlet boundaries.

c. Click **Compute**, examine the values displayed in the **Results** and **Net Results** boxes, and close the **Flux Reports** dialog box.

The net mass imbalance should be a small fraction (for example, 0.5% or less) of the total flux through the system. If a significant imbalance occurs, you should decrease your residual tolerances by at least an order of magnitude and continue iterating.

5. Display contours of surface deposition rate of **ga** (Figure 15.3: Contours of Surface Deposition Rate of Ga (p. 648)).

Postprocessing → **Graphics** → **Contours** → **Edit...**



a. Enable **Filled** in the **Options** group box.

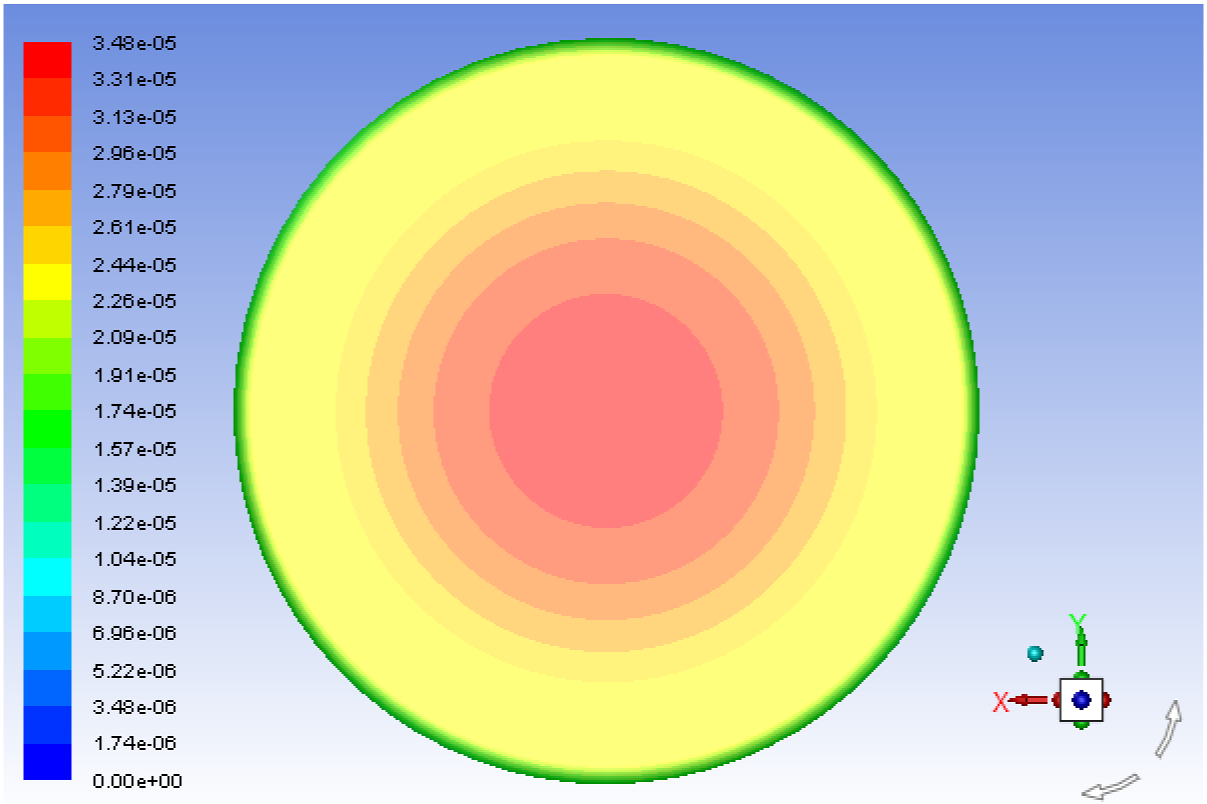
b. Select **Species...** and **Surface Deposition Rate of ga** from the **Contours of** drop-down lists.

c. Select **wall-4** from the **Surfaces** selection list.

d. Click **Display** and close the **Contours** dialog box.

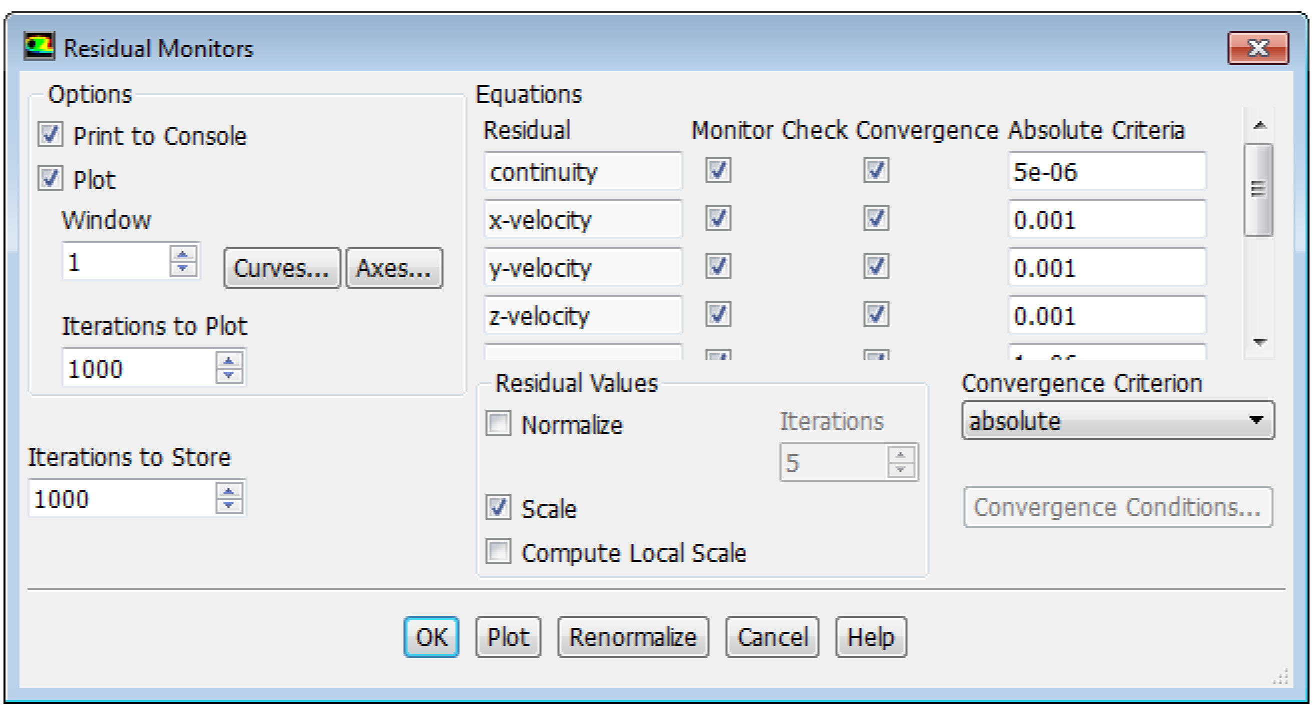
Rotate the display with the mouse to obtain the view as shown in (Figure 15.3: Contours of Surface Deposition Rate of Ga (p. 648)).

Figure 15.3: Contours of Surface Deposition Rate of Ga



6. Reduce the convergence criteria.

Solving → **Reports** → **Residuals...**



a. Enter `5e-06` for **Absolute Criteria** for **continuity**.

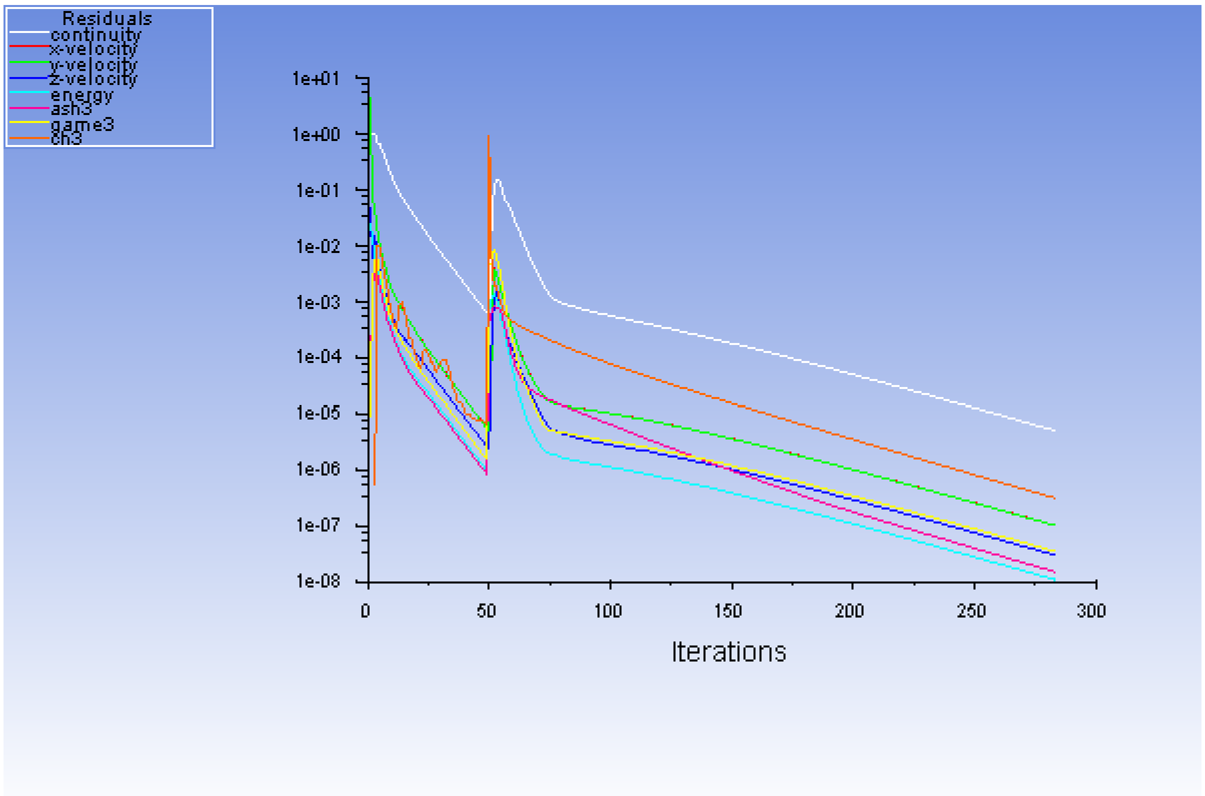
b. Click **OK** to close the **Residual Monitors** dialog box.

7. Request 200 more iterations.

Solving → **Run Calculation** → **Calculate**

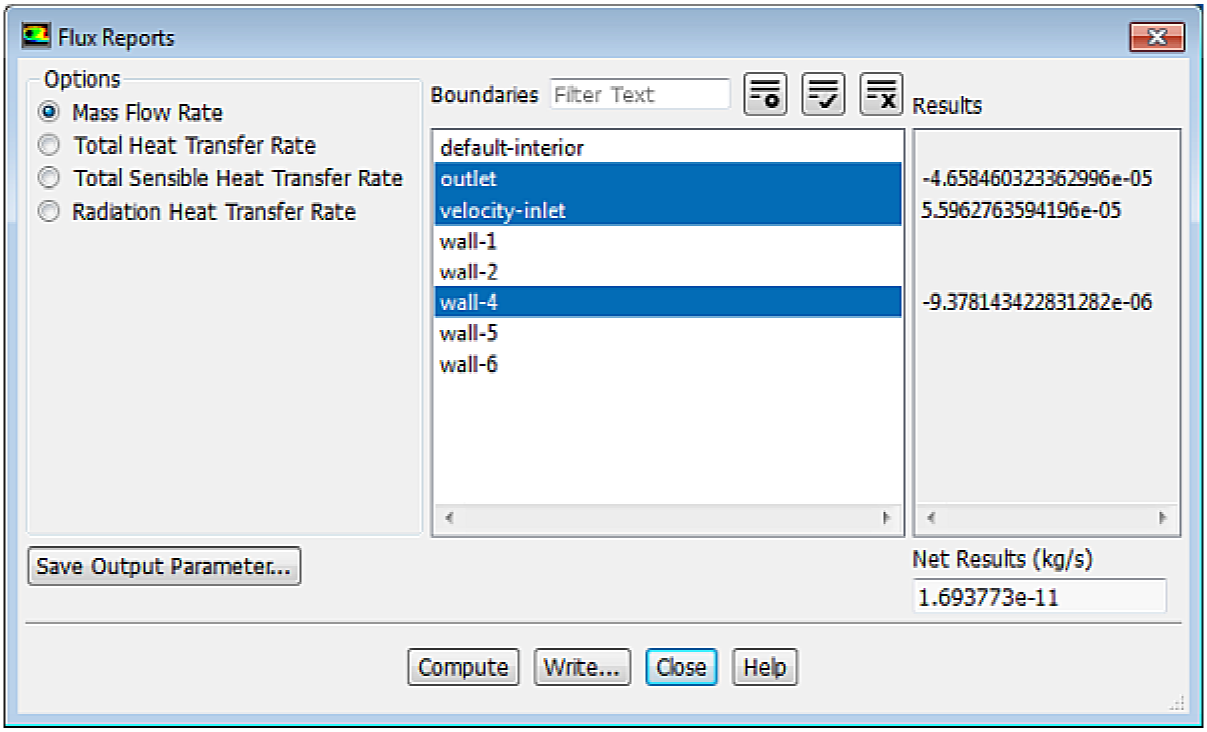
The solution will converge in approximately 175 additional iterations.

Figure 15.4: Scaled Residuals



8. Check the mass fluxes.

Postprocessing → **Reports** → **Fluxes...**



a. Retain the default selection of **Mass Flow Rate** in the **Options** group box.

b. Retain the selection of **outlet** and **velocity-inlet** and, **wall-4** from the **Boundaries** selection list.

c. Click **Compute**, examine the values displayed in the **Results** and **Net Results** boxes, and close the **Flux Reports** dialog box.

   Again, the net mass imbalance should be a small fraction (for example, 0.5% or less) of the total flux through the system.

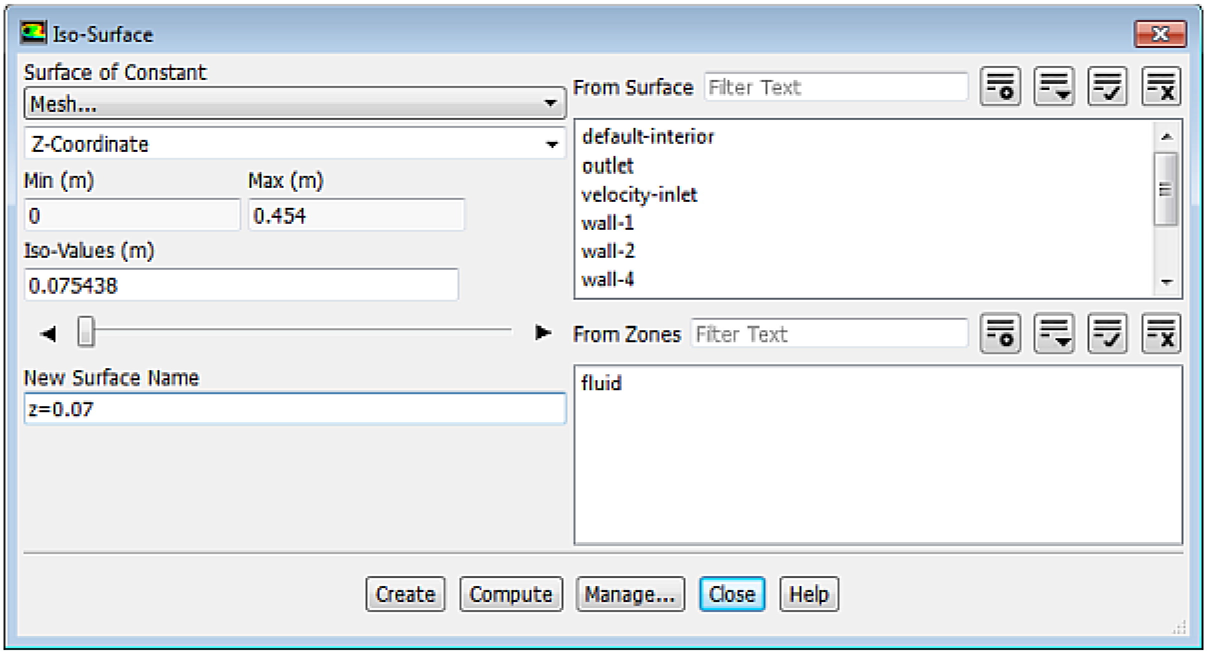
9. Save the case and data files (`surface-react1.cas.gz` and `surface-react1.dat.gz`).

   File → **Write** → **Case & Data...**

15.4.10. Postprocessing the Solution Results

1. Create an iso-surface near **wall-4**.

   Postprocessing → **Surface** → **Create** → **Iso-Surface...**



a. Select **Mesh...** and **Z-Coordinate** from the **Surface of Constant** drop-down lists.

b. Click **Compute**.

   The **Min** and **Max** fields display the z-extent of the domain.

c. Enter `0.075438` m for **Iso-Values**.

d. Enter `z=0.07` for **New Surface Name**.

   Note

   If you want to delete or otherwise manipulate any surfaces, click **Manage...** to open the **Surfaces** dialog box.

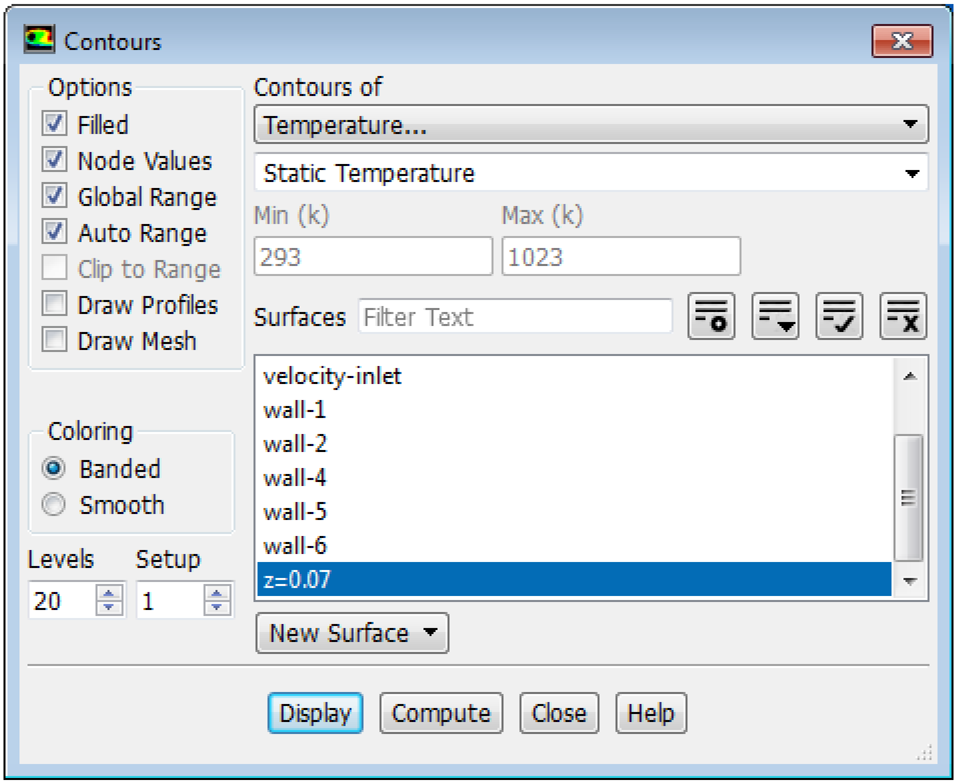
   ---

e. Click **Create** and close the **Iso-Surface** dialog box.

   The new surface `z=0.07` is added to the surfaces selection list.

2. Display contours of temperature on the plane surface created. (Figure 15.5: Temperature Contours Near wall-4 (p. 653)).

   Postprocessing → **Graphics** → **Contours** → **Edit...**



a. Ensure that **Filled** is enabled in the **Options** group box.

b. Select **Temperature...** and **Static Temperature** from the **Contours of** drop-down lists.

c. Deselect **wall-4** from the **Surfaces** selection list.

d. Select **z=0.07** from the **Surfaces** selection list.

e. Click **Display**.

Figure 15.5: Temperature Contours Near wall-4

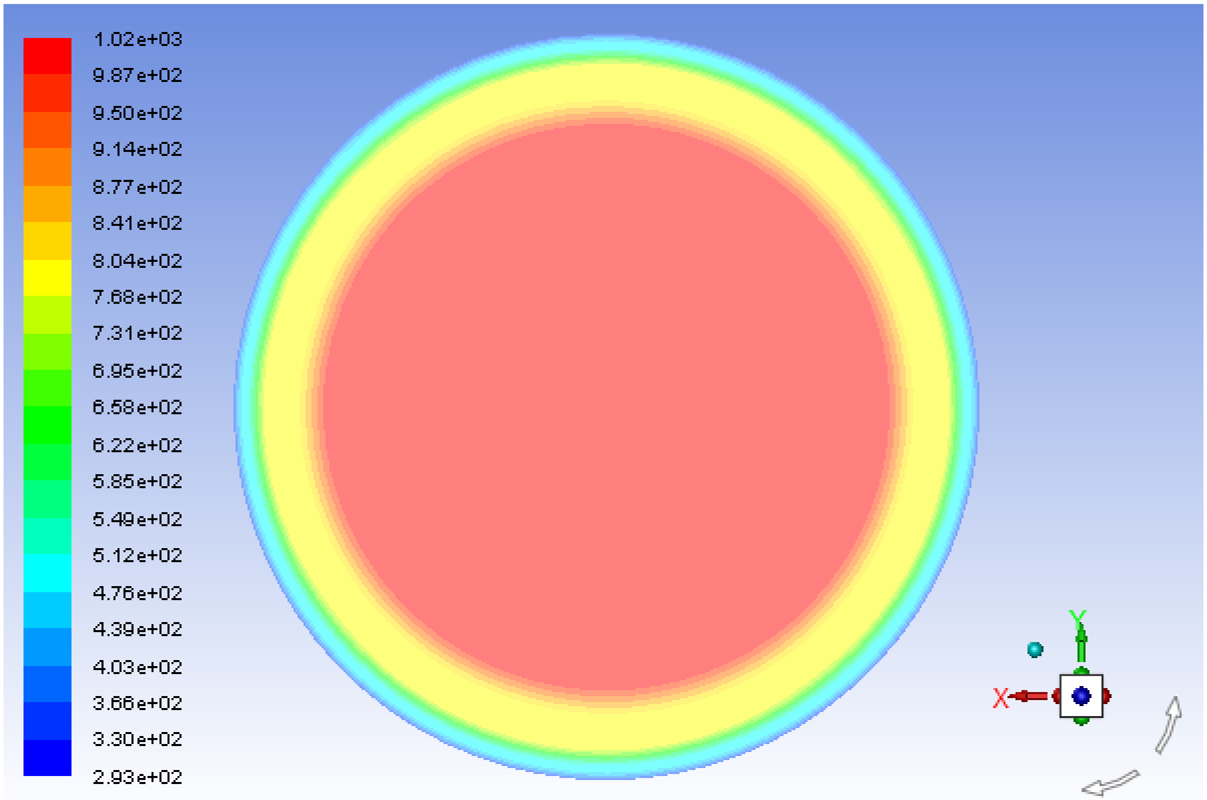


Figure 15.5: Temperature Contours Near wall-4 (p. 653) shows the temperature distribution across a plane just above the rotating disk. You can see that the disk has a temperature of 1023 K.

3. Display contours of surface deposition rates of **ga** (Figure 15.6: Contours of Surface Deposition Rate of ga (p. 654)).

   Postprocessing → **Graphics** → **Contours** → **Edit...**

   a. Select **Species...** and **Surface Deposition Rate of ga** from the **Contours of** drop-down lists.

   b. Select **wall-4** from the **Surfaces** selection list.

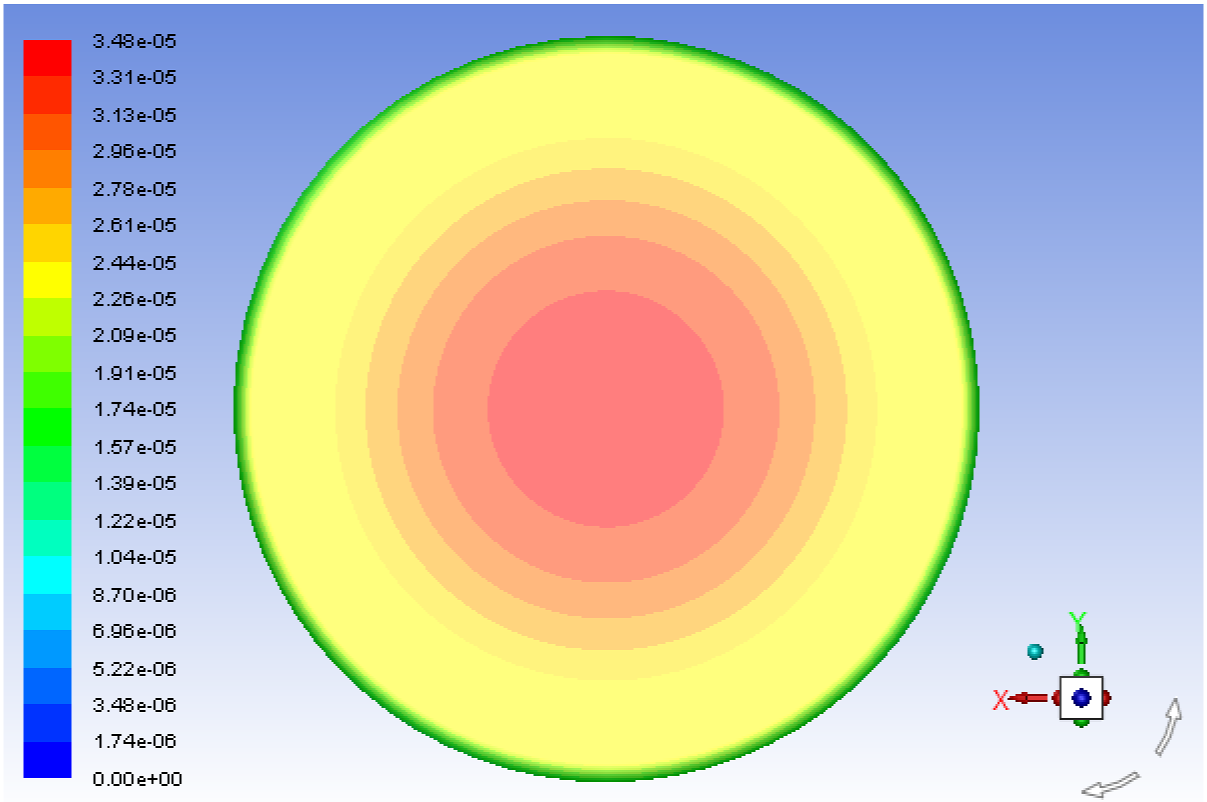
   c. Deselect **z=0.07** from the **Surfaces** selection list.

   d. Click **Display**.

   You may need to use the left mouse button to rotate the image so that you can see the contours on the top side of ***wall-4*** *where the deposition takes place.*

   Figure 15.6: Contours of Surface Deposition Rate of ga (p. 654) shows the gradient of surface deposition rate of ***ga****. The maximum deposition is seen at the center of the disk.*

Figure 15.6: Contours of Surface Deposition Rate of ga



4. Display contours of surface coverage of **ga_s** (Figure 15.7: Contours of Surface Coverage of ga_s (p. 655)).

   Postprocessing → **Graphics** → **Contours** → **Edit...**

   a. Select **Species...** and **Surface Coverage of ga_s** from the **Contours of** drop-down lists.

   b. Retain the selection of **wall-4** in the **Surfaces** selection list.

   c. Click **Display** and close the **Contours** dialog box.

Figure 15.7: Contours of Surface Coverage of ga_s

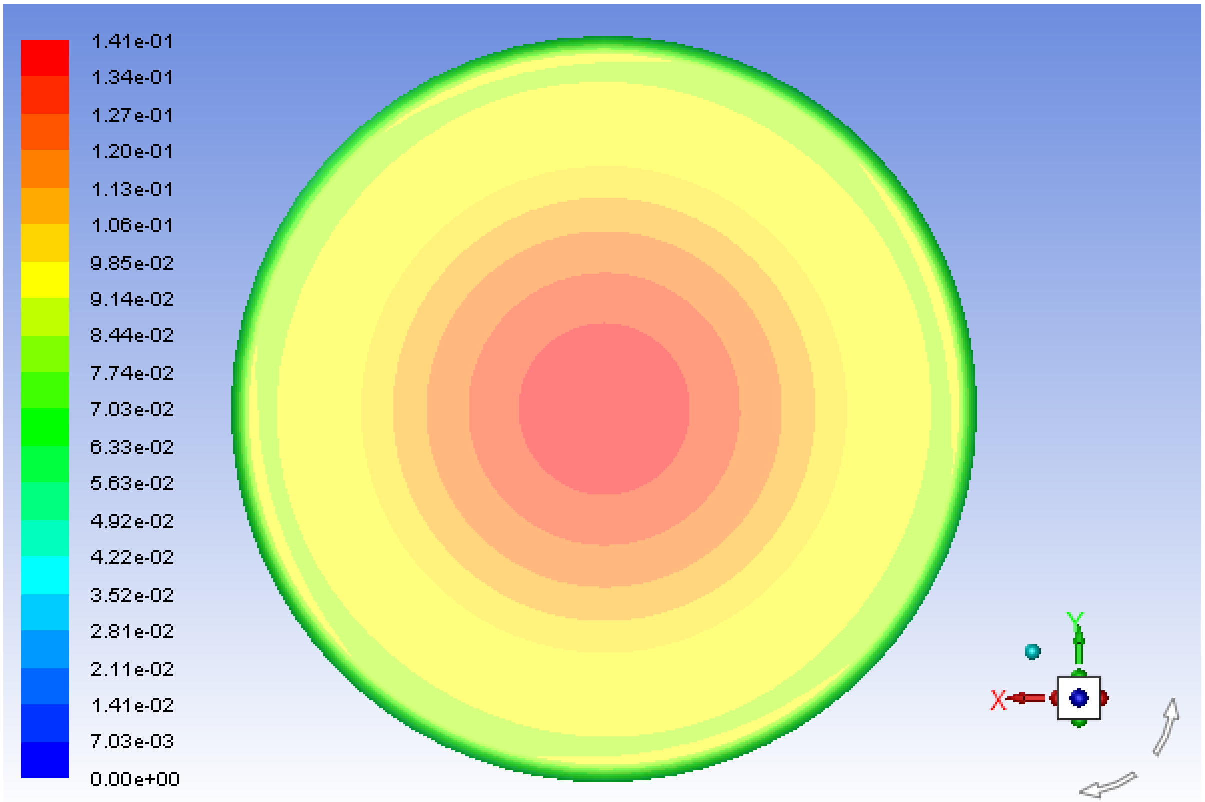
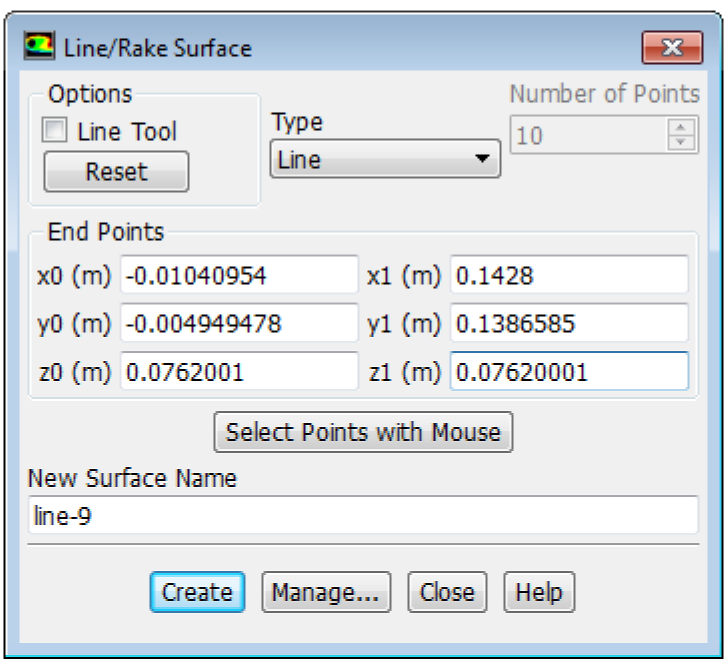


*Figure 15.7: Contours of Surface Coverage of ga_s (p. 655) shows the rate of surface coverage of the site species **ga_s**.*

5. Create a line surface from the center of **wall-4** to the edge.

Postsprocessing → Surface → Create → Line/Rake...



a. Enter the values for **x0**, **x1**, **y0**, **y1**, **z0**, and **z1** as follows:

End Points	Value
x0	-0.01040954
y0	-0.004949478
z0	0.0762001
x1	0.1428
y1	0.1386585
z1	0.07620001

*You can also select the points by clicking **Select Points with Mouse**. Then, in the graphic display, click at the center of **wall-4** and at the edge using the right mouse button.*

b. Click **Create** to accept the default name of `line-9` for the **New Surface Name**.

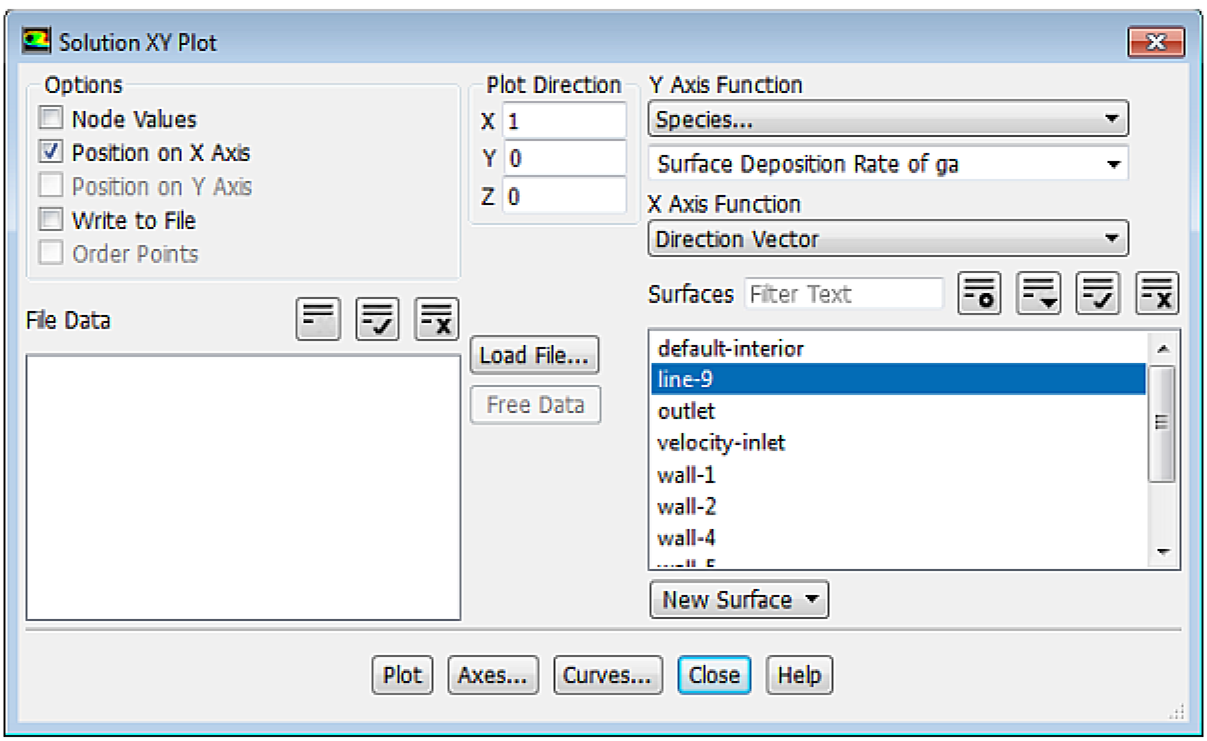
Note

If you want to delete or otherwise manipulate any surfaces, click **Manage...** to open the **Surfaces** dialog box

c. Close the **Line/Rake Surface** dialog box.

6. Plot the surface deposition rate of Ga versus radial distance (Figure 15.8: Plot of Surface Deposition Rate of Ga (p. 657)).

Postprocessing → **Plots** → **XY Plot** → **Edit...**



a. Disable **Node Values** in the **Options** group box.

b. Select **Species...** and **Surface Deposition Rate of ga** from the **Y Axis Function** drop-down lists.

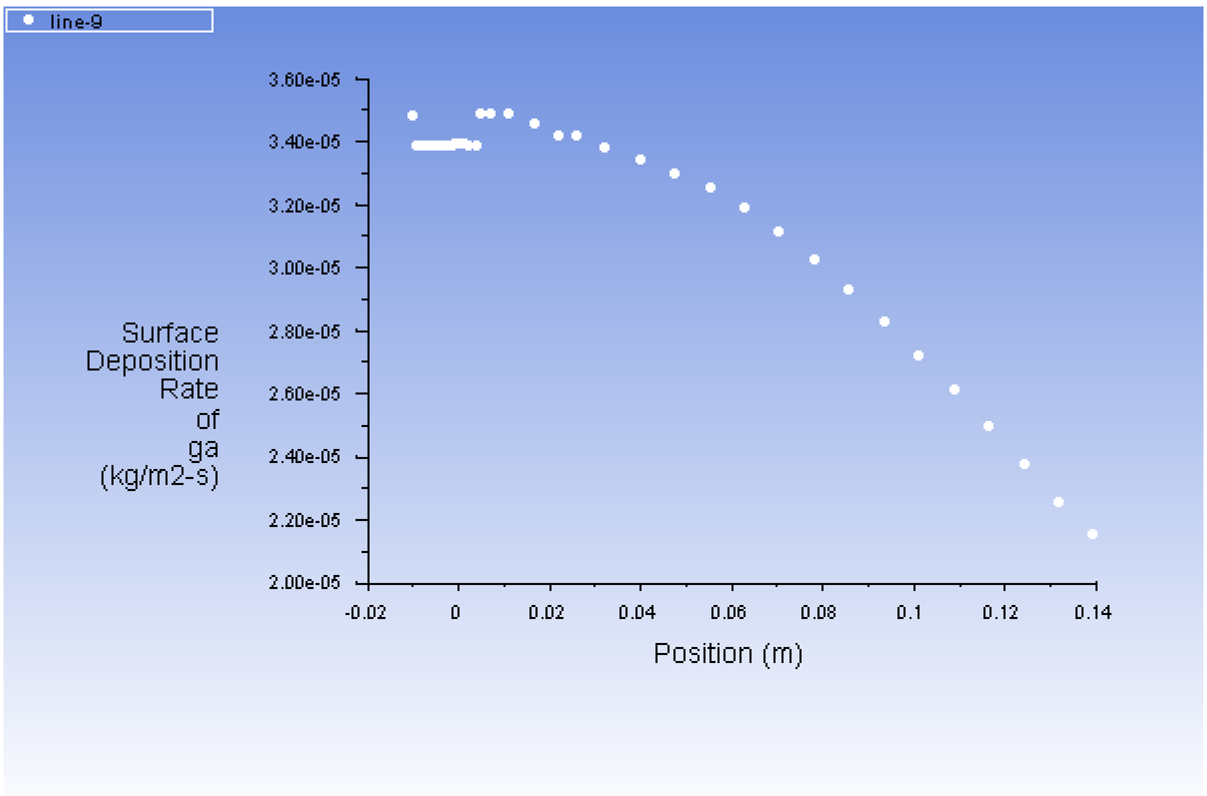
The source/sink terms due to the surface reaction are deposited in the cell adjacent to the wall cells, so it is necessary to plot the cell values and not the node values.

c. Select **line-9** you just created from the **Surfaces** selection list.

d. Click **Plot** and close the **Solution XY Plot** dialog box.

The peak surface deposition rate occurs at the center of ***wall-4*** *(where the concentration of the mixture is highest).*

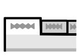
Figure 15.8: Plot of Surface Deposition Rate of Ga



Extra

You can also perform all the postprocessing steps to analyze the deposition of **As**.

7. Save the case and data files (`surface-react2.cas.gz` and `surface-react2.dat.gz`).

 **File** → **Write** → **Case & Data...**

15.5. Summary

The main focus of this tutorial is the accurate modeling of macroscopic gas flow, heat and mass transfer, species diffusion, and chemical reactions (including surface reactions) in a rotating disk CVD reactor. In this tutorial, you learned how to use the two-step surface reactions involving site species, and computed simultaneous deposition of gallium and arsenide from a mixture of precursor gases on a rotating susceptor. Note that the same approach is valid if you are simulating multi-step reactions with multiple sites/site species.

15.6. Further Improvements

This tutorial guides you through the steps to reach an initial solution. You may be able to obtain a more accurate solution by using an appropriate higher-order discretization scheme and by adapting the mesh. Mesh adaption can also ensure that the solution is independent of the mesh. These steps are demonstrated in Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121).

Chapter 16: Modeling Evaporating Liquid Spray

This tutorial is divided into the following sections:

16.1. Introduction

In this tutorial, the air-blast atomizer model in ANSYS Fluent is used to predict the behavior of an evaporating methanol spray. Initially, the air flow is modeled without droplets. To predict the behavior of the spray, the discrete phase model is used, including a secondary model for breakup.

This tutorial demonstrates how to do the following:

- Define a spray injection for an air-blast atomizer.
- Calculate a solution using the discrete phase model in ANSYS Fluent.

16.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

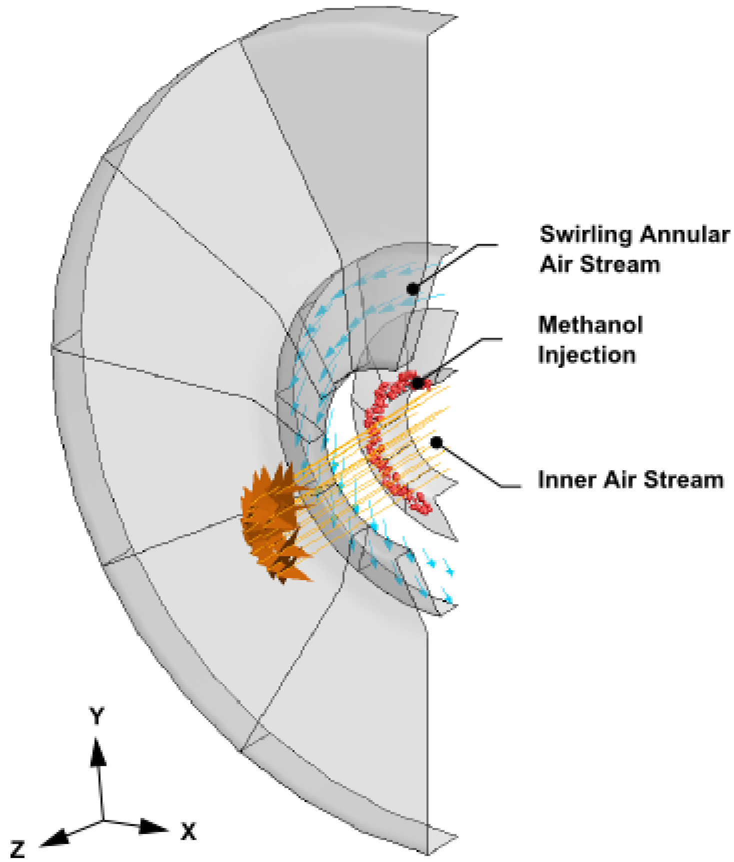
- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

and that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

16.3. Problem Description

The geometry to be considered in this tutorial is shown in Figure 16.1: Problem Specification (p. 660). Methanol is cooled to -10°C before being introduced into an air-blast atomizer. The atomizer contains an inner air stream surrounded by a swirling annular stream. To make use of the periodicity of the problem, only a 30° section of the atomizer will be modeled.

Figure 16.1: Problem Specification



16.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

16.4.1. Preparation

To prepare for running this tutorial:

1. Set up a working folder on the computer you will be using.

2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

   Note

   If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

3. Enter the name of this tutorial into the search bar.

4. Narrow the results by using the filter on the left side of the page.

   a. Click **ANSYS Fluent** under **Product**.

   b. Click **18.0** under **Version**.

5. Select this tutorial from the list.

6. Click the **evaporate_liquid_R180.zip** link to download the input files.

7. Unzip `evaporate_liquid_R180.zip` to your working folder.

   The mesh file `sector.msh` can be found in the `evaporate_liquid` directory created after unzipping the file.

8. Use the ANSYS Fluent Launcher to start the **3D** version of ANSYS Fluent.

   Fluent Launcher displays your **Display Options** preferences from the previous session.

   For more information about the ANSYS Fluent Launcher, see starting ANSYS Fluent using the Fluent Launcher in the Fluent Getting Started Guide.

9. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.

10. Ensure that the **Serial** processing option is selected.

11. Enable **Double Precision**.

12. Ensure that **Meshing Mode** is disabled.

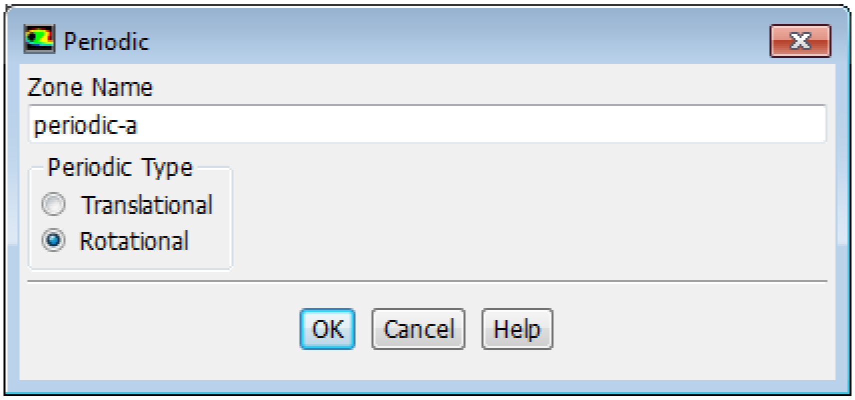
16.4.2. Mesh

1. Read in the mesh file `sector.msh`.

   File → Read → Mesh...

2. Change the periodic type of **periodic-a** to rotational.

   Setup → Boundary Conditions → periodic-a → Edit...



a. Select **Rotational** in the **Periodic Type** group box.

b. Click **OK** to close the **Periodic** dialog box.

3. In a similar manner, change the periodic type of **periodic-b** to rotational.

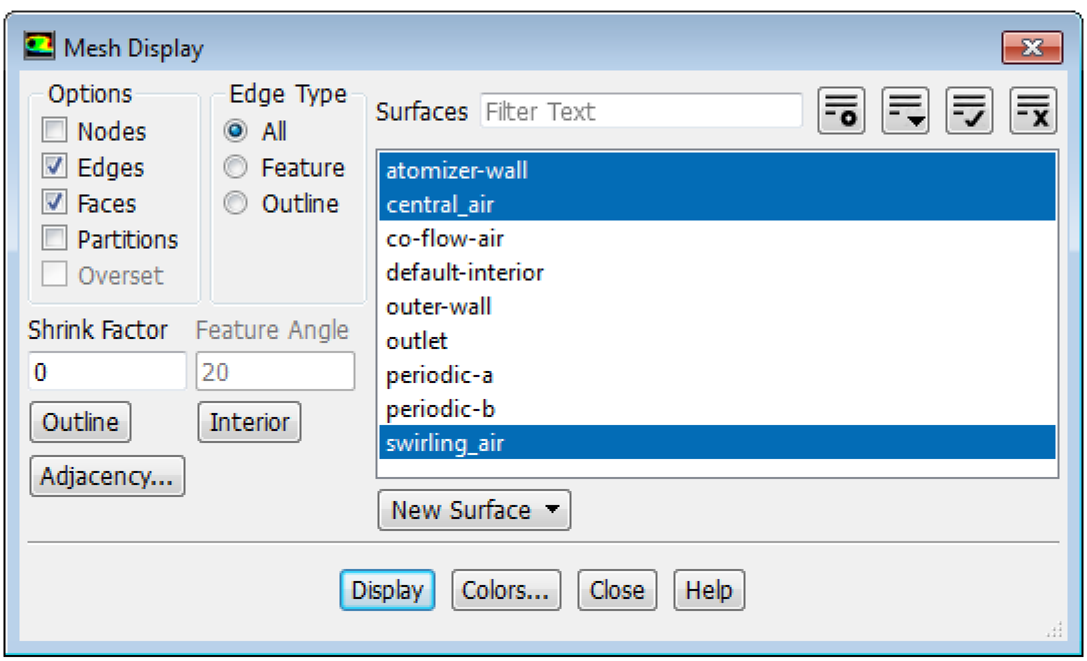
4. Check the mesh.

Setting Up Domain → **Mesh** → **Check**

ANSYS Fluent will perform various checks on the mesh and report the progress in the console. Ensure that the reported minimum volume is a positive number.

5. Display the mesh.

Setting Up Domain → **Mesh** → **Display...**



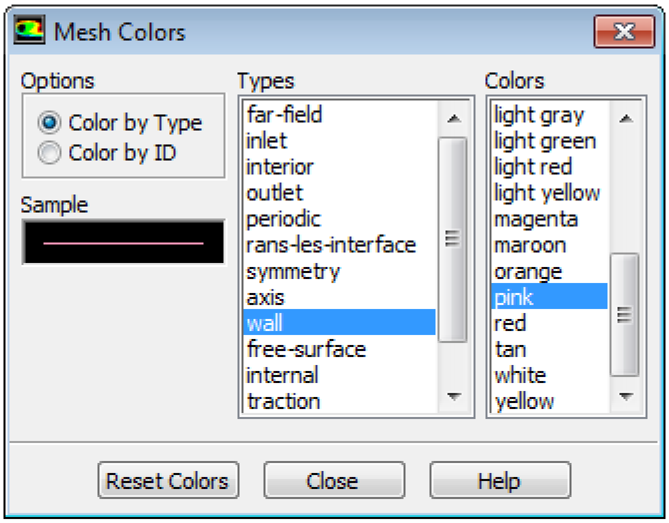
a. Ensure that **Faces** is enabled in the **Options** group box.

b. Select only **atomizer-wall**, **central_air**, and **swirling_air** from the **Surfaces** selection list.

Tip

To deselect all surfaces click the far-right button  at the top of the **Surfaces** selection list, and then select the desired surfaces from the **Surfaces** selection list.

c. Click the **Colors...** button to open the **Mesh Colors** dialog box.



i. Select **wall** from the **Types** selection list.

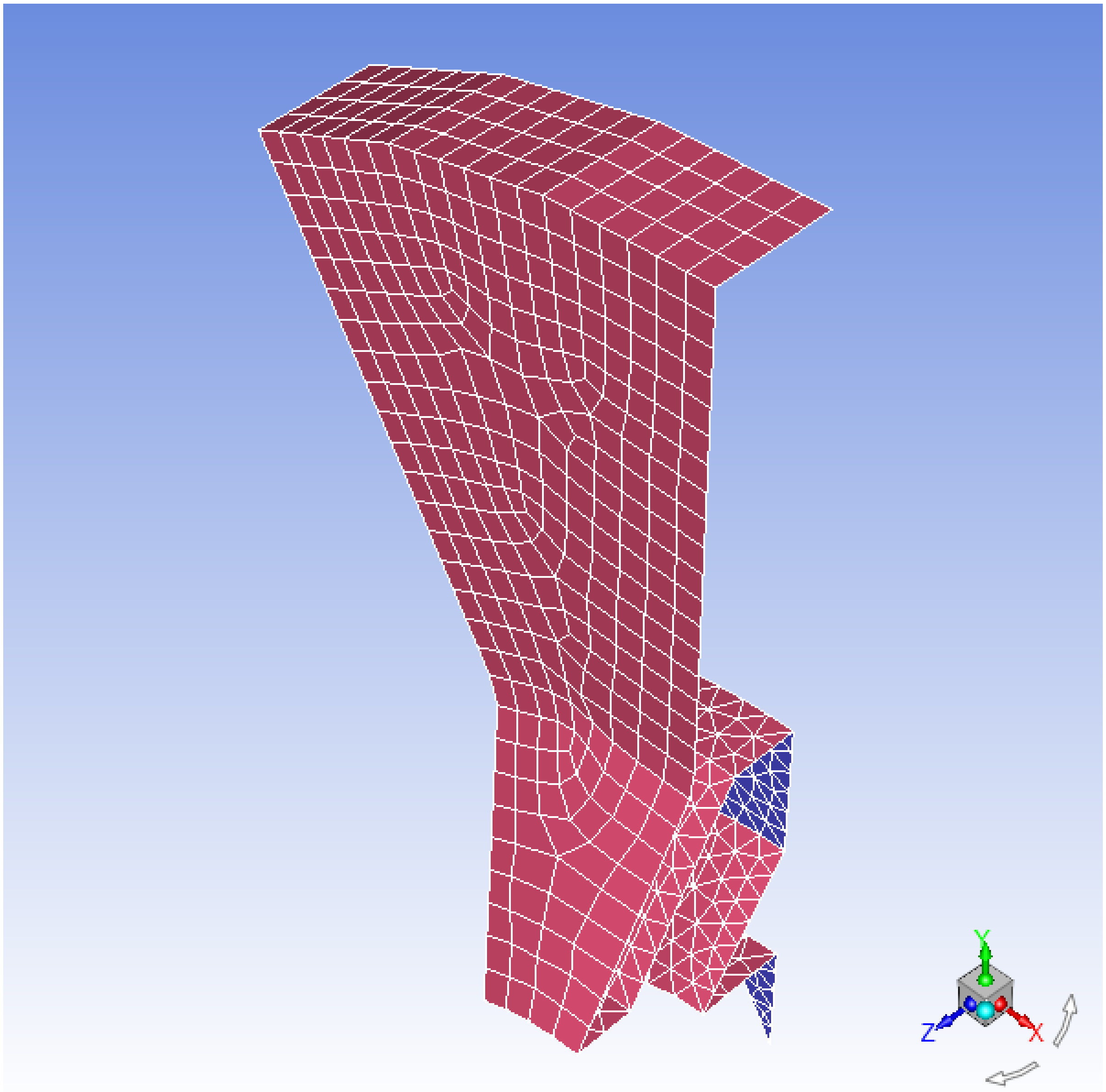
ii. Select **pink** from the **Colors** selection list.

iii. Close the **Mesh Colors** dialog box.

d. Click **Display** and close the **Mesh Display** dialog box.

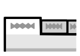
The graphics display will be updated to show the mesh. Zoom in with the mouse to obtain the view shown in *Figure 16.2: Air-Blast Atomizer Mesh Display* *(p. 664)*.

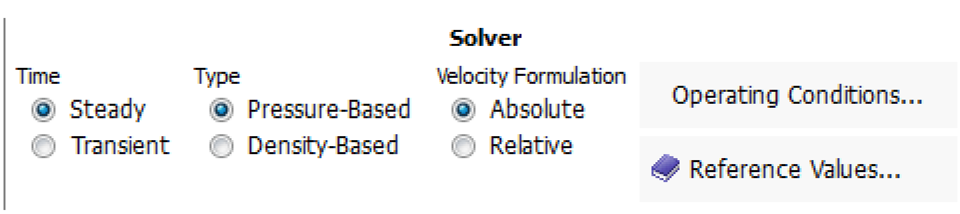
Figure 16.2: Air-Blast Atomizer Mesh Display



16.4.3. Solver

Retain the default solver settings of pressure-based steady-state solver in the **Solver** group of the **Setting Up Physics** tab.

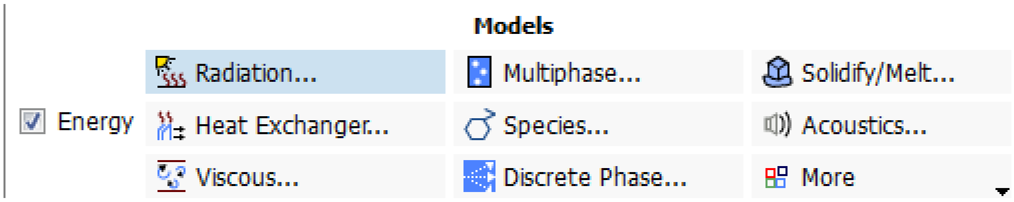
 **Setting Up Physics → Solver**



16.4.4. Models

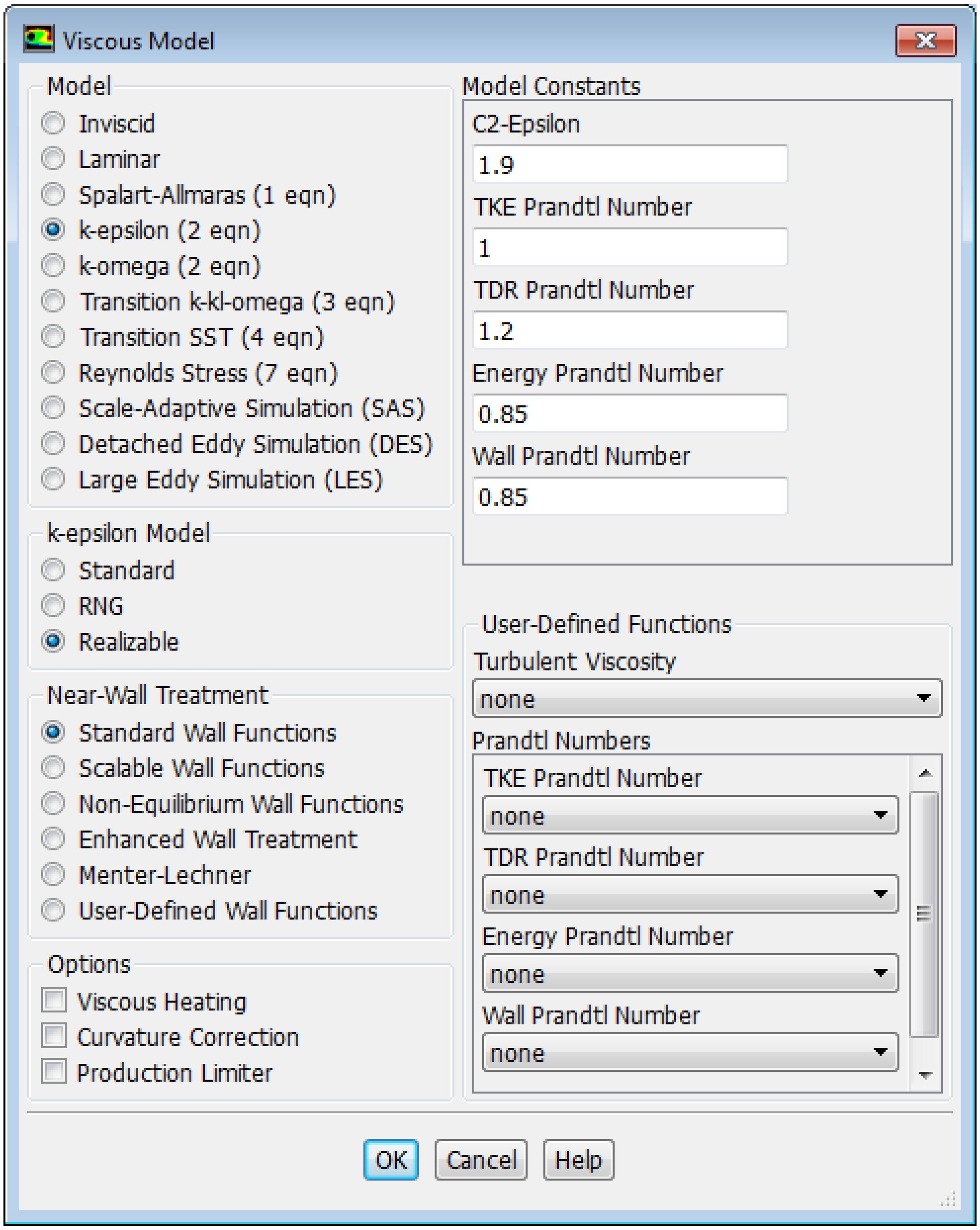
1. Enable heat transfer by enabling the energy equation.

   Setting Up Physics → Models → Energy



2. Enable the Realizable k- ε turbulence model.

   Setting Up Physics → Models → Viscous...



a. Select **k-epsilon (2 eqn)** in the **Model** list.

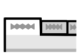
b. Select **Realizable** in the **k-epsilon Model** list.

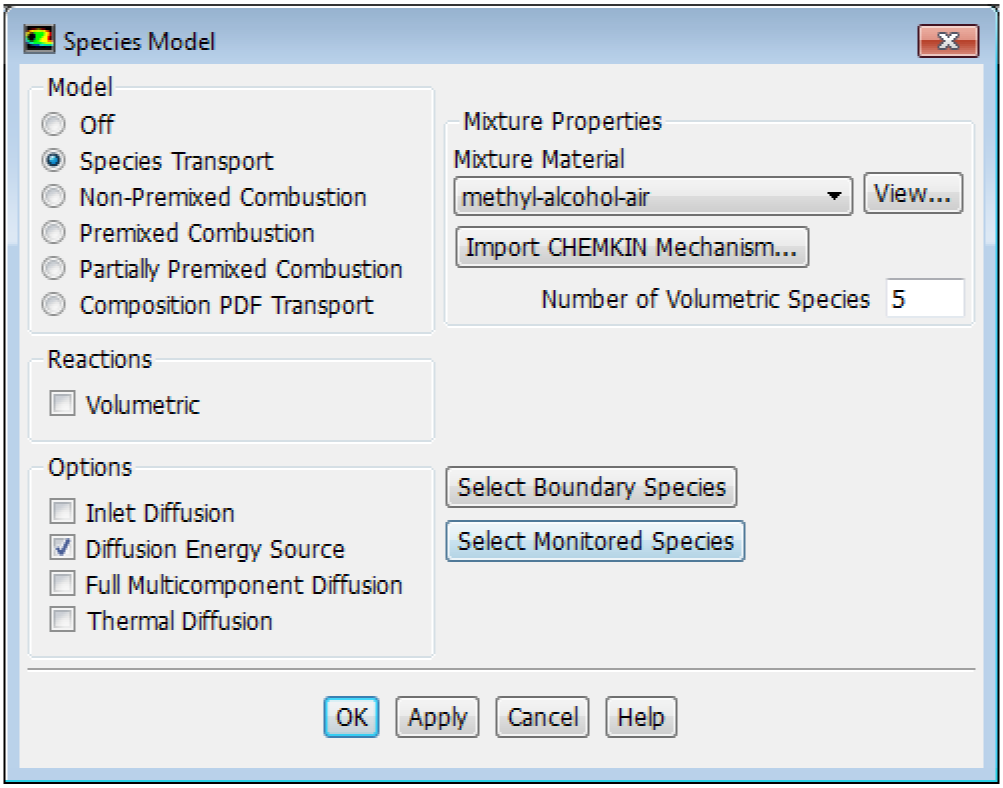
The Realizable k- ε *model gives a more accurate prediction of the spreading rate of both planar and round jets than the standard* k- ε *model.*

c. Retain the default selection of **Standard Wall Functions** in the **Near-Wall Treatment** list.

d. Click **OK** to close the **Viscous Model** dialog box.

3. Enable chemical species transport and reaction.

 **Setting Up Physics → Models → Species...**



a. Select **Species Transport** in the **Model** list.

b. Select **methyl-alcohol-air** from the **Mixture Material** drop-down list.

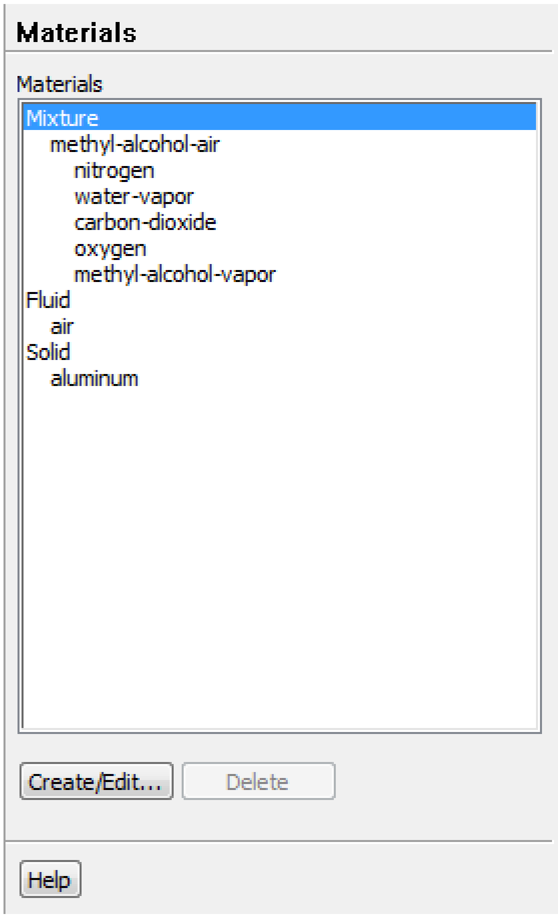
   *The **Mixture Material** list contains the set of chemical mixtures that exist in the ANSYS Fluent database. When selecting an appropriate mixture for your case, you can review the constituent species and the reactions of the predefined mixture by clicking **View...** next to the **Mixture Material** drop-down list. The chemical species and their physical and thermodynamic properties are defined by the selection of the mixture material. After enabling the **Species Transport** model, you can alter the mixture material selection or modify the mixture material properties using the **Create/Edit Materials** dialog box. You will modify your local copy of the mixture material later in this tutorial.*

c. Click **OK** to close the **Species Model** dialog box.

16.4.5. Materials

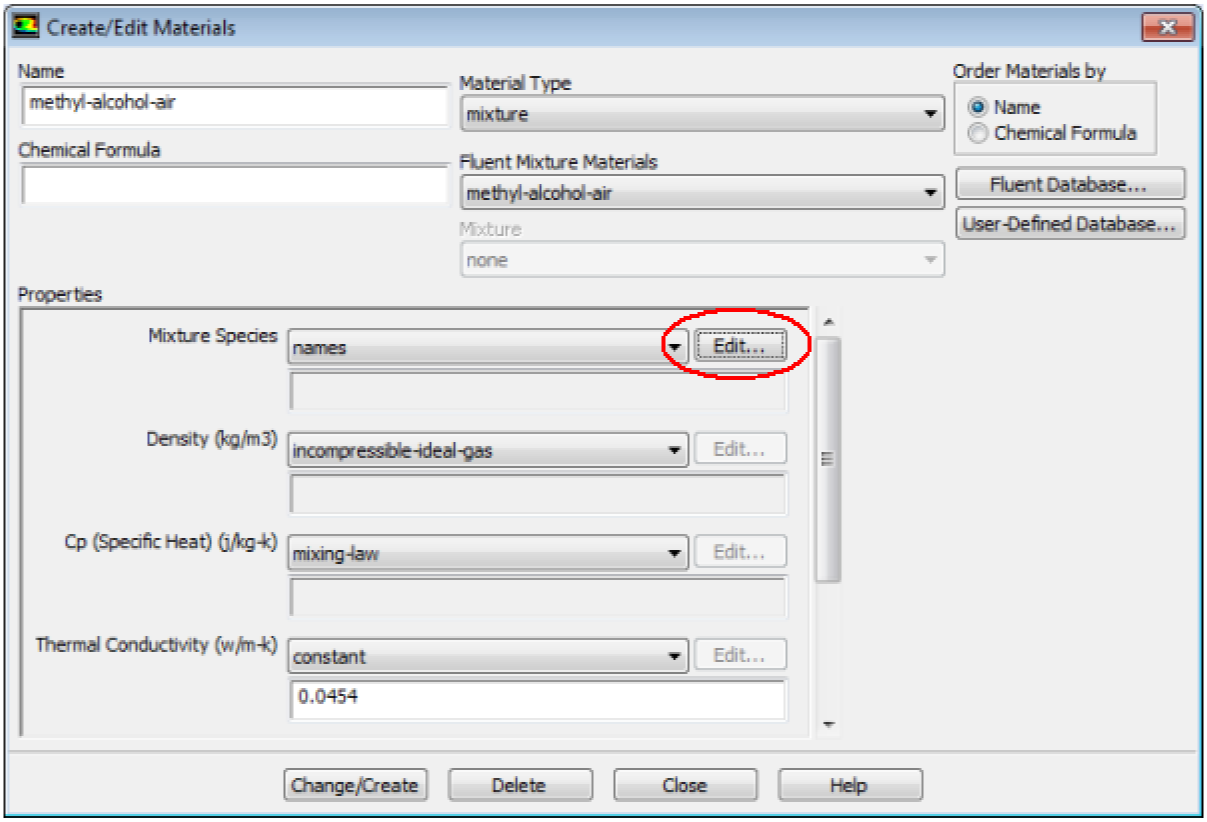
Define materials using the **Materials** task page.

Setup → **Materials**

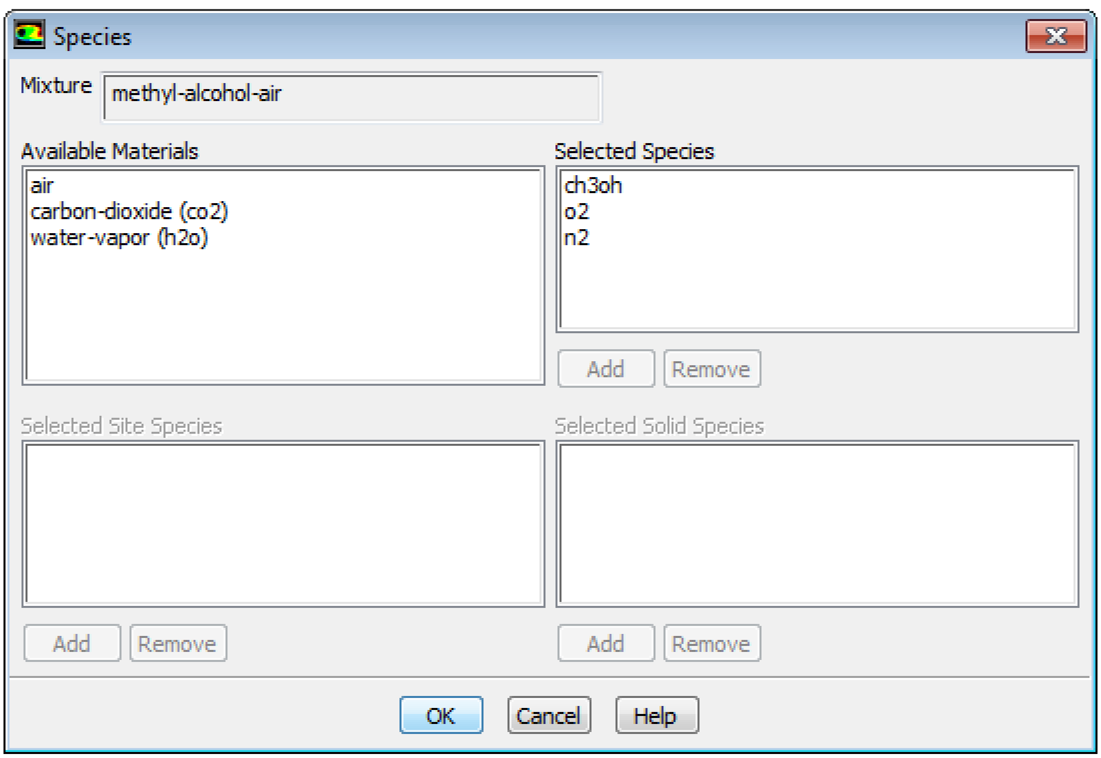


1. Remove water vapor and carbon dioxide from the **Mixture Species** list.

   Setup → **Materials** → **Mixture** → **Create/Edit...**



a. Click the **Edit** button next to the **Mixture Species** drop-down list to open the **Species** dialog box.



i. Select carbon dioxide (**co2**) from the **Selected Species** selection list.

ii. Click **Remove** to remove carbon dioxide from the **Selected Species** list.

iii. In a similar manner, remove water vapor (**h2o**) from the **Selected Species** list.

iv. Click **OK** to close the **Species** dialog box.

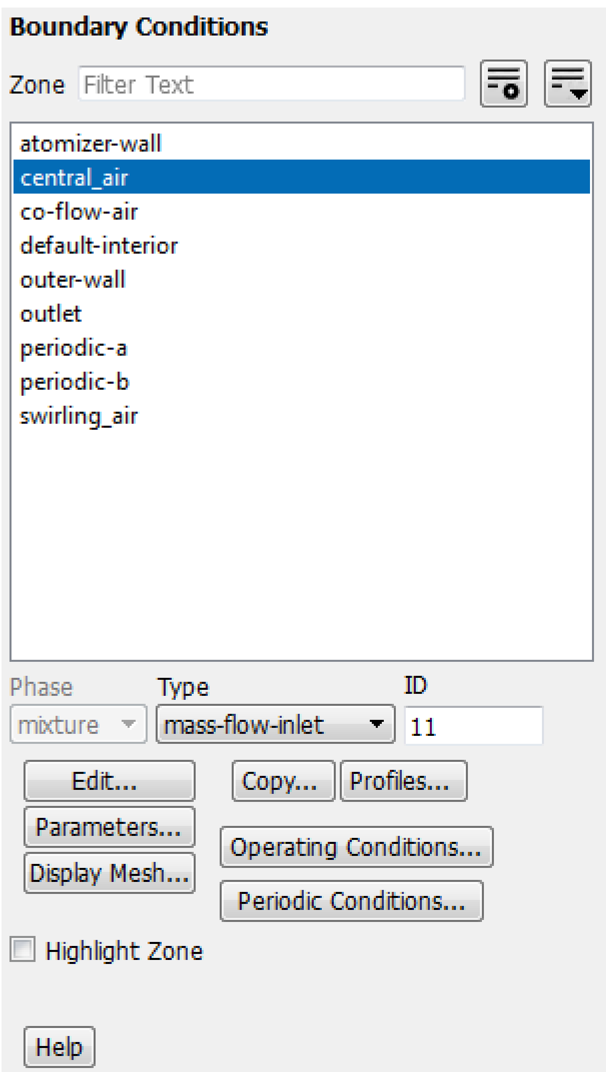
b. Click **Change/Create** and close the **Create/Edit Materials** dialog box.

> **Note**
>
> It is good practice to click the **Change/Create** button whenever changes are made to material properties even though it is not necessary in this case.

16.4.6. Boundary Conditions

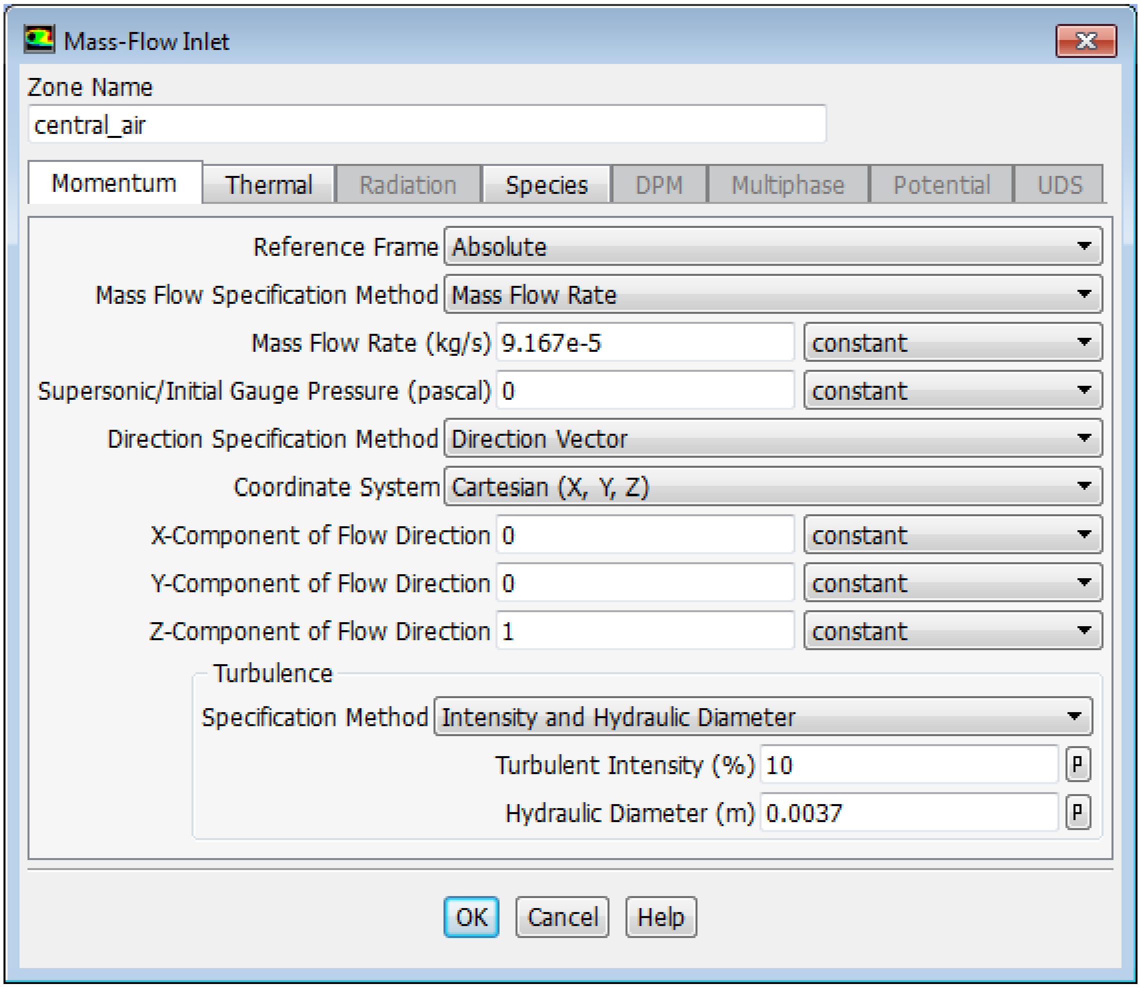
Specify boundary conditions using the **Boundary Conditions** task page.

Setup → **Boundary Conditions**



1. Set the boundary conditions for the inner air stream (**central_air**).

Setup → **Boundary Conditions** → **central_air** → **Edit...**



a. Enter `9.167e-5` kg/s for **Mass Flow Rate**.

b. Enter `0` for **X-Component of Flow Direction**.

c. Retain the default value of `0` for **Y-Component of Flow Direction**.

d. Enter `1` for **Z-Component of Flow Direction**.

e. Select **Intensity and Hydraulic Diameter** from the **Specification Method** drop-down list.

f. Enter `10` for **Turbulent Intensity**.

g. Enter `0.0037` m for **Hydraulic Diameter**.

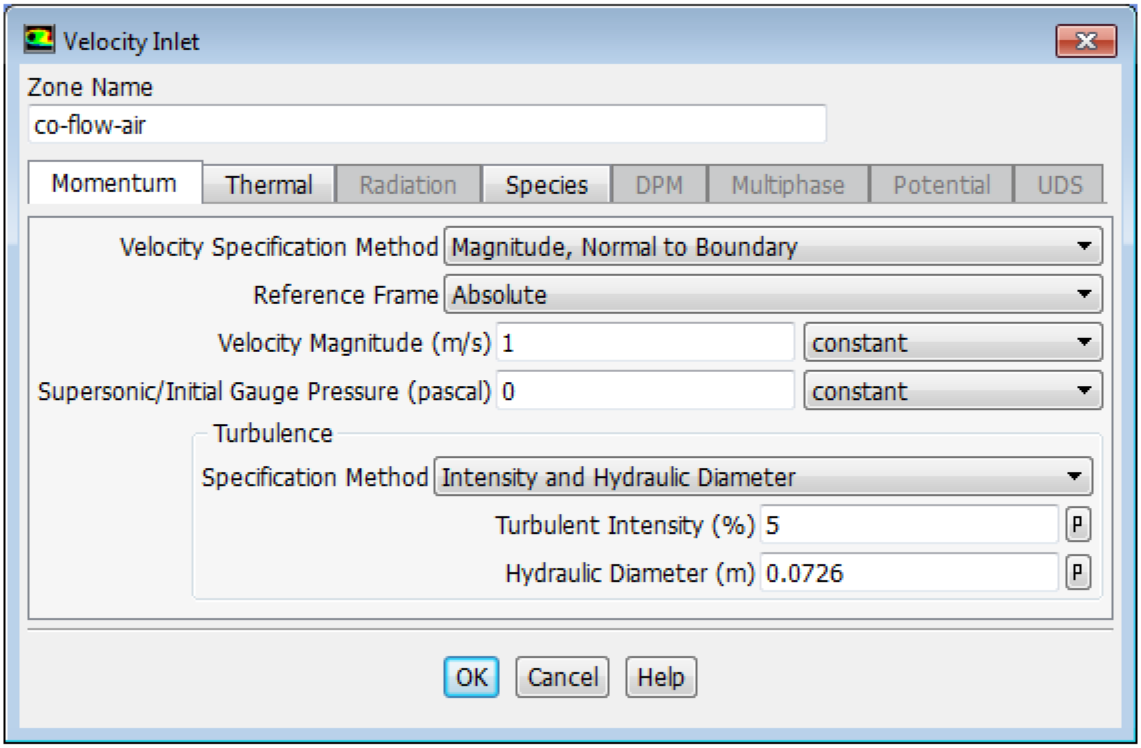
h. Click the **Thermal** tab and enter `293` K for **Total Temperature**.

i. Click the **Species** tab and enter `0.23` for **o2** in the **Species Mass Fractions** group box.

j. Click **OK** to close the **Mass-Flow Inlet** dialog box.

2. Set the boundary conditions for the air stream surrounding the atomizer (**co-flow-air**).

Setup → **Boundary Conditions** → **co-flow-air** → **Edit...**



a. Enter `1` m/s for **Velocity Magnitude**.

b. Select **Intensity and Hydraulic Diameter** from the **Specification Method** drop-down list.

c. Retain the default value of 5 for **Turbulent Intensity**.

d. Enter `0.0726` m for **Hydraulic Diameter**.

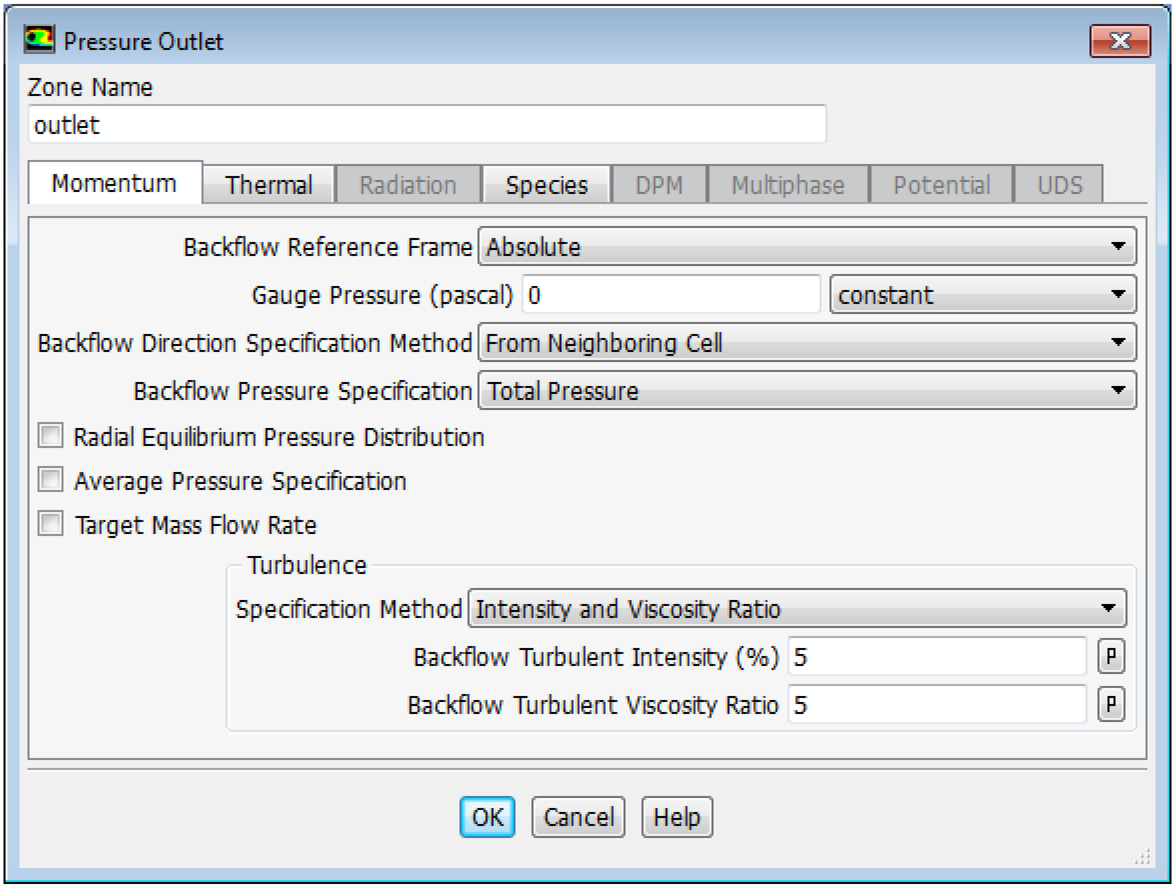
e. Click the **Thermal** tab and enter `293` K for **Temperature**.

f. Click the **Species** tab and enter `0.23` for **o2** in the **Species Mass Fractions** group box.

g. Click **OK** to close the **Velocity Inlet** dialog box.

3. Set the boundary conditions for the exit boundary (**outlet**).

Setup → **Boundary Conditions** → **outlet** → **Edit...**



a. Select **From Neighboring Cell** from the **Backflow Direction Specification Method** drop-down list.

b. Retain **Intensity and Viscosity Ratio** from the **Specification Method** drop-down list.

c. Retain the default value of 5 for **Backflow Turbulent Intensity (%)**.

d. Enter 5 for **Backflow Turbulent Viscosity Ratio**.

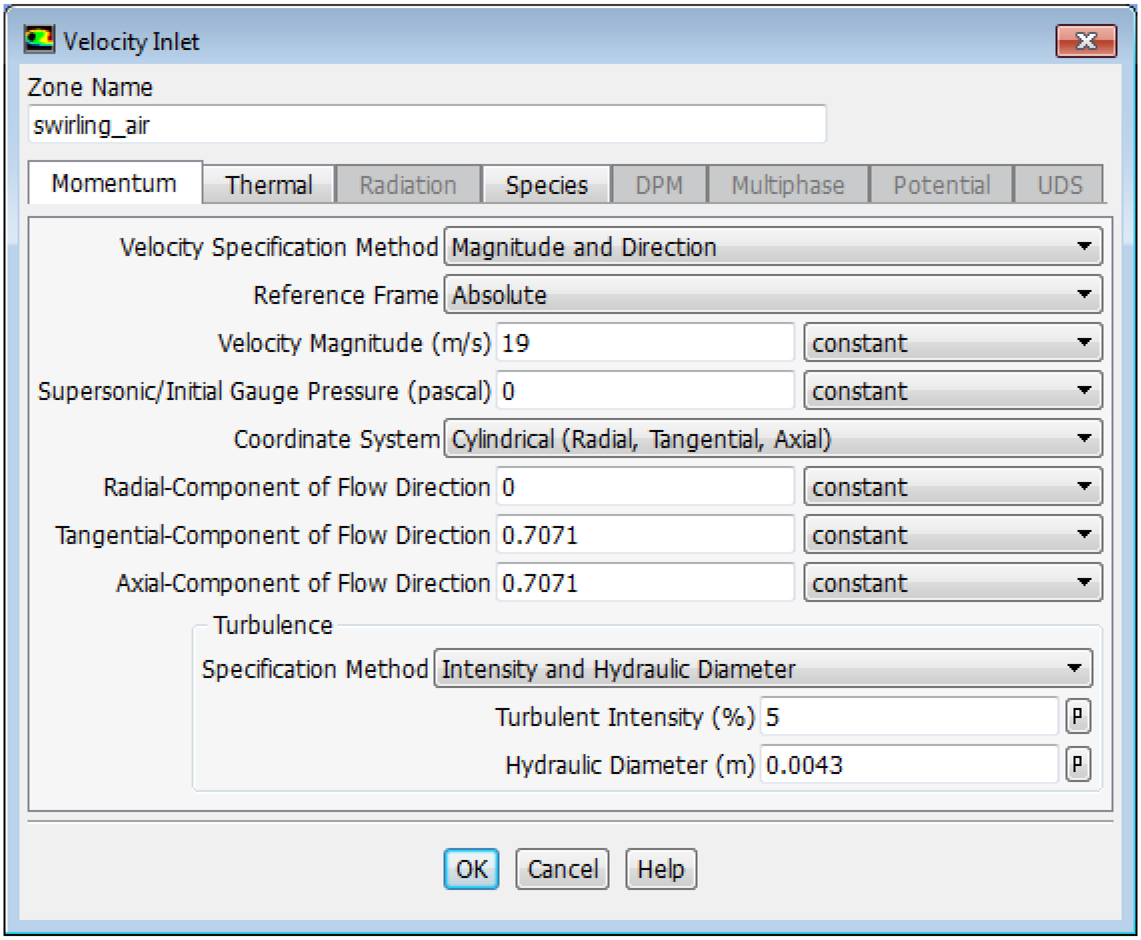
e. Click the **Thermal** tab and enter 293 K for **Backflow Total Temperature**.

f. Click the **Species** tab and enter `0.23` for **o2** in the **Species Mass Fractions** group box.

g. Click **OK** to close the **Pressure Outlet** dialog box.

4. Set the boundary conditions for the swirling annular stream (**swirling_air**).

Setup → **Boundary Conditions** → **swirling_air** → **Edit...**



a. Select **Magnitude and Direction** from the **Velocity Specification Method** drop-down list.

b. Enter `19` m/s for **Velocity Magnitude**.

c. Select **Cylindrical (Radial, Tangential, Axial)** from the **Coordinate System** drop-down list.

d. Enter `0` for **Radial-Component of Flow Direction**.

e. Enter `0.7071` for **Tangential-Component of Flow Direction**.

f. Enter `0.7071` for **Axial-Component of Flow Direction**.

g. Select **Intensity and Hydraulic Diameter** from the **Specification Method** drop-down list.

h. Retain the default value of 5 for **Turbulent Intensity**.

i. Enter `0.0043` m for **Hydraulic Diameter**.

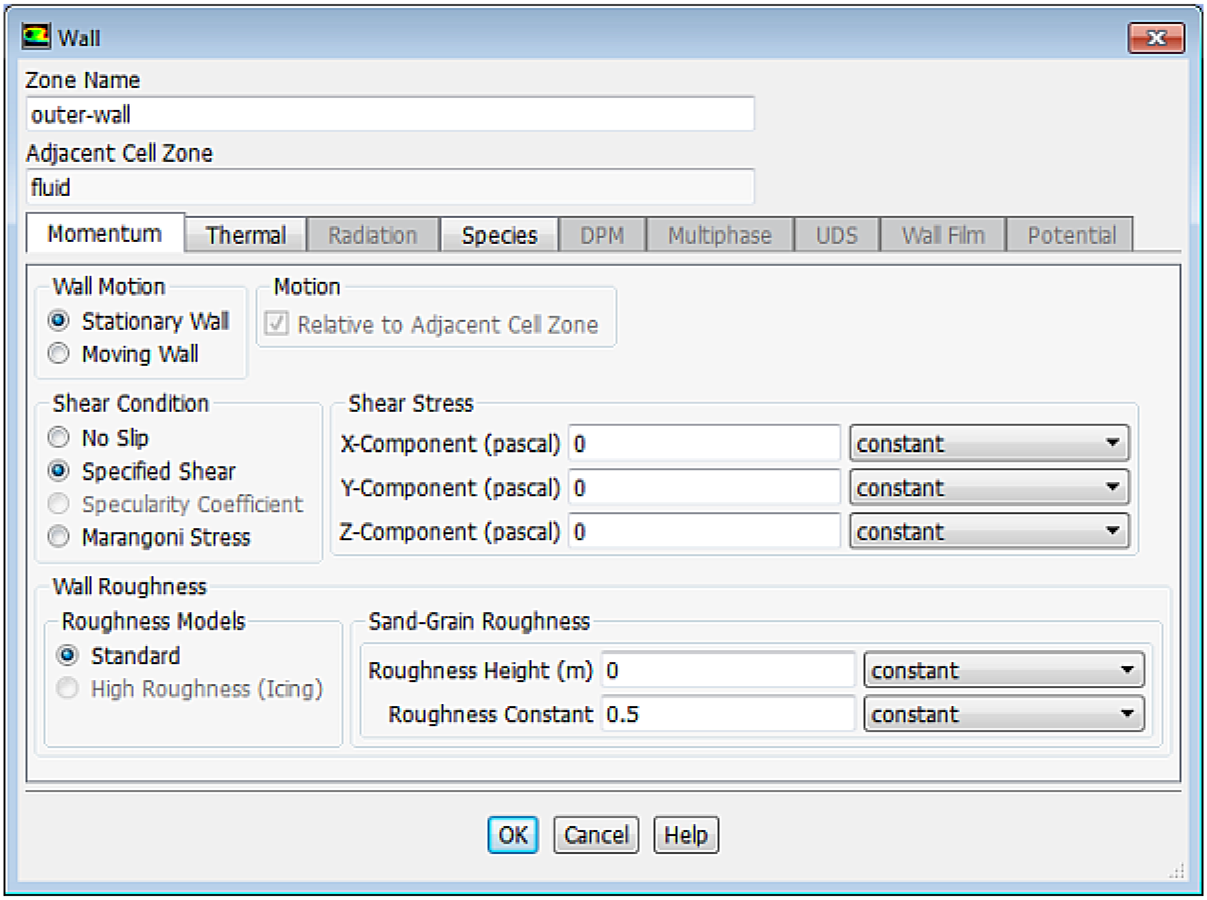
j. Click the **Thermal** tab and enter `293` K for **Temperature**.

k. Click the **Species** tab and enter `0.23` for **o2** in the **Species Mass Fractions** group box.

l. Click **OK** to close the **Velocity Inlet** dialog box.

5. Set the boundary conditions for the outer wall of the atomizer (**outer-wall**).

Setup → **Boundary Conditions** → **outer-wall** → **Edit...**



a. Select **Specified Shear** in the **Shear Condition** list.

b. Retain the default values for the remaining parameters.

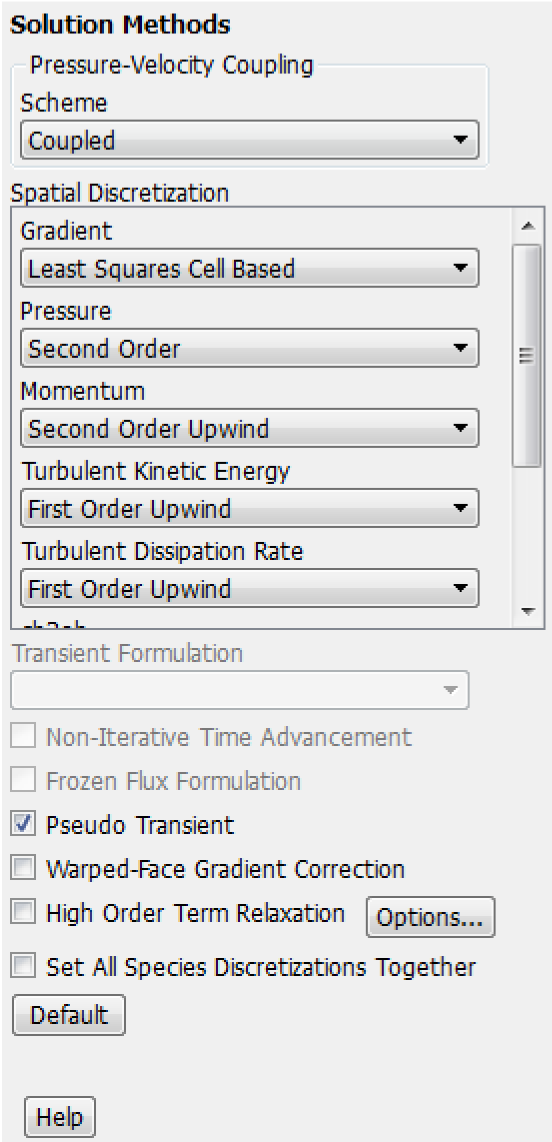
c. Click **OK** to close the **Wall** dialog box.

16.4.7. Initial Solution Without Droplets

The airflow will first be solved and analyzed without droplets.

1. Set the solution method.

Solving → **Solution** → **Methods...**

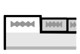


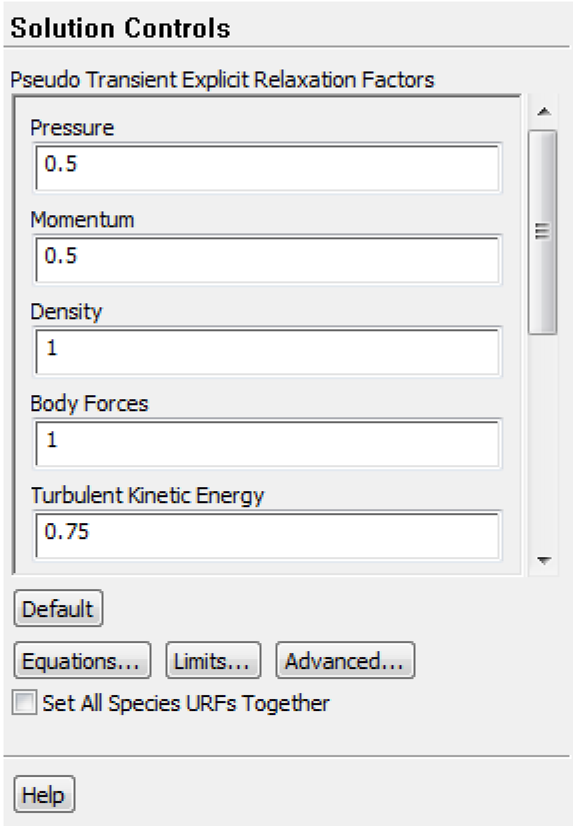
a. Select **Coupled** from the **Scheme** drop-down list in the **Pressure-Velocity Coupling** group box.

b. Enable **Pseudo Transient**.

   The message appears in the console informing you of changing AMG cycle type for Volume Fraction, Turbulent Kinetic Energy, and Turbulent Dissipation Rate to F-cycle.

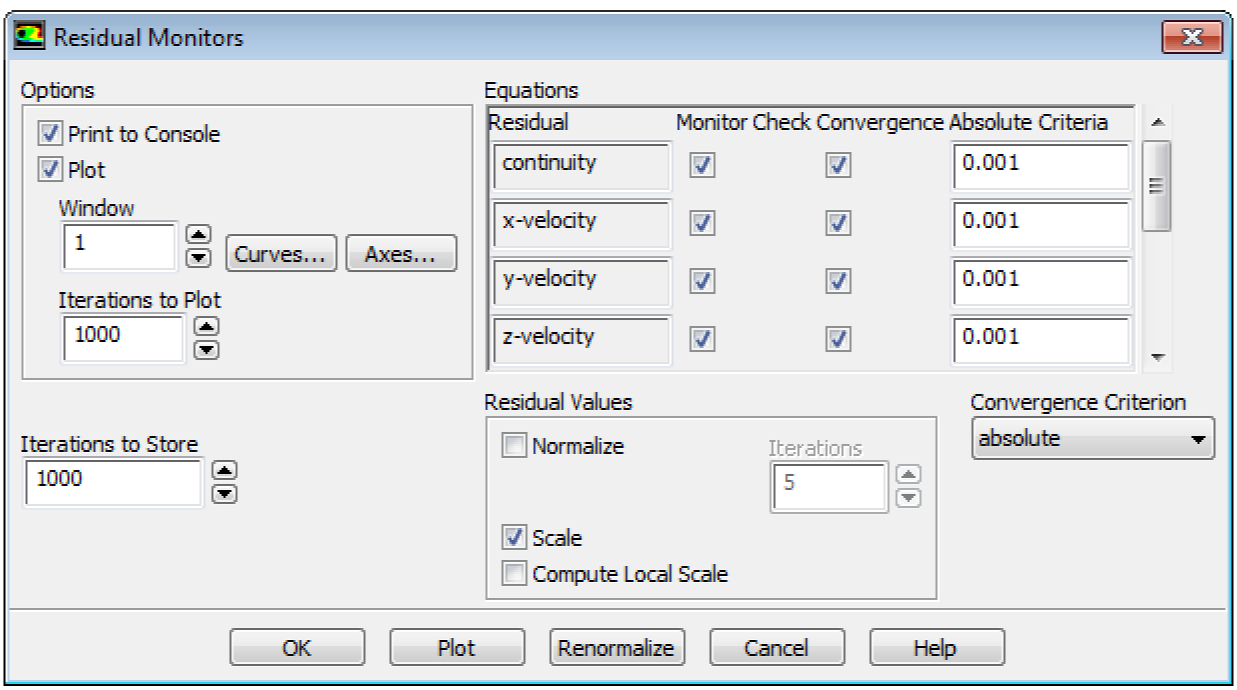
2. Retain the default under-relaxation factors.

 **Solving** → **Controls** → **Controls...**



3. Enable residual plotting during the calculation.

Solving → **Reports** → **Residuals...**



a. Ensure that **Plot** is enabled in the **Options** group box.

b. Click **OK** to close the **Residual Monitors** dialog box.

4. Initialize the flow field.

Solving → **Initialization**

a. Retain the **Method** at the default of **Hybrid**.

b. Click **Initialize** to initialize the variables.

Note

For flows in complex topologies, hybrid initialization will provide better initial velocity and pressure fields than standard initialization. This will help to improve the convergence behavior of the solver.

---

5. Save the case file (`spray1.cas.gz`).

File → **Write** → **Case...**

6. Start the calculation by requesting 150 iterations.

Solution → **Run Calculation**

a. Select **User Specified** from the **Time Step Method** group box.

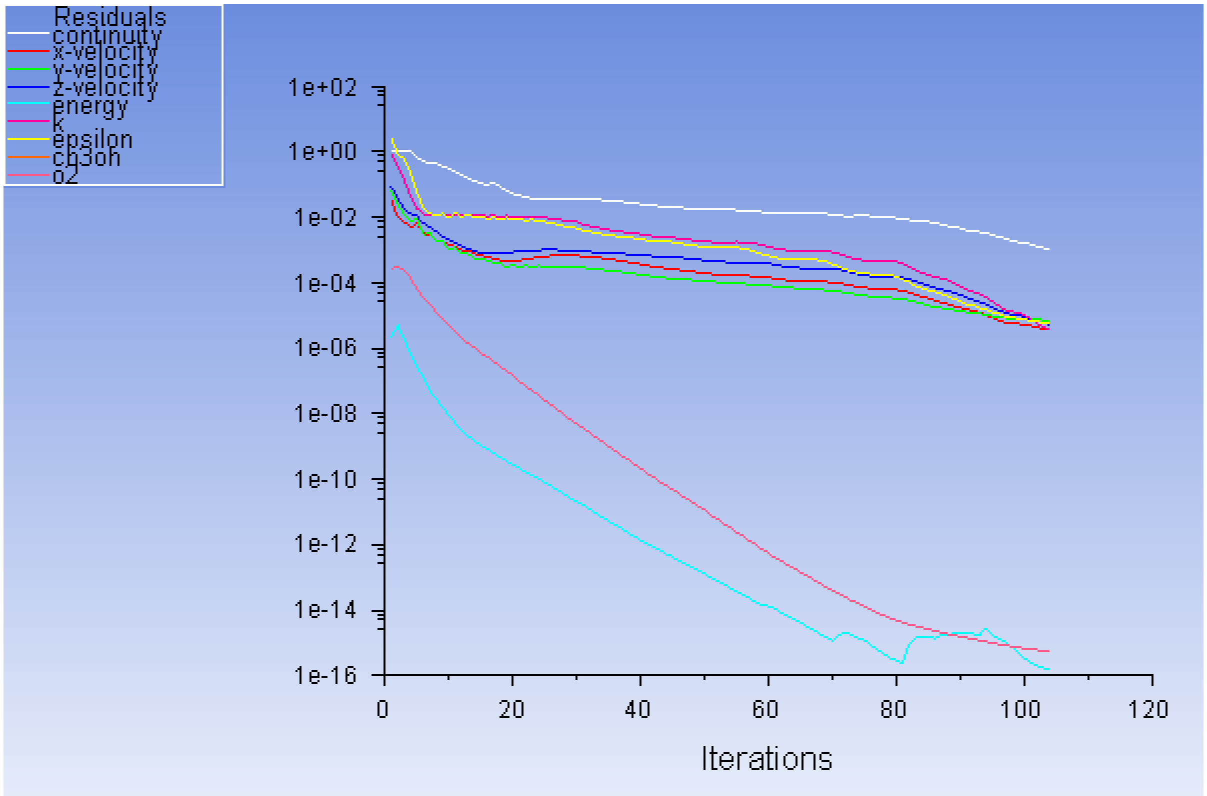
b. Retain `1 s` for **Pseudo Time Step**.

c. Enter `150` for **Number of Iterations**.

d. Click **Calculate**.

The solution will converge in approximately 105 iterations.

Figure 16.3: Scaled Residuals



7. Save the case and data files (`spray1.cas.gz` and `spray1.dat.gz`).

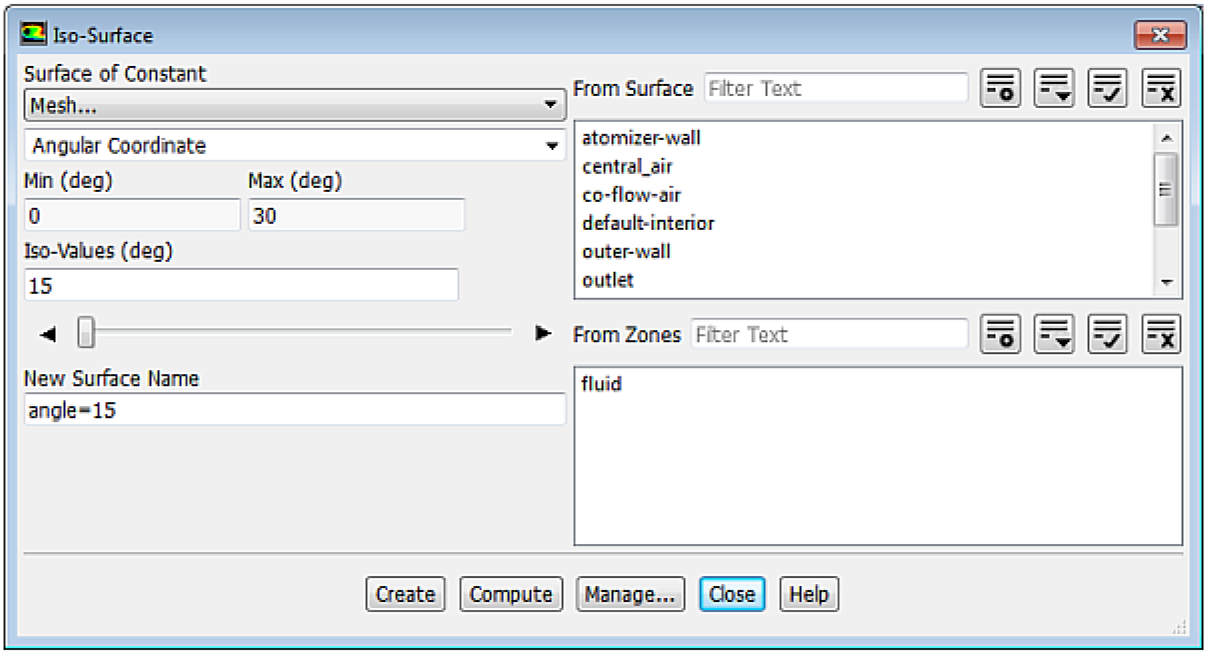
   File → Write → Case & Data...

   Note

   ANSYS Fluent will ask you to confirm that the previous case file is to be overwritten.

8. Create a clip plane to examine the flow field at the midpoint of the atomizer section.

   Postprocessing → Surface → Create → Iso-Surface...



a. Select **Mesh...** and **Angular Coordinate** from the **Surface of Constant** drop-down lists.

b. Click **Compute** to obtain the minimum and maximum values of the angular coordinate.

c. Enter `15` for **Iso-Values**.

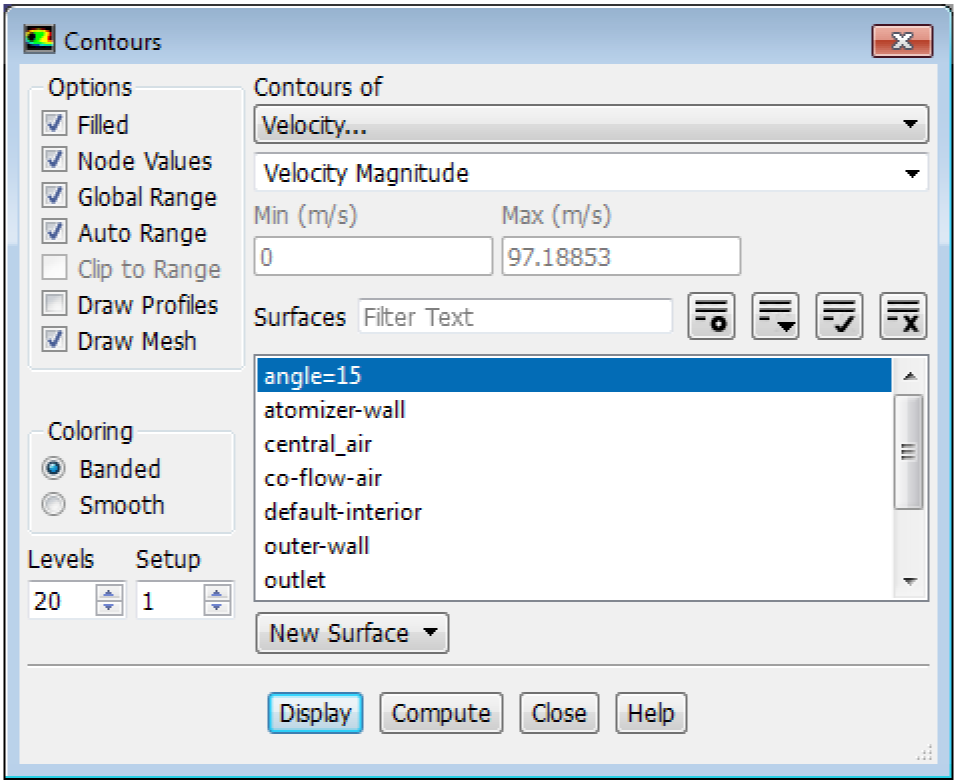
d. Enter `angle=15` for **New Surface Name**.

e. Click **Create** to create the isosurface.

f. Close the **Iso-Surface** dialog box.

9. Review the current state of the solution by examining contours of velocity magnitude (Figure 16.4: Velocity Magnitude at Mid-Point of Atomizer Section (p. 682)).

Postprocessing → **Graphics** → **Contours** → **Edit...**



a. Enable **Filled** in the **Options** group box

b. Select **Velocity...** and **Velocity Magnitude** from the **Contours of** drop-down lists.

c. Enable **Draw Mesh**.

   *The **Mesh Display** dialog box will open.*

   i. Retain the current mesh display settings.

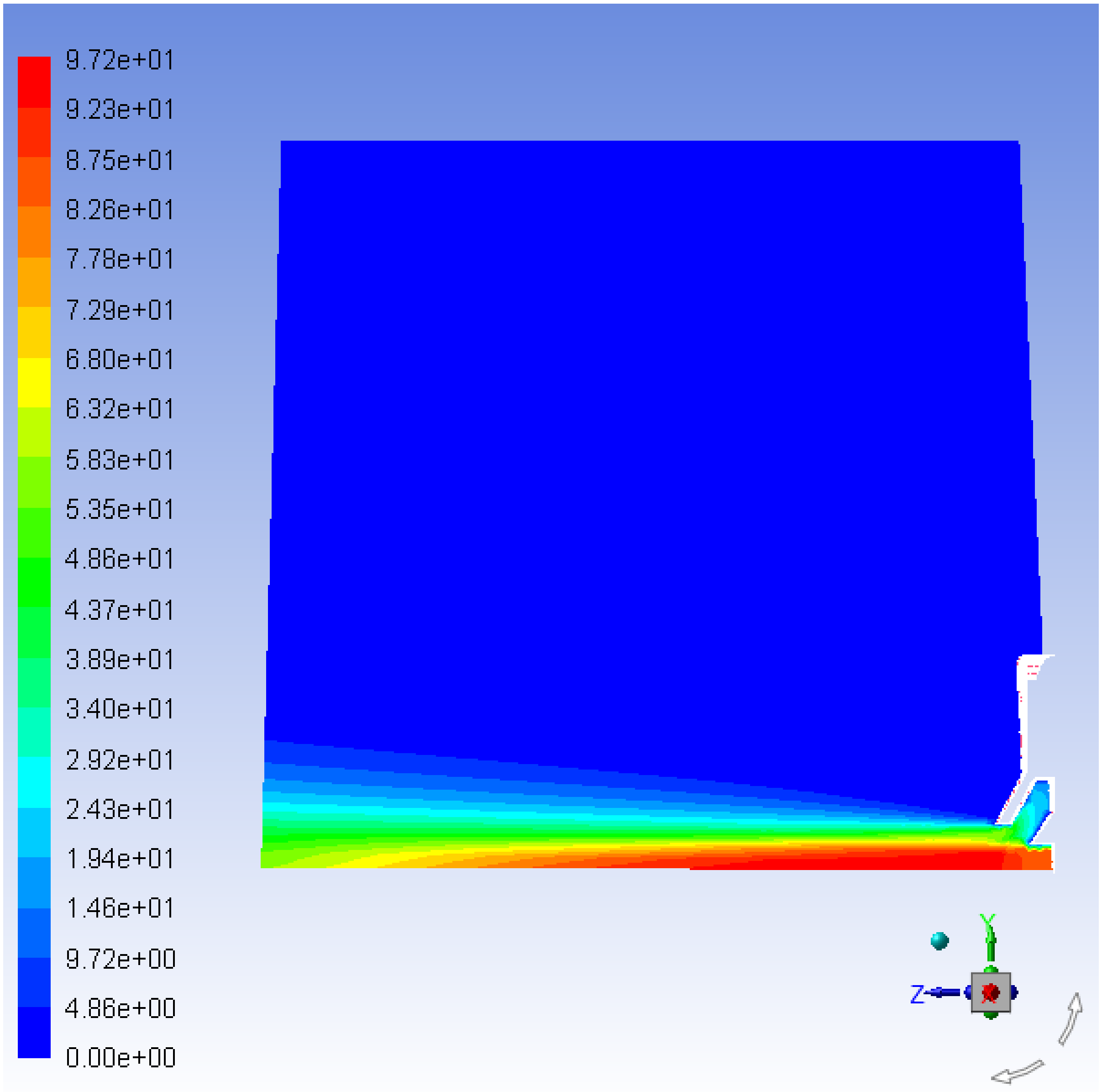
   ii. Close the **Mesh Display** dialog box.

d. Select **angle=15** from the **Surfaces** selection list.

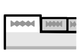
e. Click **Display** and close the **Contours** dialog box.

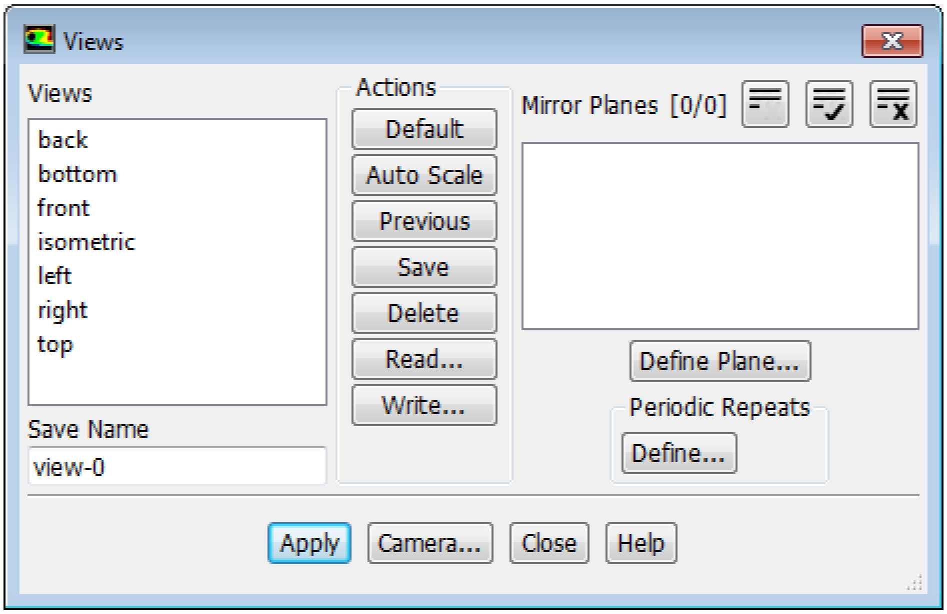
f. Use the mouse to obtain the view shown in Figure 16.4: Velocity Magnitude at Mid-Point of Atomizer Section (p. 682).

Figure 16.4: Velocity Magnitude at Mid-Point of Atomizer Section

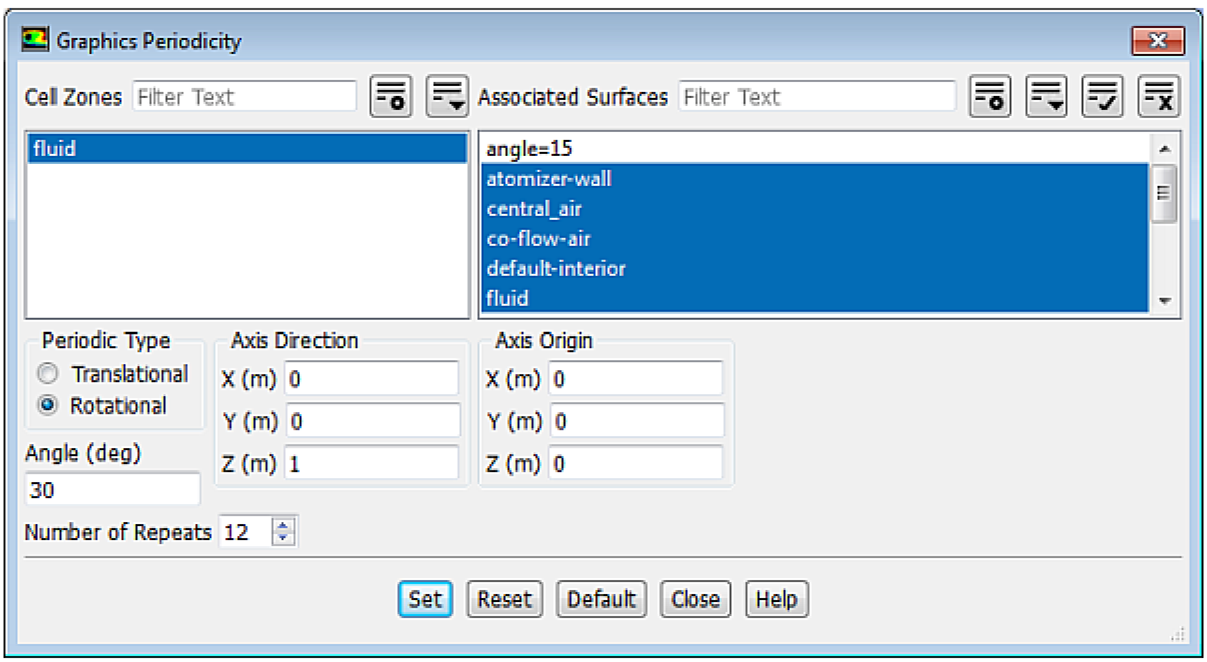


10. Modify the view to include the entire atomizer.

 **Viewing → Display → Views...**



a. Click the **Define...** button to open the **Graphics Periodicity** dialog box.



i. Select **fluid** from the **Cell Zones** selection list.

ii. Retain the selection of **Rotational** in the **Periodic Type** list.

iii. Retain the value of 12 for **Number of Repeats**.

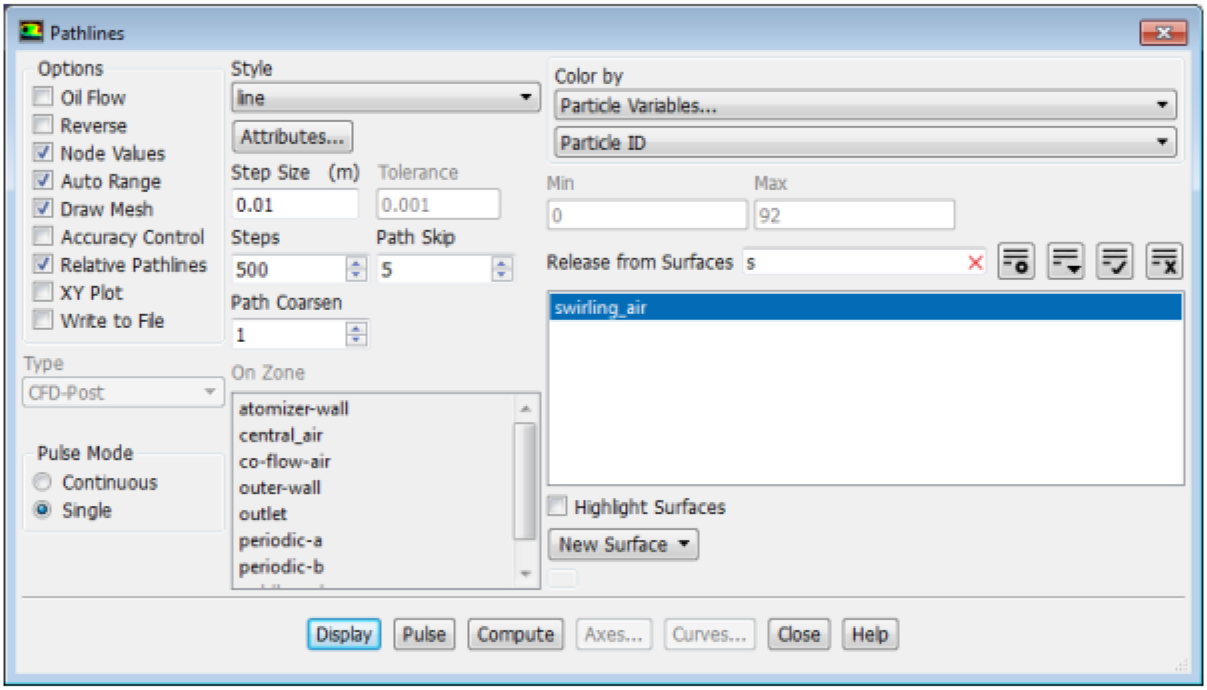
iv. Click **Set** and close the **Graphics Periodicity** dialog box.

The graphics display will be updated to show the entire atomizer.

b. Click **Apply** and close the **Views** dialog box.

11. Display pathlines of the air in the swirling annular stream (Figure 16.5: Pathlines of Air in the Swirling Annular Stream (p. 685)).

Postprocessing → **Graphics** → **Pathlines** → **Edit...**



a. Increase the **Path Skip** value to 5.

b. In the **Release from Surfaces** filter, type **s** to display the surface names that begin with **s** and select **swirling_air** from the selection list.

c. Enable **Draw Mesh** in the **Options** group box.

   *The **Mesh Display** dialog box will open.*

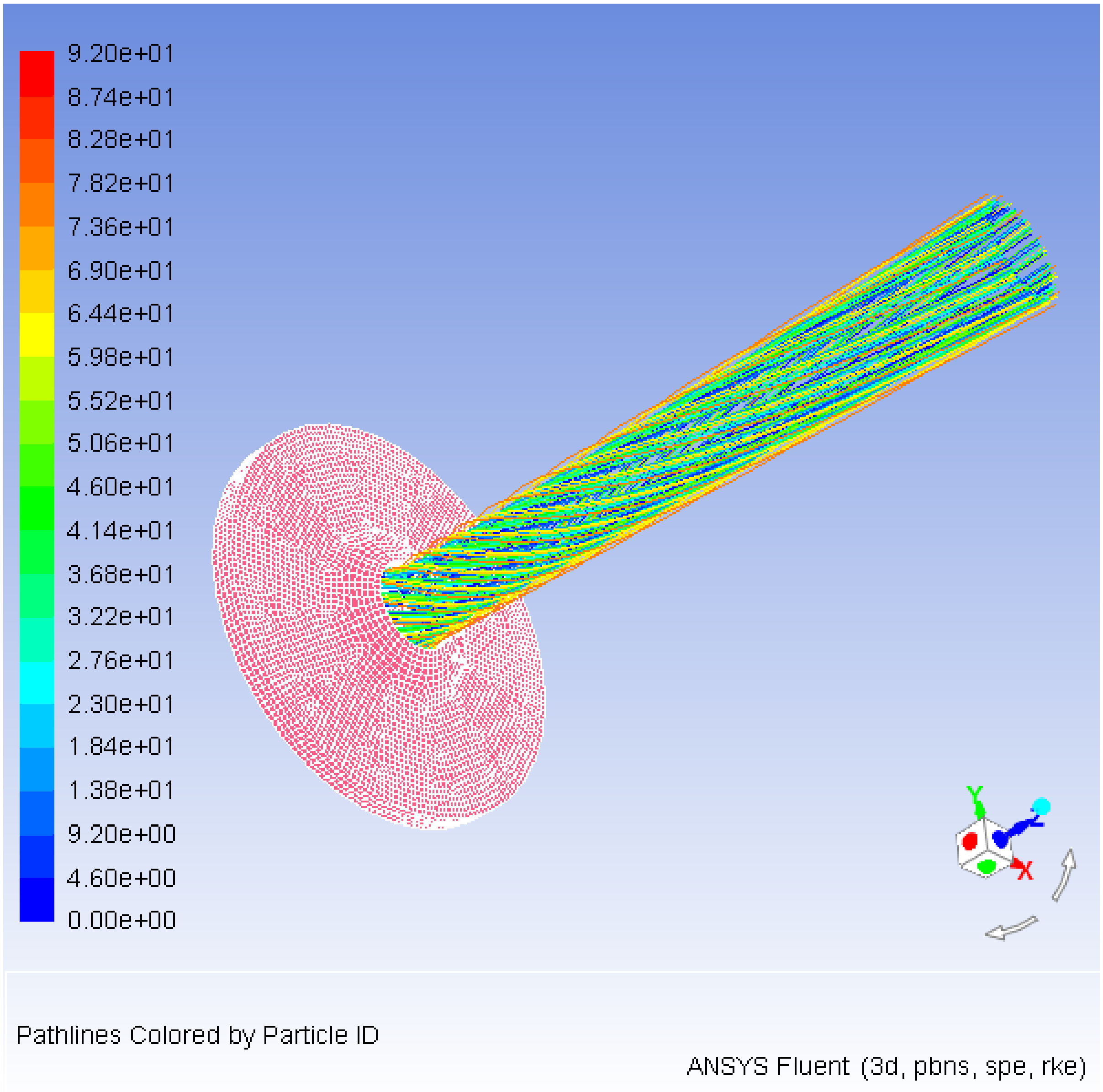
   i. Retain the current mesh display settings.

   ii. Close the **Mesh Display** dialog box.

d. Click **Display** and close the **Pathlines** dialog box.

e. Use the mouse to obtain the view shown in Figure 16.5: Pathlines of Air in the Swirling Annular Stream (p. 685).

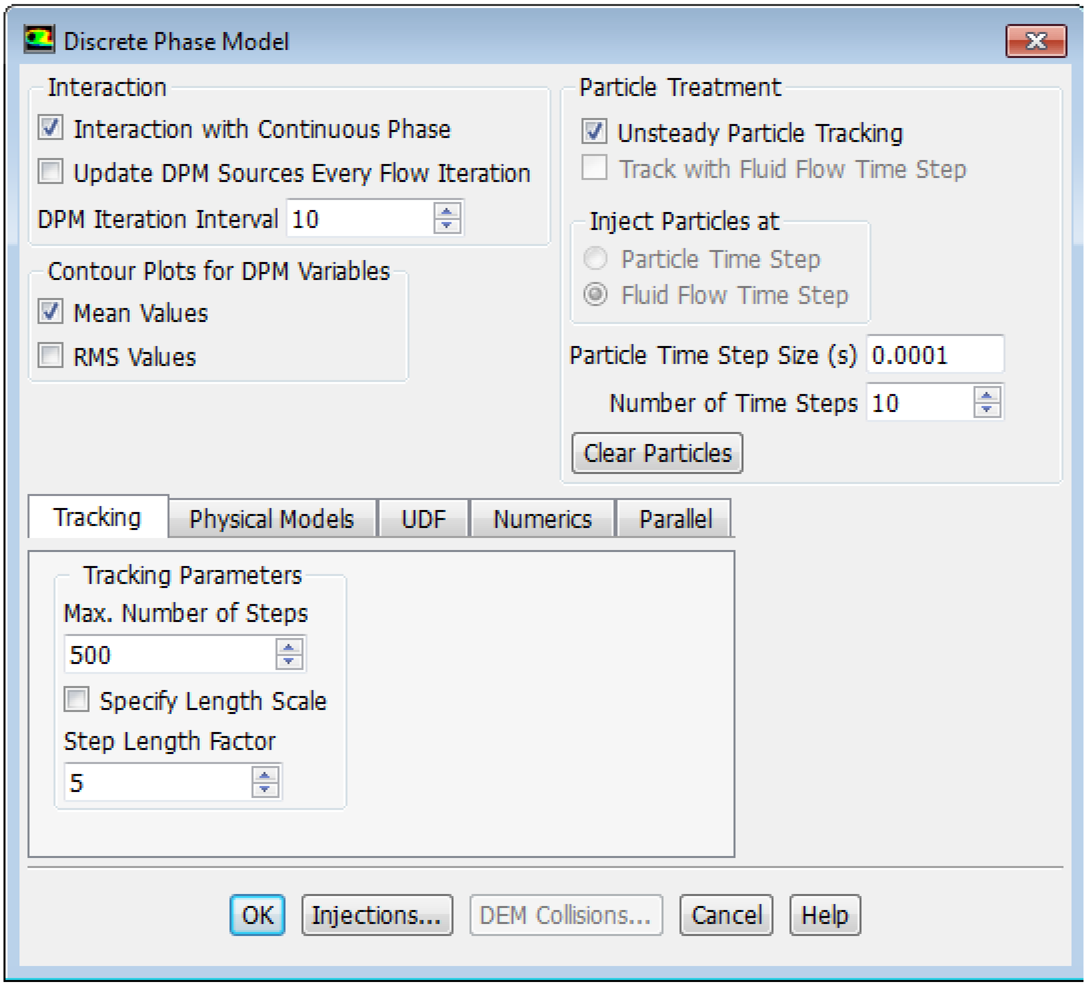
Figure 16.5: Pathlines of Air in the Swirling Annular Stream



16.4.8. Creating a Spray Injection

1. Define the discrete phase modeling parameters.

   Setting Up Physics → Models → Discrete Phase...



a. Select **Interaction with Continuous Phase** in the **Interaction** group box.

   This will include the effects of the discrete phase trajectories on the continuous phase.

b. Retain the value of `10` for **DPM Iteration Interval**.

c. Select **Mean Values** in the **Contour Plots for DPM Variables** group box.

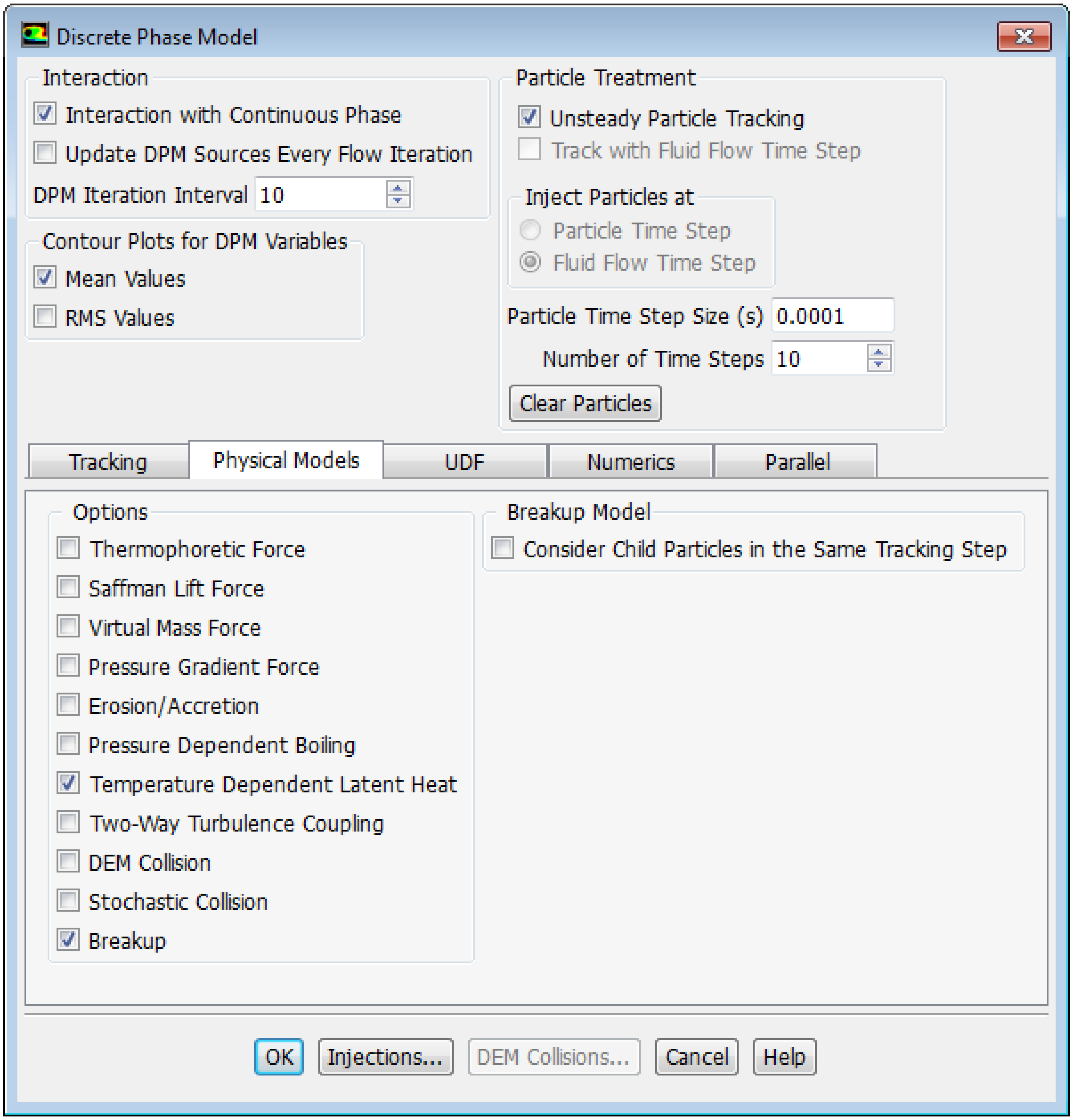
   This will make the cell-averaged variables available for postprocessing activities.

d. Select the **Unsteady Particle Tracking** option in the **Particle Treatment** group box.

e. Enter `0.0001` for **Particle Time Step Size**.

f. Enter `10` for **Number of Time Steps**.

g. Under the **Physical Models** tab, select the **Temperature Dependent Latent Heat** and **Breakup** (**Options** group).

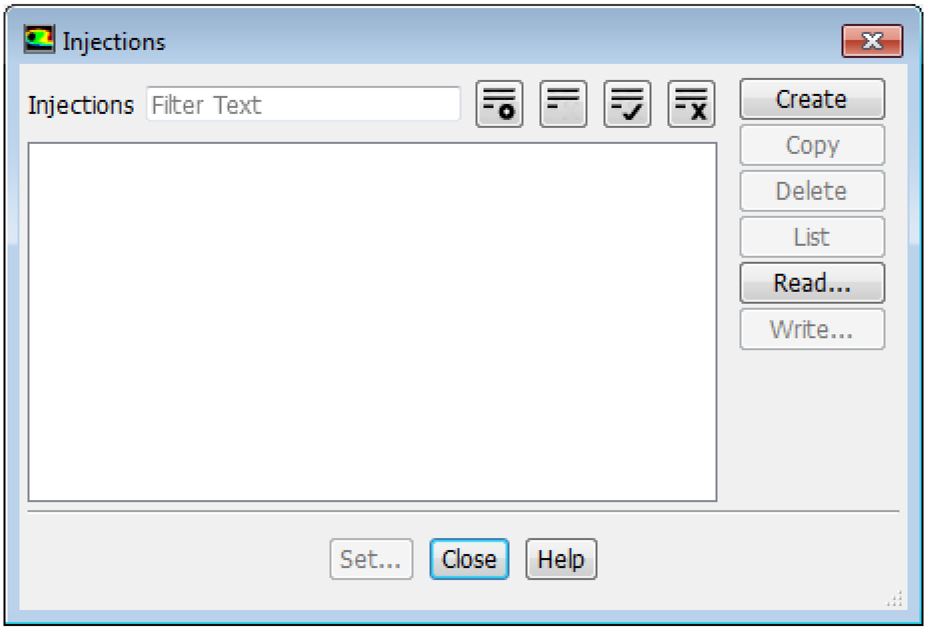


h. Under the **Numerics** tab, select **Linearize Source Terms** (**Source Terms** group).

   Enabling this option will allow you to run the simulation with more aggressive setting for the **Discrete Phase Sources** under-relaxation factor to speed up the solution convergence.

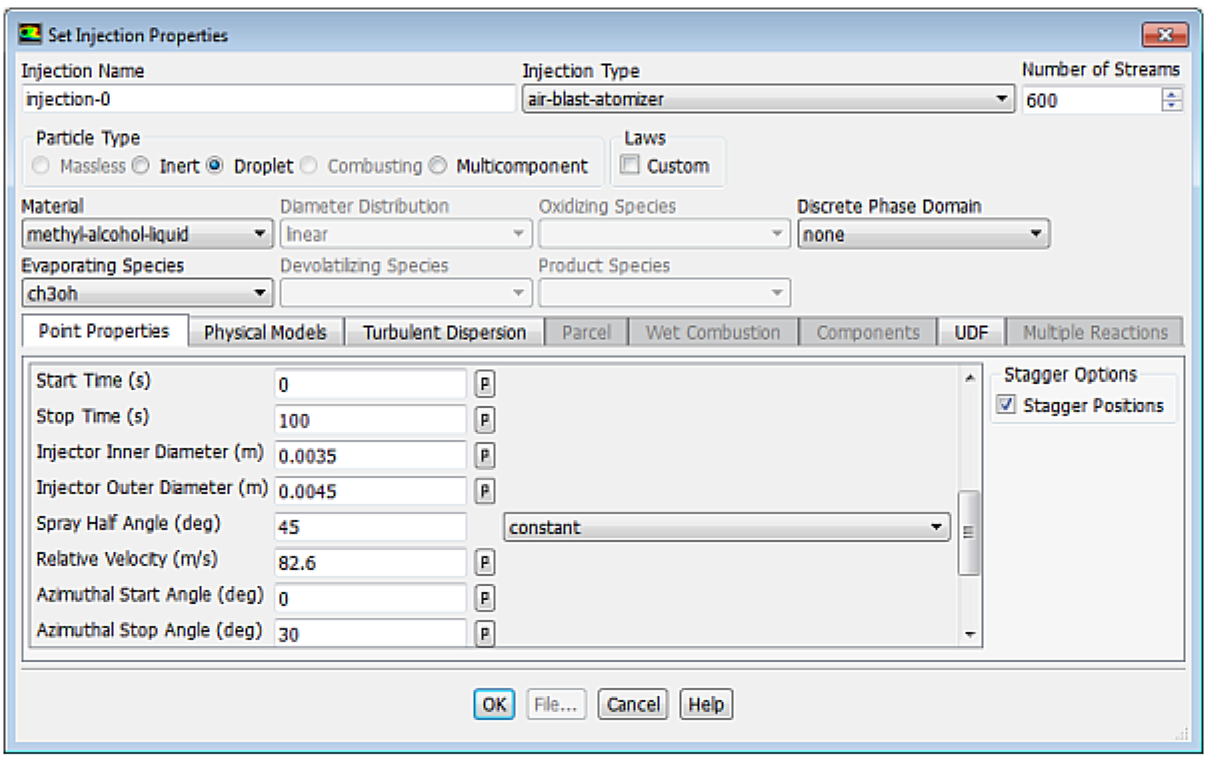
i. Click **Injections...** to open the **Injections** dialog box.

   In this step, you will define the characteristics of the atomizer.



An **Information** dialog box appears indicating that the **Max. Number of Steps** has been changed from 50000 to 500. Click **OK** in the **Information** dialog box to continue.

j. Click the **Create** button to create the spray injection.



k. In the **Set Injection Properties** dialog box, select **air-blast-atomizer** from the **Injection Type** drop-down list.

l. Enter `600` for **Number of Streams**.

This option controls the number of droplet parcels that are introduced into the domain at every time step.

m. Select **Droplet** in the **Particle Type** group box.

n. Select **methyl-alcohol-liquid** from the **Material** drop-down list.

o. In the **Point Properties** tab, specify point properties for particle injections.

   i. Retain the default values of `0` and `0` for **X-Position** and **Y-Position**.

   ii. Enter `0.0015` for **Z-Position**.

   iii. Retain the default values of `0`, `0`, and `1` for **X-Axis**, **Y-Axis**, and **Z-Axis**, respectively.

   iv. Enter `263` K for **Temperature**.

   Scroll down the list to see the remaining point properties.

   v. Enter `8.5e-5` kg/s for **Flow Rate**.

   This is the methanol flow rate for a 30-degree section of the atomizer. The actual atomizer flow rate is 12 times this value.

   vi. Retain the default **Start Time** of `0` s and enter `100` s for the **Stop Time**.

   For this problem, the injection should begin at `t=0` *and not stop until long after the time period of interest. A large value for the stop time (for example, 100 s) will ensure that the injection will essentially never stop.*

   vii. Enter `0.0035` m for the **Injector Inner Diameter** and `0.0045` m for the **Injector Outer Diameter**.

   viii. Enter `45` degrees for **Spray Half Angle**.

   The spray angle is the angle between the liquid sheet trajectory and the injector centerline.

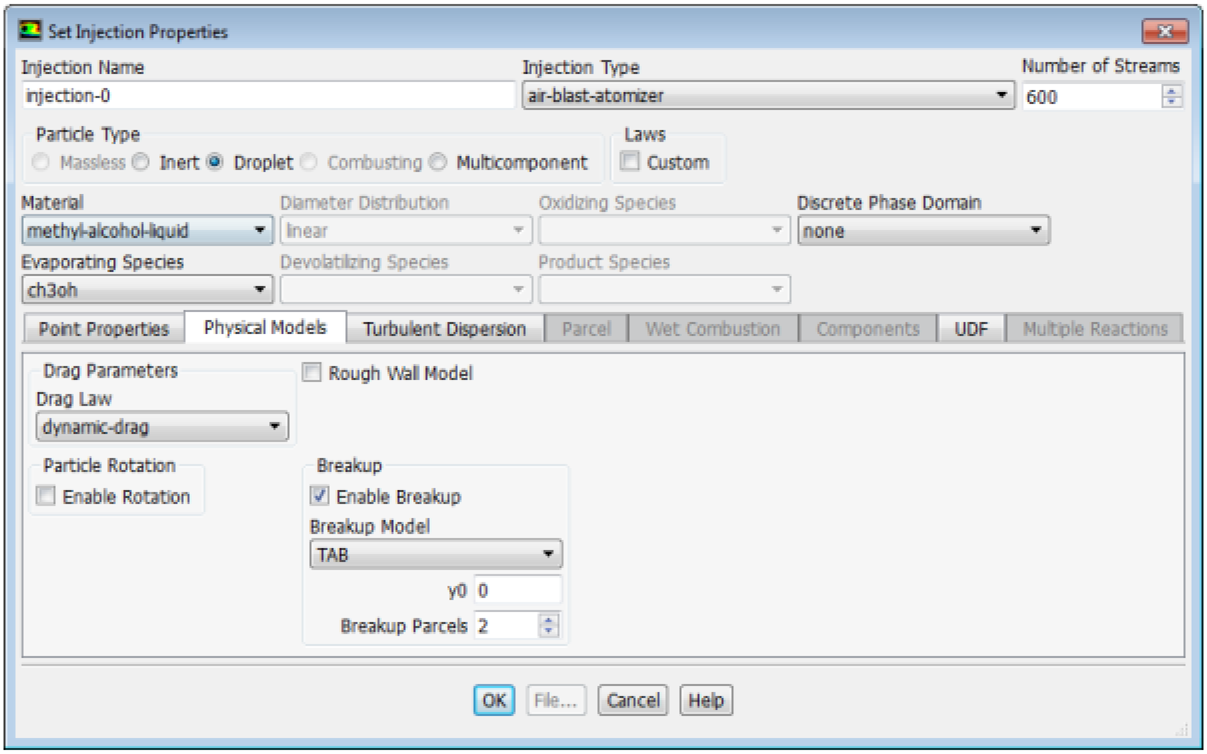
   ix. Enter `82.6` m/s for the **Relative Velocity**.

   The relative velocity is the expected relative velocity between the atomizing air and the liquid sheet.

   x. Retain the default **Azimuthal Start Angle** of `0` degrees and enter `30` degrees for the **Azimuthal Stop Angle**.

   This will restrict the injection to the 30-degree section of the atomizer that is being modeled.

p. In the **Physical Models** tab, specify the breakup model and drag parameters.



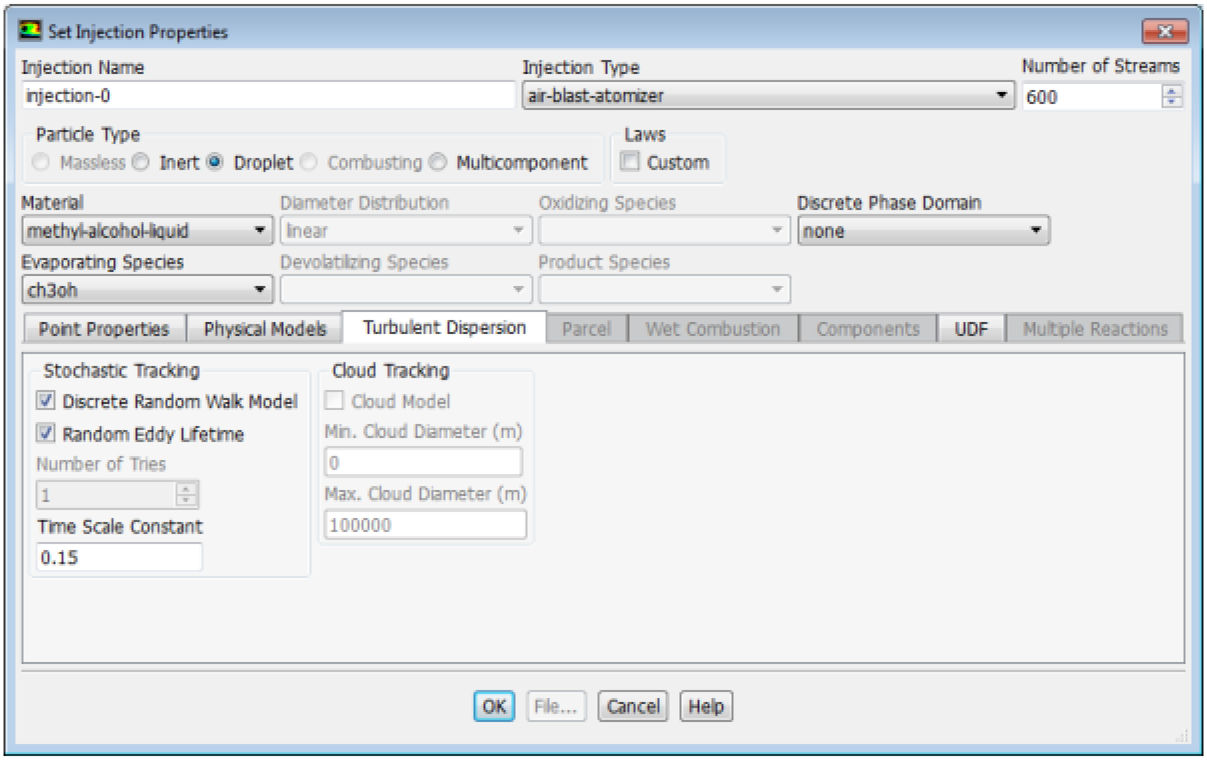
i. In the **Breakup** group, ensure that **Enable Breakup** is selected and **TAB** is selected from the **Breakup Model** drop-down list.

ii. Retain the default values of `0` for **y0** and 2 for **Breakup Parcels**.

iii. In the **Drag Parameters** group box, select **dynamic-drag** from the **Drag Law** drop-down list.

*The **dynamic-drag** law is available only when the **Breakup** model is used.*

q. In the **Turbulent Dispersion** tab, define the turbulent dispersion.



i. Enable **Discrete Random Walk Model** and **Random Eddy Lifetime** in the **Stochastic Tracking** group box.

These models will account for the turbulent dispersion of the droplets.

ii. Click **OK** to close the **Set Injection Properties** dialog box.

Note

To modify the existing injection, select its name in the **Injections** list and click **Set...**, or simply double-click the injection of interest.

---

r. Close the **Injections** dialog box.

Note

In the case that the spray injection would be striking a wall, you should specify the wall boundary conditions for the droplets. Though this tutorial does have wall zones, they are a part of the atomizer apparatus. You need not change the wall boundary conditions any further because these walls are not in the path of the spray droplets.

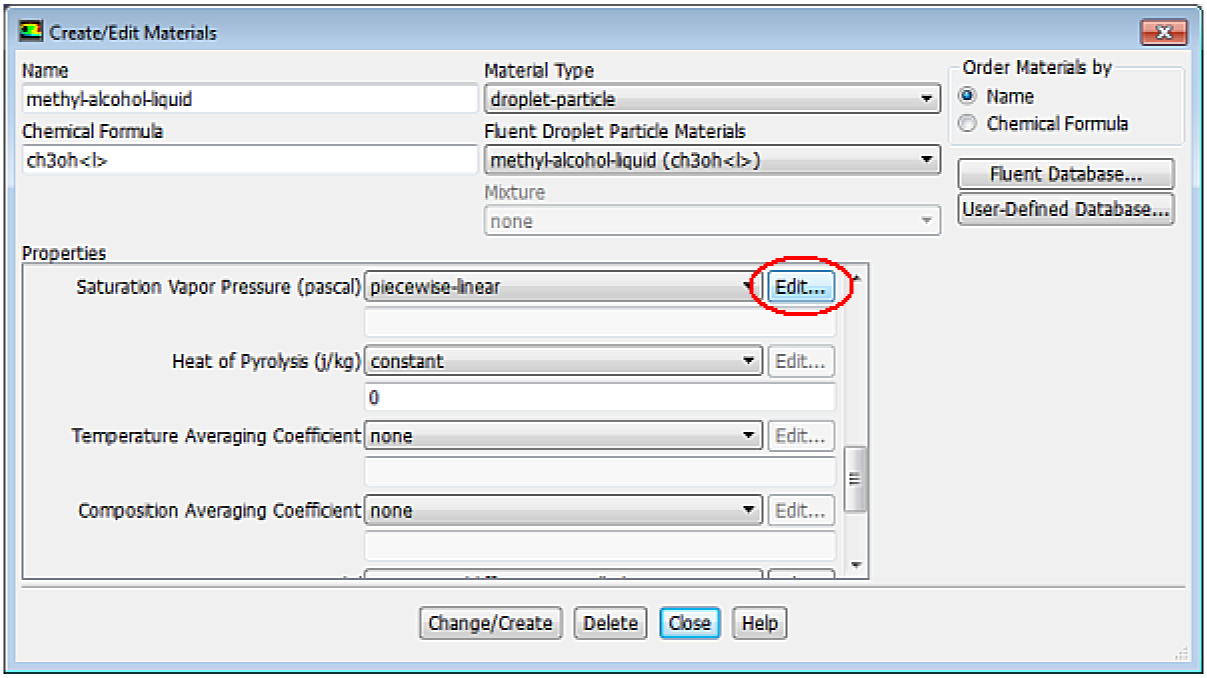
---

s. Click **OK** to close the **Discrete Phase Model** dialog box.

2. Specify the droplet material properties.

Setup → **Materials** → **methyl-alcohol-liquid** → **Create/Edit...**

*When secondary atomization models (such as **Breakup**) are used, several droplet properties need to be specified.*



a. Ensure **droplet-particle** is selected in the **Material Type** drop-down list.

b. Enter `0.0056` kg/m-s for **Viscosity** in the **Properties** group box.

c. Ensure that **piecewise-linear** is selected from the **Saturation Vapor Pressure** drop-down list.

   *Scroll down to find the **Saturation Vapor Pressure** drop-down list.*

d. Click the **Edit...** button next to **Saturation Vapor Pressure** to open the **Piecewise-Linear Profile** dialog box.

e. Retain the default values and click **OK** to close the **Piecewise-Linear Profile** dialog box.

f. Select **convection/diffusion-controlled** from the **Vaporization Model** drop-down list.

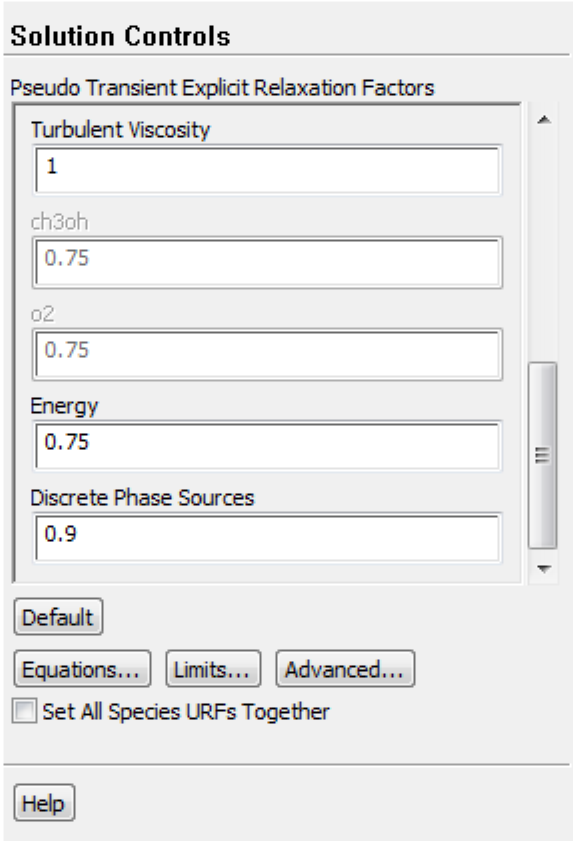
g. Click **OK** to close the **Convection/Diffusion Model** dialog box.

h. Click **Change/Create** to accept the change in properties for the methanol droplet material and close the **Create/Edit Materials** dialog box.

16.4.9. Solution

1. Increase the under-relaxation factor for **Discrete Phase Sources**.

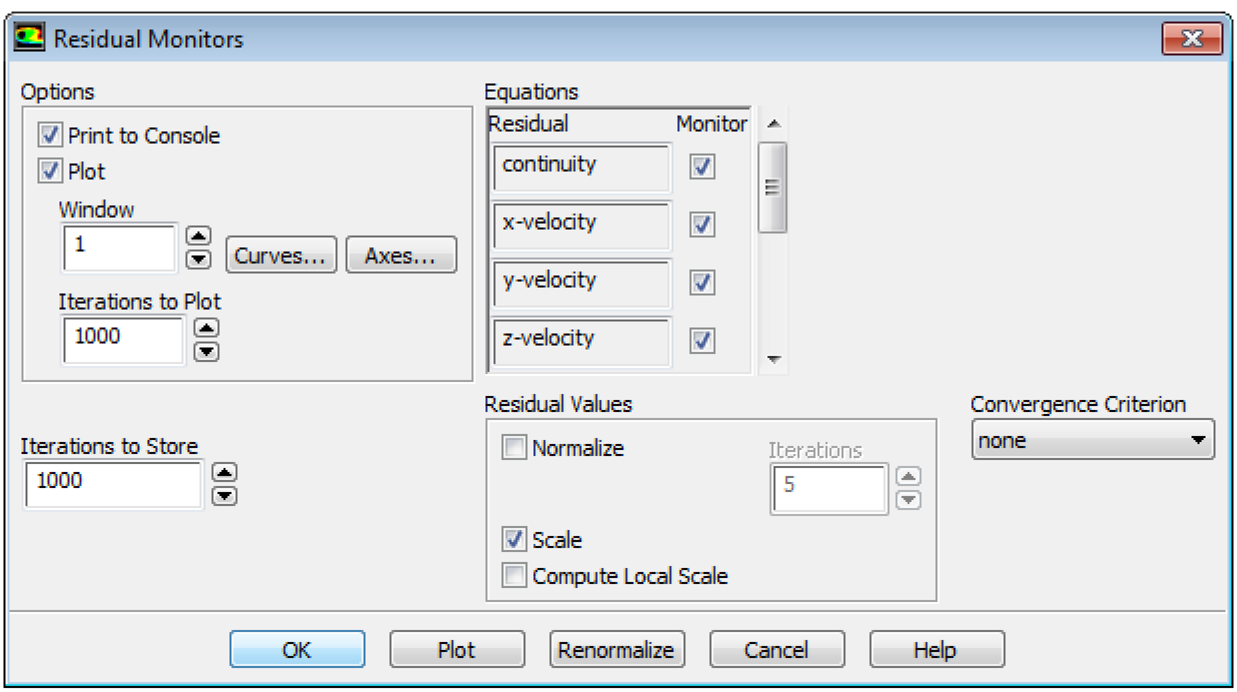
   Solving → **Controls** → **Controls...**



In the **Pseudo Transient Explicit Relaxation Factors** group box, change the under-relaxation factor for **Discrete Phase Sources** to `0.9`.

2. Remove the convergence criteria.

Solving → **Reports** → **Residuals...**

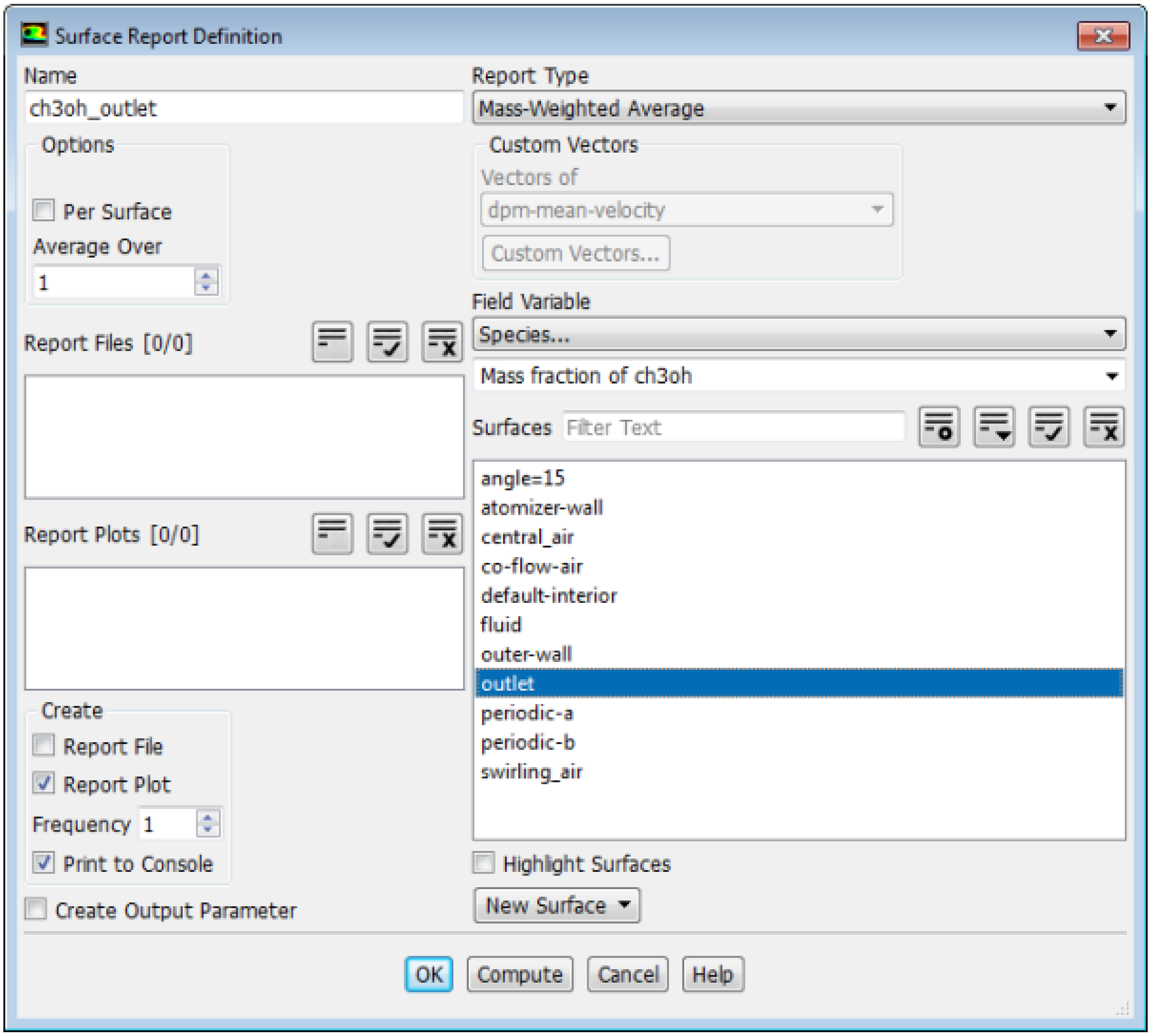


a. Select **none** from the **Convergence Criterion** drop-down list.

b. Click **OK** to close the **Residual Monitors** dialog box.

3. Enable the plotting of mass fraction of **ch3oh**.

Solving → **Reports** → **Definitions** → **New** → **Surface Report** → **Mass-Weighted Average...**



a. Enter **ch3oh_outlet** for **Name** of the surface report definition.

b. In the **Create** group box, enable **Report Plot** and **Print to Console**.

c. Select **Species...** and **Mass fraction of ch3oh** from the **Field Variable** drop-down lists.

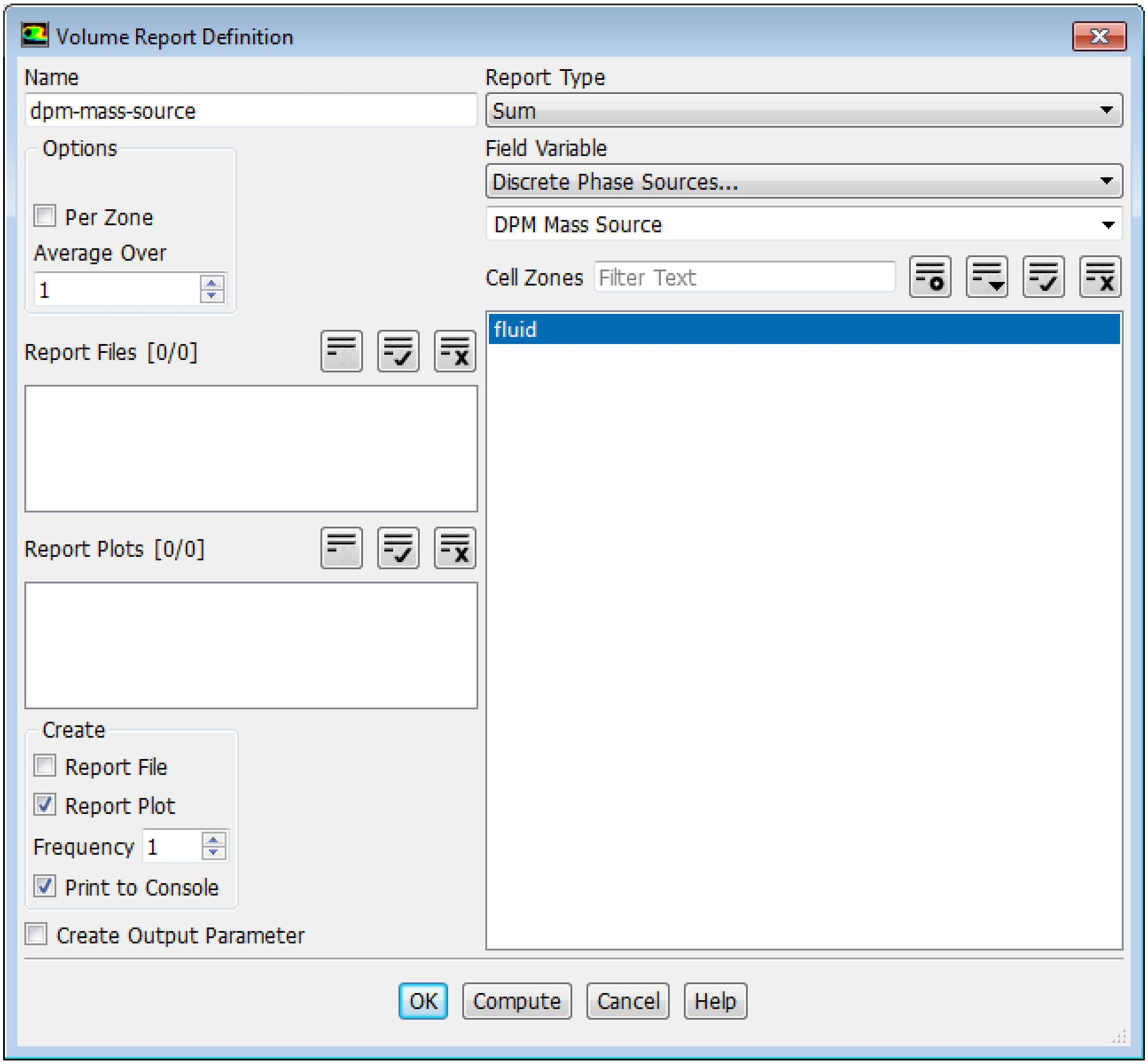
d. Select **outlet** from the **Surfaces** selection list.

e. Click **OK** to save the surface report definition settings and close the **Surface Report Definition** dialog box.

Fluent automatically generates the **ch3oh_outlet-rplot** report plot under the **Solution/Monitors/Report Plots** tree branch.

4. Enable the plotting of the sum of the **DPM Mass Source**.

   Solving → **Reports** → **Definitions** → **New** → **Volume Report** → **Sum...**



   a. Enter **dpm-mass-source** for **Name**.

   b. In the **Create** group box, enable **Report Plot** and **Print to Console**.

   c. Select **Discrete Phase Sources...** and **DPM Mass Source** from the **Field Variable** drop-down lists.

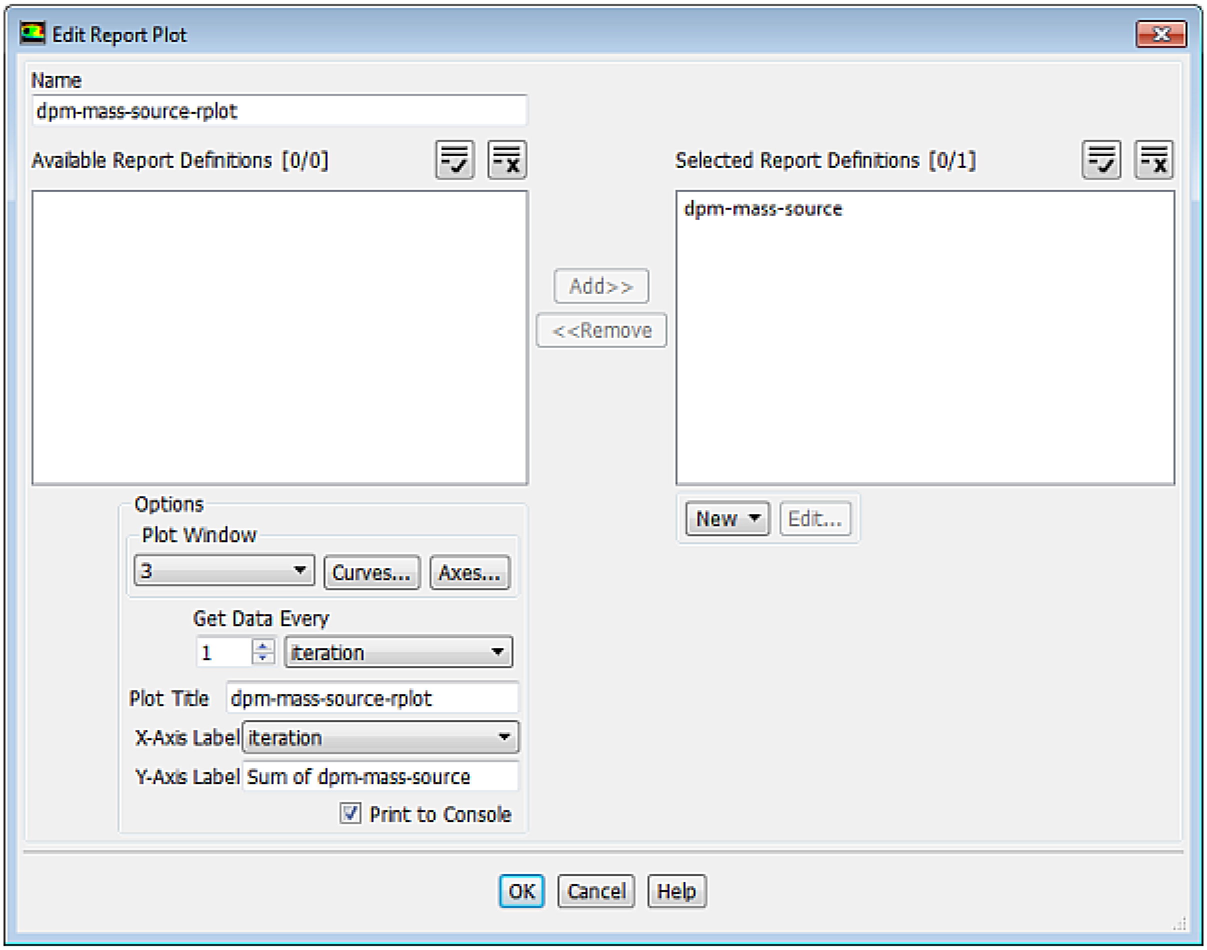
   d. Select **fluid** from the **Cell Zones** selection list.

   e. Click **OK** to save the volume report definition settings and close the **Volume Report Definition** dialog box.

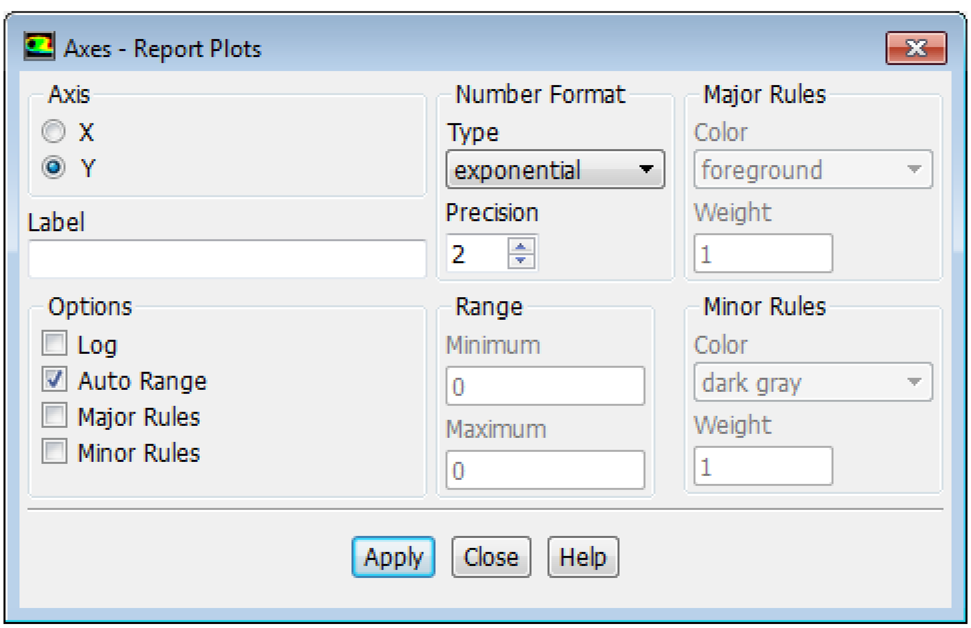
      Fluent automatically generates the **dpm-mass-source-rplot** report plot under **Solution/Monitors/Report Plots** tree branch.

f. Modify the attributes of the **dpm-mass-source-rplot** report plot axes.

Solution → **Monitors** → **Report Plots** → **dpm-mass-source-rplot** **Edit...**



i. In the **Plot Window** group box, click the **Axes...** button to open the **Axes** dialog box.



ii. Select **Y** in the **Axis** list.

iii. Select **exponential** from the **Type** drop-down list.

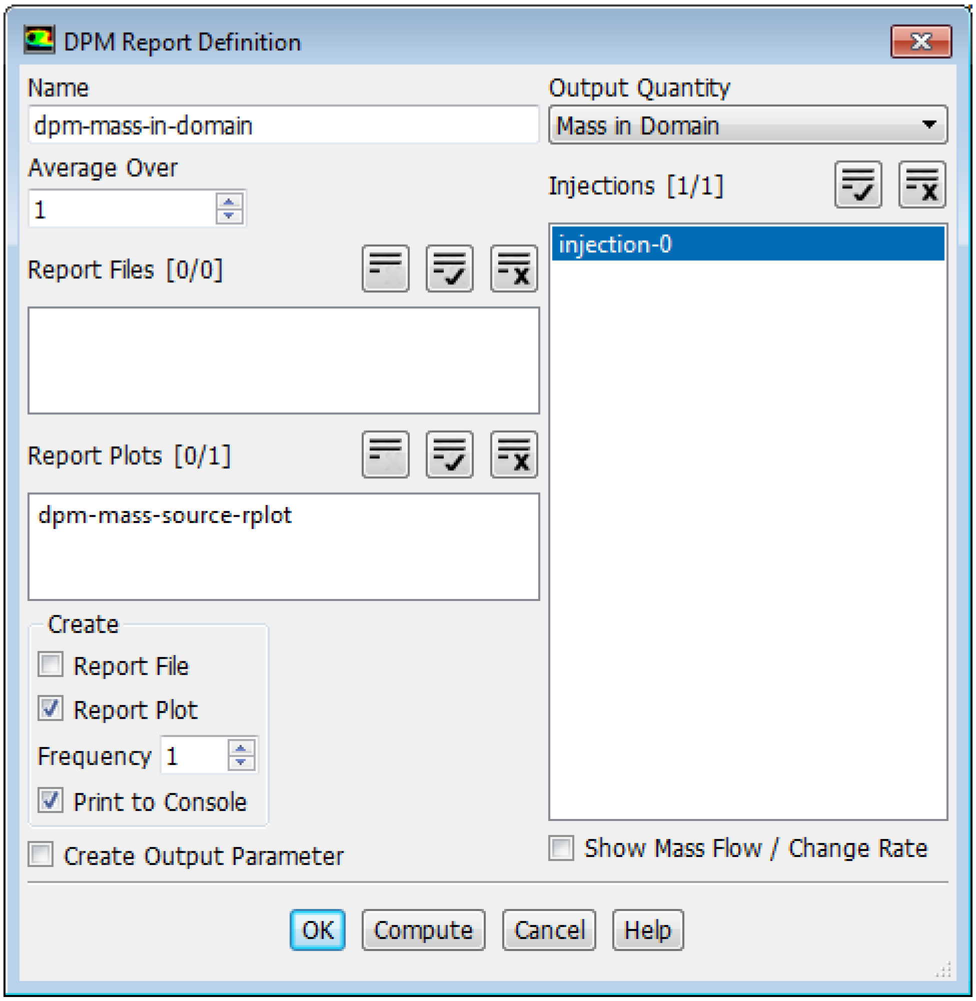
iv. Set **Precision** to 2.

v. Click **Apply** and close the **Axes** dialog box.

vi. Click OK to close the **Edit Report Plot** dialog box.

5. Create a DPM report definition for tracking the total mass present in the domain.

Solving → **Reports** → **Definitions** → **New** → **DPM Report** → **Mass in Domain...**



a. Enter **dpm-mass-in-domain** for **Name**.

b. In the **Create** group box, enable **Report Plot** and **Print to Console**.

c. Select **injection-0** from the **Injections** selection list.

d. Clear **Show Mass Flow / Change Rate**.

e. Click **OK** to save the volume report definition settings and close the **DPM Report Definition** dialog box.

Fluent automatically generates the **dpm-mass-in-domain-rplot** report plot under **Solution/Monitors/Report Plots** tree branch.

f. Modify the attributes of the **dpm-mass-in-domain-rplot** report plot axes (in a manner similar to that for the **dpm-mass-source-rplot** plot).

Solution → **Monitors** → **Report Plots** → **dpm-mass-in-domain-rplot** **Edit...**

i. In the **Plot Window** group box, click the **Axes...** button to open the **Axes** dialog box.

ii. Select **Y** in the **Axis** list.

iii. Select **exponential** from the **Type** drop-down list.

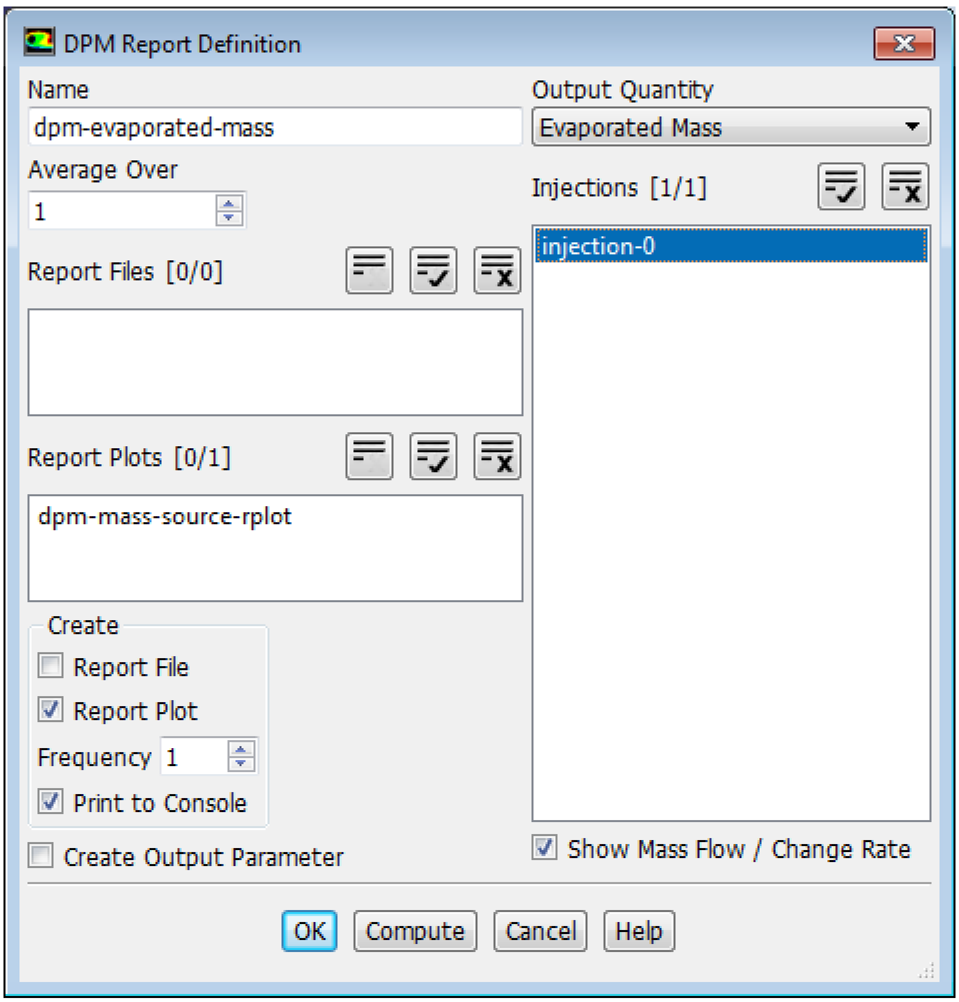
iv. Set **Precision** to 2.

v. Click **Apply** and close the **Axes** dialog box.

vi. Click OK to close the **Edit Report Plot** dialog box.

6. Create a DPM report definition for tracking the mass of the evaporated particles.

Solving → **Reports** → **Definitions** → **New** → **DPM Report** → **Evaporated Mass...**



a. Enter **dpm-evaporated-mass** for **Name**.

b. In the **Create** group box, enable **Report Plot** and **Print to Console**.

c. Select **injection-0** from the **Injections** selection list.

d. Ensure that the **Show Mass Flow / Change Rate** option is selected.

e. Click **OK** to save the volume report definition settings and close the **DPM Report Definition** dialog box.

   Fluent automatically generates the **dpm-evaporated-mass-rplot** report plot under **Solution/Monitors/Report Plots** tree branch.

f. Modify the attributes of the **dpm-evaporated-mass-rplot** report plot axes in a manner similar to that for the **dpm-mass-source-rplot** plot.

7. Request 300 more iterations (Figure 16.6: Convergence History of Mass Fraction of ch3oh on Fluid (p. 699), Figure 16.7: Convergence History of DPM Mass Source on Fluid (p. 700), Figure 16.8: Convergence History of Total Mass in Domain (p. 700), and Figure 16.9: Convergence History of Evaporated Particle Mass (p. 701)).

Solving → **Run Calculation**

It can be concluded that the solution is converged because the number of particle tracks are constant and the flow variable plots are flat.

Figure 16.6: Convergence History of Mass Fraction of ch3oh on Fluid

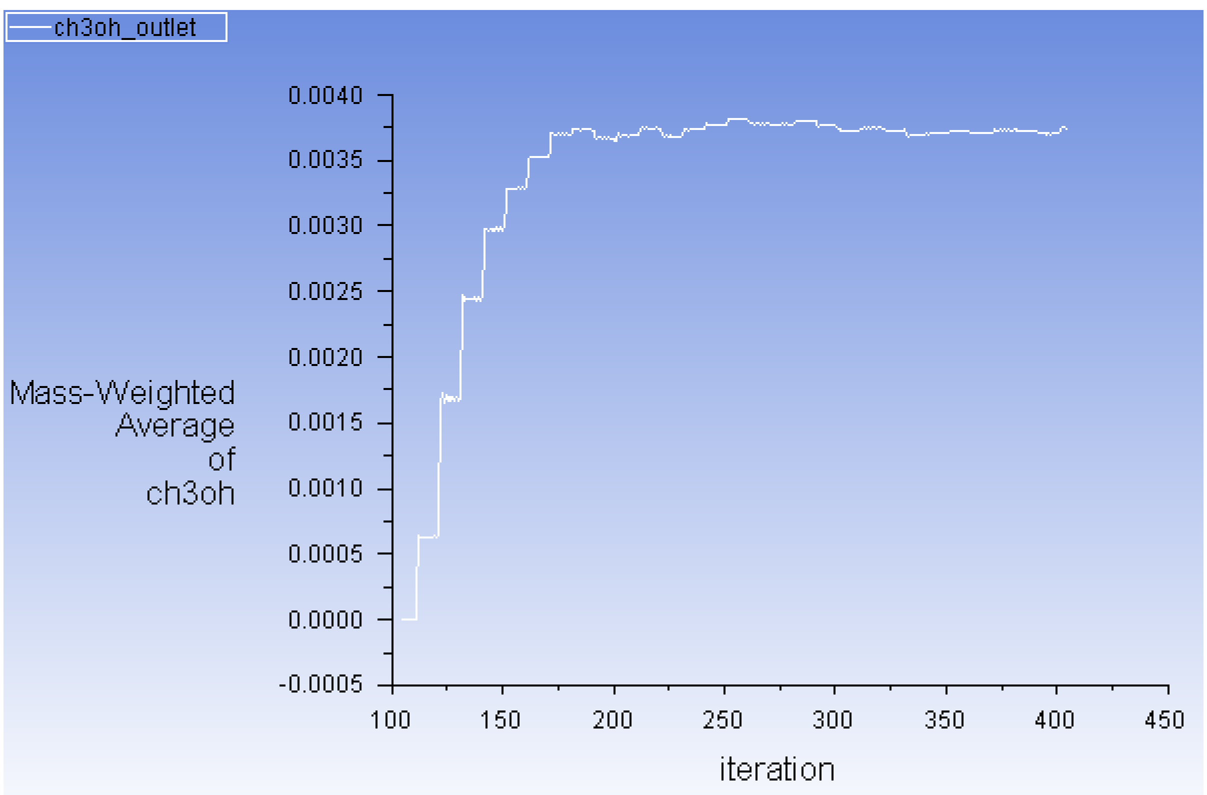


Figure 16.7: Convergence History of DPM Mass Source on Fluid

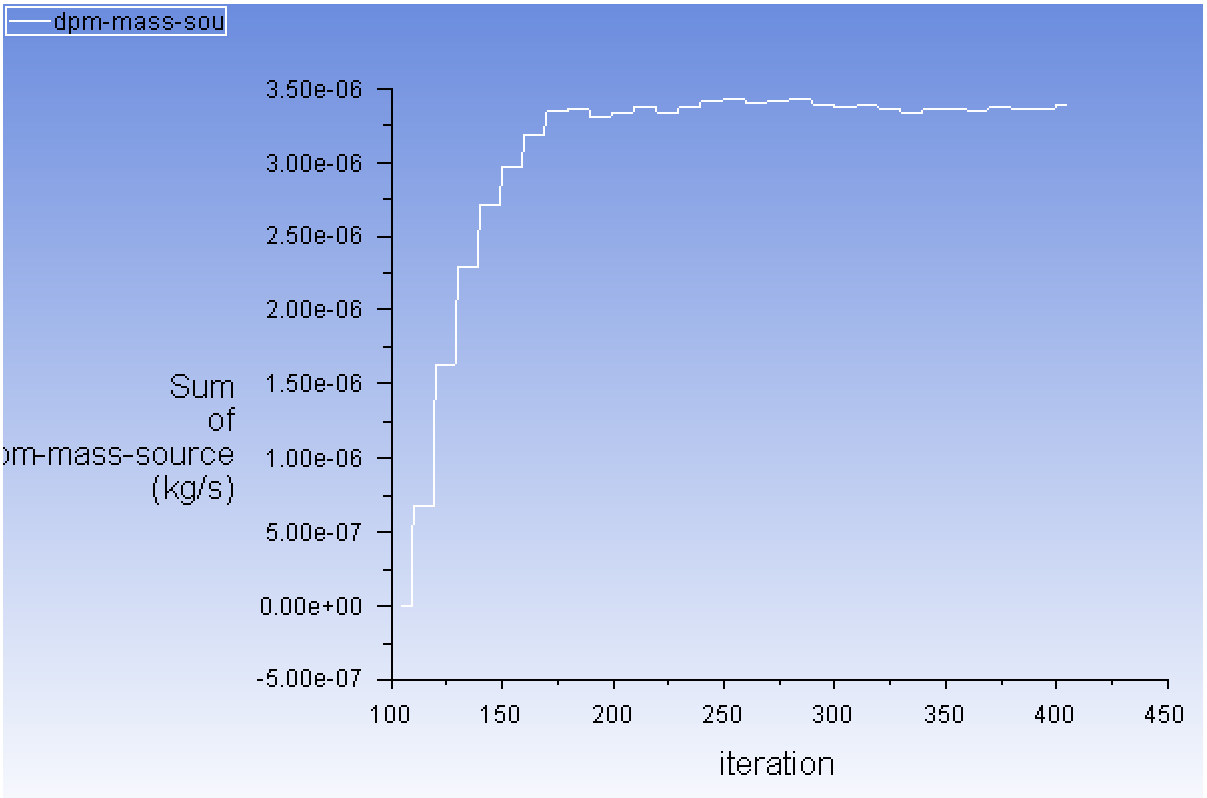


Figure 16.8: Convergence History of Total Mass in Domain

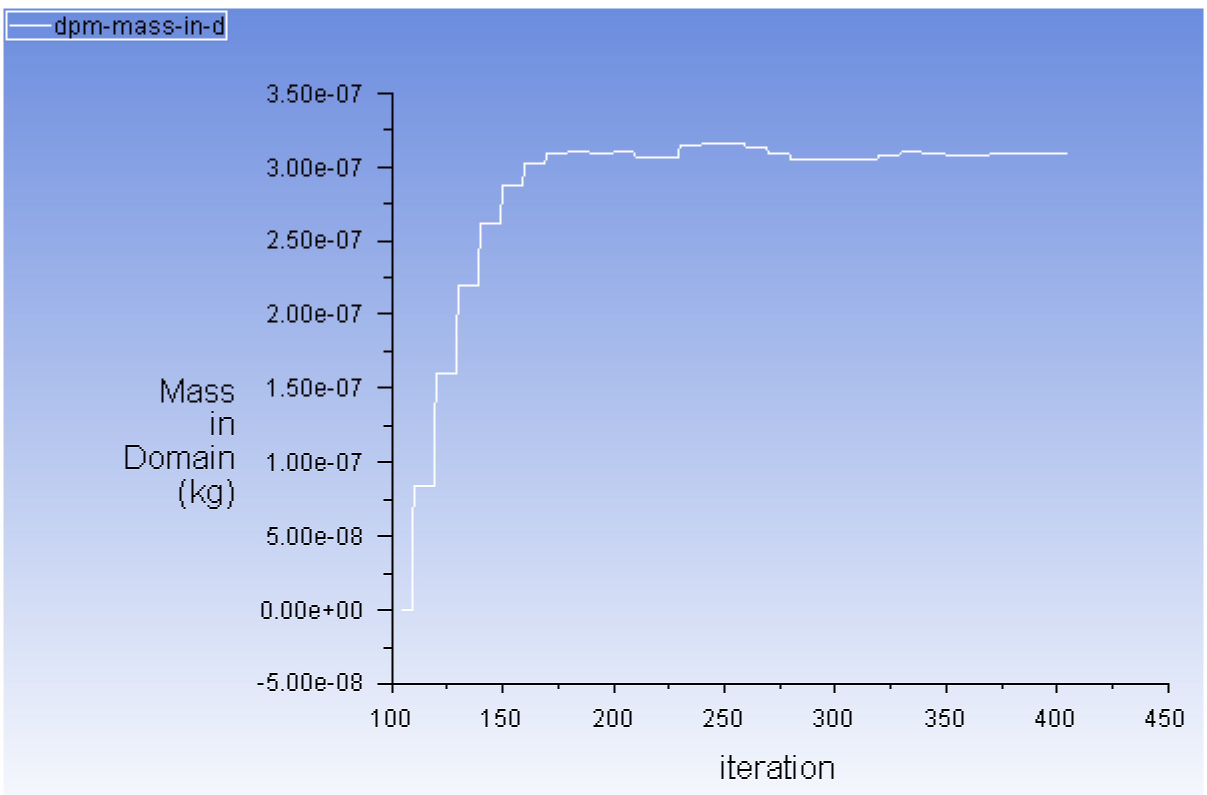
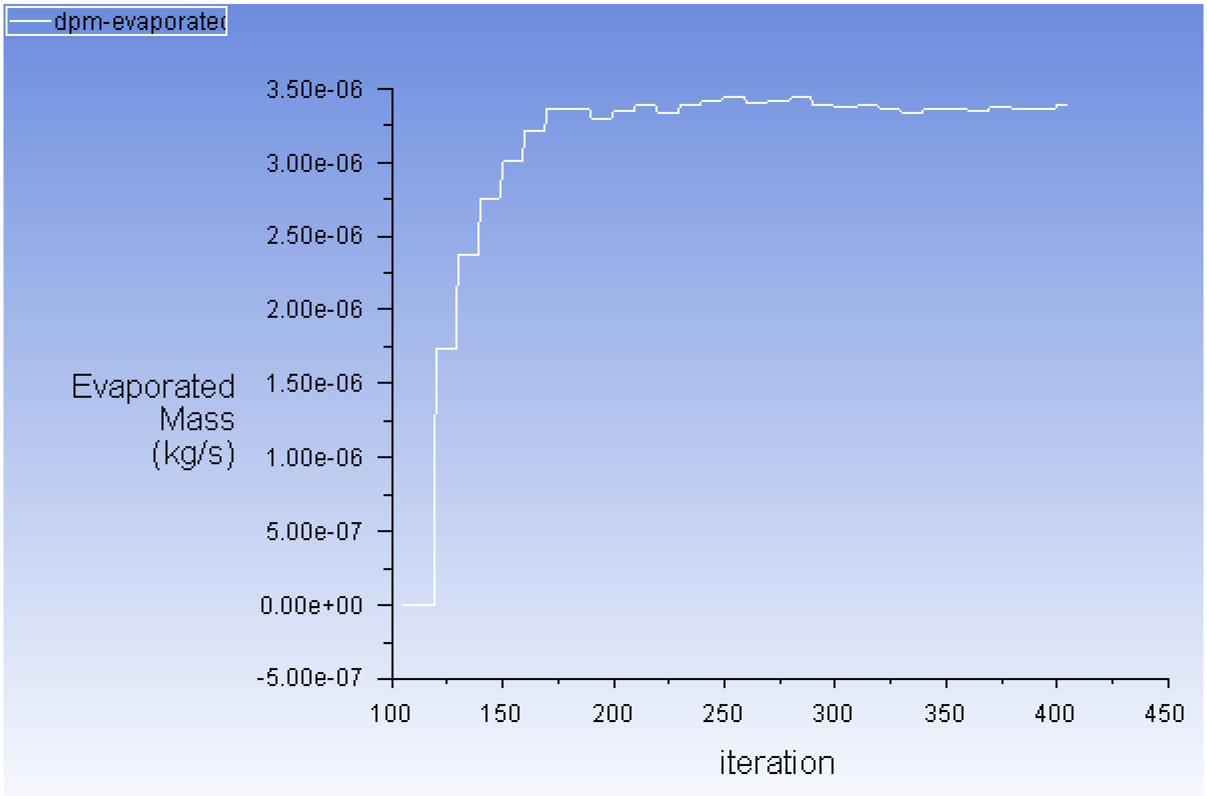


Figure 16.9: Convergence History of Evaporated Particle Mass



8. Save the case and data files (`spray2.cas.gz` and `spray2.dat.gz`).

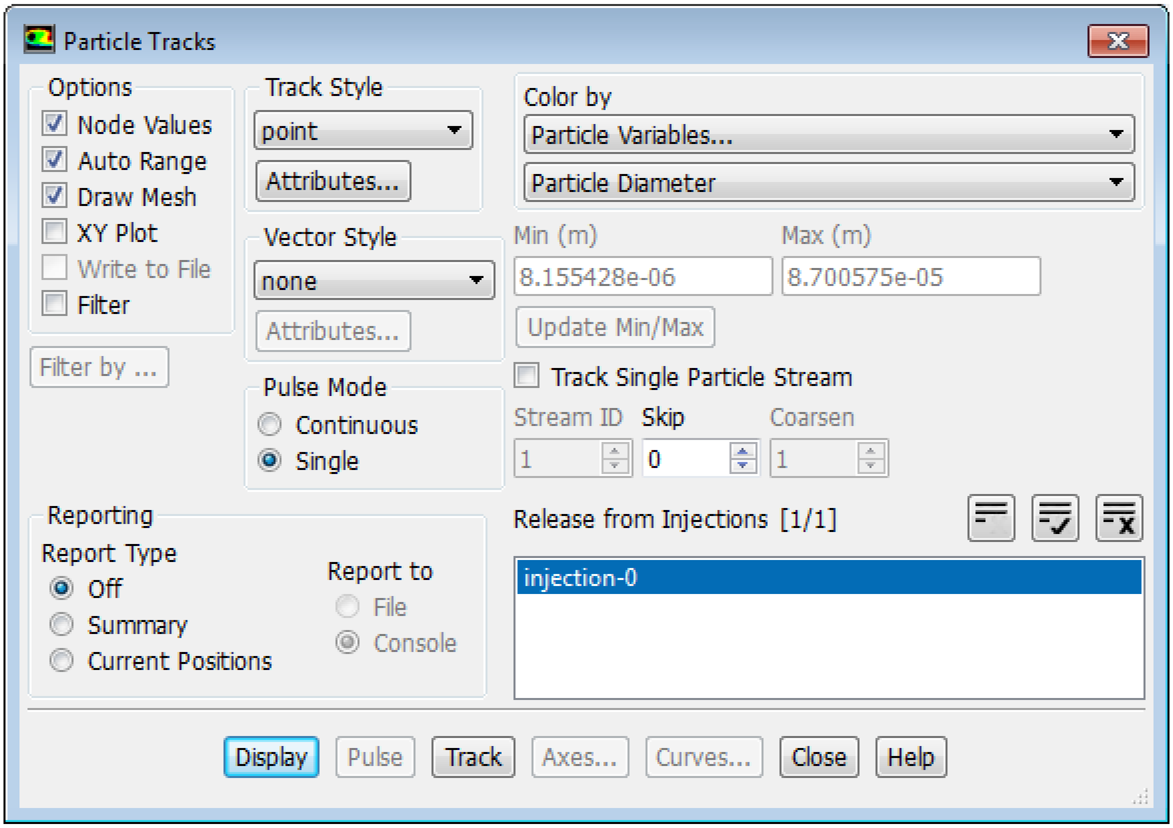
   File → Write → Case & Data...

16.4.10. Postprocessing

1. Display the trajectories of the droplets in the spray injection (Figure 16.10: Particle Tracks for the Spray Injection (p. 703)).

   This will allow you to review the location of the droplets.

   Postprocessing → Graphics → Particle Tracks → Edit...



a. Enable **Draw Mesh** in the **Options** group box.

   *The **Mesh Display** dialog box will open.*

   i. Retain the current display settings.

   ii. Close the **Mesh Display** dialog box.

b. Retain the default selection of **point** from the **Track Style** drop-down list.

c. Select **Particle Variables...** and **Particle Diameter** from the **Color by** drop-down lists.

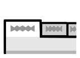
   This will display the location of the droplets colored by their diameters.

d. Select **injection-0** from the **Release from Injections** selection list.

e. Click **Display**. *As an optional exercise, you can increase the particle size by clicking the **Attributes...** button in the **Particle Tracks** dialog box and adjusting the **Marker Size** value in the **Track Style Attributes** dialog box.*

f. Close the **Particle Tracks** dialog box.

g. Restore the 30–degree section to obtain the view as shown in Figure 16.10: Particle Tracks for the Spray Injection (p. 703).

    **Viewing → Display → Views...**

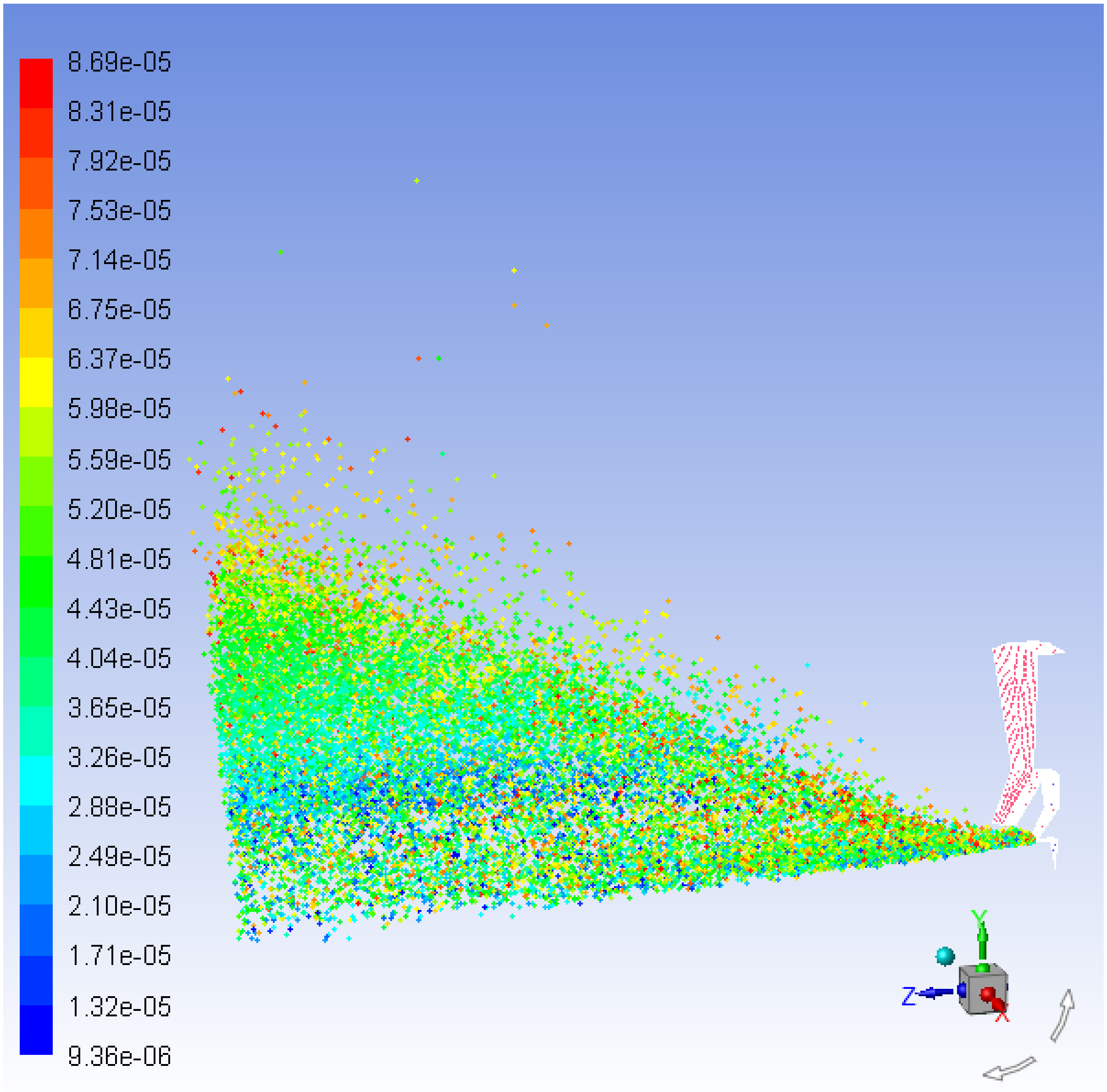
   i. Click the **Define...** button to open the **Graphics Periodicity** dialog box.

   ii. Click **Reset** and close the **Graphics Periodicity** dialog box.

iii. Close the **Views** dialog box.

h. Use the mouse to obtain the view shown in Figure 16.10: Particle Tracks for the Spray Injection (p. 703).

Figure 16.10: Particle Tracks for the Spray Injection



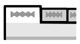
The air-blast atomizer model assumes that a cylindrical liquid sheet exits the atomizer, which then disintegrates into ligaments and droplets. Appropriately, the model determines that the droplets should be input into the domain in a ring. The radius of this disk is determined from the inner and outer radii of the injector.

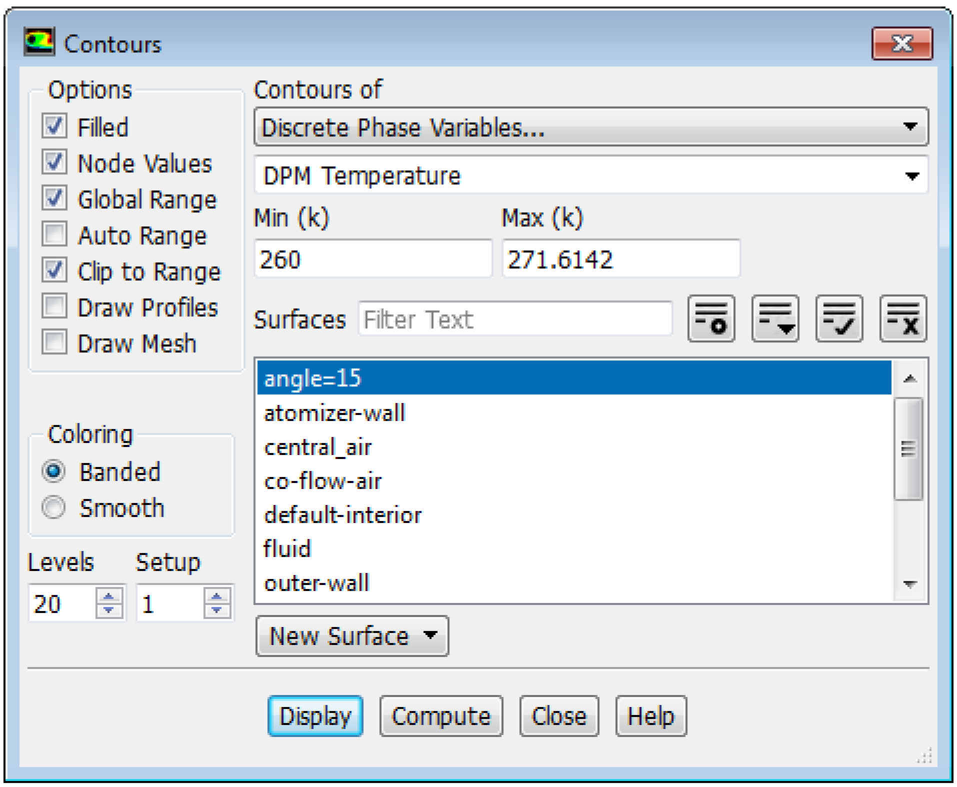
Note

The maximum diameter of the droplets is about 10^{-4} m or 0.1 mm. This is slightly smaller than the film height. The inner diameter and outer diameter of the injector are 3.5 mm and 4.5 mm, respectively. Hence the film height is 0.5 mm. The range in the droplet sizes is due to the fact that the air-blast atomizer automatically uses a distribution of droplet sizes.

Also note that the droplets are placed a slight distance away from the injector. Once the droplets are injected into the domain, their behavior will be determined by secondary models. For instance, they may collide/coalesce with other droplets depending on the secondary models employed. However, once a droplet has been introduced into the domain, the air-blast atomizer model no longer affects the droplet.

2. Display the mean particle temperature field (Figure 16.11: Contours of DPM Temperature (p. 705)).

 **Postprocessing** → **Graphics** → **Contours** → **Edit...**



a. Ensure that **Filled** is enabled in the **Options** group box

b. Disable **Draw Mesh**.

c. Select **Discrete Phase Variables...** and **DPM Temperature** from the **Contours of** drop-down lists.

d. Disable **Auto Range**.

*The **Clip to Range** option will automatically be enabled.*

e. Click **Compute** to update the **Min** and **Max** fields.

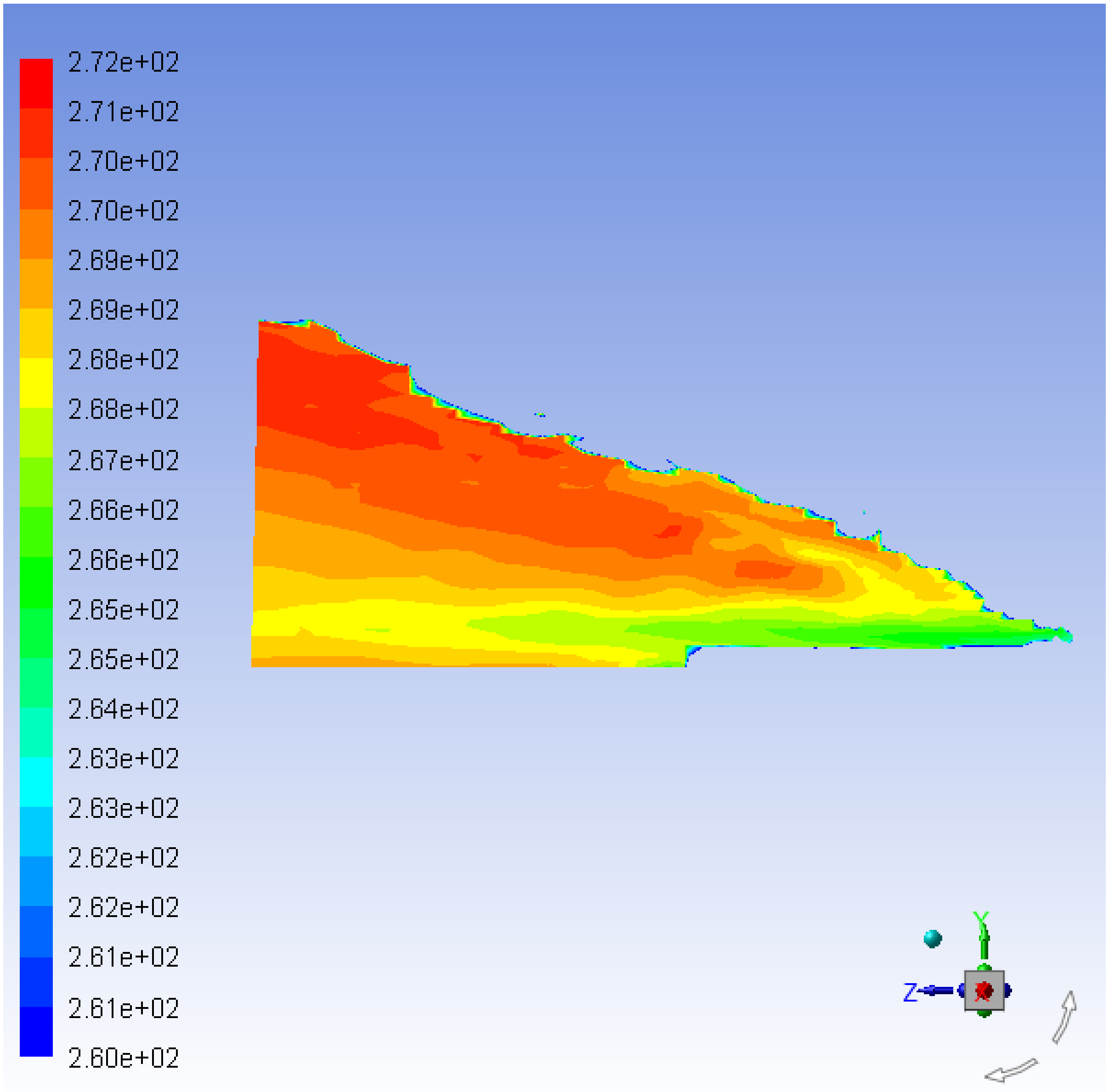
f. Enter `260` for **Min**.

g. Select **angle=15** from the **Surfaces** selection list.

h. Click **Display** and close the **Contours** dialog box.

i. Use the mouse to obtain the view shown in Figure 16.11: Contours of DPM Temperature (p. 705).

Figure 16.11: Contours of DPM Temperature



3. Display the mean Sauter diameter (Figure 16.12: Contours of DPM Sauter Diameter (p. 706)).

Postprocessing → **Graphics** → **Contours** → **Edit...**

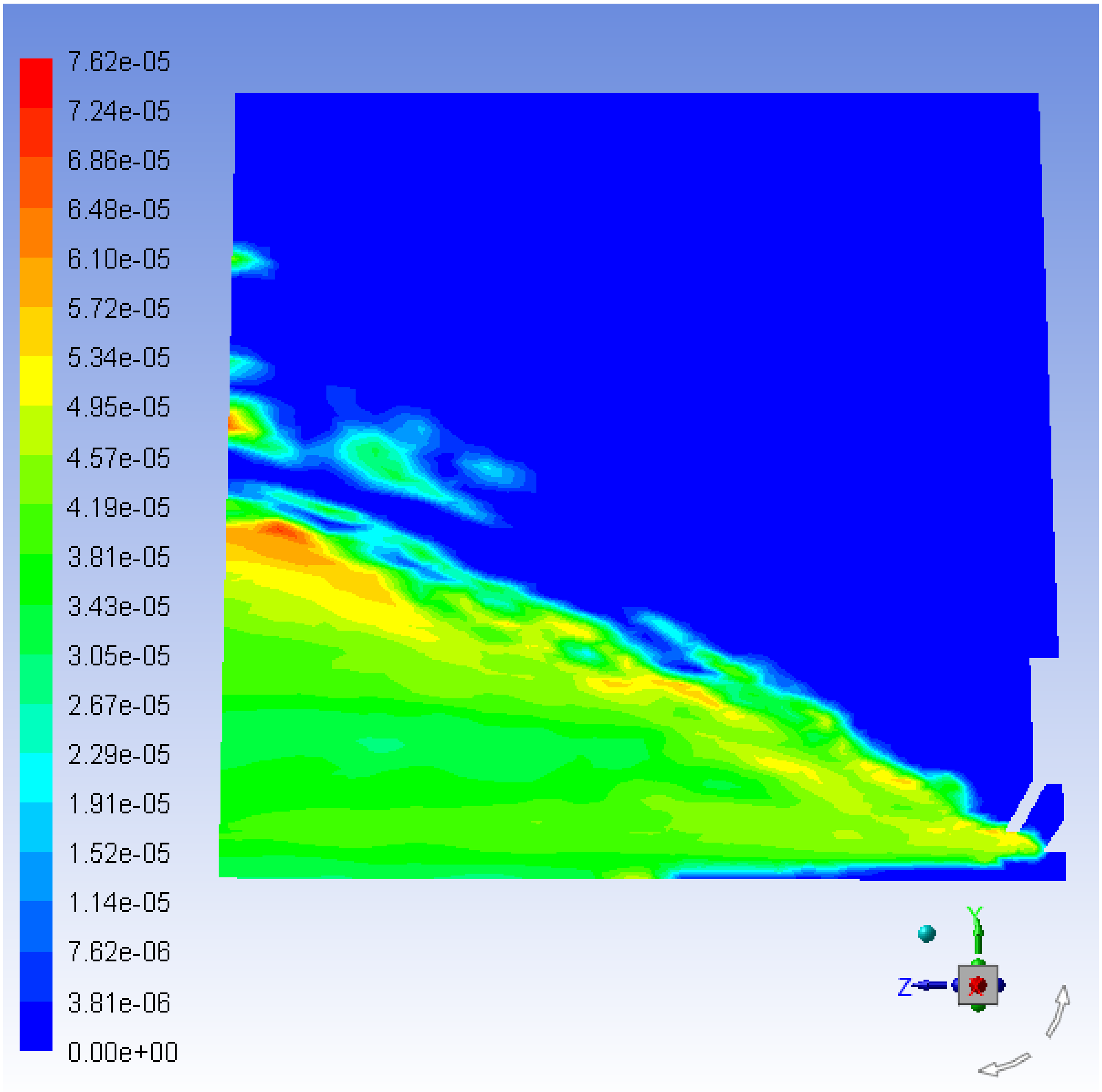
a. Enable **Autorange** in the **Options** group box.

b. Select **Discrete Phase Variables...** and **DPM D32** from the **Contours of** drop-down lists.

c. Select **angle=15** from the **Surfaces** selection list.

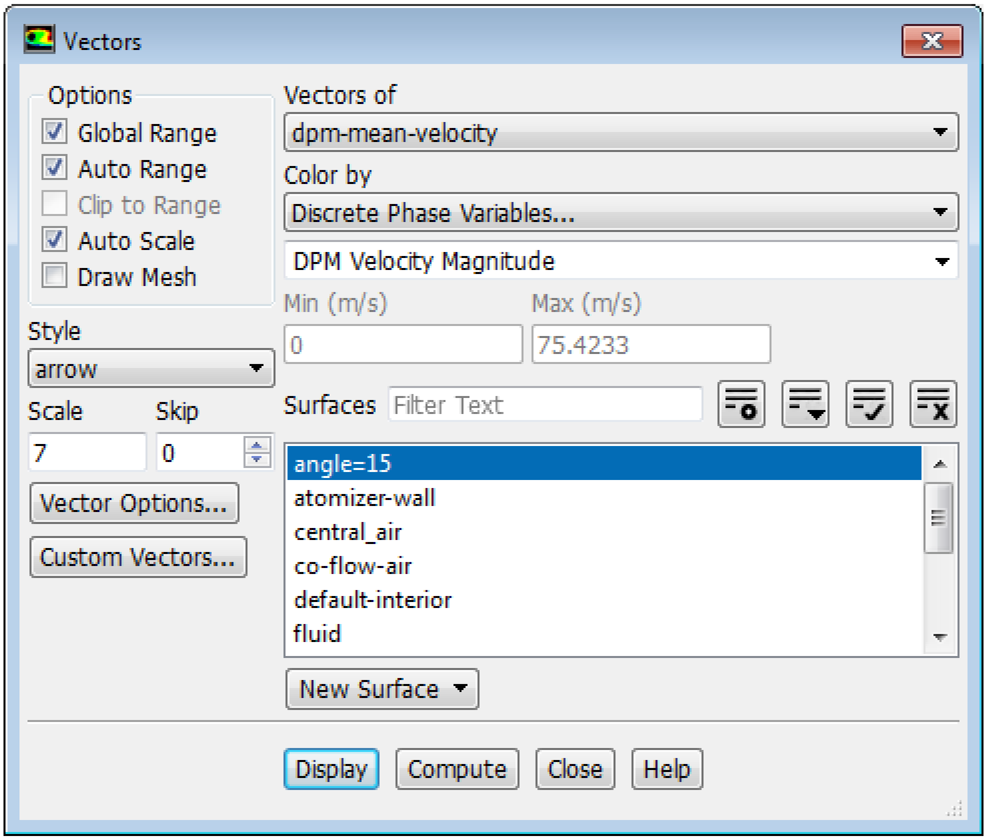
d. Click **Display** and close the **Contours** dialog box.

Figure 16.12: Contours of DPM Sauter Diameter



4. Display vectors of DPM mean velocity colored by DPM velocity magnitude (Figure 16.13: Vectors of DPM Mean Velocity Colored by DPM Velocity Magnitude (p. 708)).

   Postprocessing → **Graphics** → **Vectors** → **Edit...**



a. Select **dpm-mean-velocity** from the **Vectors of** drop-down lists.

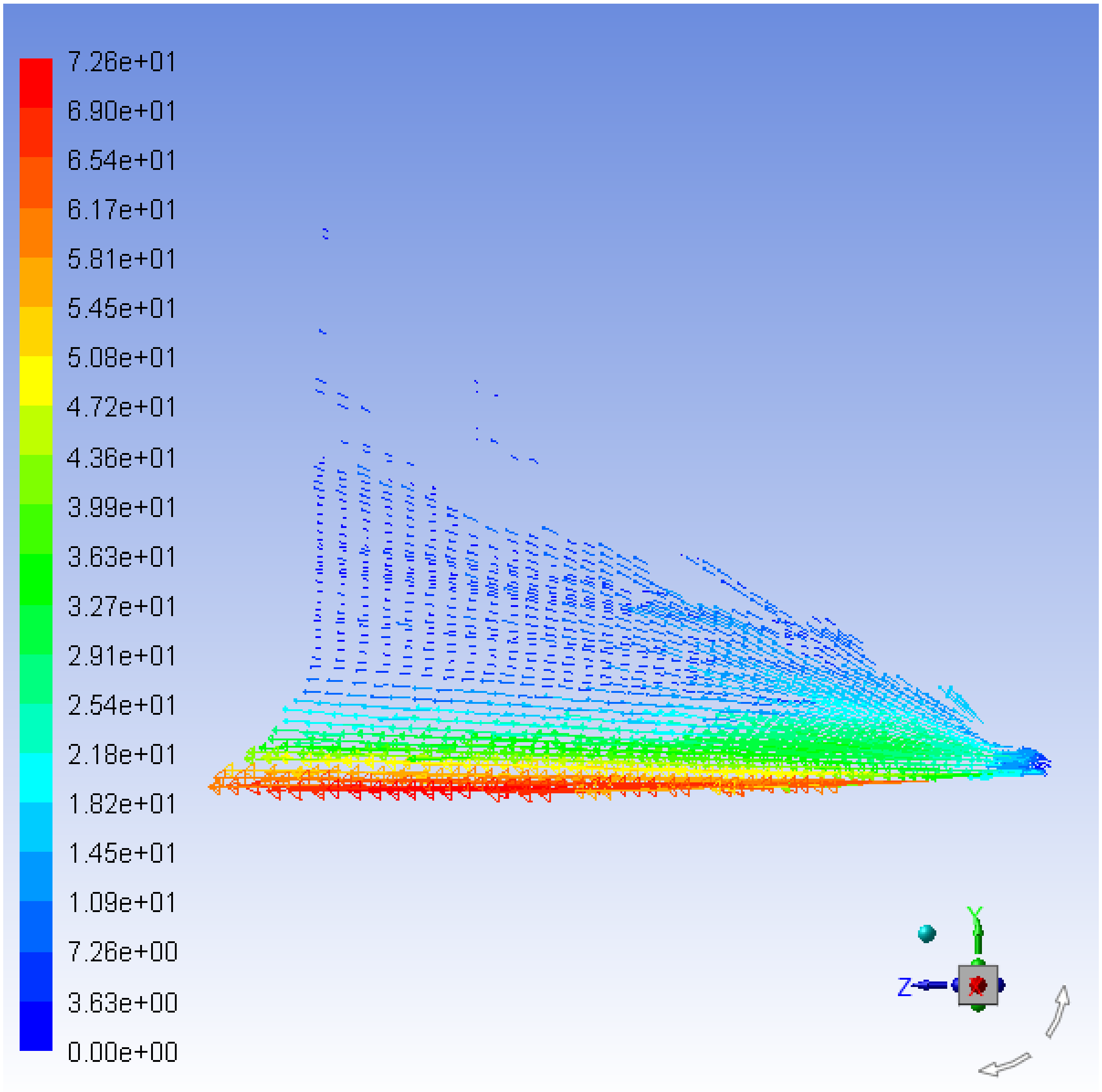
b. Select **Discrete Phase Variables...** and **DPM Velocity Magnitude** from the **Color by** drop-down lists.

c. Enter 7 for **Scale**.

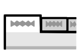
d. Select **angle=15** from the **Surfaces** selection list.

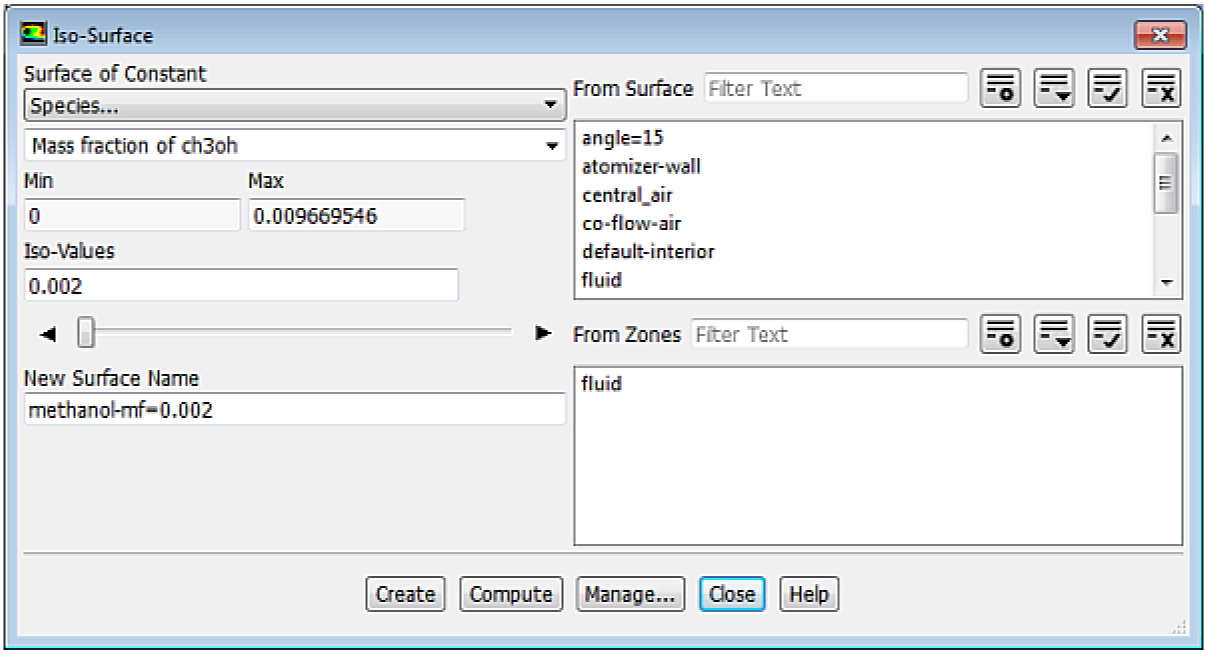
e. Click **Display** and close the **Contours** dialog box.

Figure 16.13: Vectors of DPM Mean Velocity Colored by DPM Velocity Magnitude



5. Create an isosurface of the methanol mass fraction.

    **Postprocessing → Surface → Create → Iso-Surface...**



a. Select **Species...** and **Mass fraction of ch3oh** from the **Surface of Constant** drop-down lists.

b. Click **Compute** to update the minimum and maximum values.

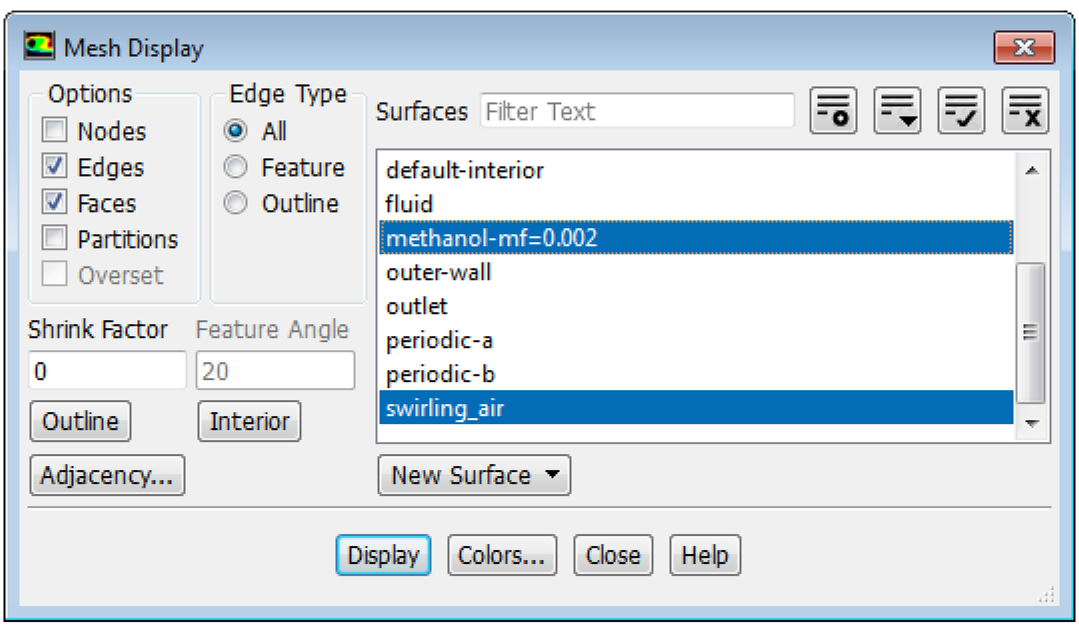
c. Enter `0.002` for **Iso-Values**.

d. Enter `methanol-mf=0.002` for the **New Surface Name**.

e. Click **Create** and then close the **Iso-Surface** dialog box.

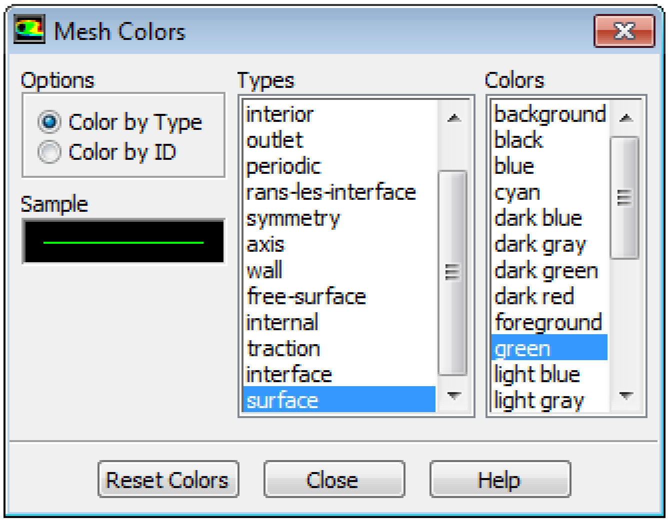
6. Display the isosurface you just created (**methanol-mf=0.002**).

Postprocessing → **Graphics** → **Mesh** → **Edit...**



a. Deselect **atomizer-wall** and select **methanol-mf=0.002** in the **Surfaces** selection list.

b. Click the **Colors...** button to open the **Mesh Colors** dialog box.



i. Select **surface** in the **Types** list and **green** in the **Colors** list.

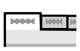
*Scroll down the **Types** list to locate **surface**. The isosurface will now be displayed in green, which contrasts better with the rest of the mesh.*

ii. Close the **Mesh Colors** dialog box.

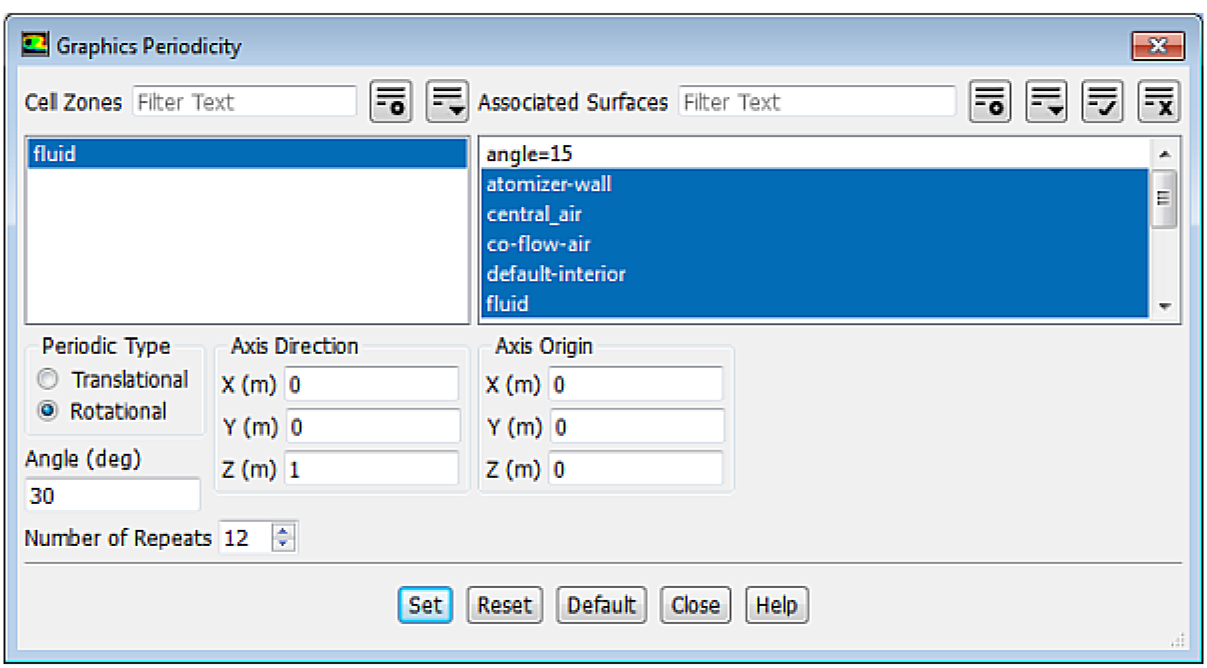
c. Click **Display** in the **Mesh Display** dialog box.

The graphics display will be updated to show the isosurface.

7. Modify the view to include the entire atomizer.

 **Viewing** → **Display** → **Views...**

a. Click **Define...** to open the **Graphics Periodicity** dialog box.



i. Select **fluid** from the **Cell Zones** list.

ii. Ensure that **Rotational** is selected from the **Periodic Type** list and the **Number of Repeats** is set to 12.

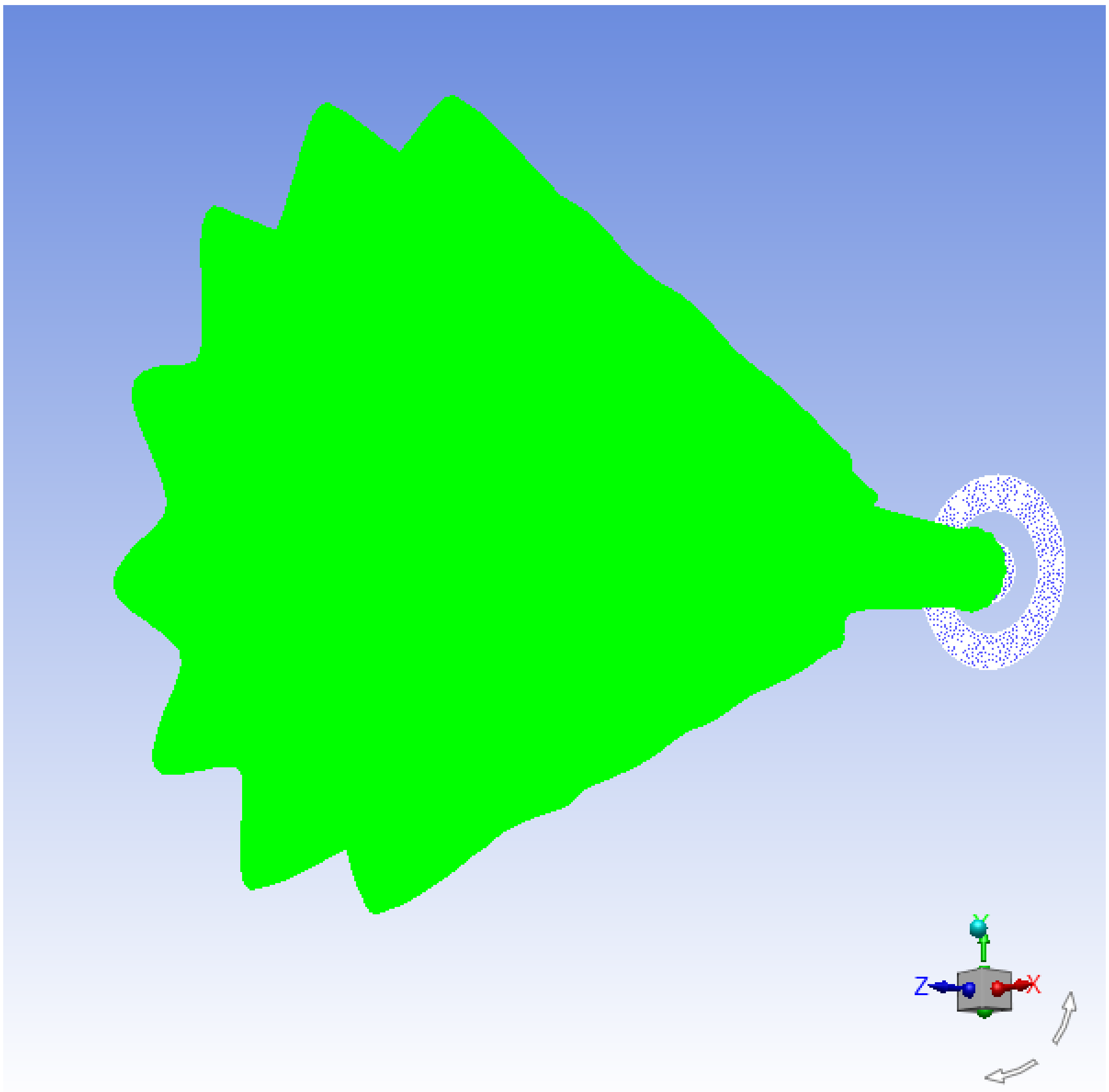
iii. Click **Set** and close the **Graphics Periodicity** dialog box.

b. Click **Apply** and close the **Views** dialog box.

c. Click **Display** and close the **Mesh Display** dialog box.

d. Use the mouse to obtain the view shown in Figure 16.14: Full Atomizer Display with Surface of Constant Methanol Mass Fraction (p. 711).

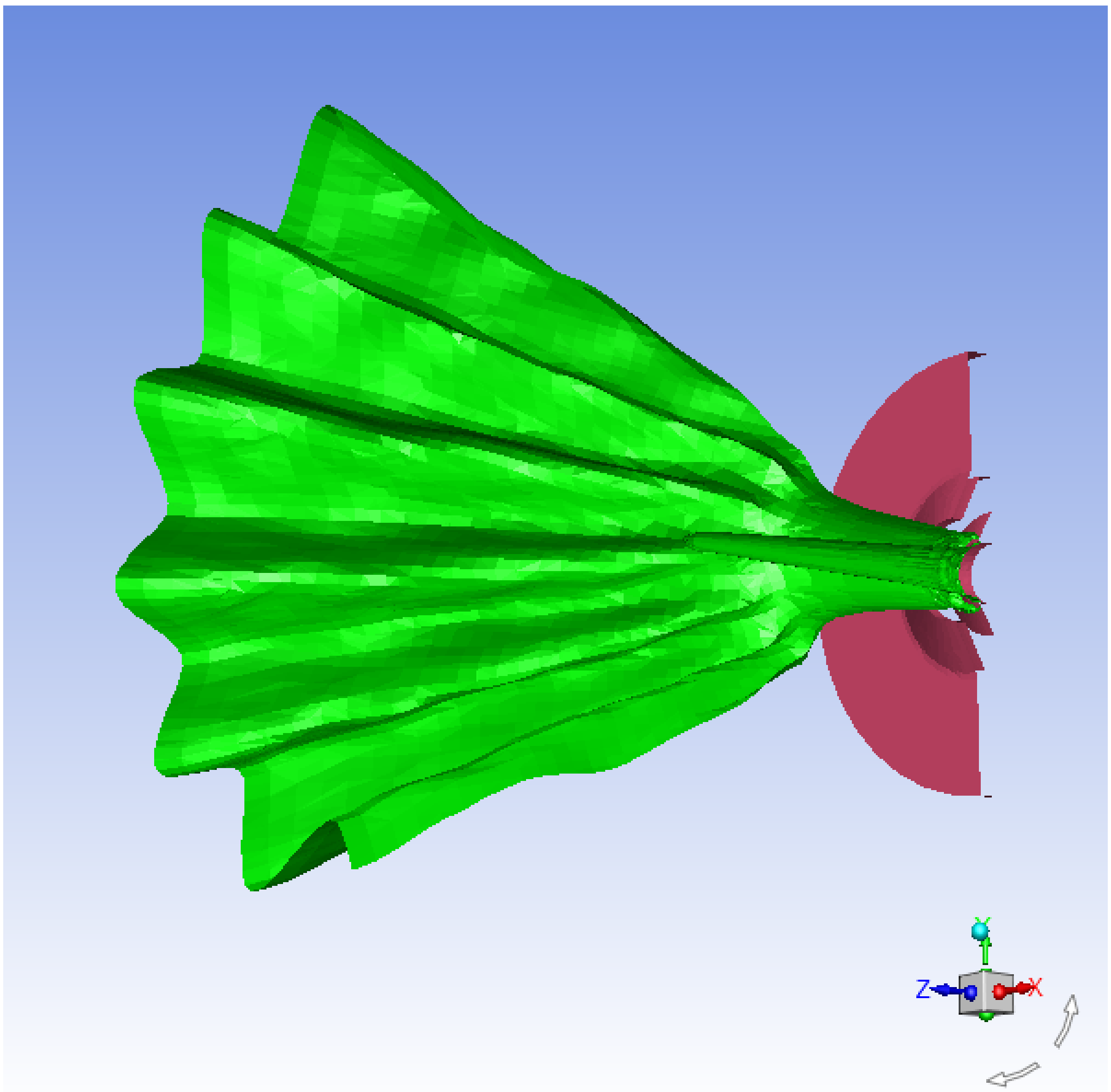
Figure 16.14: Full Atomizer Display with Surface of Constant Methanol Mass Fraction



e. This view can be improved to resemble Figure 16.15: Atomizer Display with Surface of Constant Methanol Mass Fraction Enhanced (p. 712) by changing some of the following variables:

- Disable **Edges** in the **Mesh Display** dialog box
- Select only `atomizer-wall` and `methanol-mf=0.002` in the **Surfaces** list of the **Mesh Display** dialog box
- Change the **Number of Repeats** to 6 in the **Graphics Periodicity** dialog box
- Enable **Lighting** and change it to **Flat** in the **Viewing** tab (**Display** group)
- Enable **Headlight** check in the **Viewing** tab (**Display** group)

Figure 16.15: Atomizer Display with Surface of Constant Methanol Mass Fraction Enhanced



8. Save the case and data files (`spray3.cas.gz` and `spray3.dat.gz`).

   File → **Write** → **Case & Data...**

16.5. Summary

In this tutorial, a spray injection was defined for an air-blast atomizer and the solution was calculated using the discrete phase model in ANSYS Fluent. The location of methanol droplet particles after exiting the atomizer and an isosurface of the methanol mass fraction were examined.

16.6. Further Improvements

This tutorial guides you through the steps to reach an initial solution. You may be able to obtain a more accurate solution by using an appropriate higher-order discretization scheme and by adapting the mesh. Mesh adaption can also ensure that the solution is independent of the mesh. These steps are demonstrated in Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121).

Chapter 17: Using the VOF Model

This tutorial is divided into the following sections:

17.1. Introduction

This tutorial examines the flow of ink as it is ejected from the nozzle of a printhead in an inkjet printer. Using ANSYS Fluent's volume of fluid (VOF) multiphase modeling capability, you will be able to predict the shape and motion of the resulting droplets in an air chamber.

This tutorial demonstrates how to do the following:

- Set up and solve a transient problem using the pressure-based solver and VOF model.
- Copy material from the property database.
- Define time-dependent boundary conditions with a user-defined function (UDF).
- Patch initial conditions in a subset of the domain.
- Automatically save data files at defined points during the solution.
- Examine the flow and interface of the two fluids using volume fraction contours.

17.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

and that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

17.3. Problem Description

The problem considers the transient tracking of a liquid-gas interface in the geometry shown in Figure 17.1: Schematic of the Problem (p. 716). The axial symmetry of the problem enables a 2D geometry to be used. The computation mesh consists of 24,600 cells. The domain consists of two regions: an ink chamber and an air chamber. The dimensions are summarized in Table 17.1: Ink Chamber Dimensions (p. 716).

Figure 17.1: Schematic of the Problem

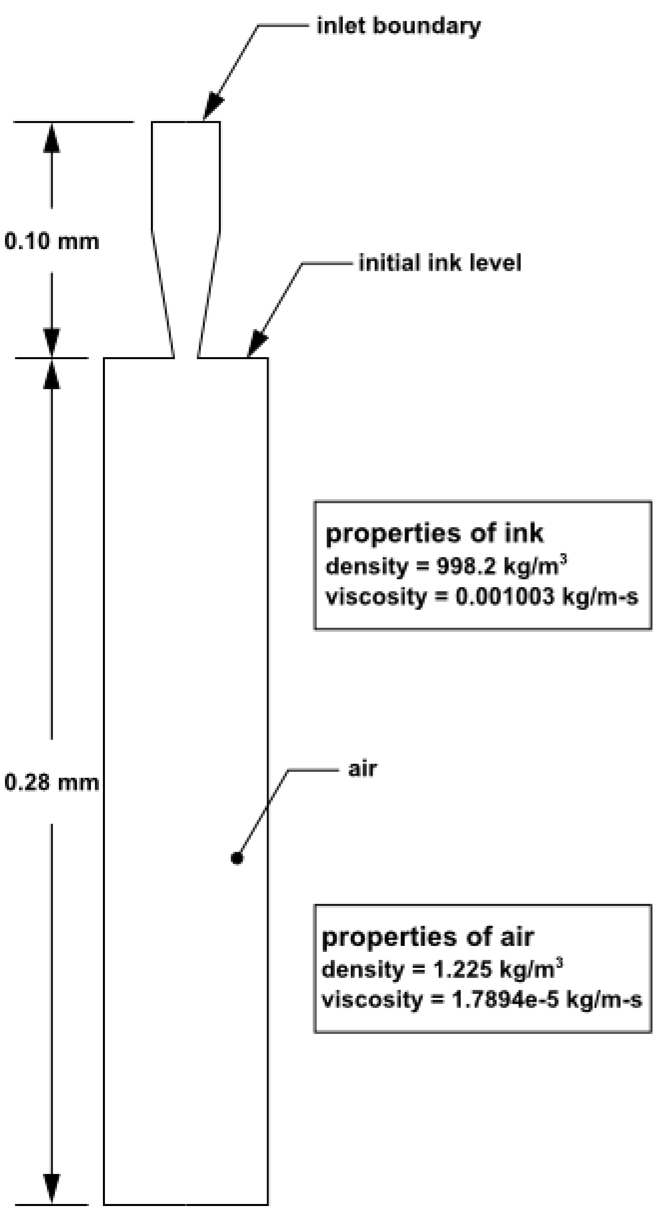


Table 17.1: Ink Chamber Dimensions

Ink Chamber, Cylindrical Region: Radius (mm)	0.015
Ink Chamber, Cylindrical Region: Length (mm)	0.050
Ink Chamber, Tapered Region: Final Radius (mm)	0.009

Ink Chamber, Tapered Region: Length (mm)	0.050
Air Chamber: Radius (mm)	0.030
Air Chamber: Length (mm)	0.280

The following is the chronology of events modeled in this simulation:

- At time zero, the nozzle is filled with ink, while the rest of the domain is filled with air. Both fluids are assumed to be at rest. To initiate the ejection, the ink velocity at the inlet boundary (which is modeled in this simulation by a user-defined function) suddenly increases from 0 to 3.58 m/s and then decreases according to a cosine law.
- After 10 microseconds, the velocity returns to zero.

The calculation is run for 30 microseconds overall, that is, three times longer than the duration of the initial impulse.

Because the dimensions are small, the double-precision version of ANSYS Fluent will be used. Air will be designated as the primary phase, and ink (which will be modeled with the properties of liquid water) will be designated as the secondary phase. Patching will be required to fill the ink chamber with the secondary phase. Gravity will not be included in the simulation. To capture the capillary effect of the ejected ink, the surface tension and prescription of the wetting angle will be specified. The surface inside the nozzle will be modeled as neutrally wettable, while the surface surrounding the nozzle orifice will be non-wettable.

17.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

17.4.1. Preparation

To prepare for running this tutorial:

1. Set up a working folder on the computer you will be using.
2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

Note

If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

3. Enter the name of this tutorial into the search bar.

4. Narrow the results by using the filter on the left side of the page.

    a. Click **ANSYS Fluent** under **Product**.

    b. Click **18.0** under **Version**.

5. Select this tutorial from the list.

6. Click the **vof_R180.zip** link to download the input files.

7. Unzip `vof_R180.zip` file you downloaded to your working folder.

    The files `inkjet.msh` and `inlet1.c` can be found in the `vof` directory created on unzipping the file.

8. Use Fluent Launcher to start the **2D** version of ANSYS Fluent.

    Fluent Launcher displays your **Display Options** preferences from the previous session.

9. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.

10. Enable **Double-Precision**.

    For more information about the Fluent Launcher, see starting ANSYS Fluent using the Fluent Launcher in the Fluent Getting Started Guide.

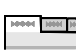
    Note

    The double precision solver is recommended for modeling multiphase flows simulation.

11. Ensure that the **Serial** processing option is selected.

17.4.2. Reading and Manipulating the Mesh

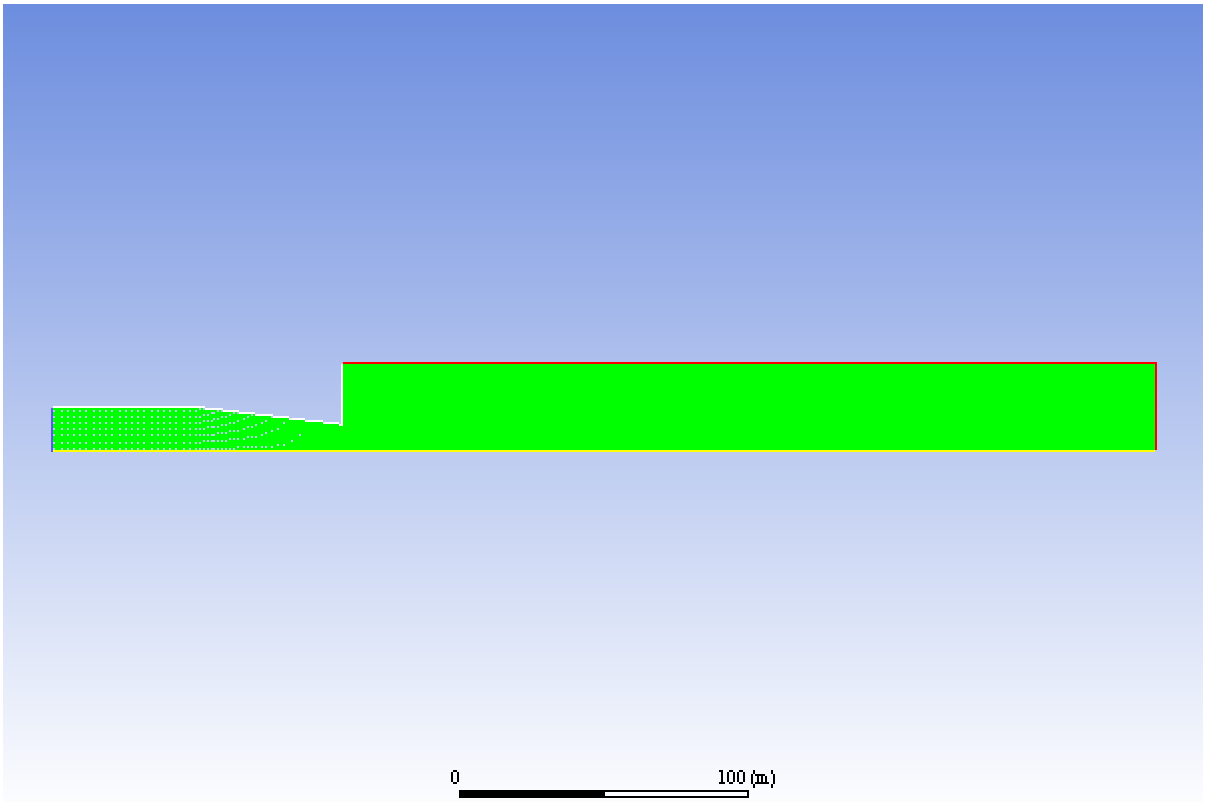
1. Read the mesh file `inkjet.msh`.

     **File → Read → Mesh...**

    A warning message will be displayed twice in the console. You need not take any action at this point, as the issue will be resolved when you define the solver settings in General Settings (p. 723).

2. Examine the mesh (Figure 17.2: Default Display of the Nozzle Mesh (p. 719)).

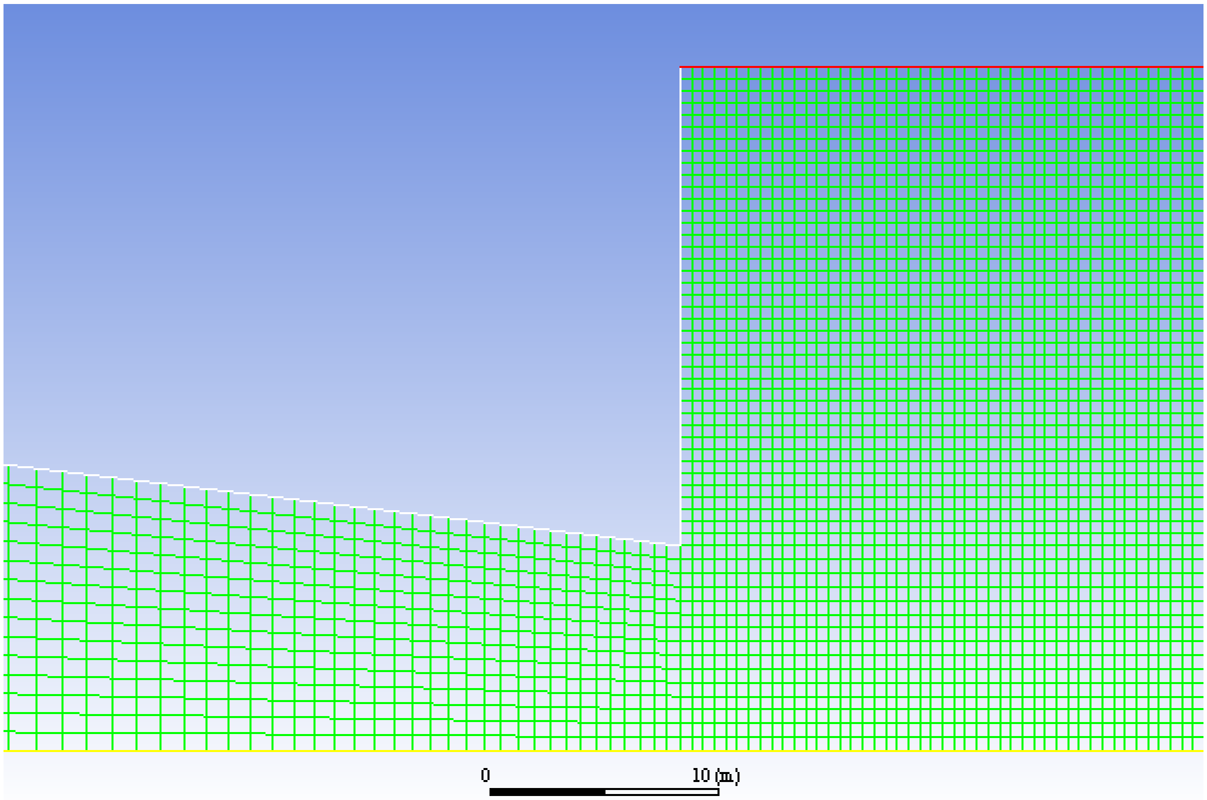
Figure 17.2: Default Display of the Nozzle Mesh



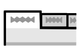
Tip

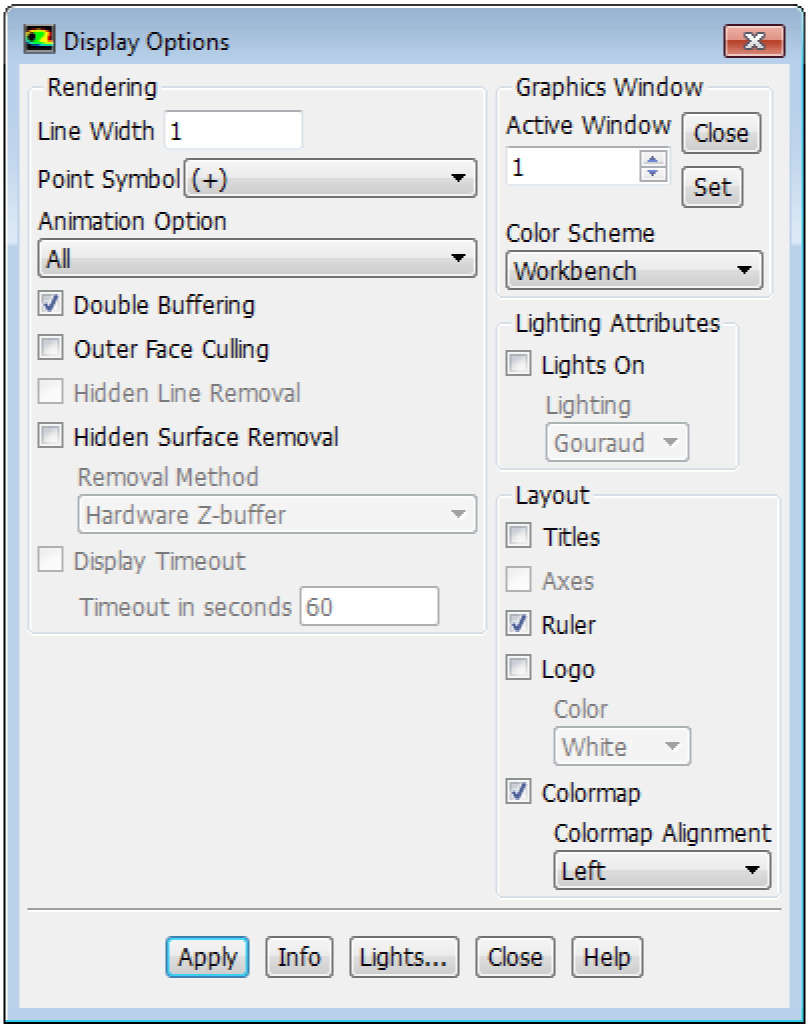
By zooming in with the middle mouse button, you can see that the interior of the model is composed of a fine mesh of quadrilateral cells (see Figure 17.3: The Quadrilateral Mesh (p. 720)).

Figure 17.3: The Quadrilateral Mesh



3. Set graphics display options

 **Viewing → Display → Options...**

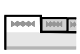


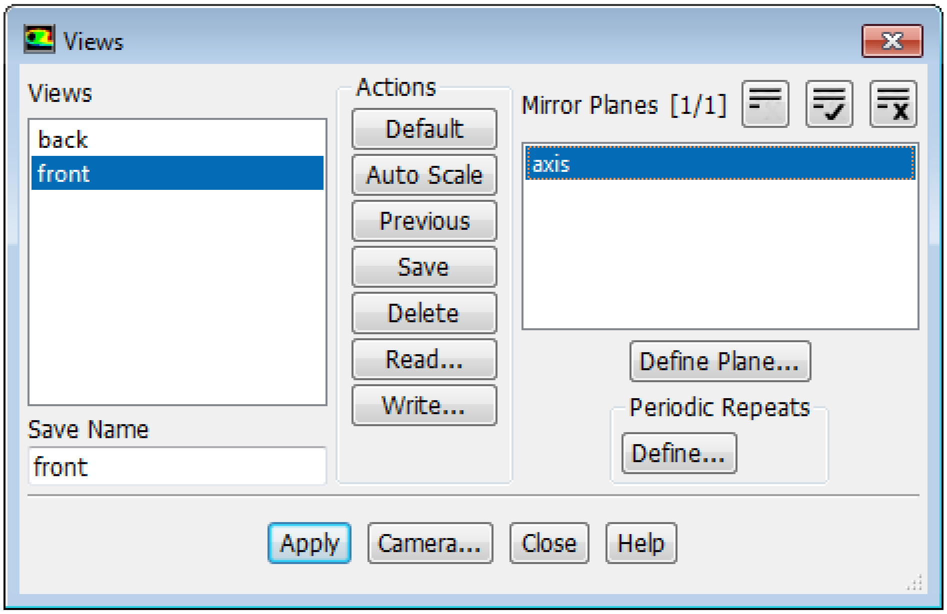
a. Ensure that **All** is selected from the **Animation Option** drop-down list.

   *Selecting **All** will allow you to see the movement of the entire mesh as you manipulate the **Camera** view in the next step.*

4. Click **Apply** and close the **Display Options** dialog box.

5. Manipulate the mesh display to show the full chamber upright.

    **Viewing** → **Display** → **Views...**



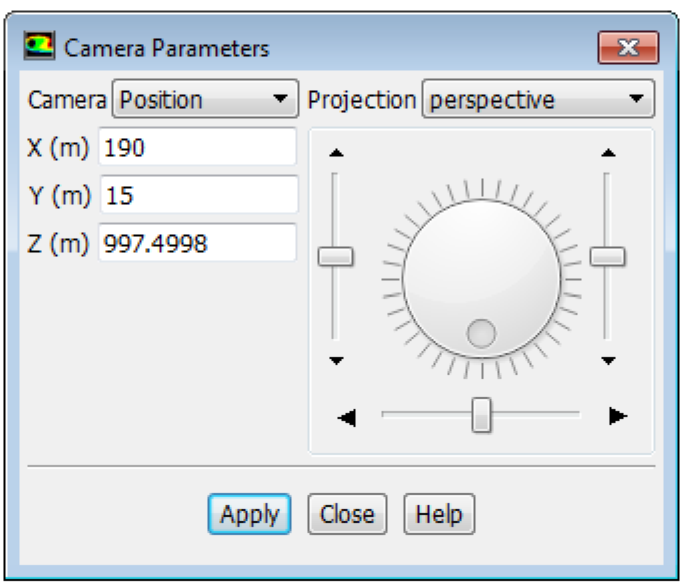
a. Select **front** from the **Views** selection list.

b. Select **axis** from the **Mirror Planes** selection list.

c. Click **Apply**.

   The mesh display is updated to show both sides of the chamber.

d. Click the **Camera...** button to open the **Camera Parameters** dialog box.



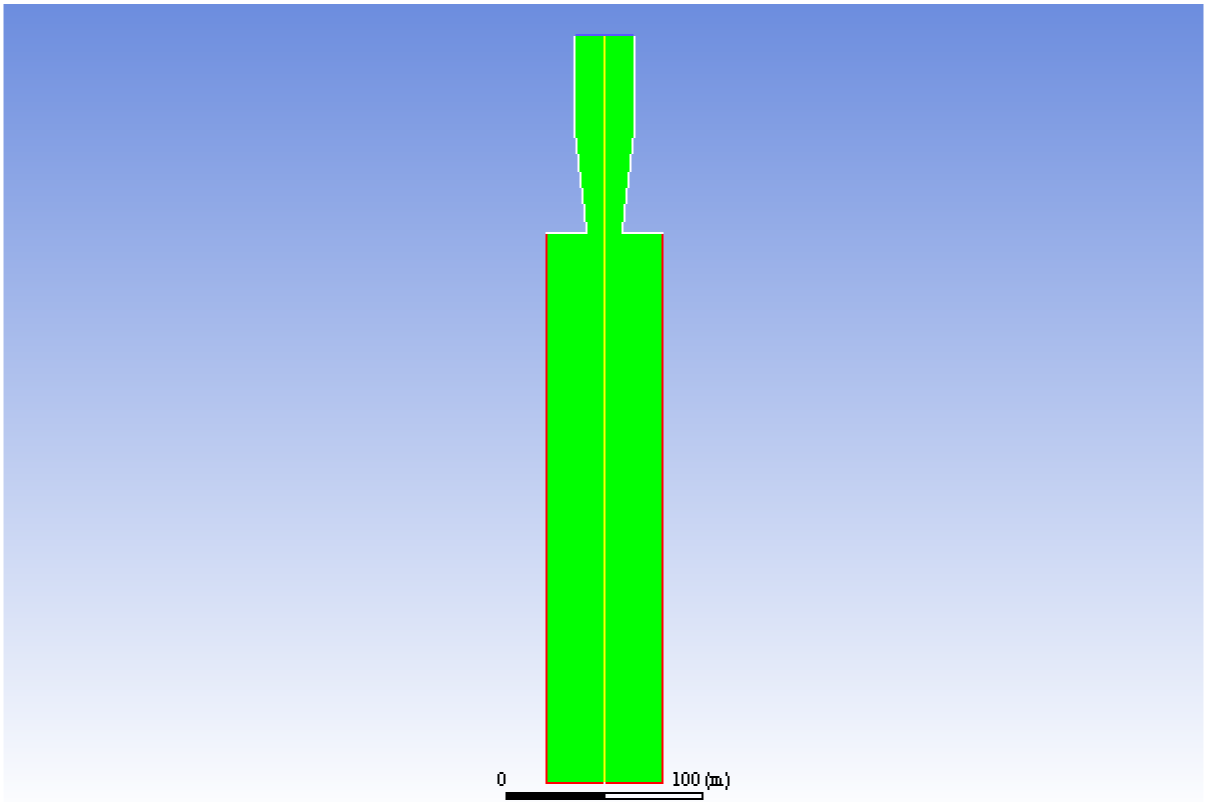
Note

You may notice that the scale of the dimensions in the Camera Parameters dialog box appear very large given the problem dimensions. This is because you have not yet scaled the mesh to the correct units. You will do this in a later step.

---

i. Drag the indicator of the dial with the left mouse button in the clockwise direction until the upright view is displayed (Figure 17.4: Mesh Display of the Nozzle Mirrored and Upright (p. 723)).

Figure 17.4: Mesh Display of the Nozzle Mirrored and Upright



ii. Close the **Camera Parameters** dialog box.

e. Close the **Views** dialog box.

17.4.3. General Settings

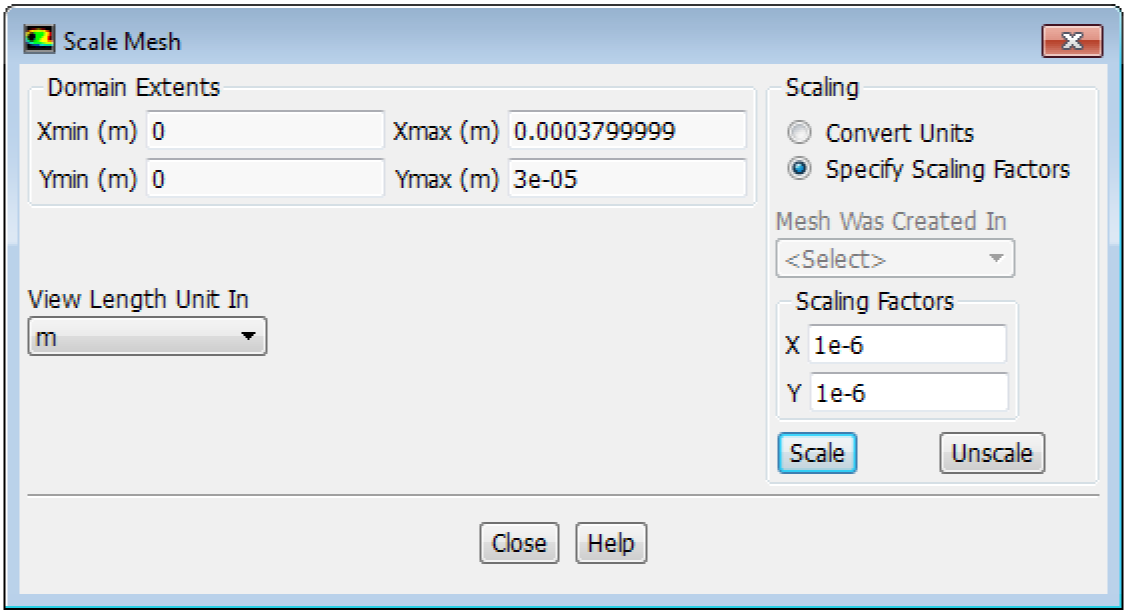
1. Check the mesh.

   Setting Up Domain → **Mesh** → **Check**

   ANSYS Fluent will perform various checks on the mesh and report the progress in the console. Make sure that the reported minimum volume is a positive number.

2. Scale the mesh.

   Setting Up Domain → **Mesh** → **Scale...**



a. Select **Specify Scaling Factors** from the **Scaling** group box.

b. Enter `1e-6` for **X** and **Y** in the **Scaling Factors** group box.

c. Click **Scale** and close the **Scale Mesh** dialog box.

3. Check the mesh.

Setting Up Domain → **Mesh** → **Check**

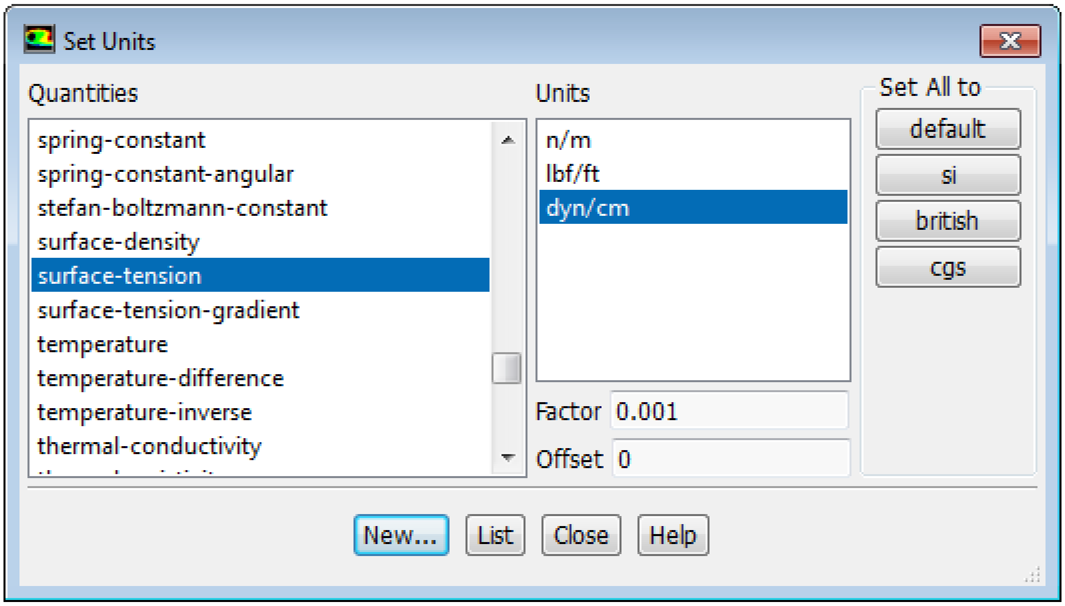
Note

It is a good idea to check the mesh after you manipulate it (that is, scale, convert to polyhedra, merge, separate, fuse, add zones, or smooth and swap.) This will ensure that the quality of the mesh has not been compromised.

---

4. Define the units for the mesh.

Setting Up Domain → **Mesh** → **Units...**



a. Select **length** from the **Quantities** list.

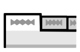
b. Select **mm** from the **Units** list.

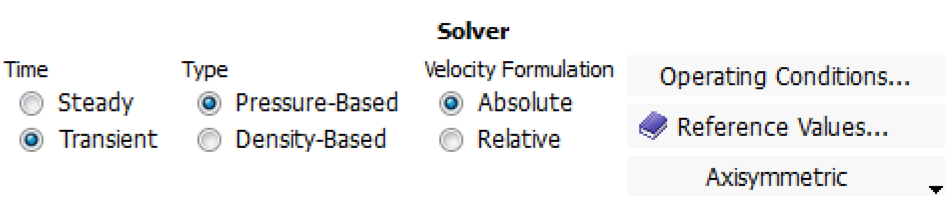
c. Select **surface-tension** from the **Quantities** list.

d. Select **dyn/cm** from the **Units** list.

e. Close the **Set Units** dialog box.

5. Retain the default setting of **Pressure-Based** in the **Solver** group box of the **Setting Up Physics** ribbon tab.

 **Setting Up Physics → Solver**

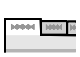


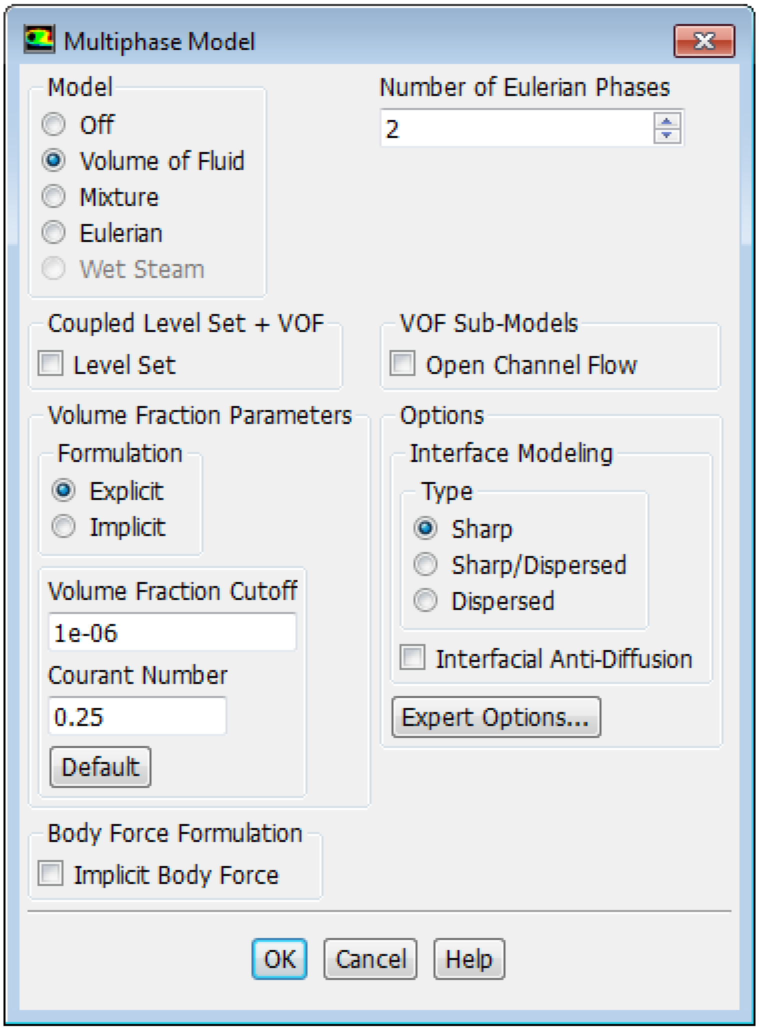
6. Select **Transient** from the **Time** list.

7. Select **Axisymmetric** from the drop-down list in the **Solver** group box (below **Reference Values**).

17.4.4. Models

1. Enable the **Volume of Fluid** multiphase model.

 **Setting Up Physics → Models → Multiphase...**



a. Select **Volume of Fluid** from the **Model** list.

   *The **Multiphase Model** dialog box expands to show related inputs.*

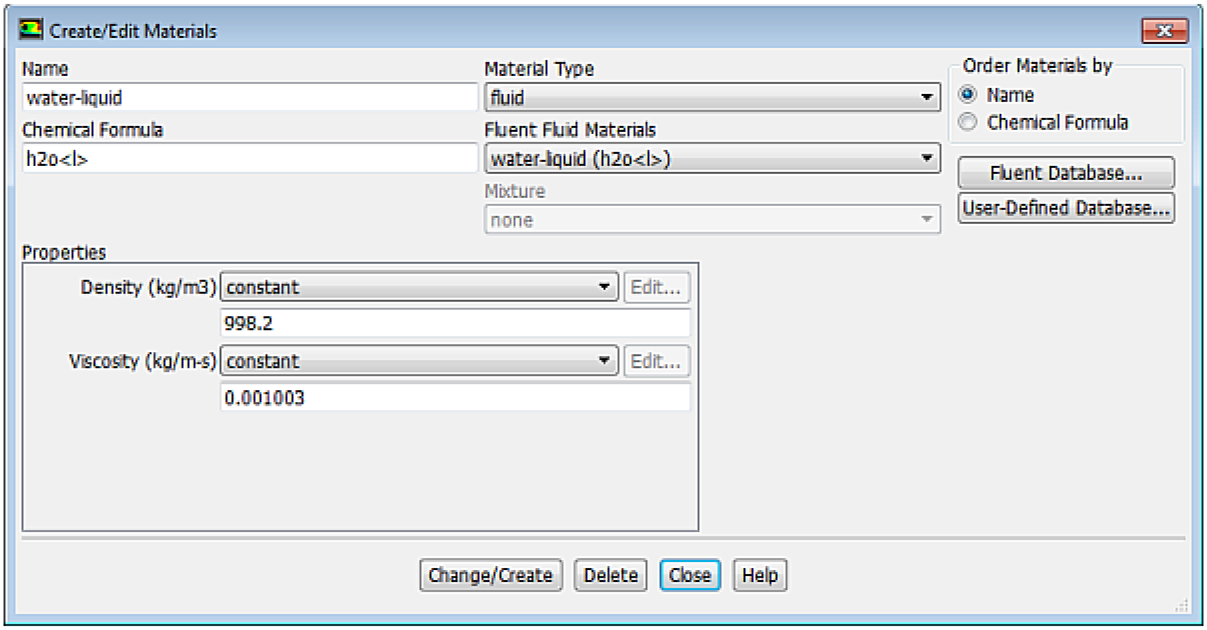
b. Retain the default settings and click **OK** to close the **Multiphase Model** dialog box.

17.4.5. Materials

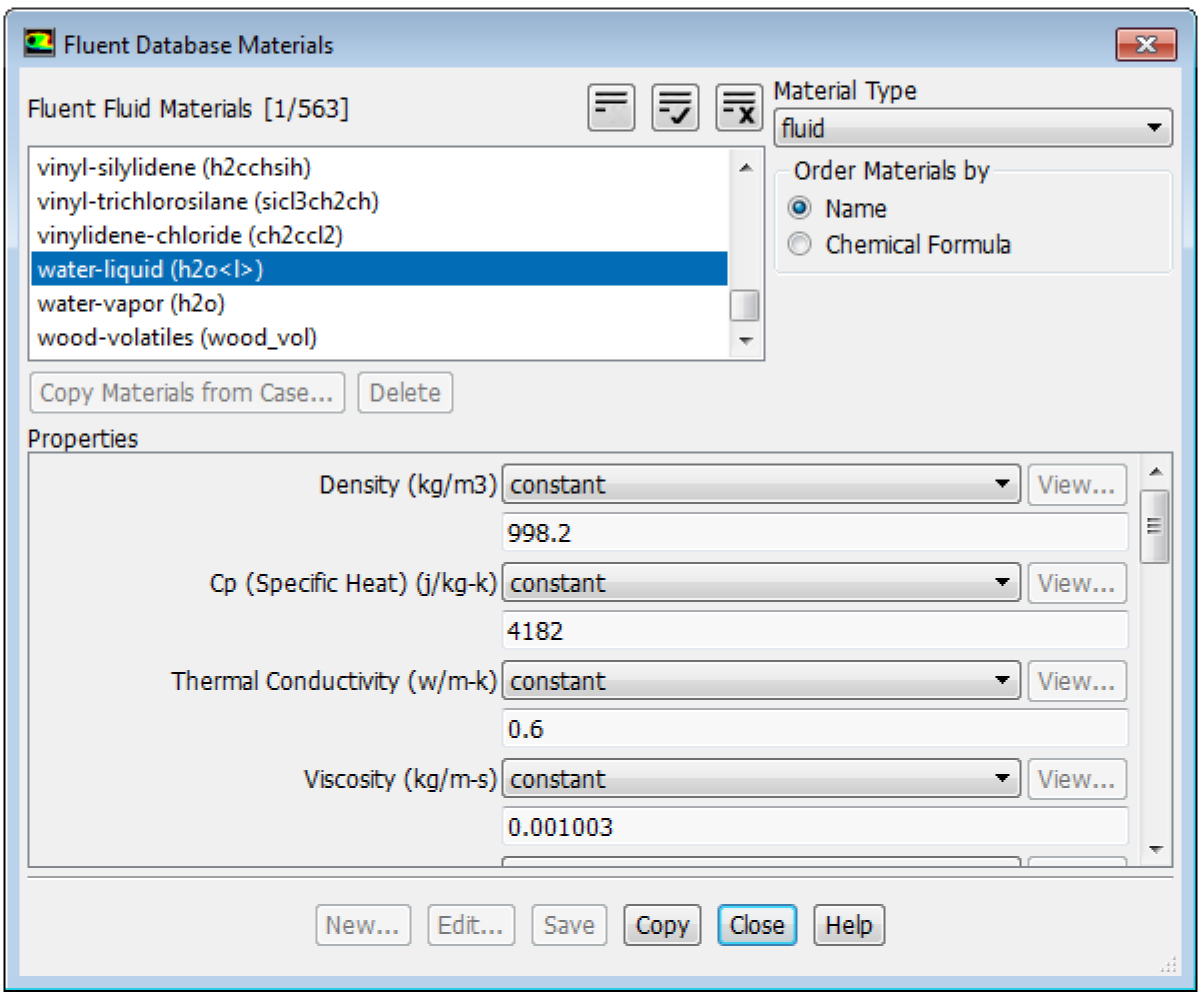
The default properties of air and water defined in ANSYS Fluent are suitable for this problem. In this step, you will make sure that both materials are available for selection in later steps.

1. Add water to the list of fluid materials by copying it from the ANSYS Fluent materials database.

   Setting Up Physics → Materials → Create/Edit...



a. Click **Fluent Database...** in the **Create/Edit Materials** dialog box to open the **Fluent Database Materials** dialog box.



i. Select **water-liquid (h2o < l >)** from the **Fluent Fluid Materials** selection list.

*Scroll down the **Fluent Fluid Materials** list to locate **water-liquid (h2o < l >)**.*

ii. Click **Copy** to copy the information for water to your list of fluid materials.

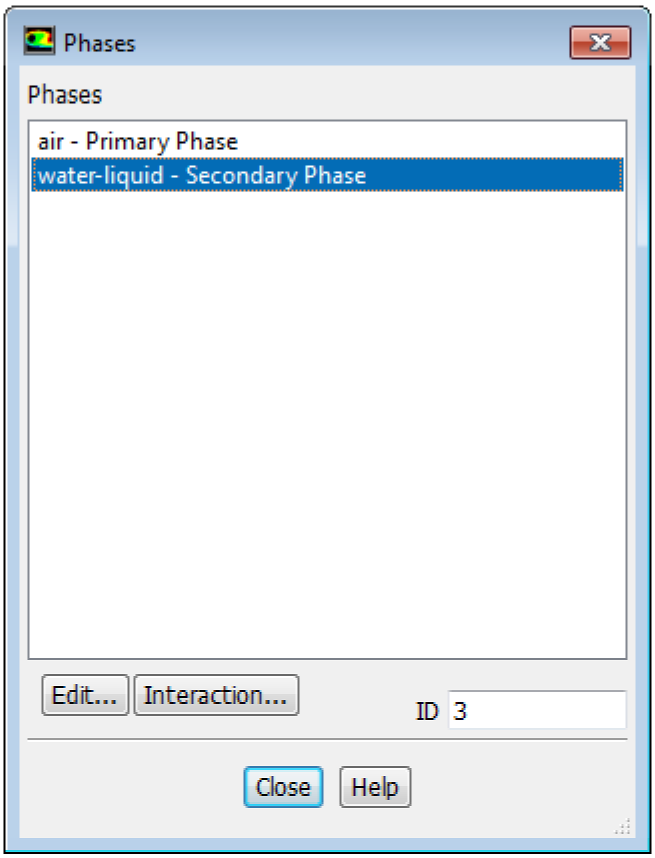
iii. Close the **Fluent Database Materials** dialog box.

b. Click **Change/Create** and close the **Create/Edit Materials** dialog box.

17.4.6. Phases

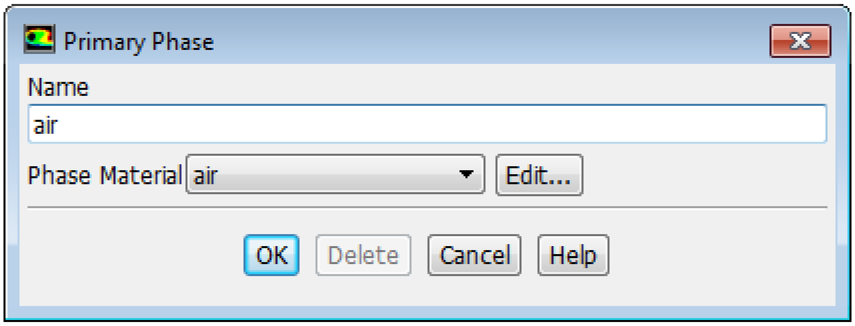
In the following steps, you will define water as the secondary phase. When you define the initial solution, you will patch water in the nozzle region. In general, you can specify the primary and secondary phases whichever way you prefer. It is a good idea to consider how your choice will affect the ease of problem setup, especially with more complicated problems.

Setting Up Physics → Phases → List/Show All...



1. Specify air (**air**) as the primary phase.

   In the **Phases** dialog box, select **phase 1 – Primary Phase** and click **Edit...** to open the **Primary Phase** dialog box.



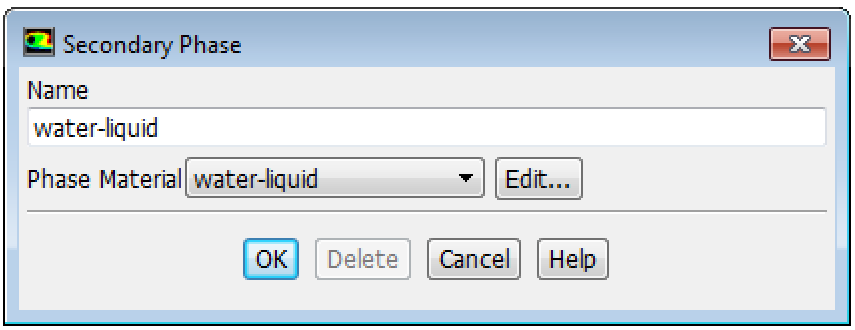
a. Enter `air` for **Name**.

b. Retain the default selection of **air** in the **Phase Material** drop-down list.

c. Click **OK** to close the **Primary Phase** dialog box.

2. Specify water (**water-liquid**) as the secondary phase.

   In the **Phases** dialog box, select **phase 2 – Secondary Phase** and click **Edit...** to open the **Secondary Phase** dialog box.



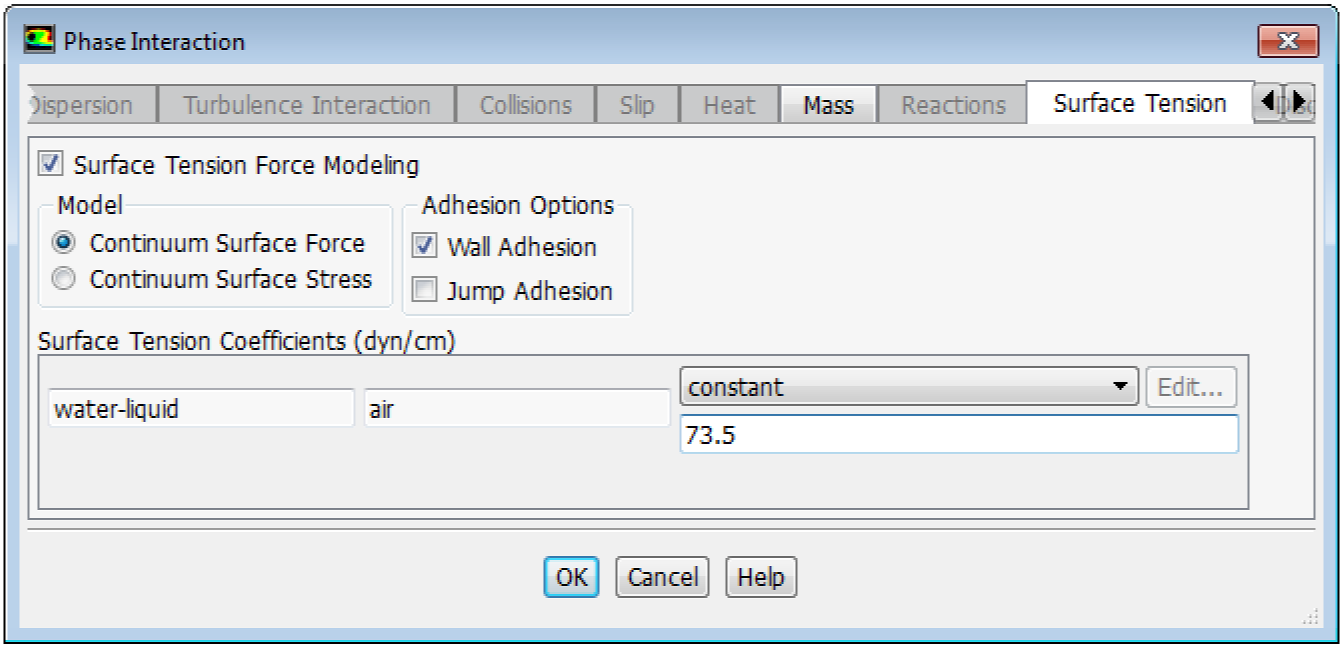
a. Enter `water-liquid` for **Name**.

b. Select **water-liquid** from the **Phase Material** drop-down list.

c. Click **OK** to close the **Secondary Phase** dialog box.

3. Specify the interphase interaction.

   In the **Phases** dialog box, click **Interaction...** to open the **Phase Interaction** dialog box.



a. Click the **Surface Tension** tab.

b. Enable **Surface Tension Force Modeling**.

The surface tension inputs is displayed and the ***Continuum Surface Force*** *model is set as the default.*

c. Enable **Wall Adhesion** so that contact angles can be prescribed.

d. Select **constant** from the **Surface Tension Coefficient** drop-down list.

e. Enter 73.5 dyn/cm for **Surface Tension Coefficient**.

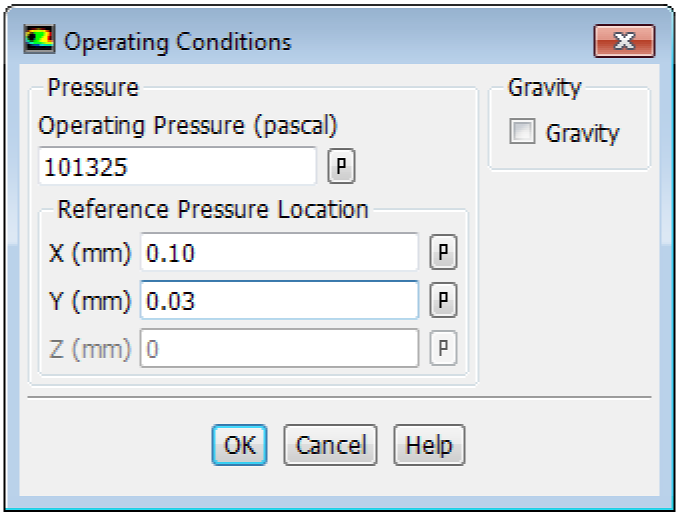
f. Click **OK** to close the **Phase Interaction** dialog box.

4. Close the **Phases** dialog box.

17.4.7. Operating Conditions

1. Set the operating reference pressure location.

Setup → **Boundary Conditions** → **Operating Conditions...**



*You will set the **Reference Pressure Location** to be a point where the fluid will always be 100 % air.*

a. Enter `0.10` mm for **X**.

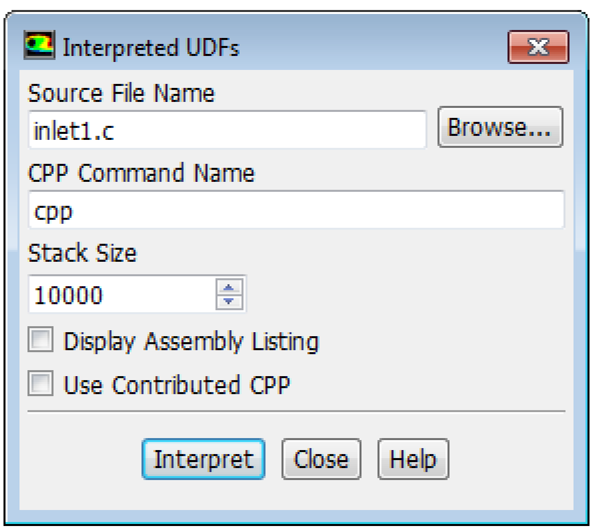
b. Enter `0.03` mm for **Y**.

c. Click **OK** to close the **Operating Conditions** dialog box.

17.4.8. User-Defined Function (UDF)

1. Interpret the UDF source file for the ink velocity distribution (`inlet1.c`).

   User Defined → **User Defined** → **Functions** → **Interpreted...**



a. Enter `inlet1.c` for **Source File Name**.

   *If the UDF source file is not in your working directory, then you must enter the entire directory path for **Source File Name** instead of just entering the file name. Alternatively, click the **Browse...** button and select **inlet1.c** in the* `vof` *directory that was created after you unzipped the original file.*

b. Click **Interpret**.

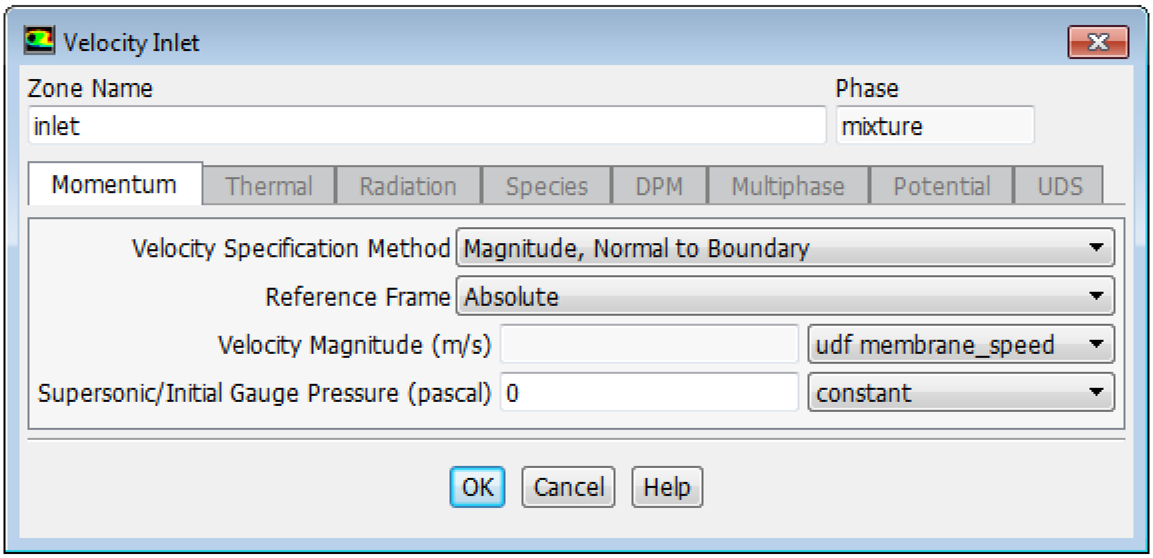
*The UDF defined in inlet1.c is now visible and available for selection as **udf membrane_speed** in the drop-down lists of relevant graphical user interface dialog boxes.*

c. Close the **Interpreted UDFs** dialog box.

17.4.9. Boundary Conditions

1. Set the boundary conditions at the inlet (**inlet**) for the mixture by selecting **mixture** from the **Phase** drop-down list in the **Boundary Conditions** task page.

Setup → **Boundary Conditions** → **inlet** **Edit...**

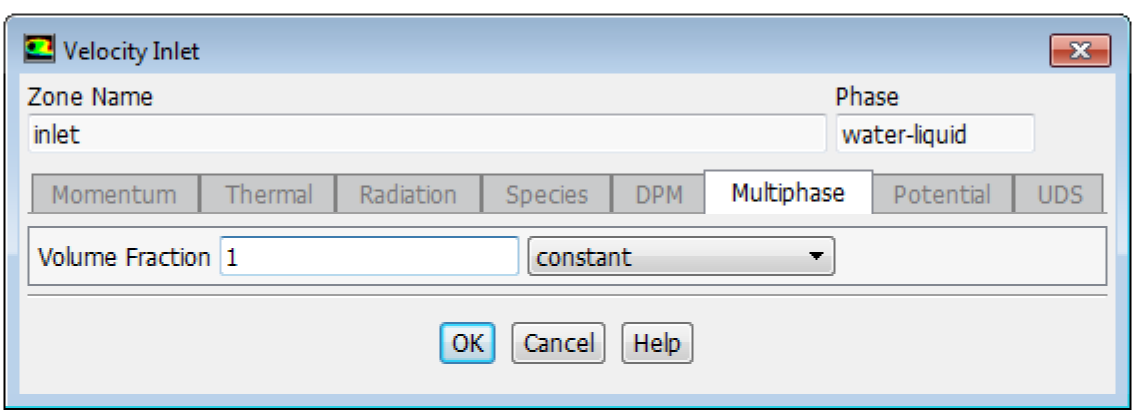


a. Select **udf membrane_speed** from the **Velocity Magnitude** drop-down list.

b. Click **OK** to close the **Velocity Inlet** dialog box.

2. Set the boundary conditions at the inlet (**inlet**) for the secondary phase by selecting **water-liquid** from the **Phase** drop-down list in the **Boundary Conditions** task page.

Setup → **Boundary Conditions** → **inlet** → **water-liquid** **Edit...**

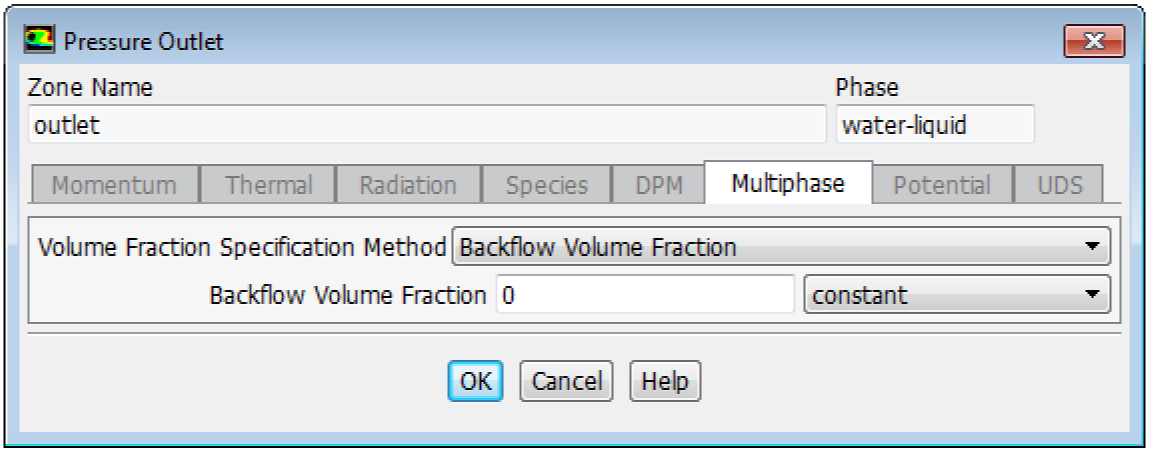


a. Click the **Multiphase** tab and enter 1 for the **Volume Fraction**.

b. Click **OK** to close the **Velocity Inlet** dialog box.

3. Set the boundary conditions at the outlet (**outlet**) for the secondary phase by selecting **water-liquid** from the **Phase** drop-down list in the **Boundary Conditions** task page.

Setup → **Boundary Conditions** → **outlet** → **water-liquid** **Edit...**

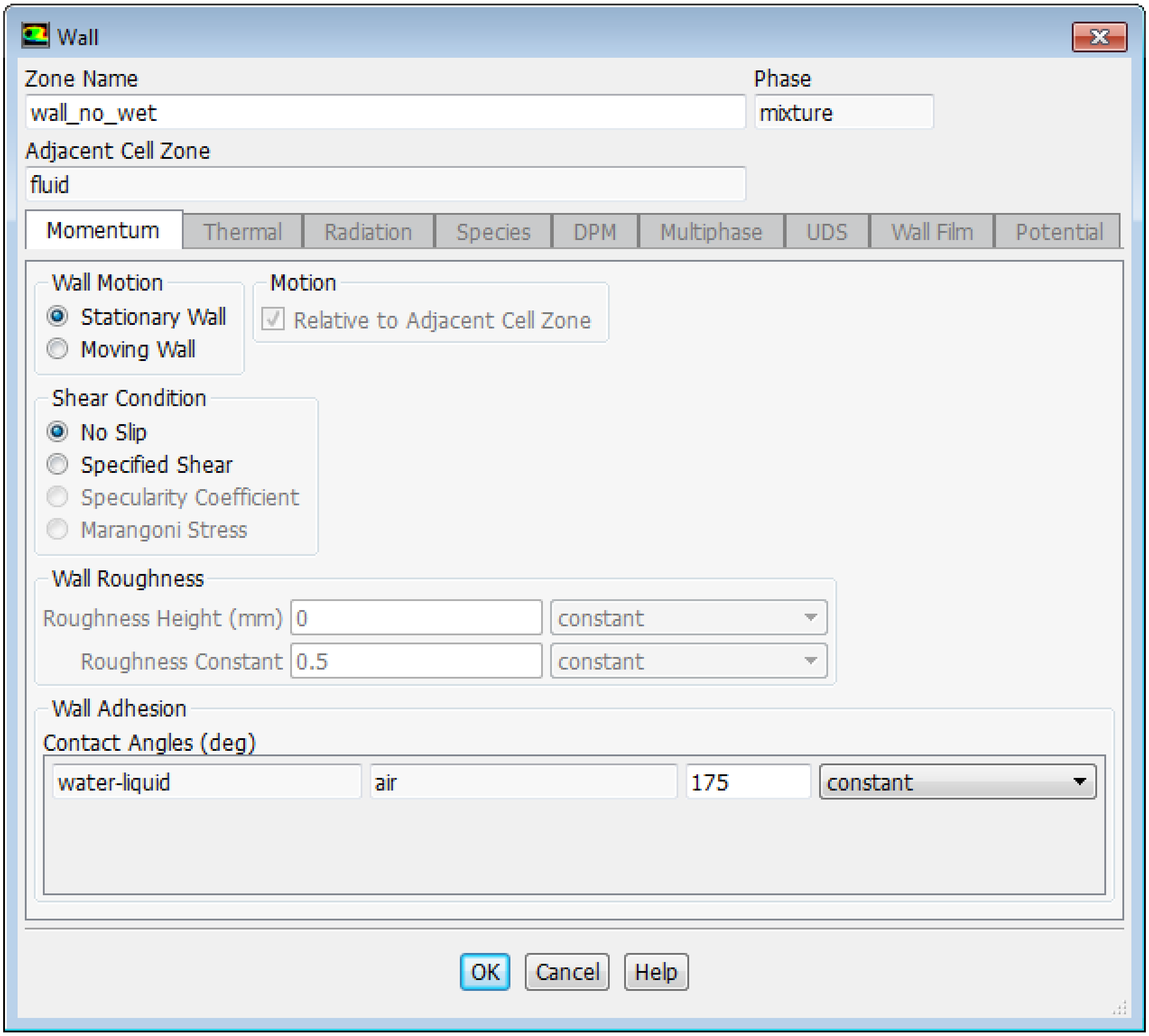


a. Click the **Multiphase** tab and retain the default setting of **0** for the **Backflow Volume Fraction**.

b. Click **OK** to close the **Pressure Outlet** dialog box.

4. Set the conditions at the top wall of the air chamber (**wall_no_wet**) for the mixture by selecting **mixture** from the **Phase** drop-down list in the **Boundary Conditions** task page.

Setup → **Boundary Conditions** → **wall_no_wet** **Edit...**



a. Enter 175 degrees for **Contact Angles**.

b. Click **OK** to close the **Wall** dialog box.

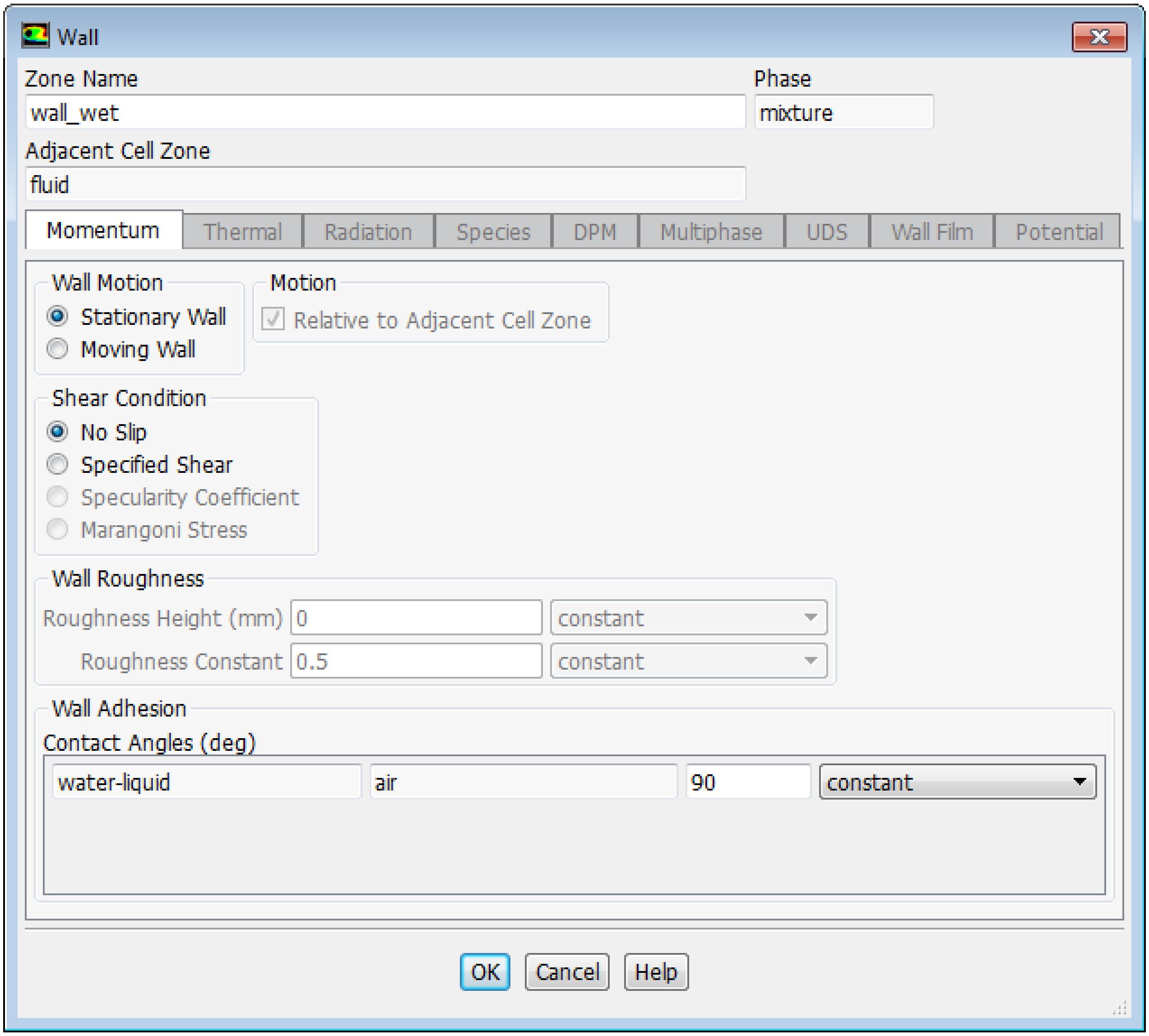
Note

This angle affects the dynamics of droplet formation. You can repeat this simulation to find out how the result changes when the wall is hydrophilic (that is, using a small contact angle, say 10 degrees).

---

5. Set the conditions at the side wall of the ink chamber (**wall_wet**) for the mixture.

Setup → **Boundary Conditions** → **wall_wet** **Edit...**

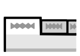


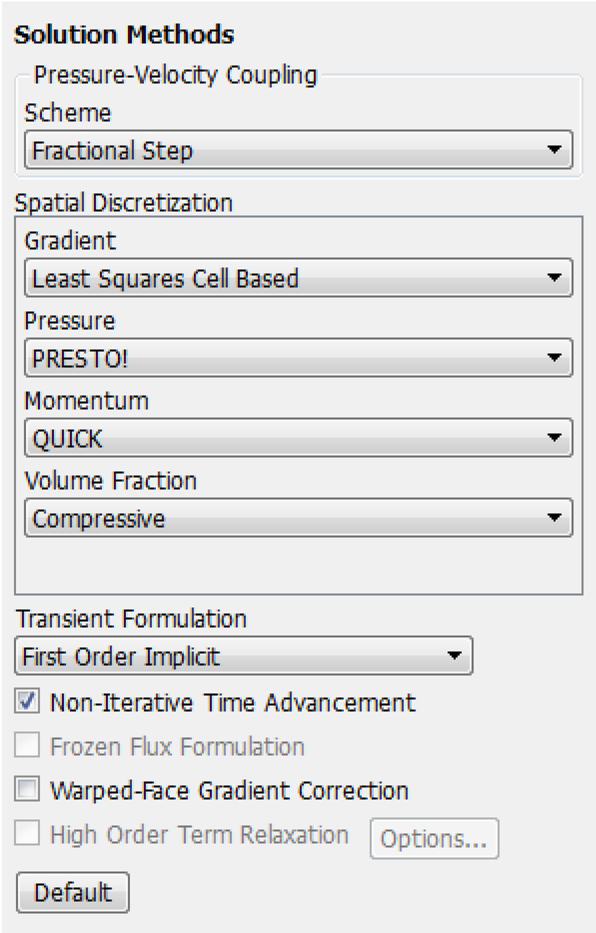
a. Retain the default setting of **90** degrees for **Contact Angles**.

b. Click **OK** to close the **Wall** dialog box.

17.4.10. Solution

1. Set the solution methods.

 **Solving** → **Solution** → **Methods...**



a. Enable **Non-Iterative Time Advancement**.

   The non-iterative time advancement (NITA) scheme is often advantageous compared to the iterative schemes as it is less CPU intensive. Although smaller time steps must be used with NITA compared to the iterative schemes, the total CPU expense is often smaller. If the NITA scheme leads to convergence difficulties, then the iterative schemes (for example, PISO, SIMPLE) should be used instead.

b. Select **Fractional Step** from the **Scheme** drop-down list in the **Pressure-Velocity Coupling** group box.

c. Retain the default selection of **Least Squares Cell Based** from the **Gradient** drop-down list in the **Spatial Discretization** group box.

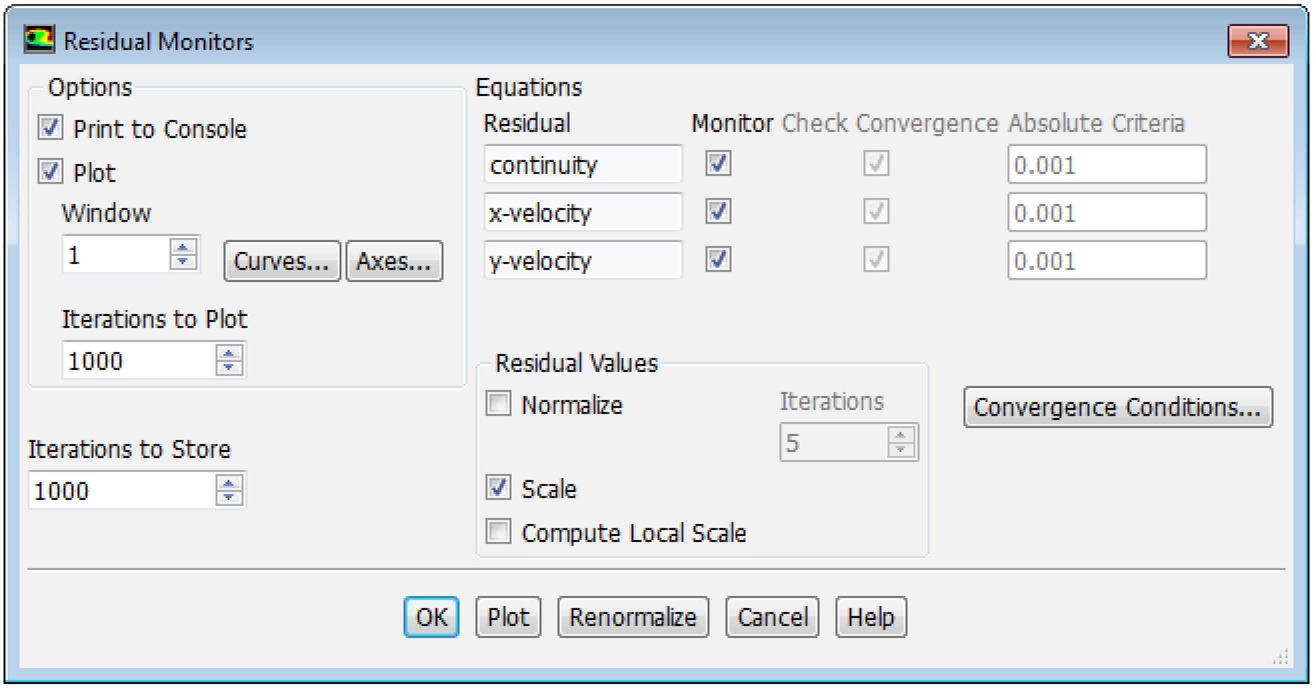
d. Retain the default selection of **PRESTO!** from the **Pressure** drop-down list.

e. Select **QUICK** from the **Momentum** drop-down list.

f. Select **Compressive** from the **Volume-Fraction** drop-down list.

2. Enable the plotting of residuals during the calculation.

   Solving → **Reports** → **Residuals...**

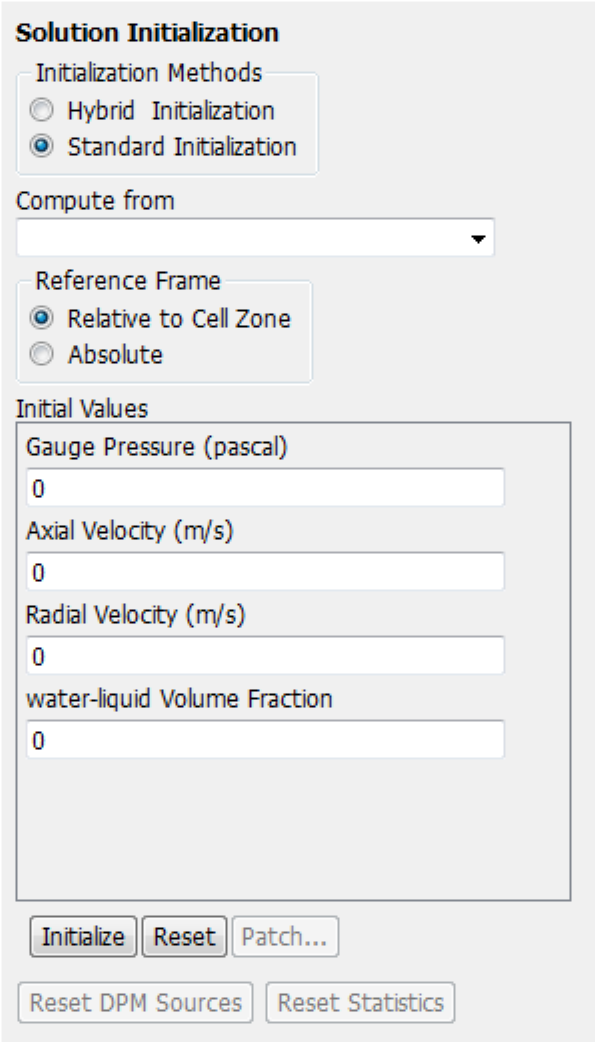


a. Ensure **Plot** is selected in the **Options** group box.

b. Click **OK** to close the **Residual Monitors** dialog box.

3. Initialize the solution after reviewing the default initial values.

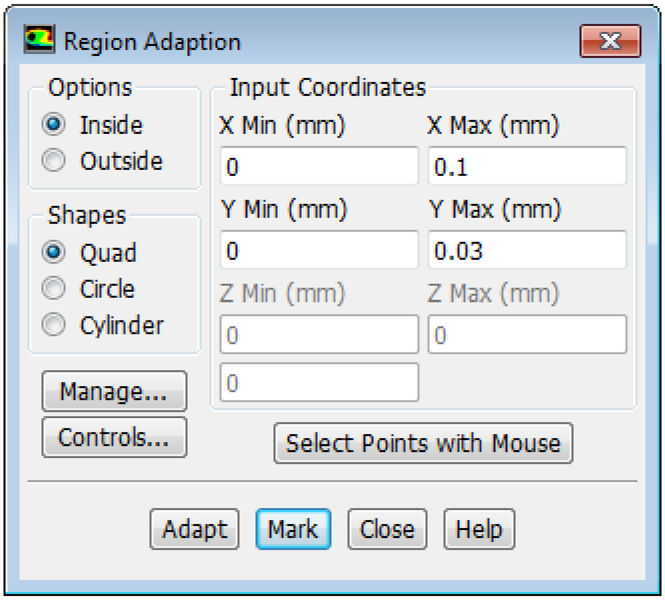
Solving → **Initialization** → **Options...**



a. Retain the default settings for all the parameters and click **Initialize** (either in the ribbon or in the **Solution Initialization** task page.

4. Define a register for the ink chamber region.

Setting Up Domain → **Adapt** → **Mark/Adapt Cells** → **Region...**



a. Retain the default setting of **0** mm for **X Min** and **Y Min** in the **Input Coordinates** group box.

b. Enter `0.10` mm for **X Max**.

c. Enter `0.03` mm for **Y Max**.

d. Click **Mark**.

ANSYS Fluent will report in the console that 1500 cells were marked for refinement while zero cells were marked for coarsening.

Tip

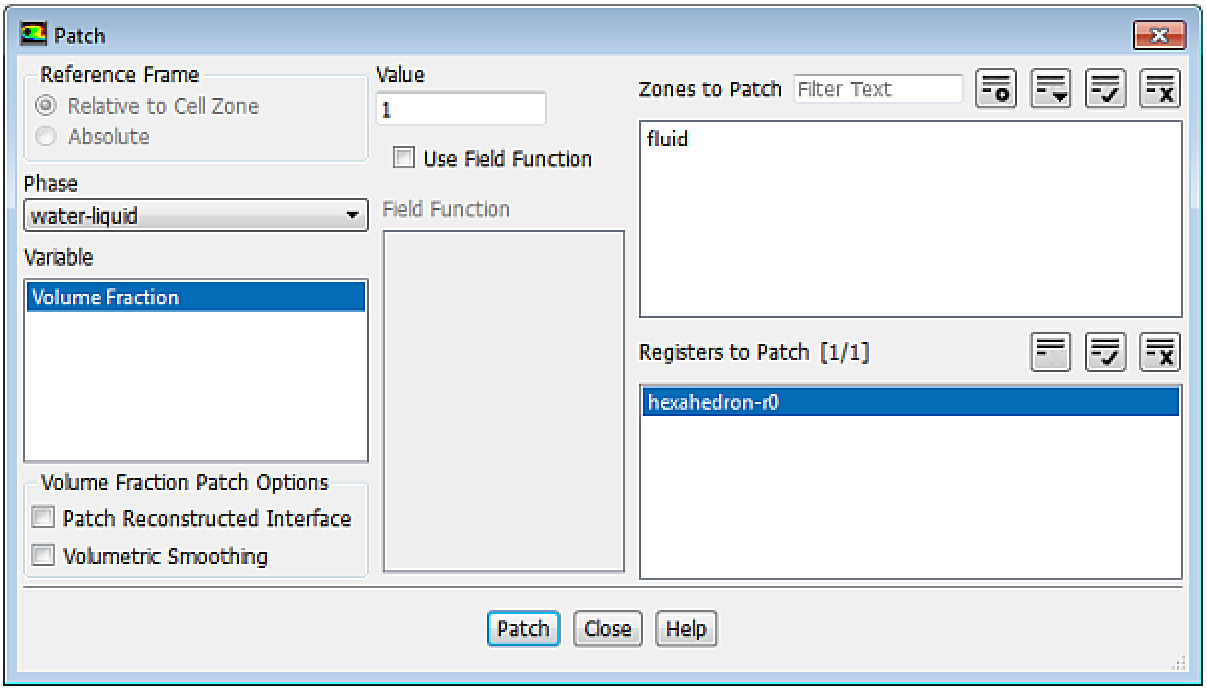
You can display and manipulate adaption registers, which are generated using the **Mark** command, using the **Manage Adaption Registers** dialog box. Click the **Manage...** button in the **Region Adaption** dialog box to open the **Manage Adaption Registers** dialog box. For details, see Adapting the Mesh (p. 174).

---

e. Close the **Region Adaption** dialog box.

5. Patch the initial distribution of the secondary phase (**water-liquid**).

Solving → **Initialization** → **Patch...**



a. Select **water-liquid** from the **Phase** drop-down list.

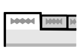
b. Select **Volume Fraction** from the **Variable** list.

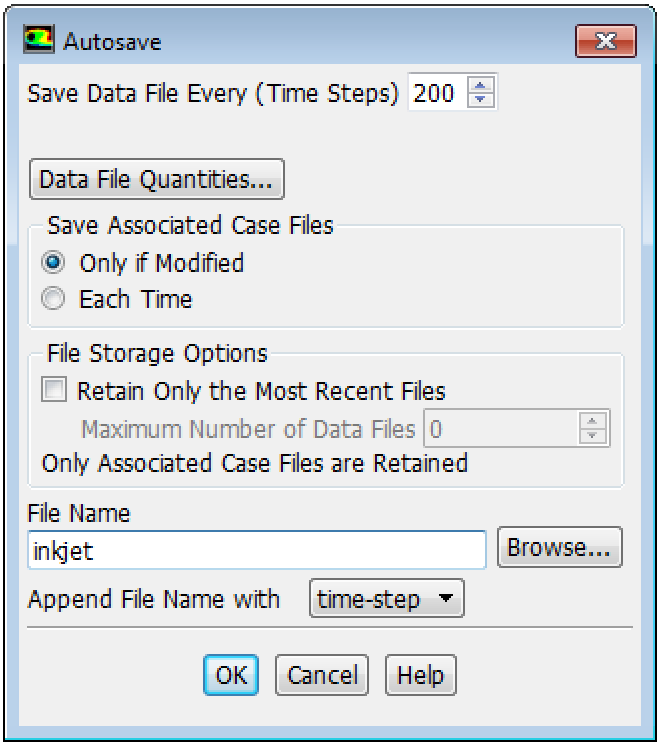
c. Enter 1 for **Value**.

d. Select **hexahedron-r0** from the **Registers to Patch** selection list.

e. Click **Patch** and close the **Patch** dialog box.

6. Request the saving of data files every 200 steps.

 **Solving** → **Activities** → **Autosave...**



a. Enter `200` for **Save Data File Every (Time Steps)**.

b. Ensure that **time-step** is selected from the **Append File Name with** drop-down list.

c. Enter `inkjet` for the **File Name**.

ANSYS Fluent will append the time step value to the file name prefix (`inkjet`). The standard `.dat` extension will also be appended. This will yield file names of the form `inkjet-1-00200.dat`, where 200 is the time step number.

Optionally, you can add the extension `.gz` to the end of the file name (for example, `inkjet.gz`), which instructs ANSYS Fluent to save the data files in a compressed format, yielding file names of the form `inkjet-1-00200.dat.gz`.

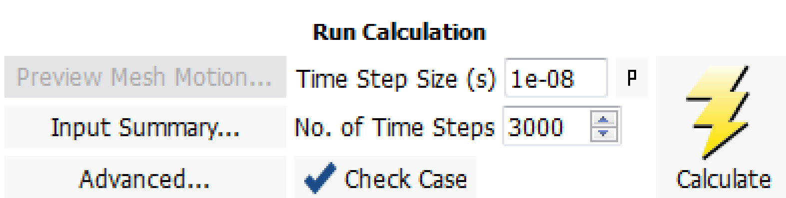
d. Click **OK** to close the **Autosave** dialog box.

7. Save the initial case file (`inkjet.cas.gz`).

File → Write → Case...

8. Run the calculation.

Solving → Run Calculation



a. Enter `1.0e-8` seconds for the **Time Step Size (s)**.

Note

Small time steps are required to capture the oscillation of the droplet interface and the associated high velocities. Failure to use sufficiently small time steps may cause differences in the results between platforms.

---

b. Enter `3000` for the **Number of Time Steps**.

c. Click **Calculate**.

The solution will run for 3000 iterations.

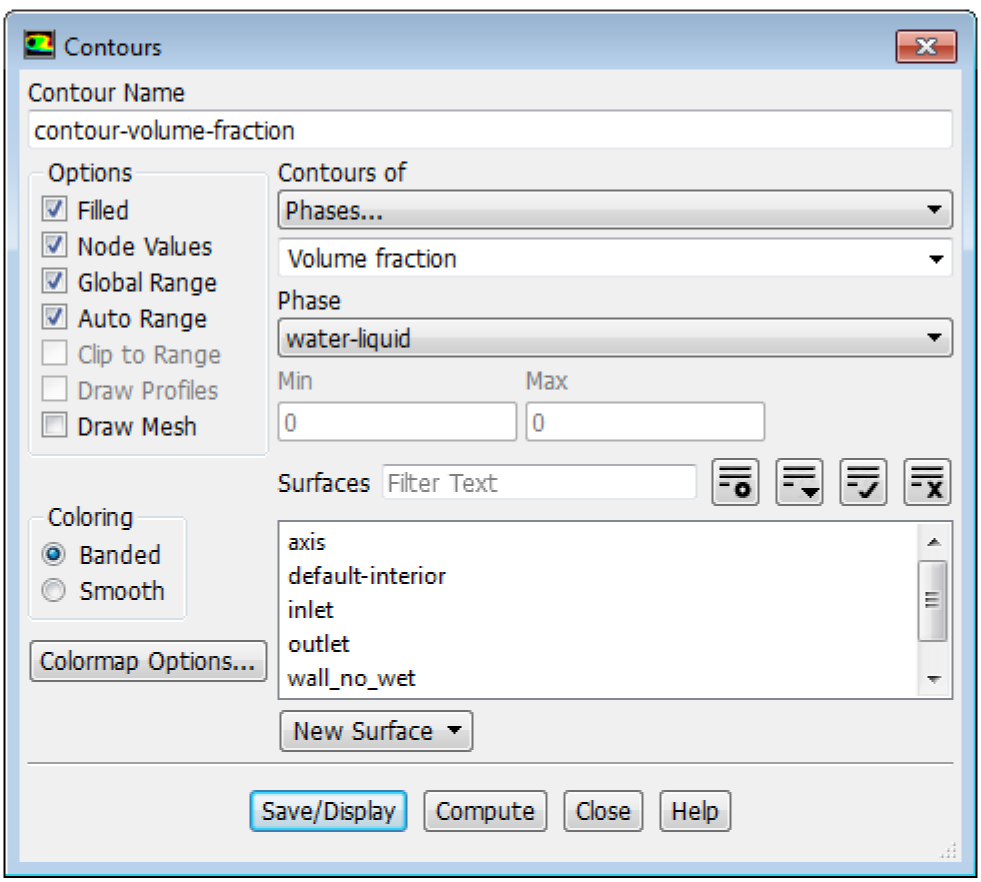
17.4.11. Postprocessing

1. Read the data file for the solution after 6 microseconds (`inkjet-1-00600.dat.gz`).

   File → **Read** → **Data...**

2. Create and display a filled contour of water volume fraction after 6 microseconds (Figure 17.5: Contours of Water Volume Fraction After 6 μs (p. 744)).

   Postprocessing → **Graphics** → **Contours** → **New...**



a. Change the **Contour Name** to `contour-volume-fraction`.

b. Enable **Filled** in the **Options** group box.

c. Select **Phases...** and **Volume fraction** from the **Contours of** drop-down lists.

d. Select **water-liquid** from the **Phase** drop-down list.

e. Click **Save/Display**.

Tip

In order to display the contour plot in the graphics window, you may need to click the **Fit to Window** button.

---

3. Display contours of water volume fraction after 12, 18, 24, and 30 microseconds (Figure 17.6: Contours of Water Volume Fraction After 12 μs (p. 744) — Figure 17.9: Contours of Water Volume Fraction After 30 μs (p. 746)).

   a. Read the data file for the solution after 12 microseconds (`inkjet-1-01200.dat.gz`).

   File → **Read** → **Data...**

   b. Reload the contour graphic saved in the previous step.

   Results → **Graphics** → **Contours** → **contour-volume-fraction** **Display**

   c. Repeat these steps for the 18, 24, and 30 microseconds files.

Figure 17.5: Contours of Water Volume Fraction After 6 μs

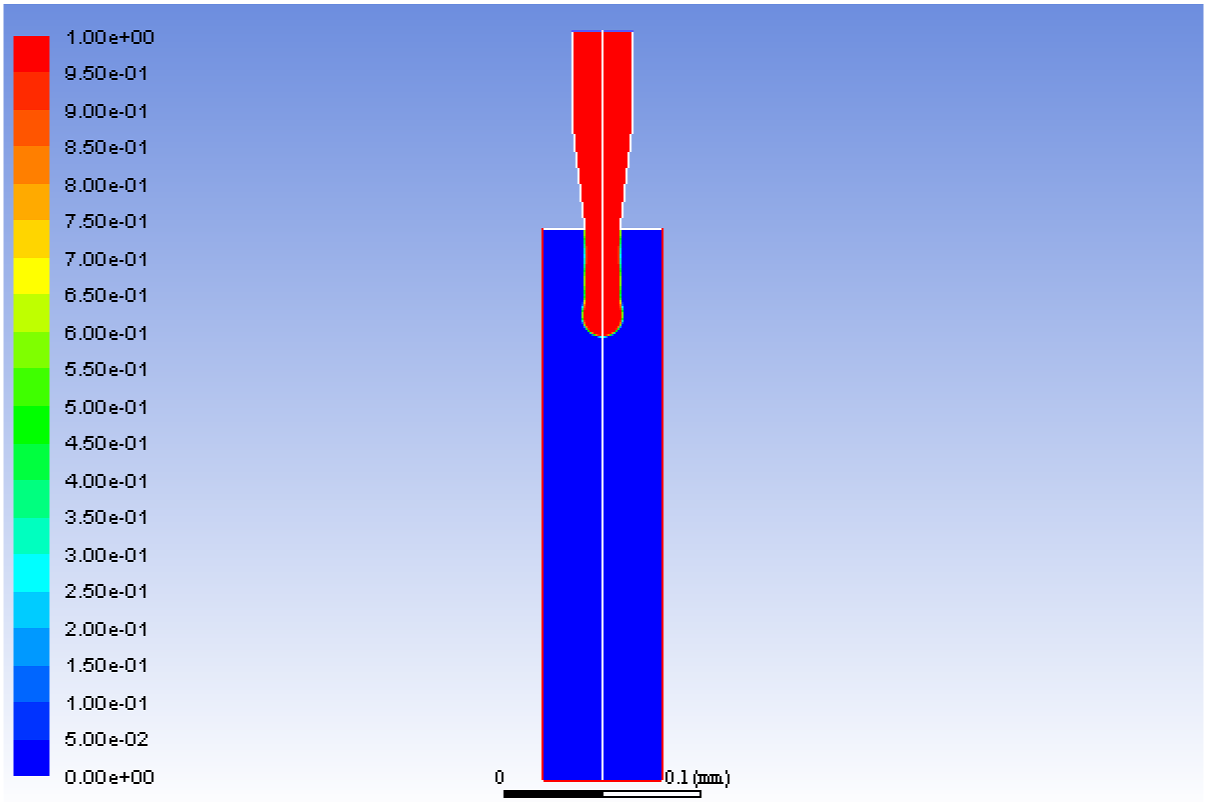


Figure 17.6: Contours of Water Volume Fraction After 12 μs

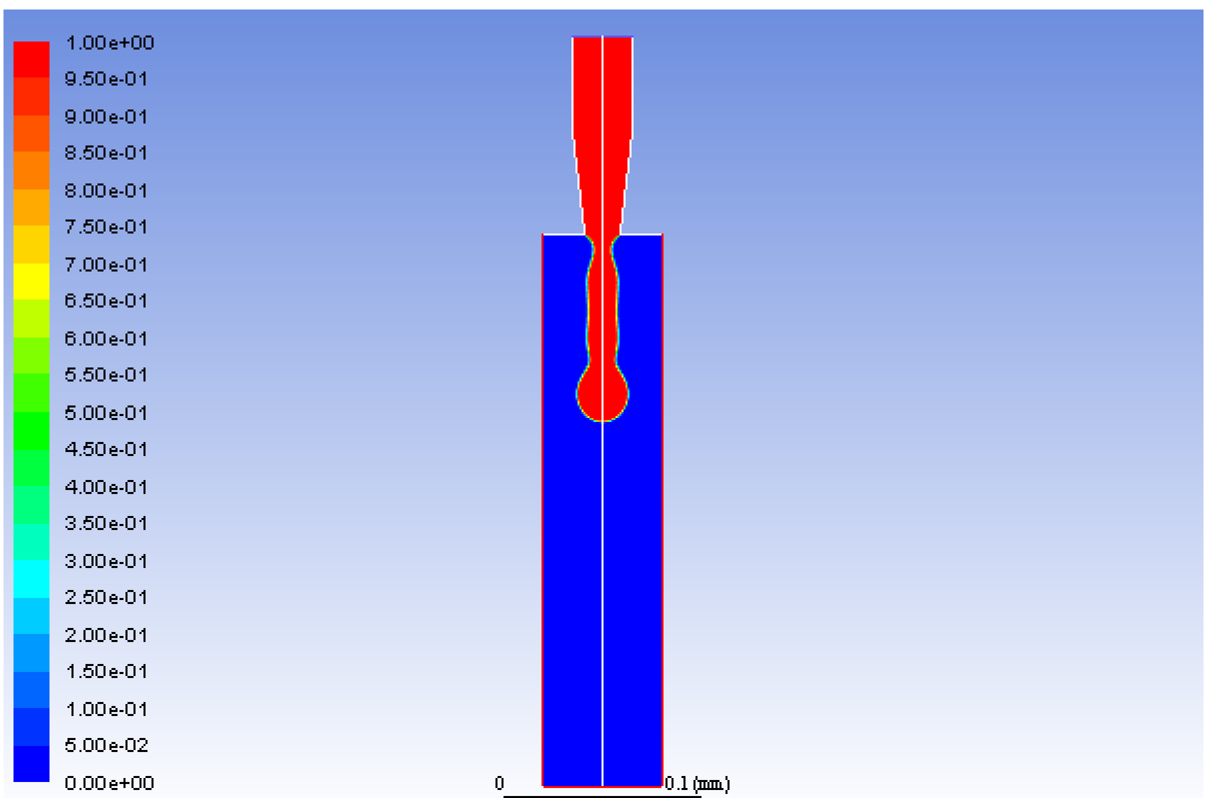


Figure 17.7: Contours of Water Volume Fraction After 18 μs

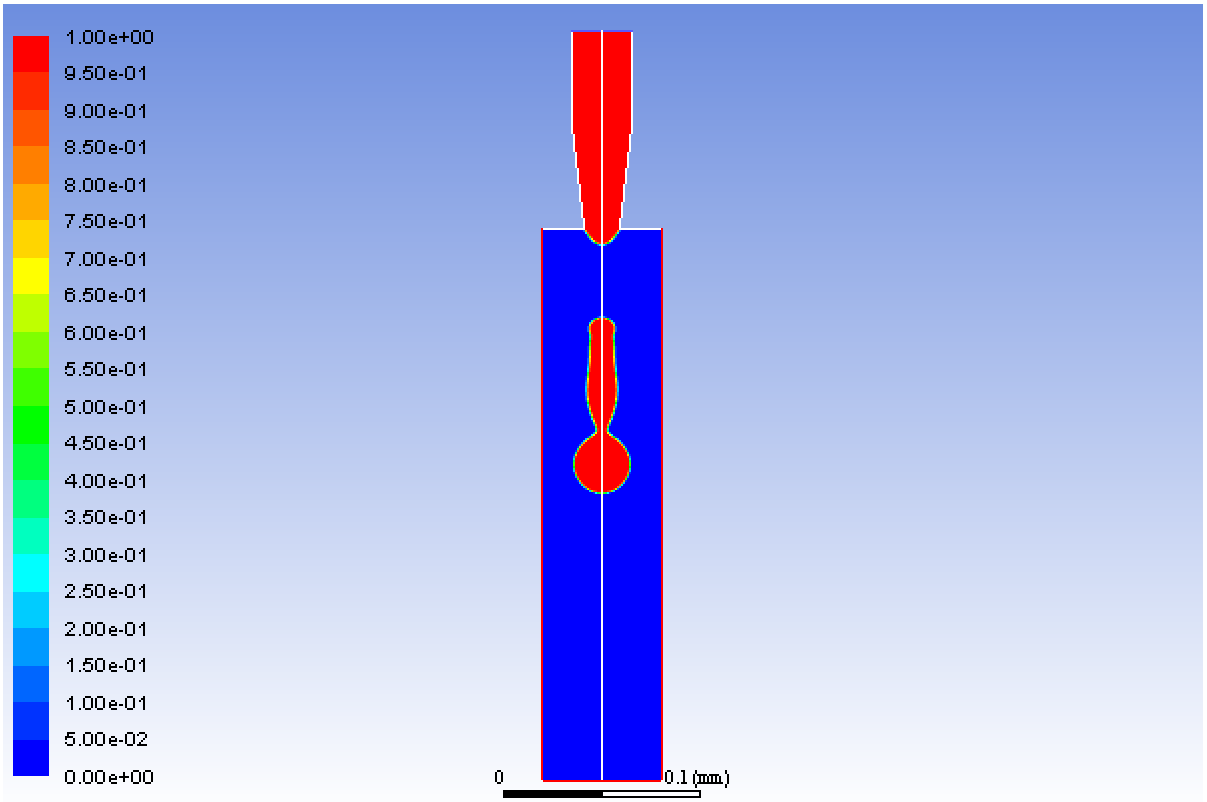


Figure 17.8: Contours of Water Volume Fraction After 24 μs

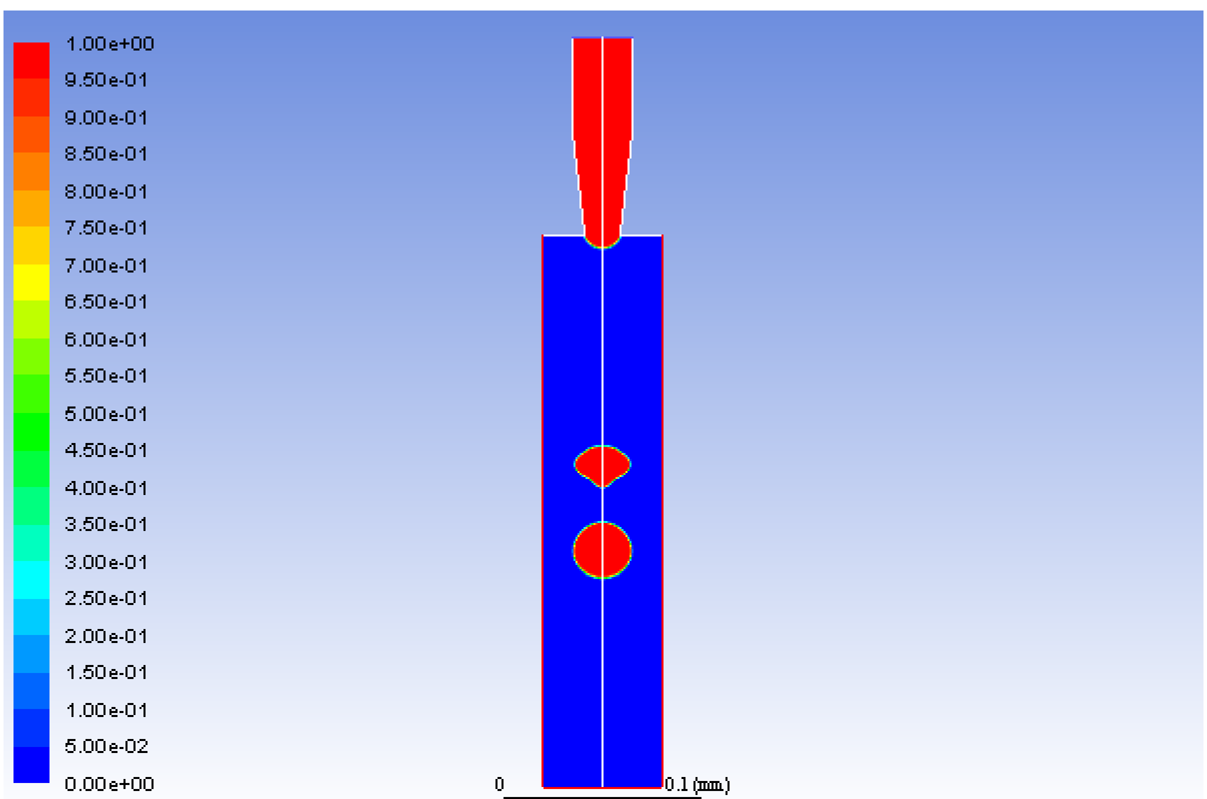
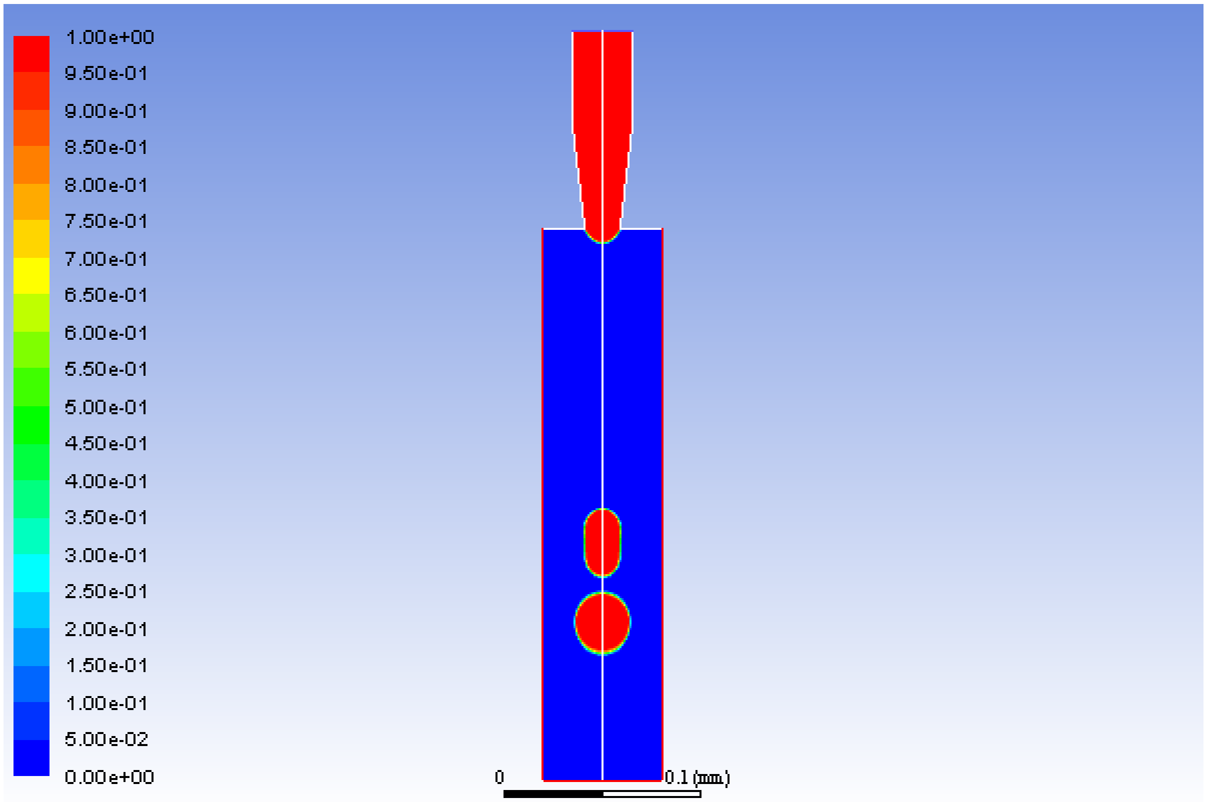


Figure 17.9: Contours of Water Volume Fraction After 30 μs



17.5. Summary

This tutorial demonstrated the application of the volume of fluid method with surface tension effects. The problem involved the 2D axisymmetric modeling of a transient liquid-gas interface, and postprocessing showed how the position and shape of the surface between the two immiscible fluids changed over time.

For additional details about VOF multiphase flow modeling, see the Fluent Theory Guide.

17.6. Further Improvements

This tutorial guides you through the steps to reach an initial solution. You may be able to obtain a more accurate solution by using an appropriate higher-order discretization scheme and by adapting the mesh. Mesh adaption can also ensure that the solution is independent of the mesh. These steps are demonstrated in Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121).

Chapter 18: Modeling Cavitation

This tutorial is divided into the following sections:

18.1. Introduction

This tutorial examines the pressure-driven cavitating flow of water through a sharp-edged orifice. This is a typical configuration in fuel injectors, and brings a challenge to the physics and numerics of cavitation models because of the high pressure differentials involved and the high ratio of liquid to vapor density. Using the multiphase modeling capability of ANSYS Fluent, you will be able to predict the strong cavitation near the orifice after flow separation at a sharp edge.

This tutorial demonstrates how to do the following:

- Set boundary conditions for internal flow.
- Use the mixture model with cavitation effects.
- Calculate a solution using the pressure-based coupled solver.

18.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

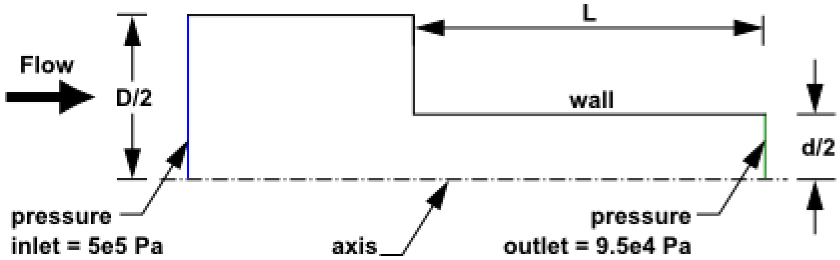
and that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

18.3. Problem Description

The problem considers the cavitation caused by the flow separation after a sharp-edged orifice. The flow is pressure driven, with an inlet pressure of 5×10^5 Pa and an outlet pressure of 9.5×10^4 Pa. The orifice diameter is 4×10^{-3} m, and the geometrical parameters of the orifice are D/d = 2.88 and L/d =

4, where D, d, and L are the inlet diameter, orifice diameter, and orifice length respectively. The geometry of the orifice is shown in Figure 18.1: Problem Schematic (p. 748).

Figure 18.1: Problem Schematic



18.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

18.4.1. Preparation

To prepare for running this tutorial:

1. Set up a working folder on the computer you will be using.

2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

> **Note**
>
> If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

3. Enter the name of this tutorial into the search bar.

4. Narrow the results by using the filter on the left side of the page.

   a. Click **ANSYS Fluent** under **Product**.

   b. Click **18.0** under **Version**.

5. Select this tutorial from the list.

6. Click the **cavitation_R180.zip** link to download the input files.

7. Unzip the `cavitation_R180.zip` file you downloaded to your working folder.

   The mesh file `cav.msh` can be found in the `cavitation` directory created after unzipping the file.

8. Use the Fluent Launcher to start the **2D** version of ANSYS Fluent.

   Fluent Launcher displays your **Display Options** preferences from the previous session.

   For more information about Fluent Launcher, see starting ANSYS Fluent using the Fluent Launcher in the Fluent Getting Started Guide.

9. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.

10. Ensure that the **Serial** processing option is selected.

11. Enable **Double Precision**.

Note

The double precision solver is recommended for modeling multiphase flows simulation.

18.4.2. Reading and Checking the Mesh

1. Read the mesh file `cav.msh`.

   File → Read → Mesh...

   As ANSYS Fluent reads the mesh file, it will report the progress in the console. You can disregard the warnings about the use of axis boundary conditions, as you will make the appropriate change to the solver settings in the next step.

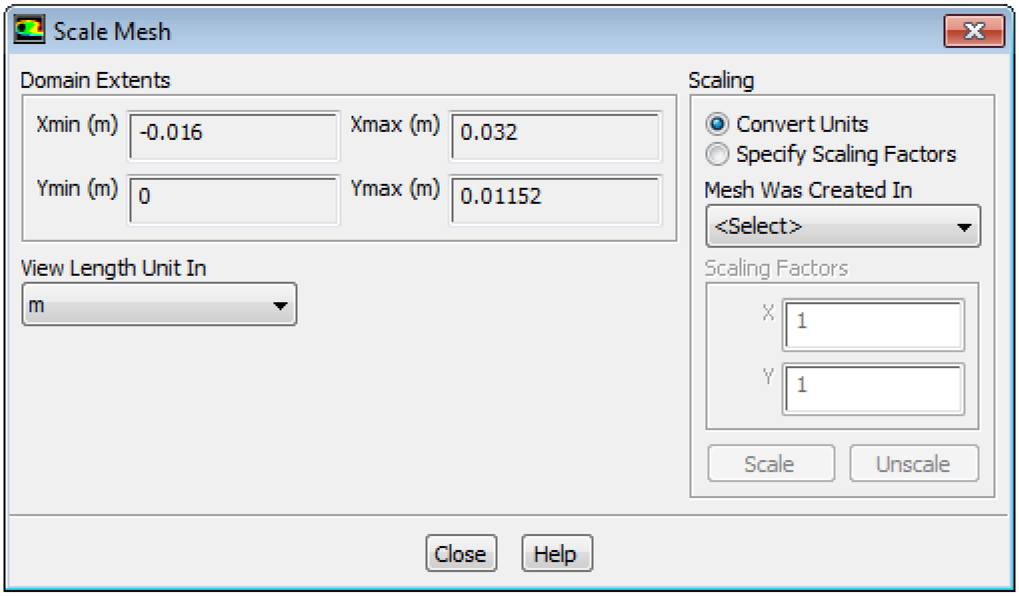
2. Check the mesh.

   Setting Up Domain → Mesh → Check

   ANSYS Fluent will perform various checks on the mesh and will report the progress in the console. Ensure that the reported minimum volume is a positive number.

3. Check the mesh scale.

   Setting Up Domain → Mesh → Transform → Scale...

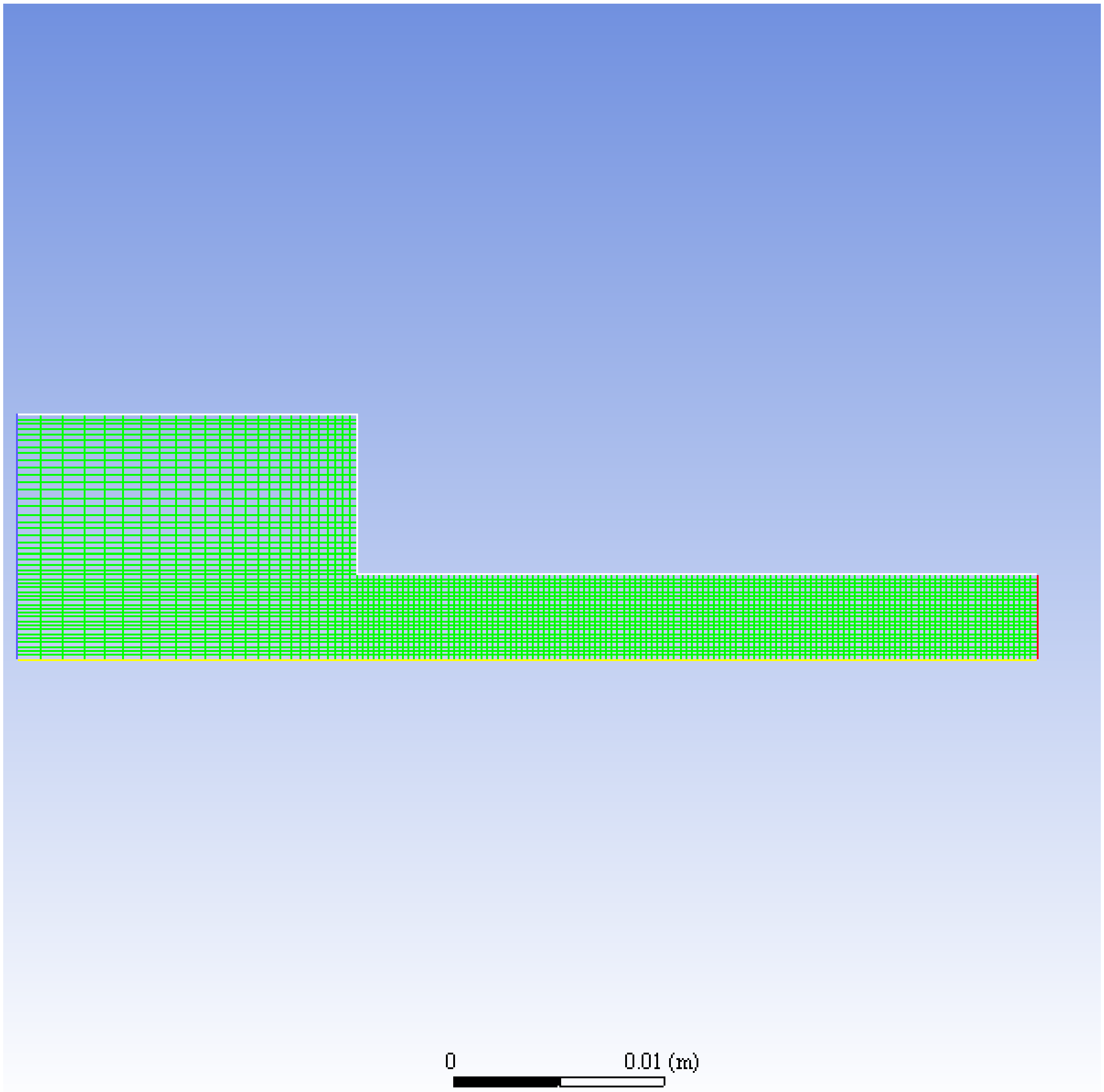


a. Retain the default settings.

b. Close the **Scale Mesh** dialog box.

4. Examine the mesh (Figure 18.2: The Mesh in the Orifice (p. 751)).

Figure 18.2: The Mesh in the Orifice



As seen in Figure 18.2: The Mesh in the Orifice (p. 751), half of the problem geometry is modeled, with an axis boundary (consisting of two separate lines) at the centerline. The quadrilateral mesh is slightly graded in the plenum to be finer toward the orifice. In the orifice, the mesh is uniform with aspect ratios close to 1, as the flow is expected to exhibit two-dimensional gradients.

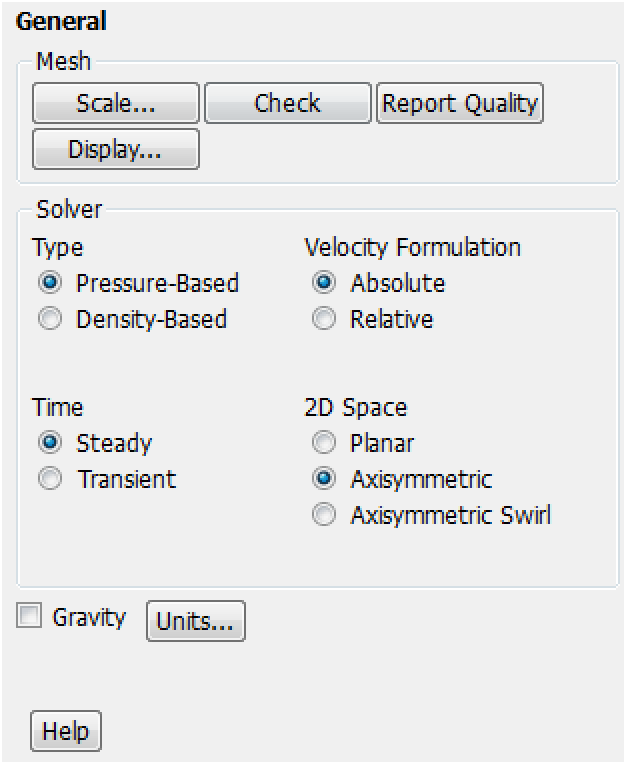
When you display data graphically in a later step, you will mirror the view across the centerline to obtain a more realistic view of the model.

Since the bubbles are small and the flow is high speed, gravity effects can be neglected and the problem can be reduced to axisymmetrical. If gravity could not be neglected and the direction of gravity were not coincident with the geometrical axis of symmetry, you would have to solve a 3D problem.

18.4.3. Solver Settings

1. Specify an axisymmetric model.

Seup → General



a. Retain the default selection of **Pressure-Based** in the **Type** list.

   The pressure-based solver must be used for multiphase calculations.

b. Select **Axisymmetric** in the **2D Space** list.

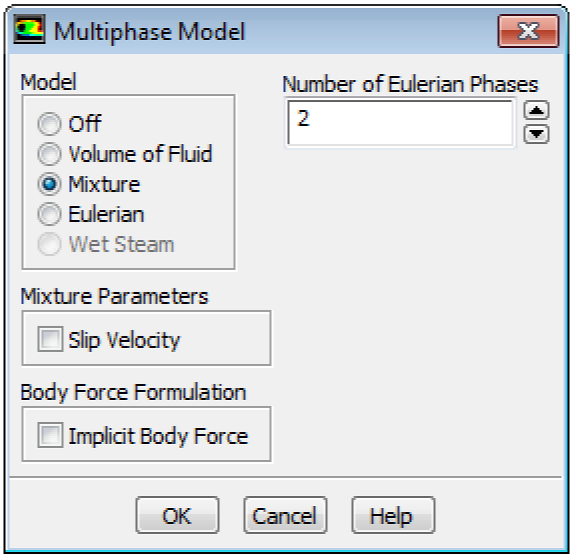
Note

A computationally intensive, transient calculation is necessary to accurately simulate the irregular cyclic process of bubble formation, growth, filling by water jet re-entry, and break-off. In this tutorial, you will perform a steady-state calculation to simulate the presence of vapor in the separation region in the time-averaged flow.

18.4.4. Models

1. Enable the multiphase mixture model.

   Setting Up Physics → Models → Multiphase...



a. Select **Mixture** in the **Model** list.

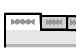
   *The **Multiphase Model** dialog box will expand.*

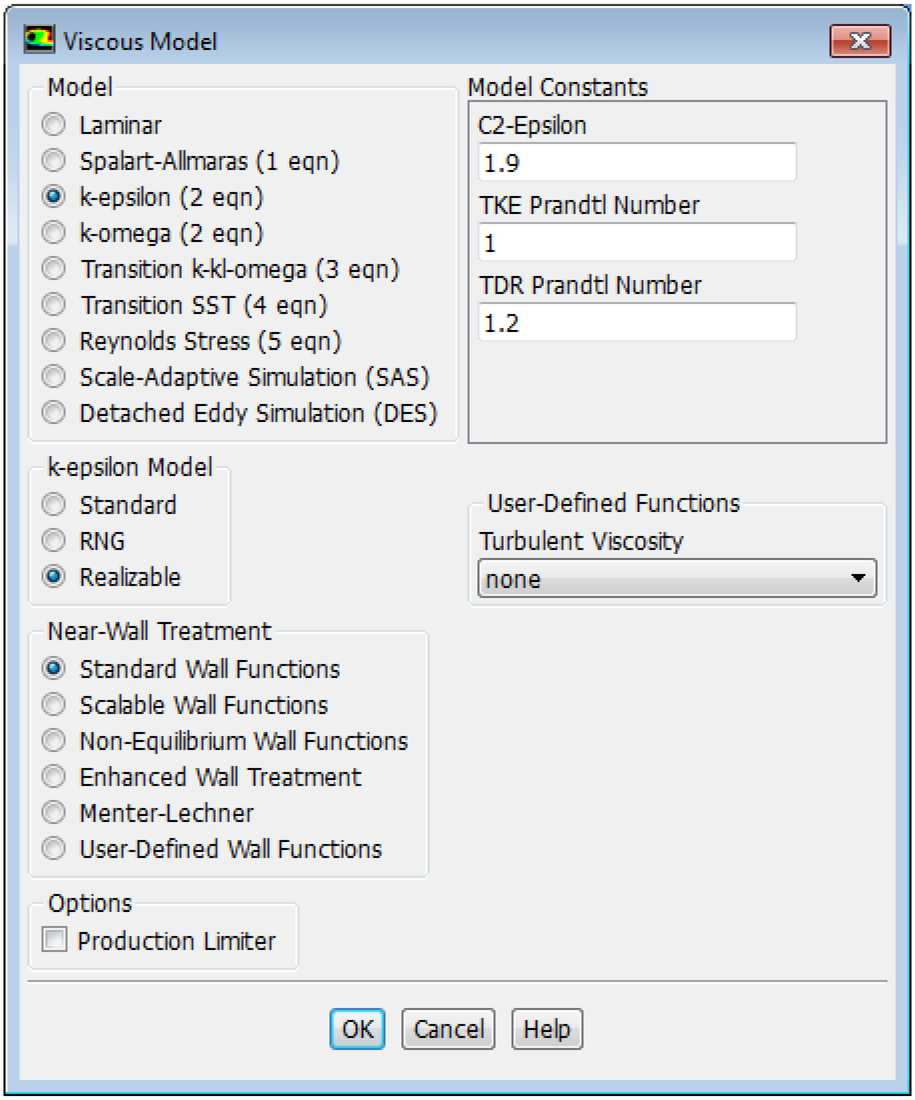
b. Clear **Slip Velocity** in the **Mixture Parameters** group box.

   In this flow, the high level of turbulence does not allow large bubble growth, so gravity is not important. It is also assumed that the bubbles have same velocity as the liquid. Therefore, there is no need to solve for the slip velocity.

c. Click **OK** to close the **Multiphase Model** dialog box.

2. Enable the realizable k- ε turbulence model with standard wall functions.

    **Setting Up Physics → Models → Viscous...**



a. Select **k-epsilon (2 eqn)** in the **Model** list.

b. Select **Realizable** in the **k-epsilon Model** list.

c. Retain the default of **Standard Wall Functions** in the **Near-Wall Treatment** list.

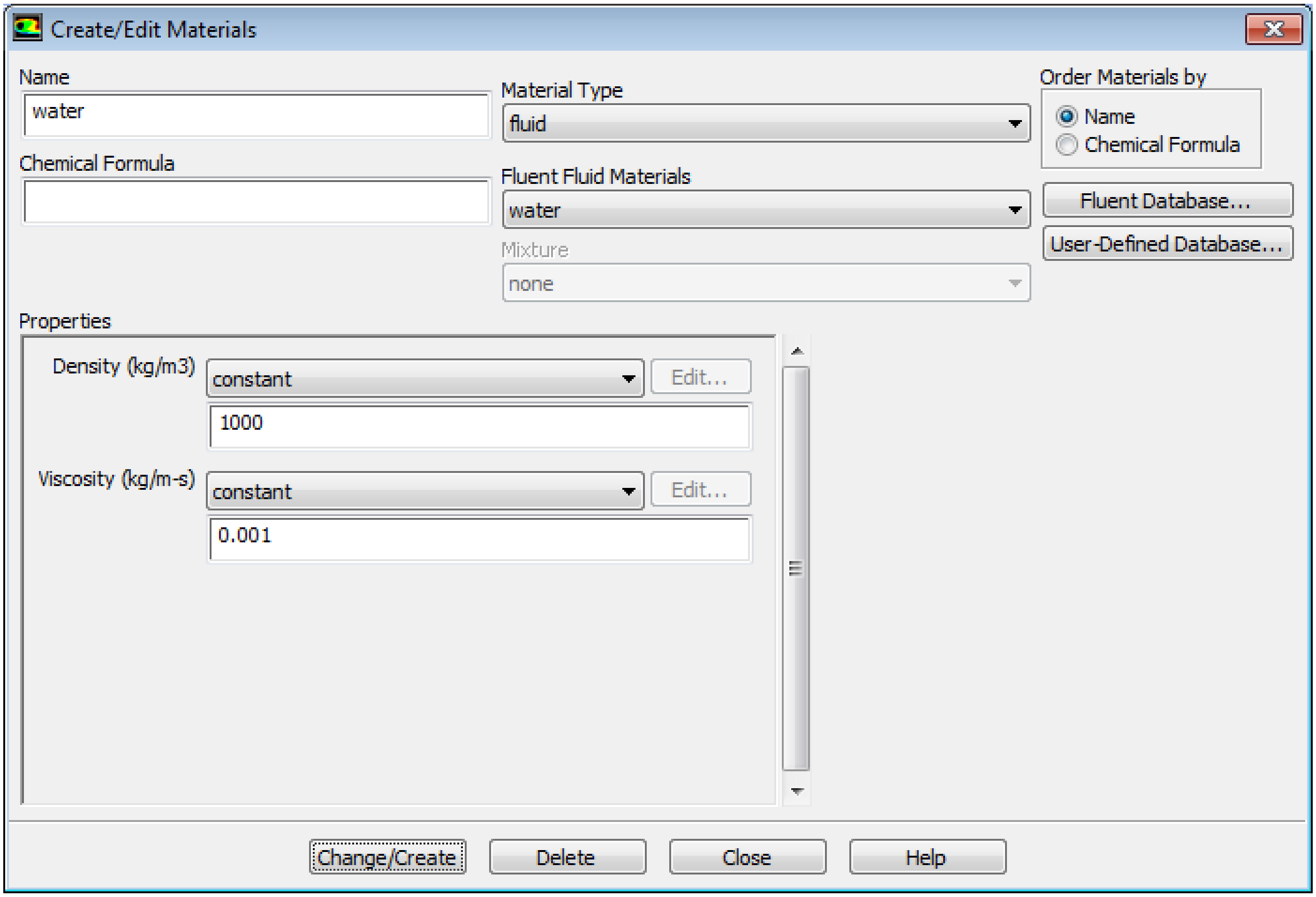
d. Click **OK** to close the **Viscous Model** dialog box.

18.4.5. Materials

For the purposes of this tutorial, you will be modeling the liquid and vapor phases as incompressible. Note that more comprehensive models are available for the densities of these phases, and could be used to more fully capture the affects of the pressure changes in this problem.

1. Create a new material to be used for the primary phase.

Setup → **Materials** → **Fluid** **New...**



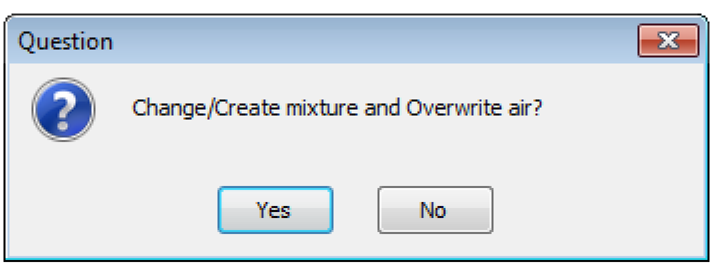
a. Enter `water` for **Name**.

b. Enter `1000` kg/m^3 for **Density**.

c. Enter `0.001` kg/m–s for **Viscosity**.

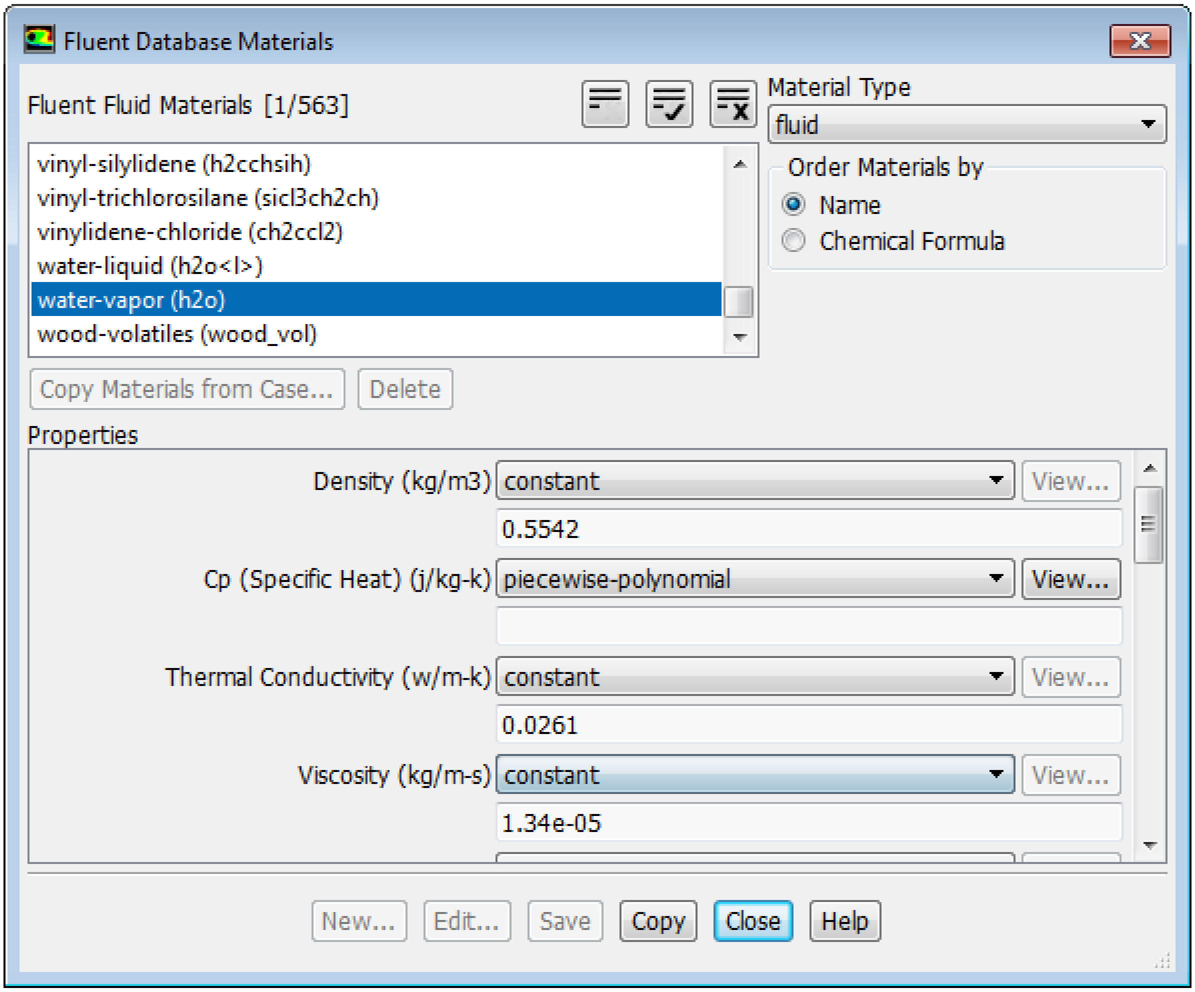
d. Click **Change/Create**.

*A **Question** dialog box will open, asking if you want to overwrite **air**. Click **Yes**.*



2. Copy water vapor from the materials database and modify the properties of your local copy.

   a. In the **Create/Edit Materials** dialog box, click the **Fluent Database...** button to open the **Fluent Database Materials** dialog box.



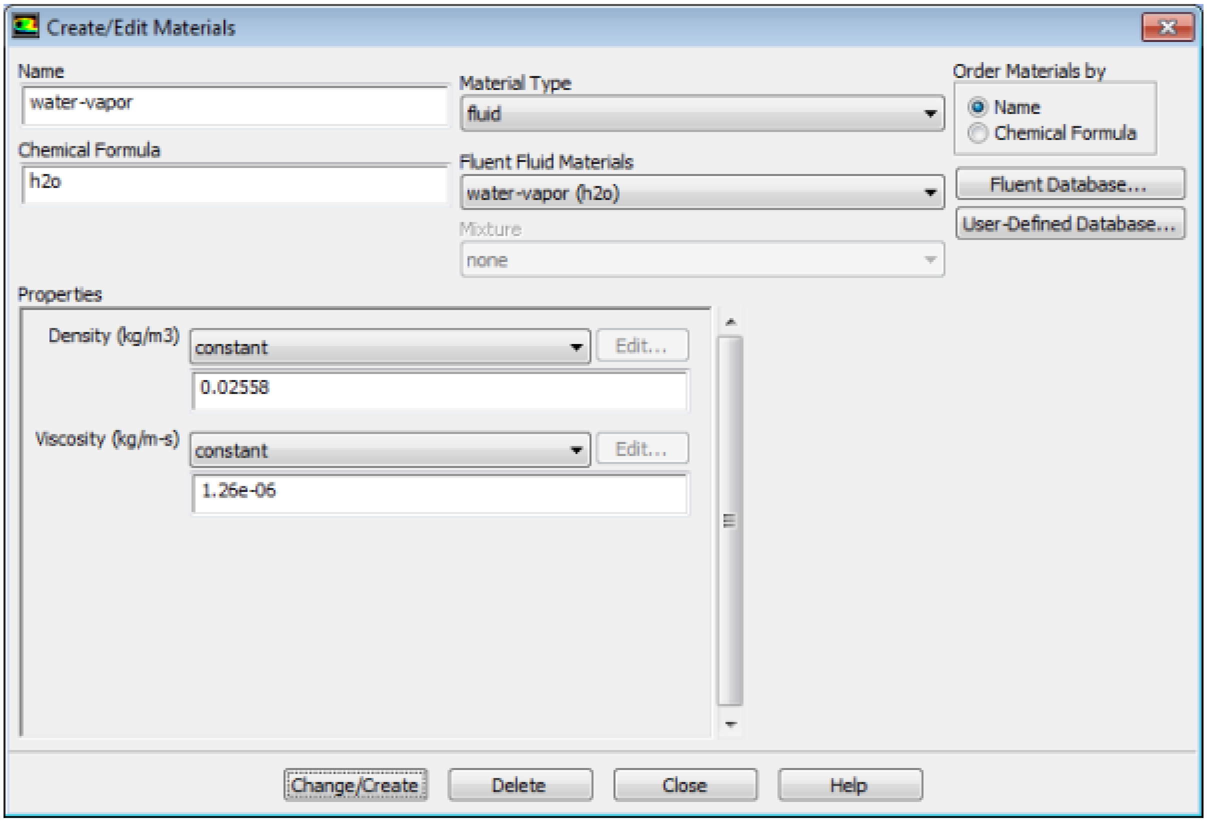
i. Select **water-vapor (h2o)** from the **Fluent Fluid Materials** selection list.

   *Scroll down the list to find **water-vapor (h2o)**.*

ii. Click **Copy** to include water vapor in your model.

   water-vapor appears under **Fluid** in the **Materials** task page

iii. Close the **Fluent Database Materials** dialog box.



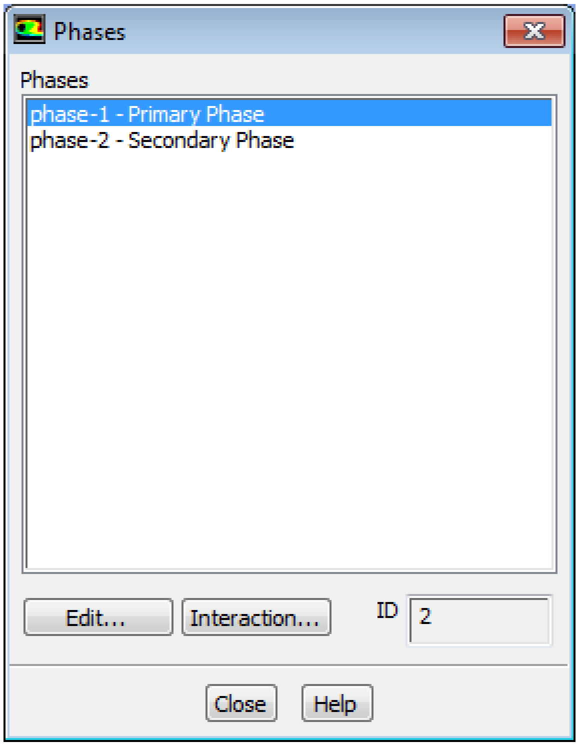
b. Enter `0.02558` kg/m^3 for **Density**.

c. Enter `1.26e-06` kg/m–s for **Viscosity**.

d. Click **Change/Create** and close the **Create/Edit Materials** dialog box.

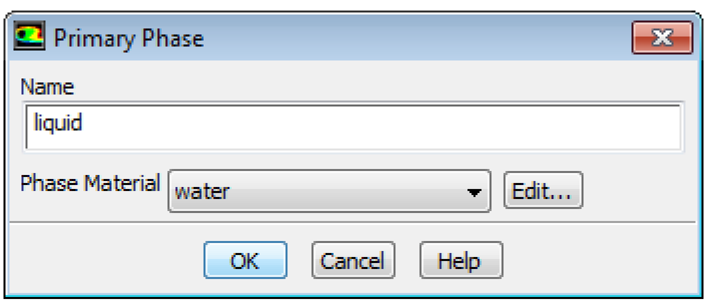
18.4.6. Phases

Setup → **Models** → **Multiphase (Mixture)** → **Phases** **Edit...**



1. Specify liquid water as the primary phase.

   a. In the **Phases** dialog box, select **phase 1 – Primary Phase** and click **Edit...** to open the **Primary Phase** dialog box.

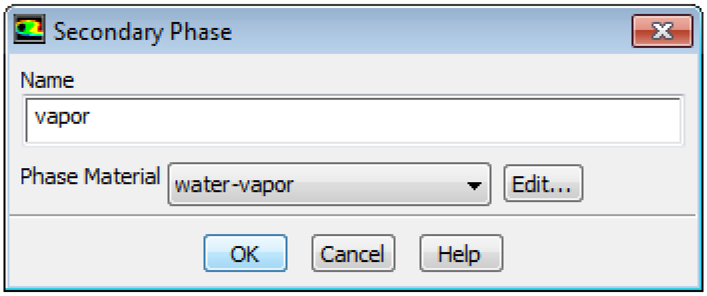
   b. Enter `liquid` for **Name**.

   c. Retain the default selection of **water** from the **Phase Material** drop-down list.

   d. Click **OK** to close the **Primary Phase** dialog box.

2. Specify water vapor as the secondary phase.

   a. In the **Phases** dialog box, select **phase 2 – Secondary Phase** and click **Edit...** to open the **Secondary Phase** dialog box.



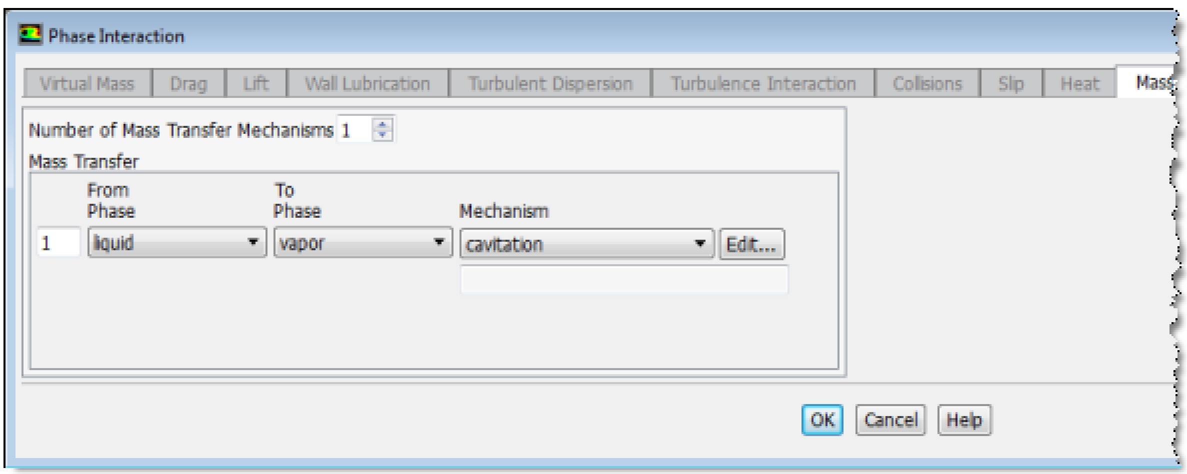
b. Enter `vapor` for **Name**.

c. Select **water-vapor** from the **Phase Material** drop-down list.

d. Click **OK** to close the **Secondary Phase** dialog box.

3. Enable the cavitation model.

a. In the **Phases** dialog box, click **Interaction...** to open the **Phase Interaction** dialog box.



b. In the **Mass** tab, set **Number of Mass Transfer Mechanisms** to 1.

The dialog box expands to show relevant modeling parameters.

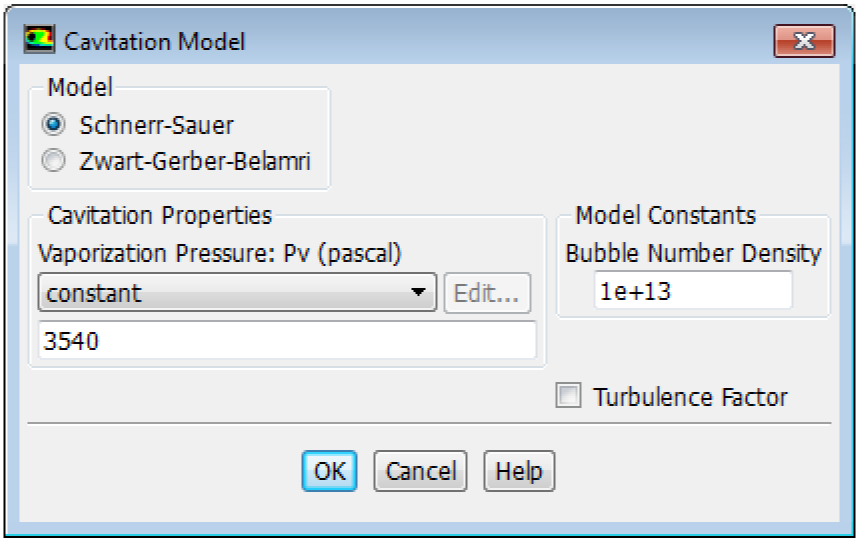
c. Click **OK** in the dialog box that appears.

d. Ensure that **liquid** is selected from the **From Phase** drop-down list in the **Mass Transfer** group box.

e. Select **vapor** from the **To Phase** drop-down list.

f. Select **cavitation** from the **Mechanism** drop-down list.

*The **Cavitation Model** dialog box will open to show the cavitation inputs.*



i. Retain the default settings.

ii. Retain the value of `3540` Pa for **Vaporization Pressure**.

The vaporization pressure is a property of the working liquid, which depends mainly on the temperature and pressure. The default value is the vaporization pressure of water at 1 atmosphere and a temperature of 300 K.

iii. Click **OK** to close the **Cavitation Model** dialog box.

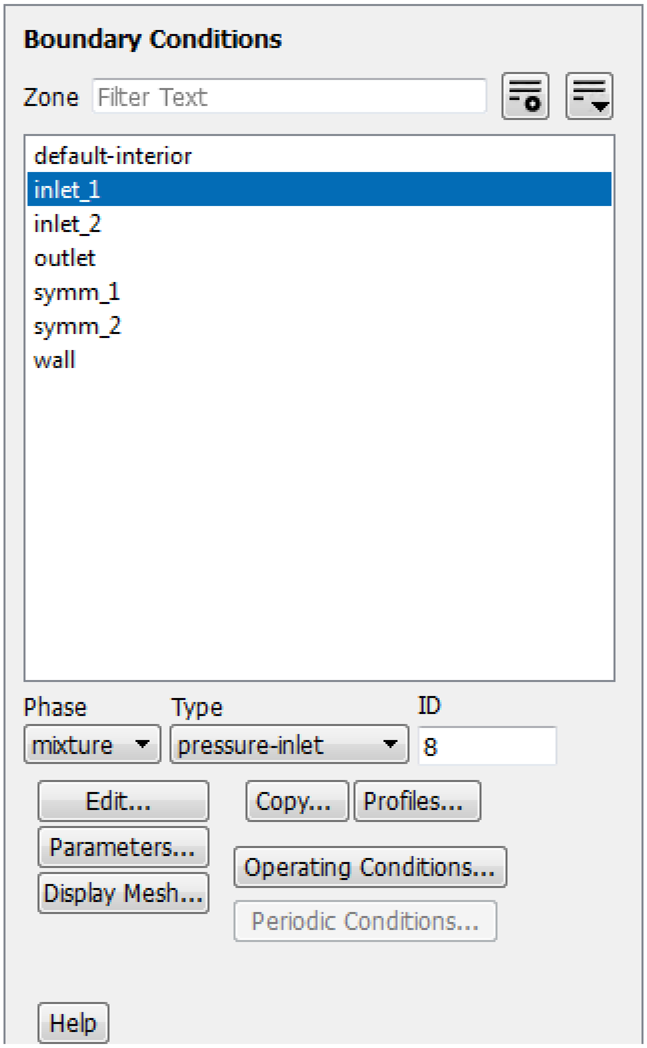
g. Click **OK** to close the **Phase Interaction** dialog box.

4. Close the **Phases** dialog box.

18.4.7. Boundary Conditions

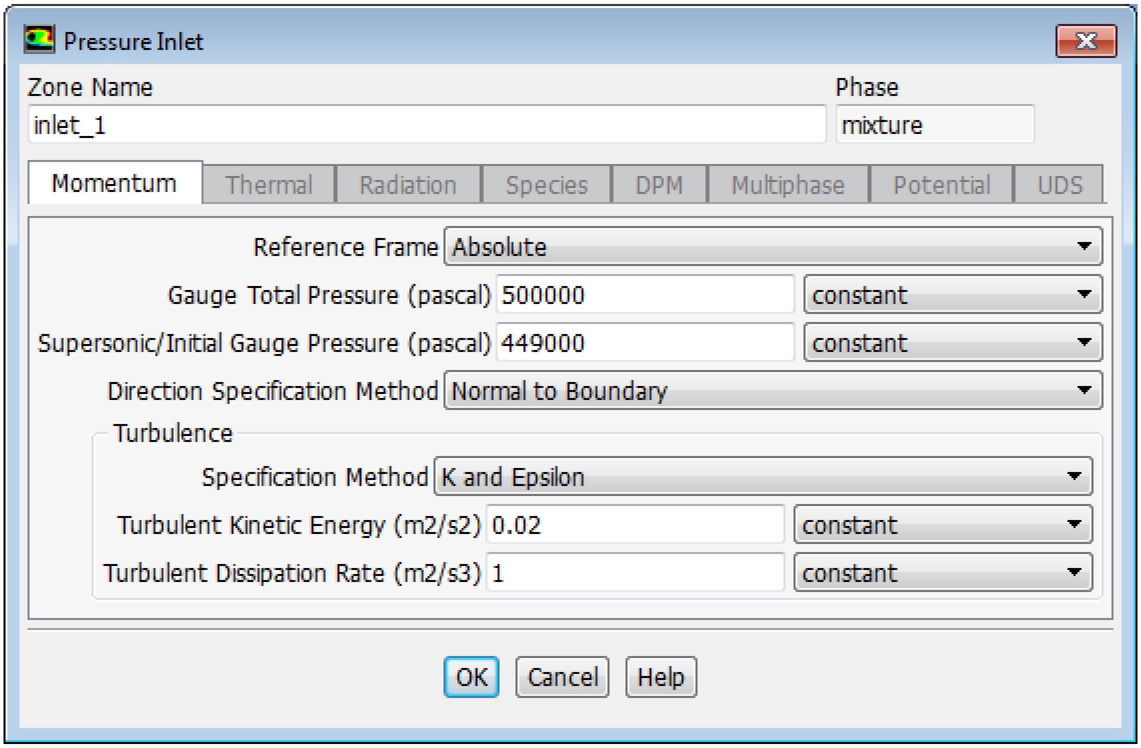
For the multiphase mixture model, you will specify conditions for the mixture (that is, conditions that apply to all phases) and the conditions that are specific to the primary and secondary phases. In this tutorial, boundary conditions are required only for the mixture and secondary phase of two boundaries: the pressure inlet (consisting of two boundary zones) and the pressure outlet. The pressure outlet is the downstream boundary, opposite the pressure inlets.

Setup → **Boundary Conditions**



1. Set the boundary conditions at **inlet_1** for the mixture. Ensure that **mixture** is selected from the **Phase** drop-down list in the **Boundary Conditions** task page.

   Setup → **Boundary Conditions** → **inlet_1** → **Edit...**



a. Enter `500000` Pa for **Gauge Total Pressure**.

b. Enter `449000` Pa for **Supersonic/Initial Gauge Pressure**.

*If you choose to initialize the solution based on the pressure-inlet conditions, the **Supersonic/Initial Gauge Pressure** will be used in conjunction with the specified stagnation pressure (the **Gauge Total Pressure**) to compute initial values according to the isentropic relations (for compressible flow) or Bernoulli's equation (for incompressible flow). Otherwise, in an incompressible flow calculation, ANSYS Fluent will ignore the **Supersonic/Initial Gauge Pressure** input.*

c. Retain the default selection of **Normal to Boundary** from the **Direction Specification Method** drop-down list.

d. Select **K and Epsilon** from the **Specification Method** drop-down list in the **Turbulence** group box.

e. Enter `0.02` m^2/s^2 for **Turbulent Kinetic Energy**.

f. Retain the value of `1` m^2/s^3 for **Turbulent Dissipation Rate**.

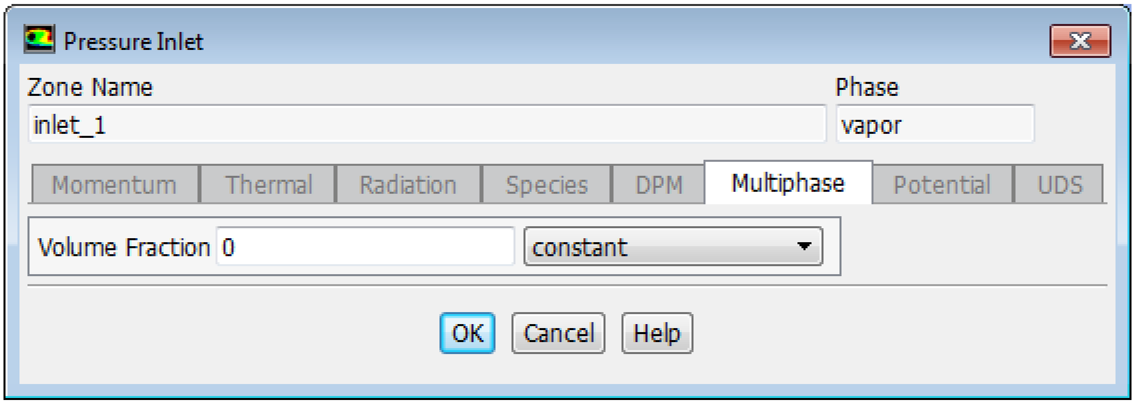
g. Click **OK** to close the **Pressure Inlet** dialog box.

2. Set the boundary conditions at **inlet-1** for the secondary phase.

Setup → **Boundary Conditions** → **inlet_1**

a. Select **vapor** from the **Phase** drop-down list.

b. Click **Edit...** to open the **Pressure Inlet** dialog box.



i. In the **Multiphase** tab, retain the default value of `0` for **Volume Fraction**.

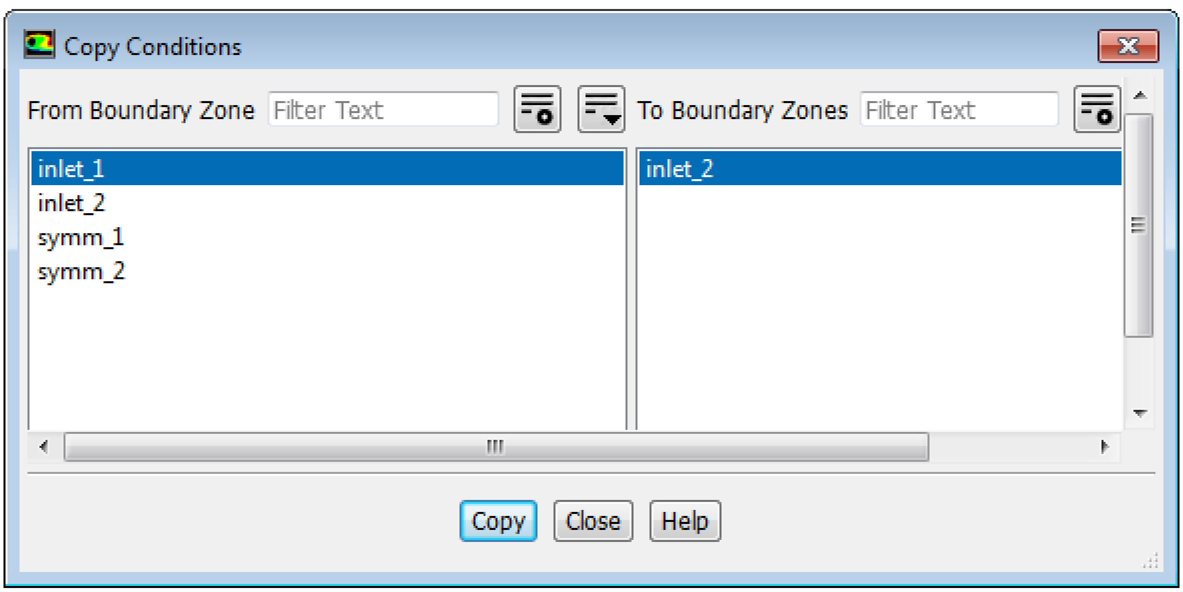
ii. Click **OK** to close the **Pressure Inlet** dialog box.

3. Copy the boundary conditions defined for the first pressure inlet zone (**inlet_1**) to the second pressure inlet zone (**inlet_2**).

Setup → **Boundary Conditions** → **inlet_1**

a. Select **mixture** from the **Phase** drop-down list.

b. Click **Copy...** to open the **Copy Conditions** dialog box.

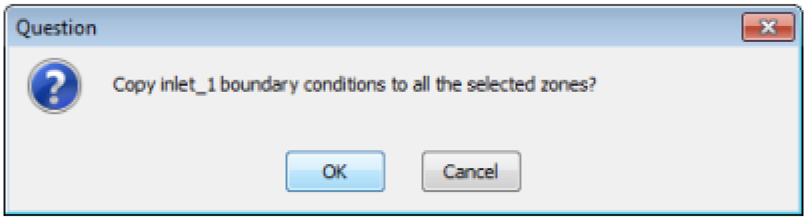


i. Select **inlet_1** from the **From Boundary Zone** selection list.

ii. Select **inlet_2** from the **To Boundary Zones** selection list.

iii. Click **Copy**.

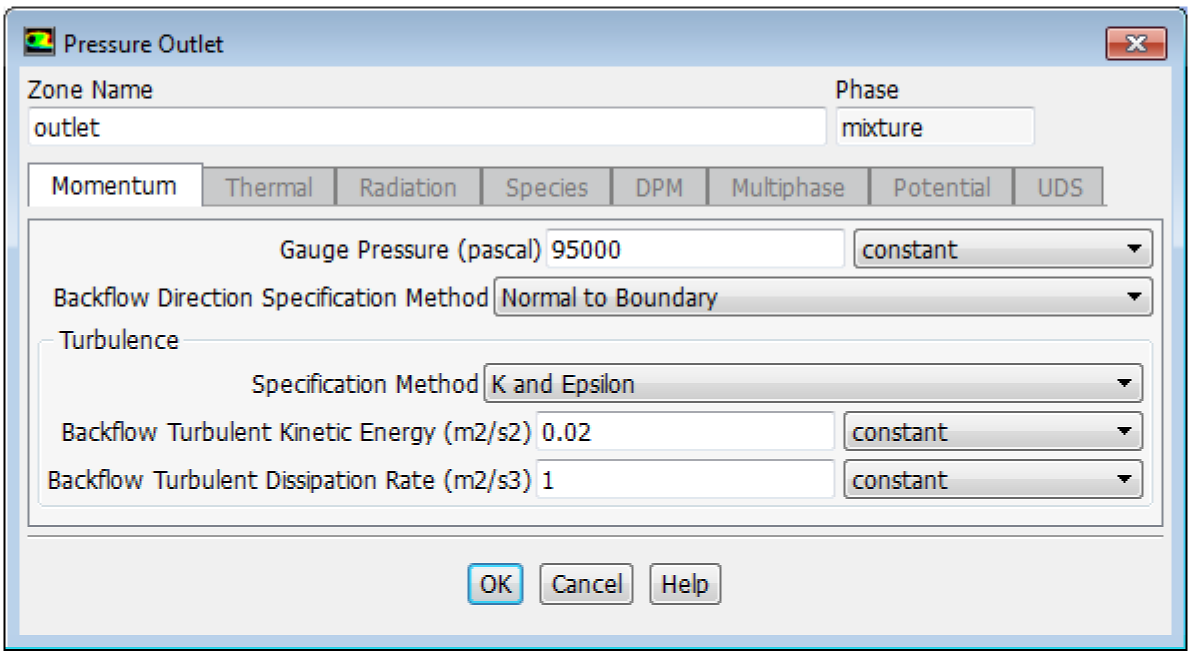
*A **Question** dialog box will open, asking if you want to copy **inlet_1** boundary conditions to **inlet_2**. Click **OK**.*



iv. Close the **Copy Conditions** dialog box.

4. Set the boundary conditions at **outlet** for the mixture.

Setup → **Boundary Conditions** → **outlet** → **Edit...**



a. Enter `95000` Pa for **Gauge Pressure**.

b. Select **K and Epsilon** from the **Specification Method** drop-down list in the **Turbulence** group box.

c. Enter `0.02` m^2/s^2 for **Backflow Turbulent Kinetic Energy**.

d. Retain the value of `1` m^2/s^3 for **Backflow Turbulent Dissipation Rate**.

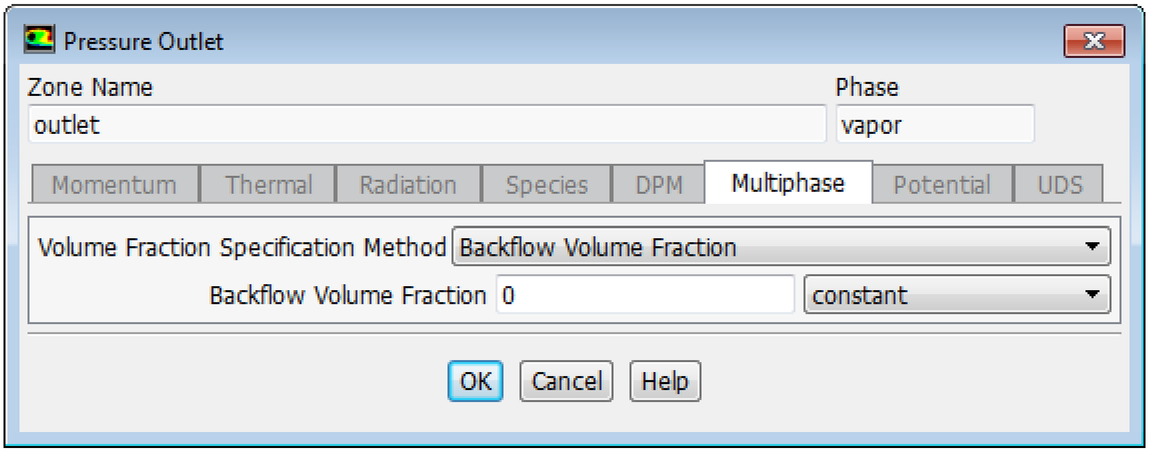
e. Click **OK** to close the **Pressure Outlet** dialog box.

5. Set the boundary conditions at **outlet** for the secondary phase.

Setup → **Boundary Conditions** → **outlet**

a. Select **vapor** from the **Phase** drop-down list.

b. Click **Edit...** to open the **Pressure Outlet** dialog box.



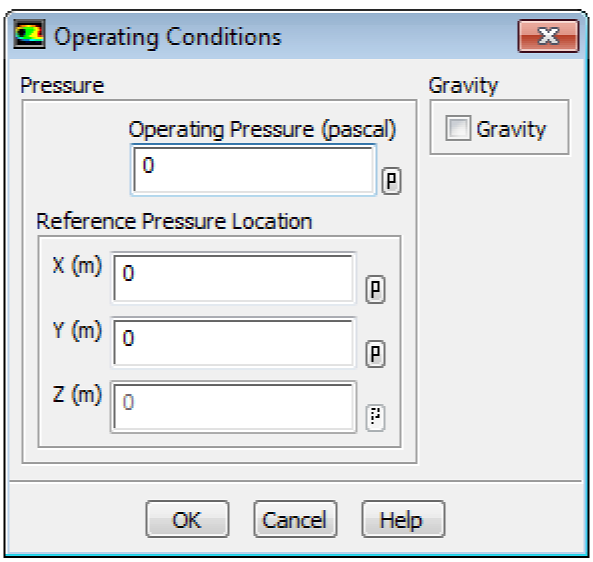
i. In the **Multiphase** tab, retain the default value of `0` for **Backflow Volume Fraction**.

ii. Click **OK** to close the **Pressure Outlet** dialog box.

18.4.8. Operating Conditions

1. Set the operating pressure.

Setup → **Boundary Conditions** → **Operating Conditions...**



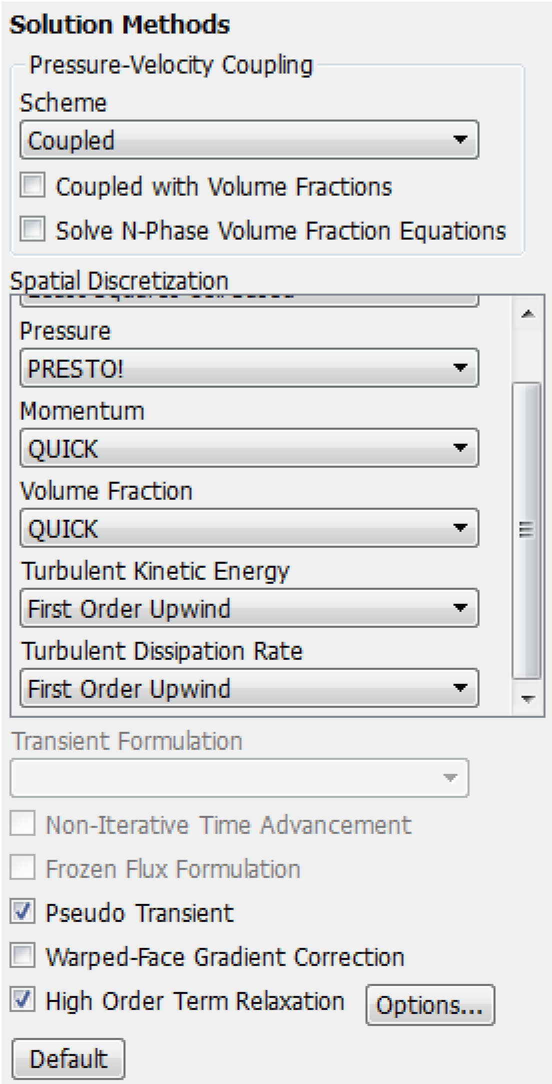
a. Enter `0` Pa for **Operating Pressure**.

b. Click **OK** to close the **Operating Conditions** dialog box.

18.4.9. Solution

1. Set the solution parameters.

Solving → **Solution** → **Methods...**



a. Select **Coupled** from the **Scheme** drop-down list in the **Pressure-Velocity Coupling** group box.

b. Retain the selection of **PRESTO!** from the **Pressure** drop-down list in the **Spatial Discretization** group box.

c. Select **QUICK** for **Momentum** and **Volume Fraction**.

d. Retain **First Order Upwind** for **Turbulent Kinetic Energy** and **Turbulent Dissipation Rate**.

e. Enable **Pseudo Transient**.

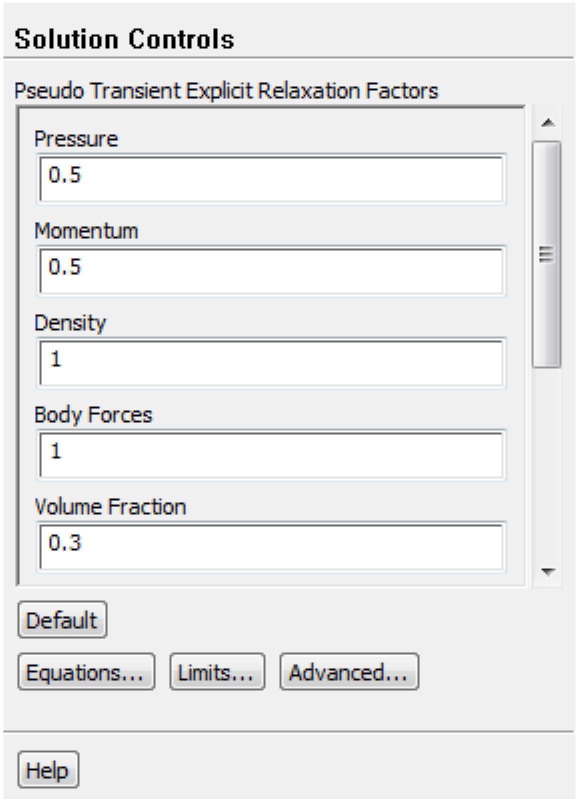
f. Enable **High Order Term Relaxation**.

   The message appears in the console informing you of changing AMG cycle type for Volume Fraction, Turbulent Kinetic Energy, and Turbulent Dissipation Rate to F-cycle.

   The relaxation of high order terms will help to improve the solution behavior of flow simulations when higher order spatial discretizations are used (higher than first).

2. Set the solution controls.

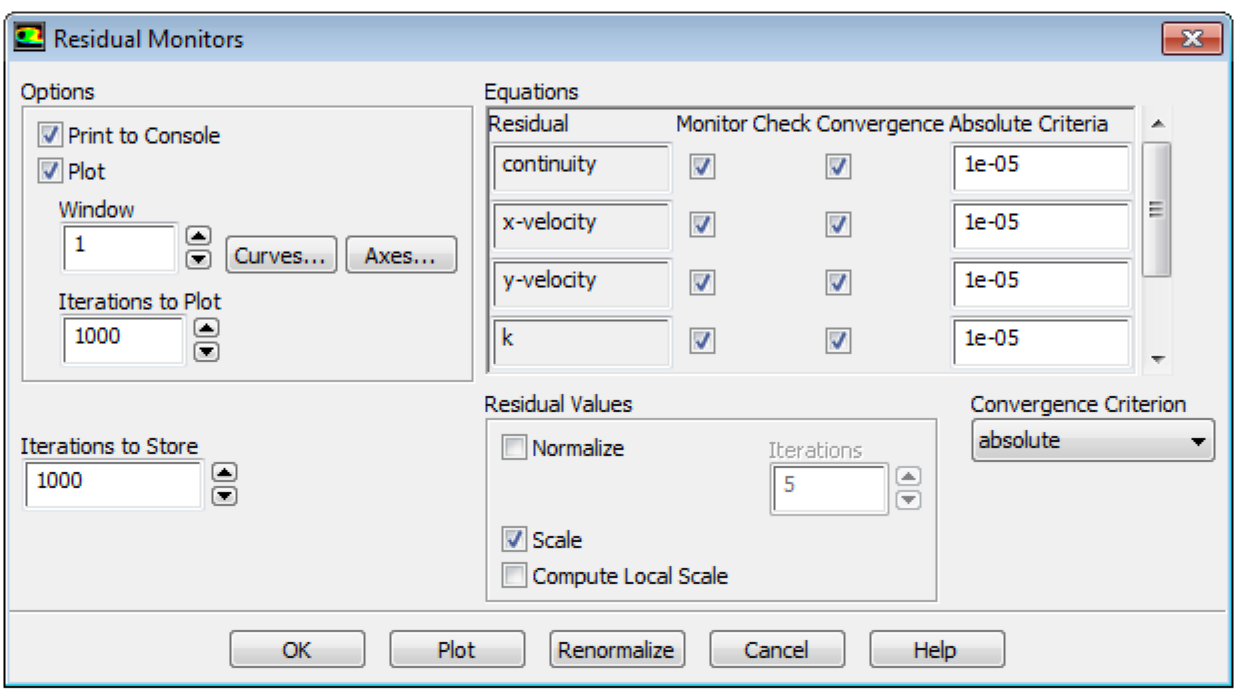
Solving → Controls → Controls...



a. Set the pseudo transient explicit relaxation factor for **Volume Fraction** to 0.3.

3. Enable the plotting of residuals during the calculation.

Solving → Reports → Residuals...



a. Ensure that **Plot** is enabled in the **Options** group box.

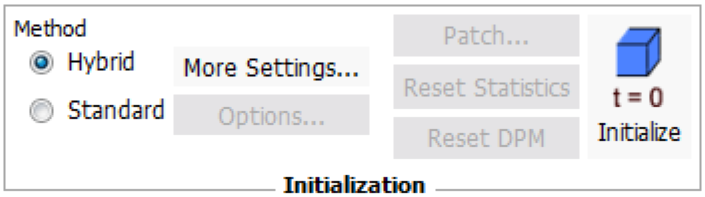
b. Enter `1e-05` for the **Absolute Criteria** of **continuity**, **x-velocity**, **y-velocity**, **k**, and **epsilon**.

Decreasing the criteria for these residuals will improve the accuracy of the solution.

c. Click **OK** to close the **Residual Monitors** dialog box.

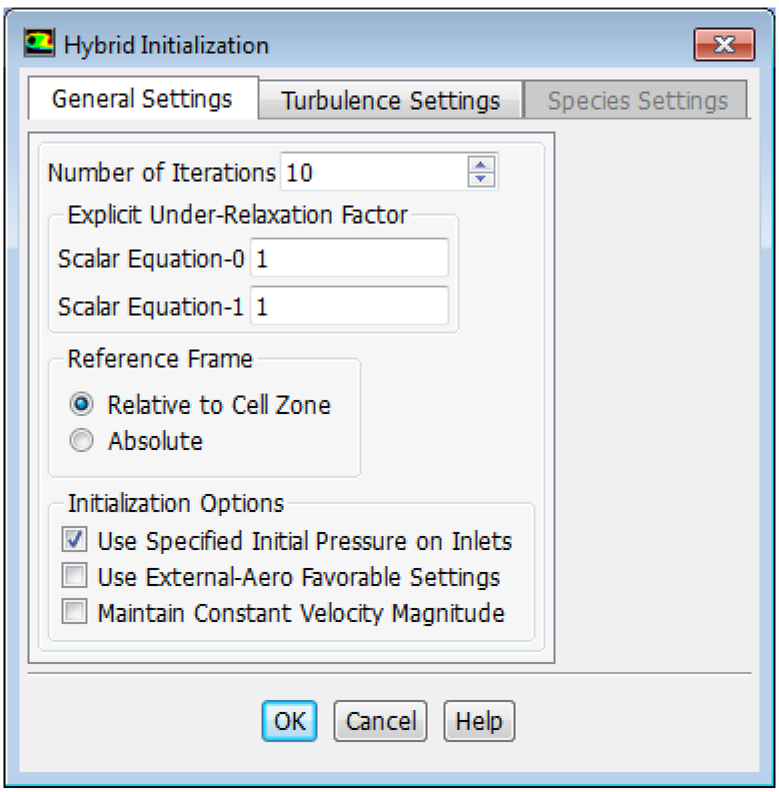
4. Initialize the solution.

Solving → Initialization



a. Select **Hybrid** initialization method from the **Initialization** group.

b. Click **More Settings...** to open the **Hybrid Initialization** dialog box.



c. Enable **Use Specified Initial Pressure on Inlets** in the **Initialization Options** group box. The velocity will now be initialized to the **Initial Gauge Pressure** value that you set in the **Pressure Inlet** boundary condition dialog box. For more information on initialization options, see hybrid initialization in the Fluent User's Guide.

d. Click **OK** to close the **Hybrid Initialization** dialog box.

e. Click **Initialize** to initialize the solution.

Note

For flows in complex topologies, hybrid initialization will provide better initial velocity and pressure fields than standard initialization. This will help to improve the convergence behavior of the solver.

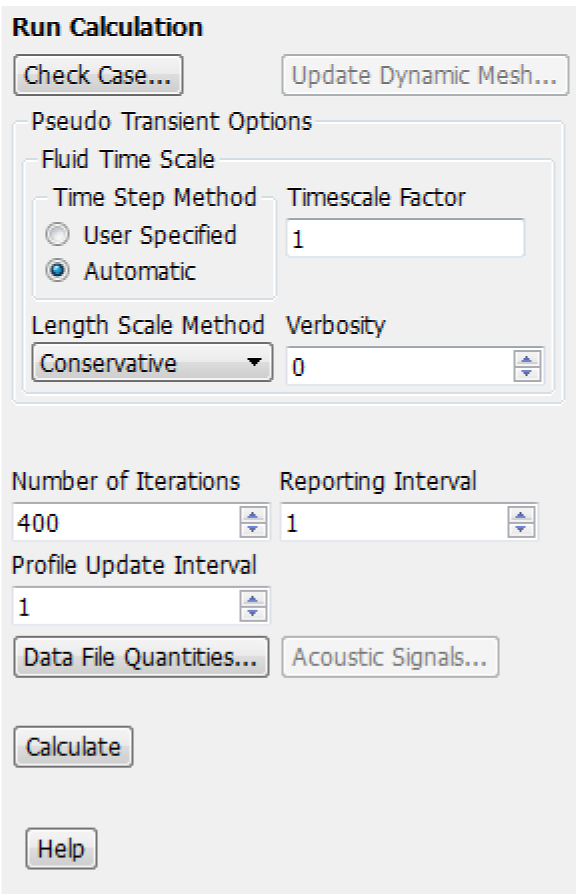
---

5. Save the case file (`cav.cas.gz`).

   File → **Write** → **Case...**

6. Start the calculation by requesting 400 iterations.

   Solving → **Run Calculation** → **Advanced...**



   a. Enter `400` for **Number of Iterations**.

   b. Click **Calculate**.

      The solution will converge in approximately 340 iterations.

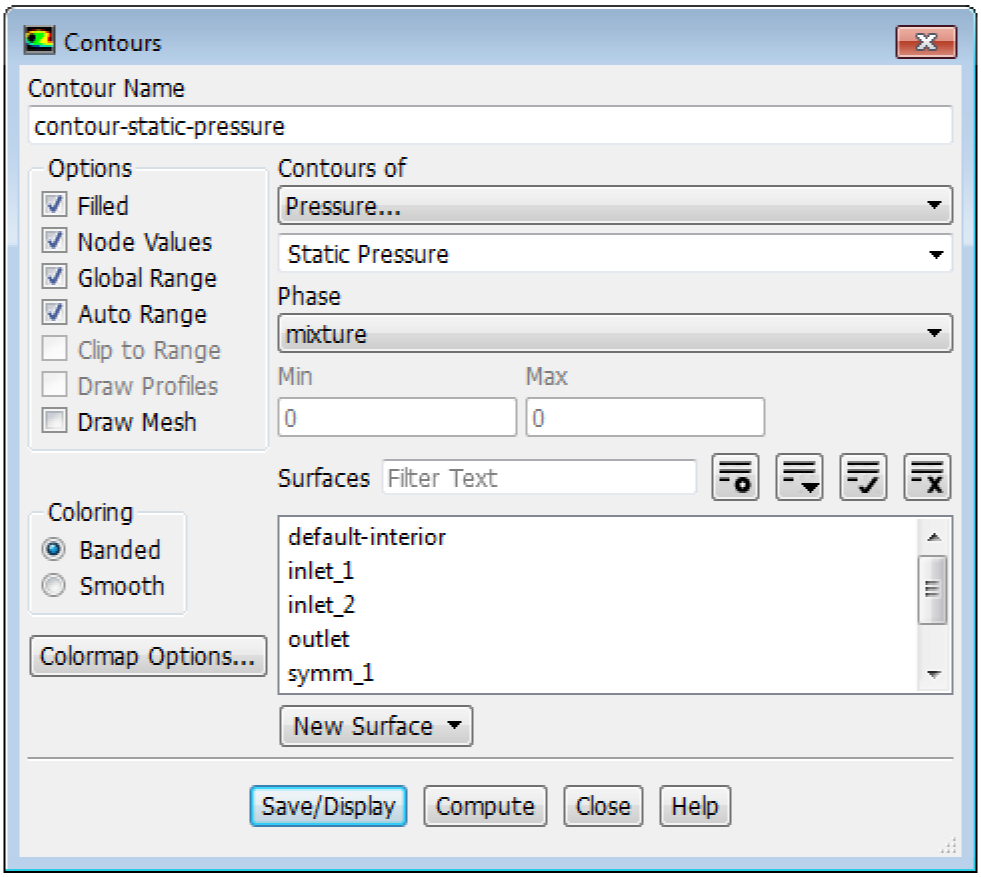
7. Save the data file (`cav.dat.gz`).

File → Write → Data...

18.4.10. Postprocessing

1. Create and plot a definition of pressure contours in the orifice (Figure 18.3: Contours of Static Pressure (p. 771)).

   Postprocessing → Graphics → Contours → New...



   a. Change **Contour Name** to `contour-static-pressure`

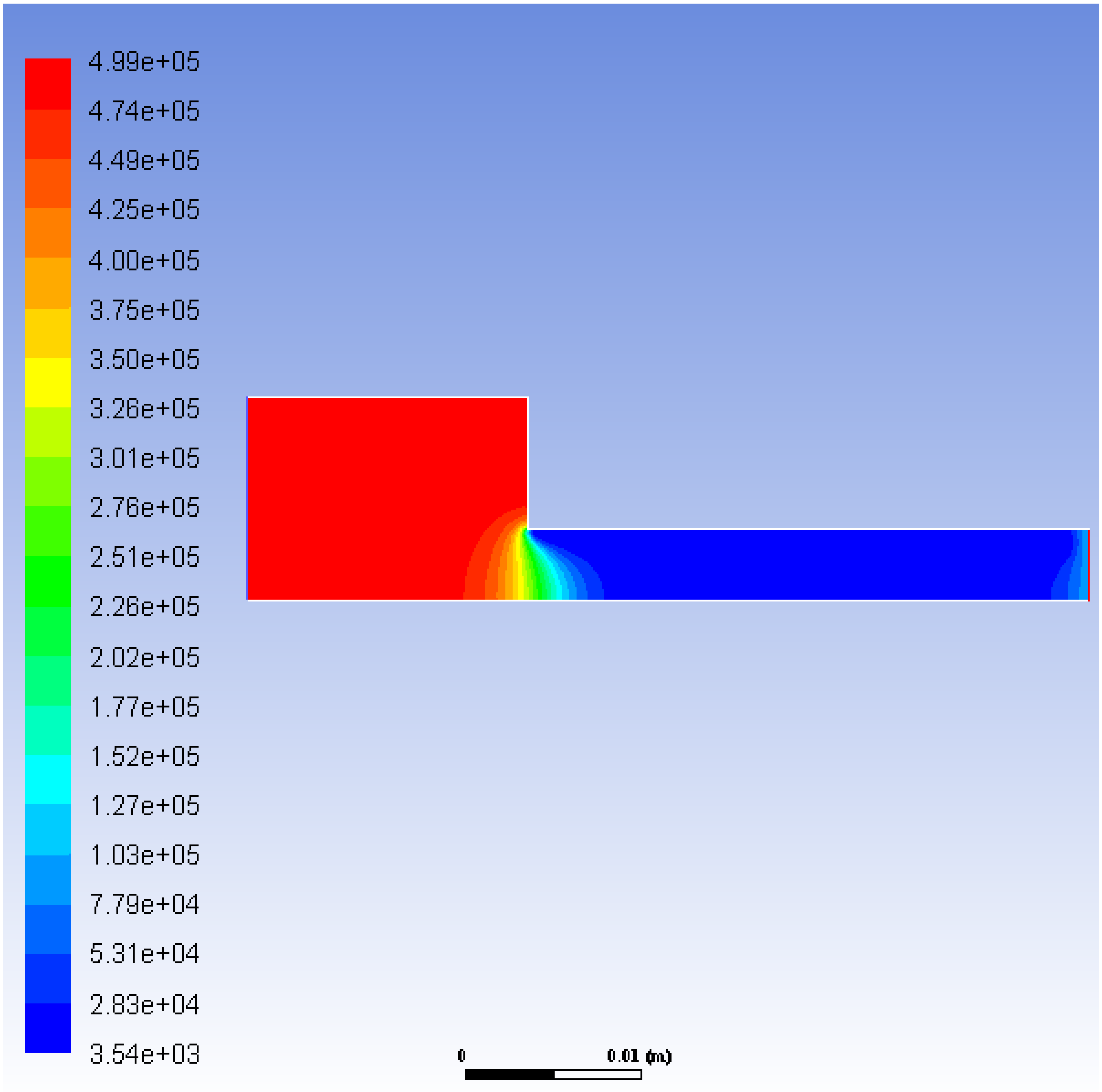
   b. Enable **Filled** in the **Options** group box.

   c. Retain the default selection of **Pressure...** and **Static Pressure** from the **Contours of** drop-down lists.

   d. Click **Save/Display** and close the **Contours** dialog box.

      The `contour-static-pressure` contour definition appears under the **Results/Graphics/Contours** tree branch. Once you create a plot definition, you can use a right-click menu to display this definition at a later time, for instance, in subsequent simulations with different settings ;or in combination with other plot definitions.

Figure 18.3: Contours of Static Pressure

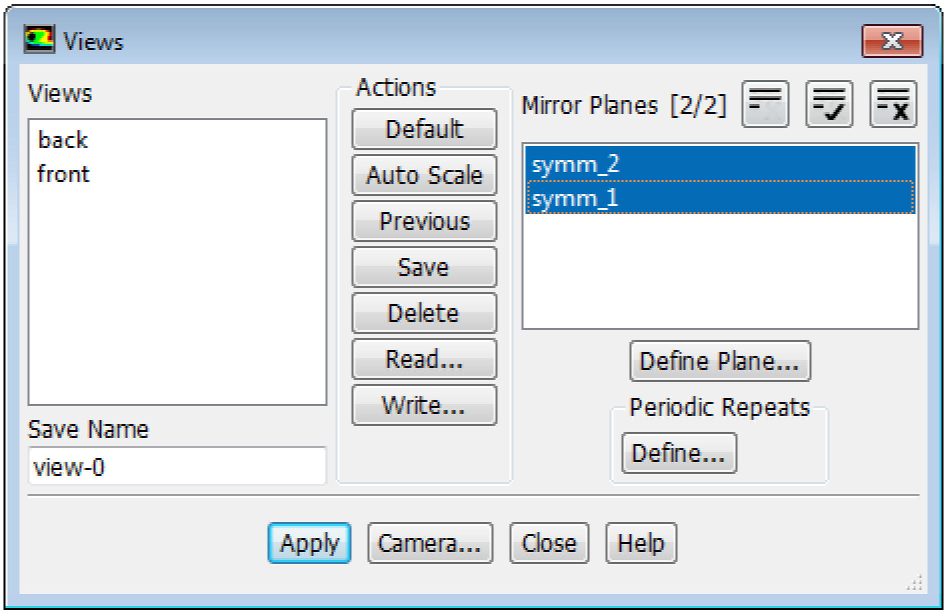


Note the dramatic pressure drop at the flow restriction in Figure 18.3: Contours of Static Pressure (p. 771). Low static pressure is the major factor causing cavitation. Additionally, turbulence contributes to cavitation due to the effect of pressure fluctuation (Figure 18.4: Mirrored View of Contours of Static Pressure (p. 773)) and turbulent diffusion (Figure 18.5: Contours of Turbulent Kinetic Energy (p. 774)).

2. Mirror the display across the centerline (Figure 18.4: Mirrored View of Contours of Static Pressure (p. 773)).

 Viewing → **Display** → **Views...**

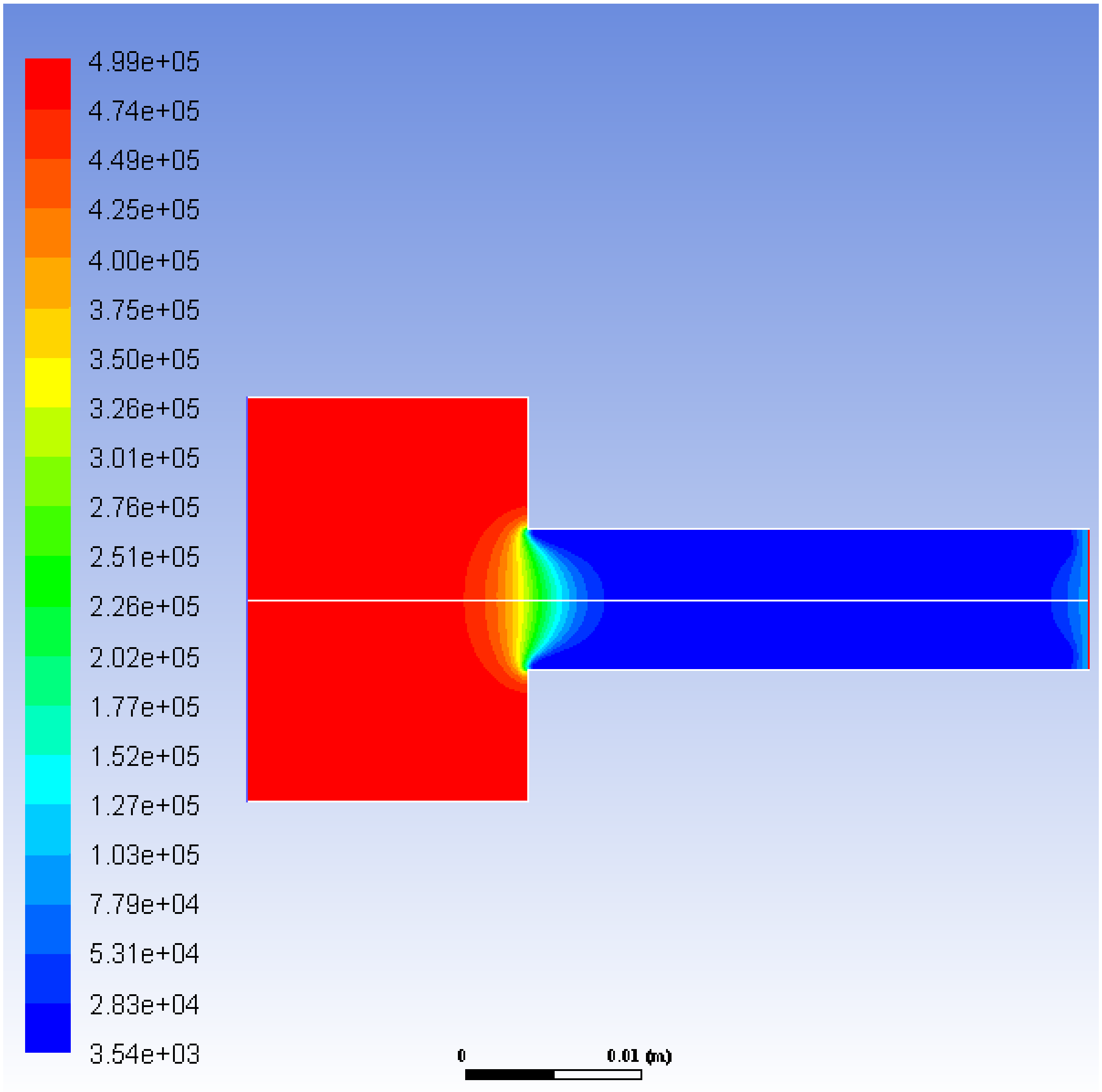
 Mirroring the display across the centerline gives a more realistic view.



a. Select **symm_2** and **symm_1** from the **Mirror Planes** selection list.

b. Click **Apply** and close the **Views** dialog box.

Figure 18.4: Mirrored View of Contours of Static Pressure



3. Create and plot a contour definition of the turbulent kinetic energy (Figure 18.5: Contours of Turbulent Kinetic Energy (p. 774)).

   Postprocessing → **Graphics** → **Contours** → **New...**

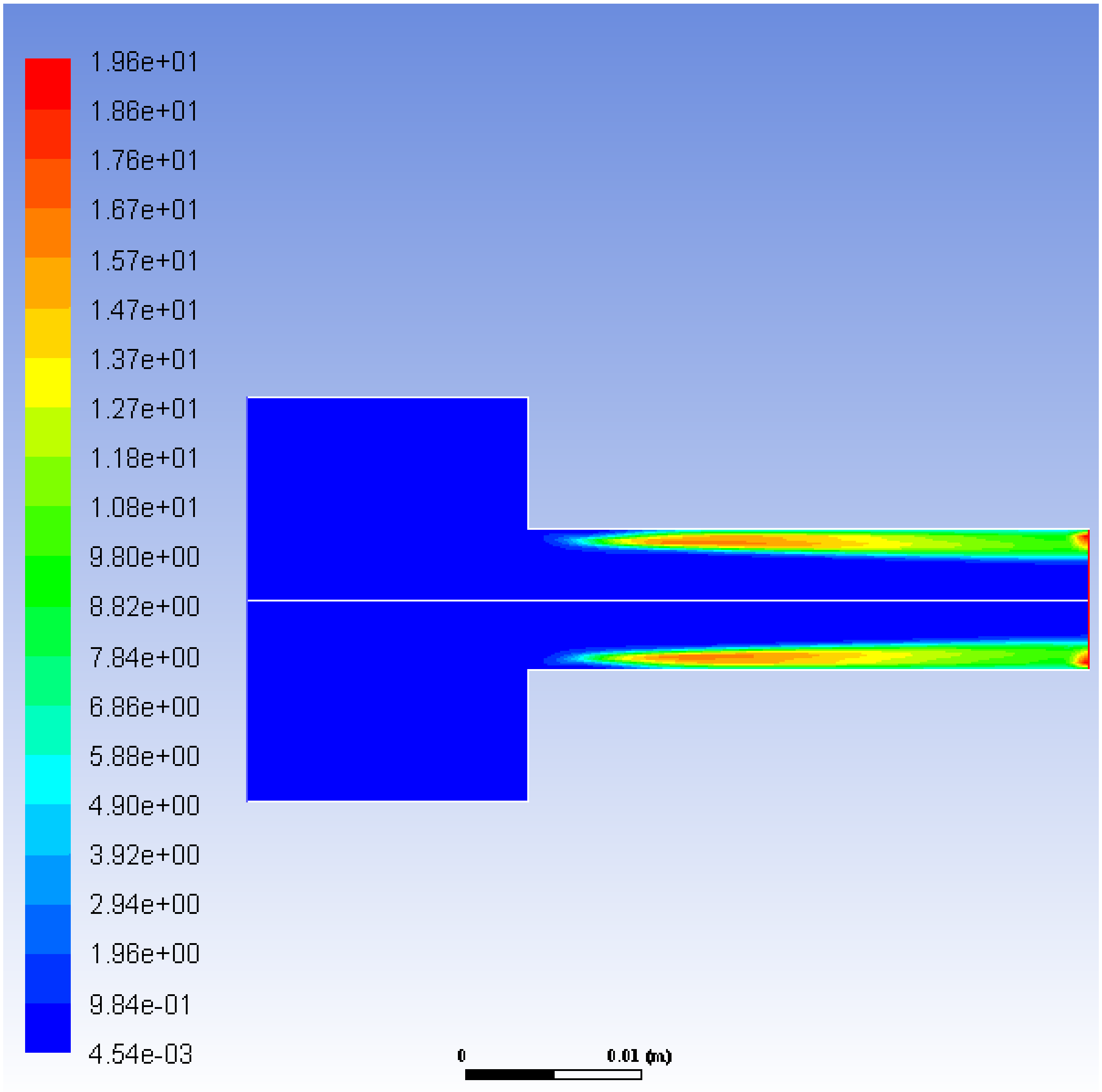
   a. Change **Contour Name** to `contour-tke`

   b. Enable **Filled** in the **Options** group box.

   c. Select **Turbulence...** and **Turbulent Kinetic Energy(k)** from the **Contours of** drop-down lists.

   d. Click **Save/Display**.

Figure 18.5: Contours of Turbulent Kinetic Energy



In this example, the mesh used is fairly coarse. However, in cavitating flows the pressure distribution is the dominant factor, and is not very sensitive to mesh size.

4. Create and plot a contour definition of the volume fraction of water vapor (Figure 18.6: Contours of Vapor Volume Fraction (p. 775)).

   Postprocessing → **Graphics** → **Contours** → **New...**

   a. Change **Contour Name** to `contour-vf-vapor`

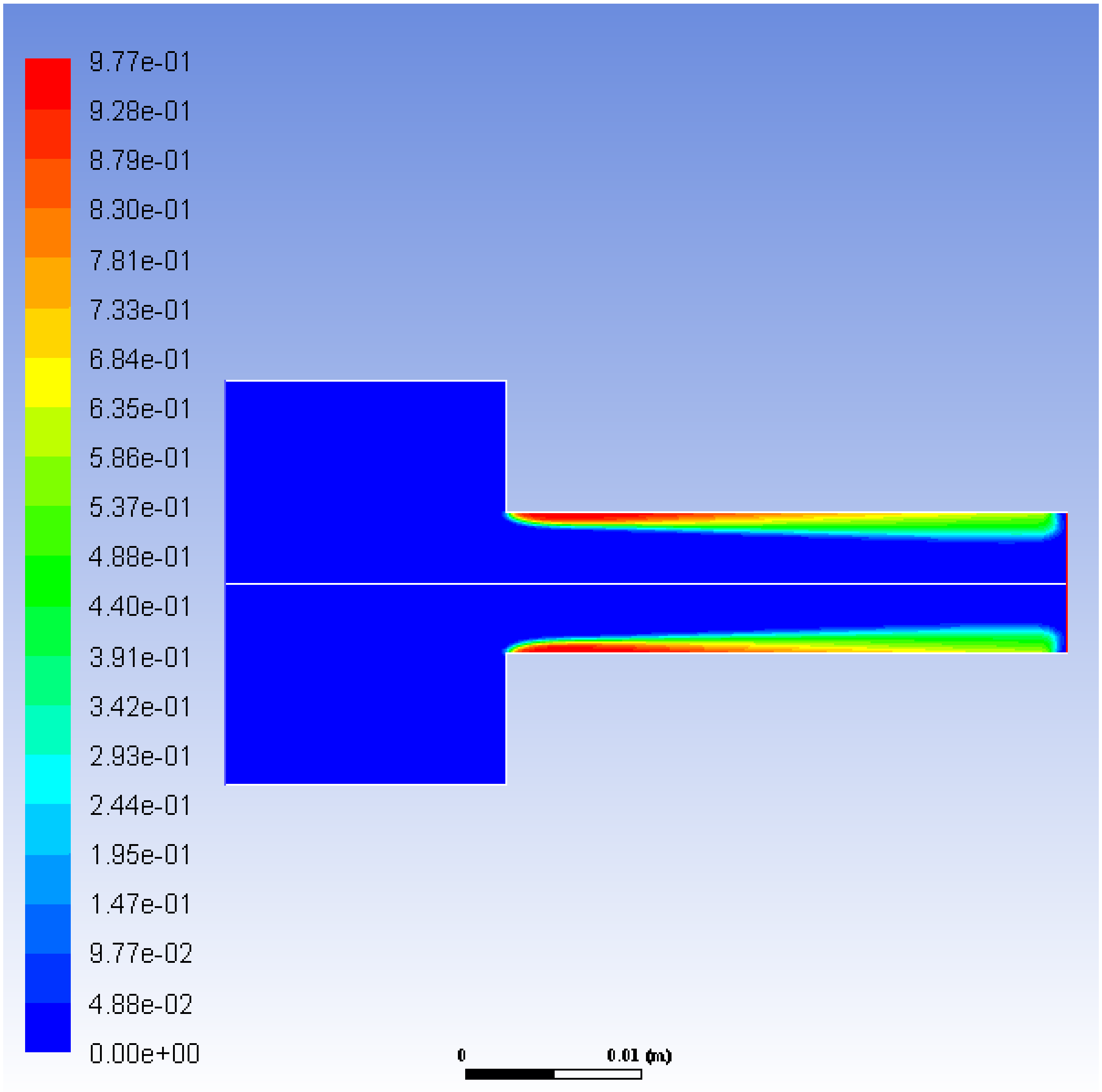
   b. Enable **Filled** in the **Options** group box.

   c. Select **Phases...** and **Volume fraction** from the **Contours of** drop-down lists.

   d. Select **vapor** from the **Phase** drop-down list.

e. Click **Save/Display** and close the **Contours** dialog box.

Figure 18.6: Contours of Vapor Volume Fraction



The high turbulent kinetic energy region near the neck of the orifice in Figure 18.5: Contours of Turbulent Kinetic Energy (p. 774) coincides with the highest volume fraction of vapor in Figure 18.6: Contours of Vapor Volume Fraction (p. 775). This indicates the correct prediction of a localized high phase change rate. The vapor then gets convected downstream by the main flow.

18.5. Summary

This tutorial demonstrated how to set up and resolve a strongly cavitating pressure-driven flow through an orifice, using multiphase mixture model of ANSYS Fluent with cavitation effects. You learned how to set the boundary conditions for an internal flow. A steady-state solution was calculated to simulate the formation of vapor in the neck of the flow after the section restriction at the orifice. A more computationally intensive transient calculation is necessary to accurately simulate the irregular cyclic process of bubble formation, growth, filling by water jet re-entry, and break-off.

18.6. Further Improvements

This tutorial guides you through the steps to reach an initial solution. You may be able to obtain a more accurate solution by using an appropriate higher-order discretization scheme and by adapting the mesh. Mesh adaption can also ensure that the solution is independent of the mesh. These steps are demonstrated in Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121).

Chapter 19: Using the Mixture and Eulerian Multiphase Models

This tutorial is divided into the following sections:

19.1. Introduction

This tutorial examines the flow of water and air in a tee junction. Initially you will solve the problem using the less computationally intensive mixture model. You will then switch to the more accurate Eulerian model and compare the results of these two approaches.

This tutorial demonstrates how to do the following:

- Use the mixture model with slip velocities.
- Set boundary conditions for internal flow.
- Calculate a solution using the pressure-based coupled solver with the mixture model.
- Use the Eulerian model.
- Calculate a solution using the multiphase coupled solver with the Eulerian model.
- Display the results obtained using the two approaches for comparison.

19.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

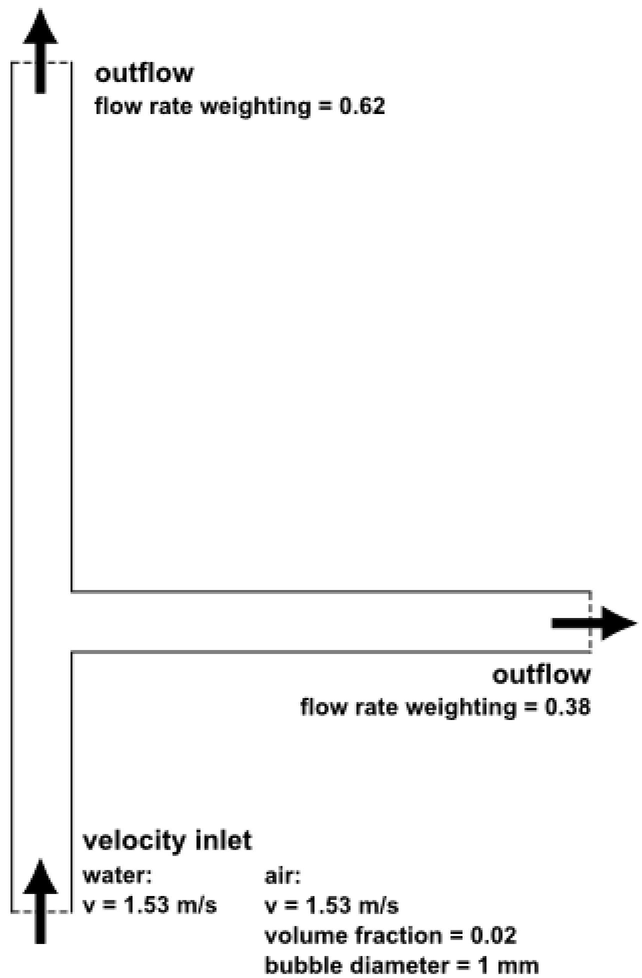
- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

and that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

19.3. Problem Description

This problem considers an air-water mixture flowing upwards in a duct and then splitting in a tee junction. The ducts are 25 mm in width, the inlet section of the duct is 125 mm long, and the top and the side ducts are 250 mm long. The schematic of the problem is shown in Figure 19.1: Problem Specification (p. 778).

Figure 19.1: Problem Specification



19.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

19.4.1. Preparation

To prepare for running this tutorial:

1. Set up a working folder on the computer you will be using.
2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

   Note

   If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

3. Enter the name of this tutorial into the search bar.
4. Narrow the results by using the filter on the left side of the page.
   a. Click **ANSYS Fluent** under **Product**.
   b. Click **18.0** under **Version**.
5. Select this tutorial from the list.
6. Click the **mix_eulerian_multiphase_R180.zip** link to download the input files.
7. Unzip `mix_eulerian_multiphase_R180.zip` to your working folder.

   The file `tee.msh` can be found in the `mix_eulerian_multiphase` folder created after unzipping the file.

8. Use Fluent Launcher to enable **Double Precision** and start the **2D** version of ANSYS Fluent.

   Fluent Launcher displays your **Display Options** preferences from the previous session.

   For more information about Fluent Launcher, see starting ANSYS Fluent using the Fluent Launcher in the Fluent Getting Started Guide.

9. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.
10. Run in **Serial** under **Processing Options**.

Note

The double precision solver is recommended for modeling multiphase flow simulations.

19.4.2. Mesh

1. Read the mesh file `tee.msh`.

File → Read → Mesh...

As ANSYS Fluent reads the mesh file, it will report the progress in the console.

19.4.3. General Settings

1. Check the mesh.

   Setting Up Domain → Mesh → Check

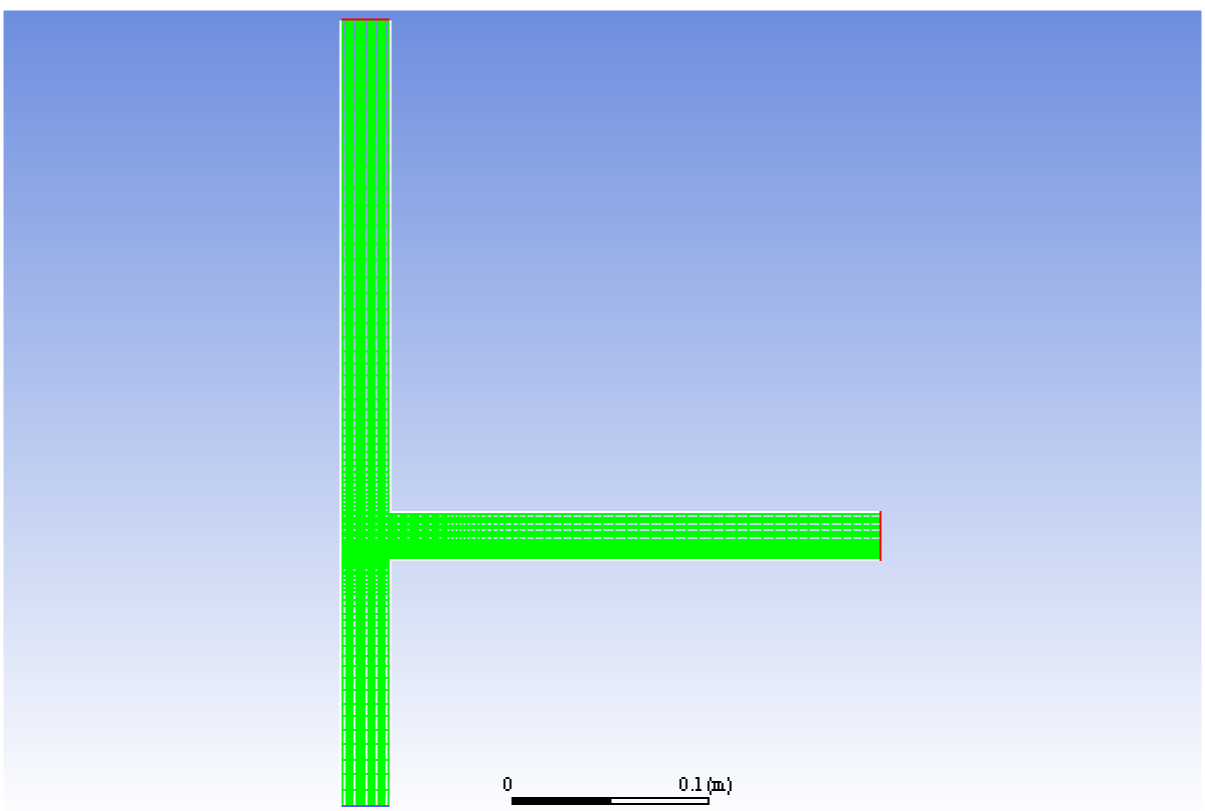
   ANSYS Fluent will perform various checks on the mesh and will report the progress in the console. Ensure that the reported minimum volume is a positive number.

2. Examine the mesh (Figure 19.2: Mesh Display (p. 780)).

Extra

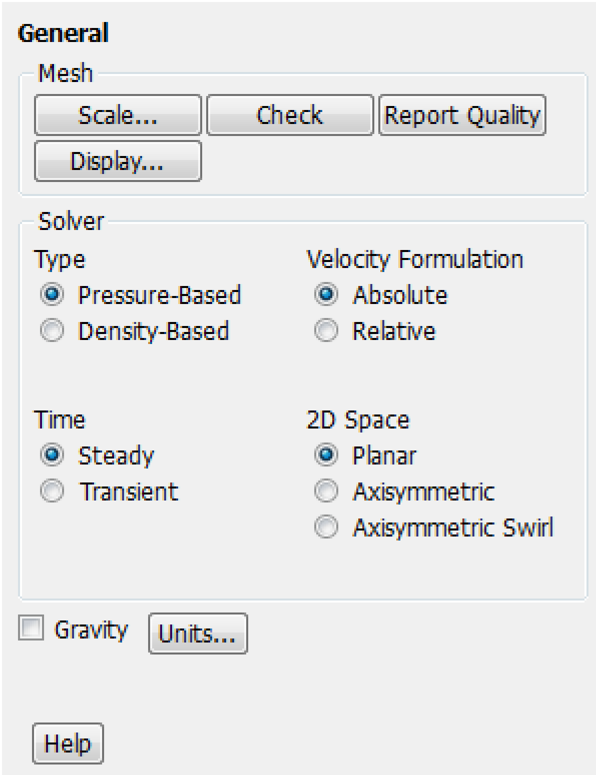
You can use the right mouse button to probe for mesh information in the graphics window. If you click the right mouse button on any node in the mesh, information will be displayed in the ANSYS Fluent console about the associated zone, including the name of the zone. This feature is especially useful when you have several zones of the same type and you want to distinguish between them quickly.

Figure 19.2: Mesh Display



3. Retain the default settings for the pressure-based solver.

Setup → General



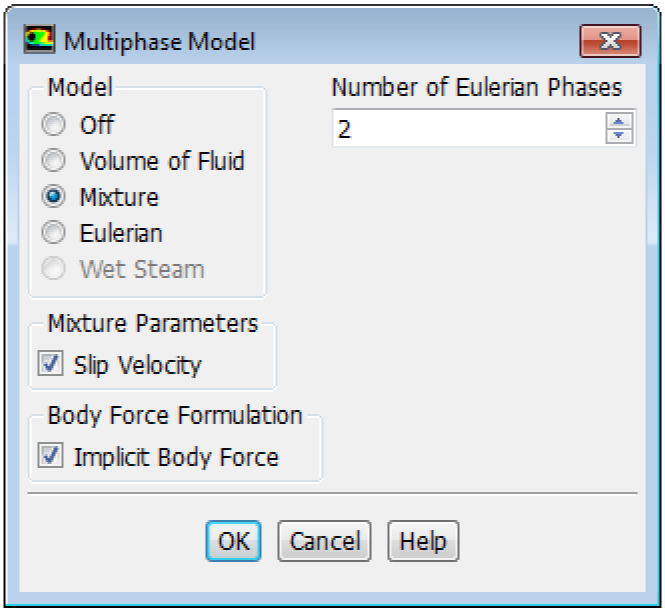
19.4.4. Models

1. Select the mixture multiphase model with slip velocities.

   Setting Up Physics → Models → Multiphase...

   a. Select **Mixture** in the **Model** list.

      *The **Multiphase Model** dialog box will expand to show the inputs for the mixture model.*



b. Ensure that **Slip Velocity** is enabled in the **Mixture Parameters** group box.

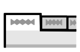
   You need to solve the slip velocity equation because there will be significant difference in velocities for the different phases.

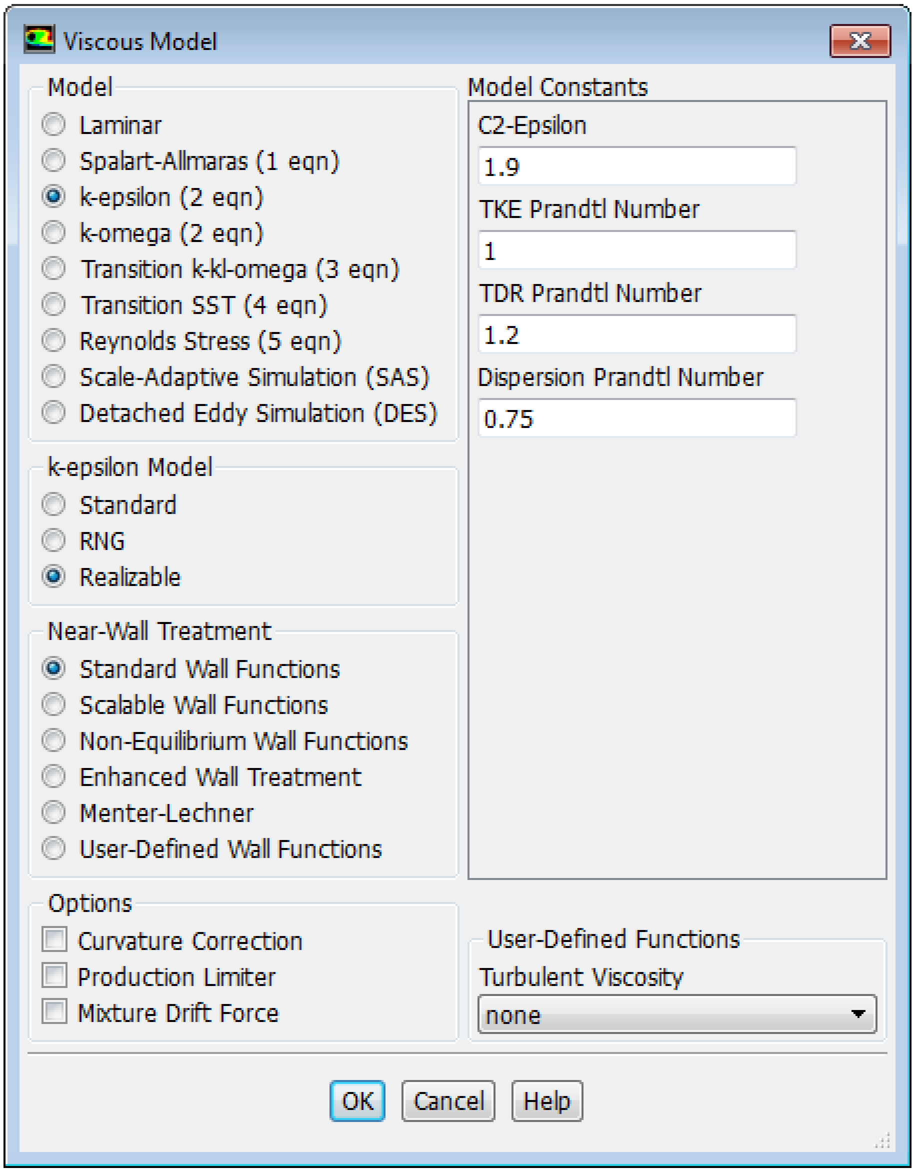
c. Enable **Implicit Body Force** in the **Body Force Formulation** group box.

   This treatment improves solution convergence by accounting for the partial equilibrium of the pressure gradient and body forces in the momentum equations. It is used in VOF and mixture problems, where body forces are large in comparison to viscous and convective forces.

d. Click **OK** to close the **Multiphase Model** dialog box.

2. Select the realizable k- ε turbulence model with standard wall functions.

 **Setting Up Physics → Models → Viscous...**



a. Select **k-epsilon** in the **Model** list.

b. Select **Realizable** under in the **k-epsilon Model** list.

   The realizable k- ε model is recommended in cases where flow separation around sharp corners or over bluff bodies can be expected.

c. Retain **Standard Wall Functions** in the **Near-Wall Treatment** list.

   This problem does not require a particularly fine mesh, and standard wall functions will be used.

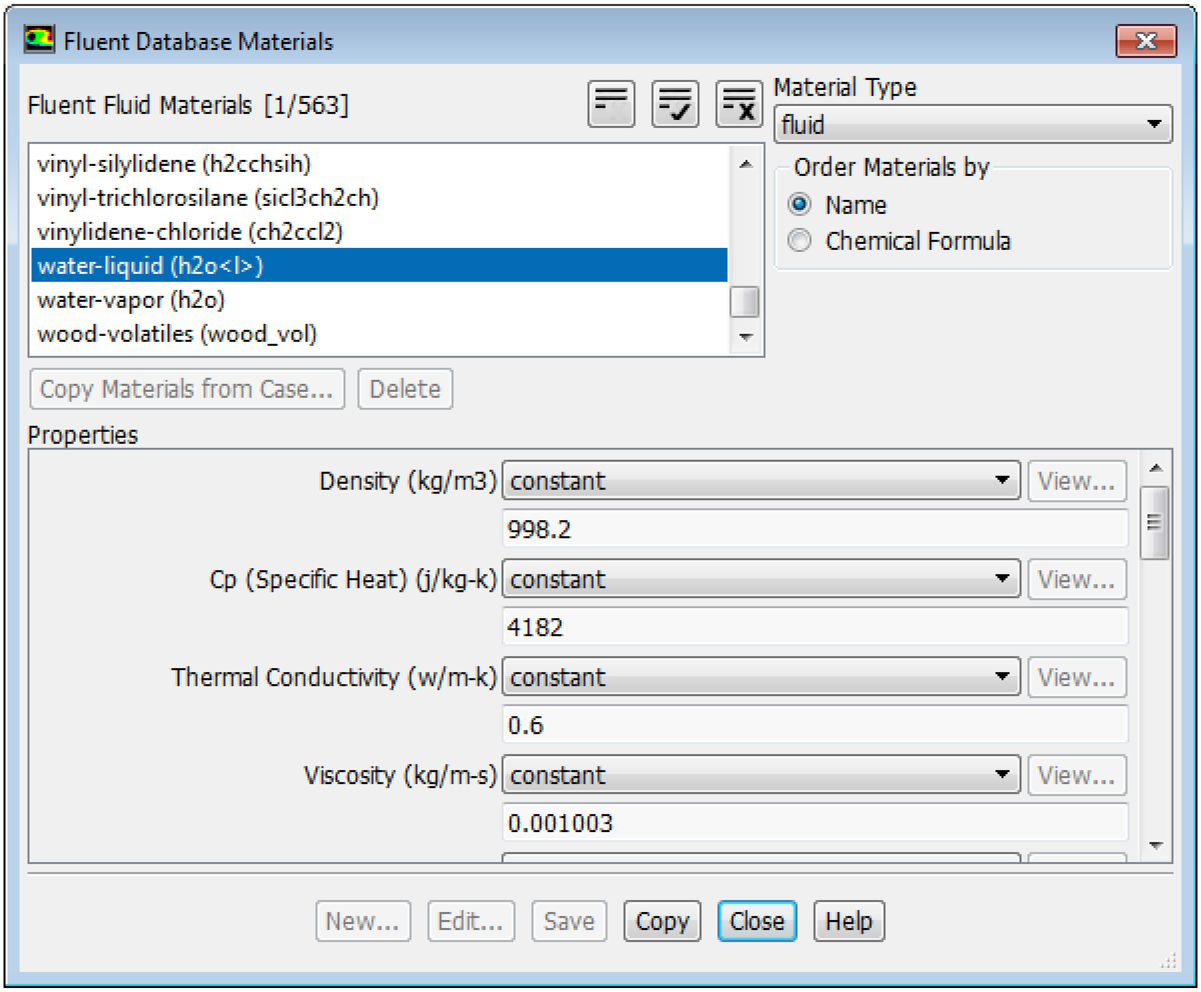
d. Click **OK** to close the **Viscous Model** dialog box.

19.4.5. Materials

1. Copy the properties for liquid water from the materials database so that it can be used for the primary phase.

Setup → **Materials** → **Fluid** → **air** **Edit...**

a. Click the **Fluent Database...** button to open the **Fluent Database Materials** dialog box.



i. Select **water-liquid (h2o < l >)** from the **Fluent Fluid Materials** selection list.

*Scroll down the list to find **water-liquid (h2o < l >)**.*

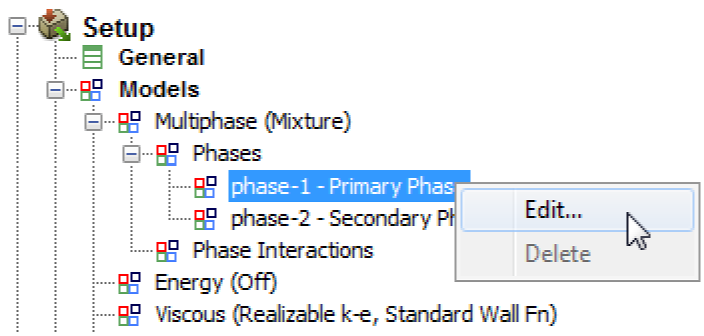
ii. Click **Copy** to copy the properties for liquid water to your model.

iii. Close the **Fluent Database Materials** dialog box.

b. Close the **Create/Edit Materials** dialog box.

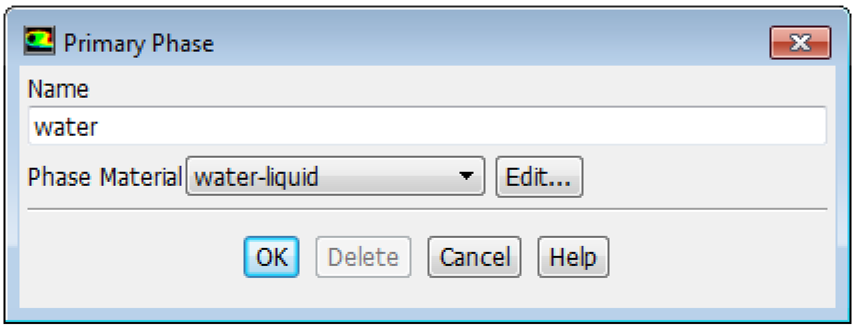
19.4.6. Phases

In the following steps you will define the liquid water and air phases that flow in the tee junction.



1. Specify liquid water as the primary phase.

   Setup → **Models** → **Multiphase** → **Phases** → **phase-1** **Edit...**

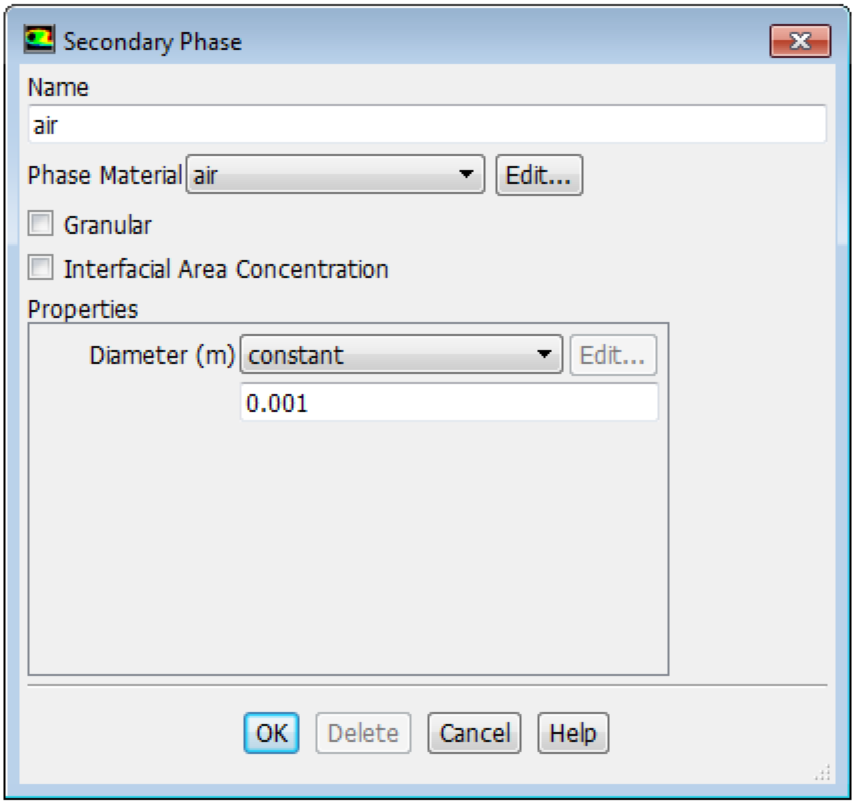
   a. Enter `water` for **Name**.

   b. Select **water-liquid** from the **Phase Material** drop-down list.

   c. Click **OK** to close the **Primary Phase** dialog box.

2. Specify air as the secondary phase.

   Setup → **Models** → **Multiphase** → **Phases** → **phase-2** **Edit...**



a. Enter `air` for **Name**.

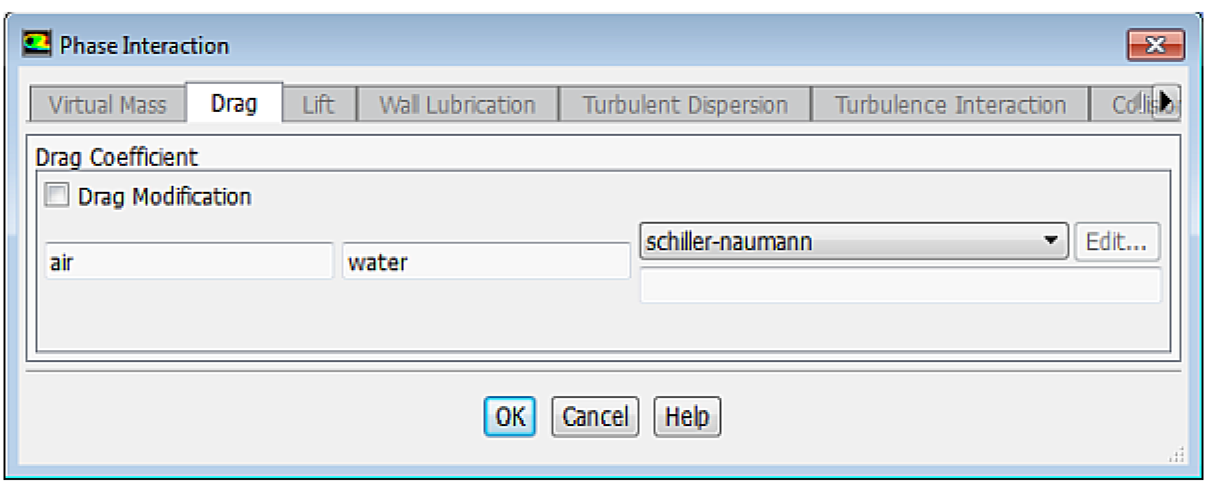
b. Retain the default selection of **air** from the **Phase Material** drop-down list.

c. Enter `0.001` m for **Diameter**.

d. Click **OK** to close the **Secondary Phase** dialog box.

3. Check that the drag coefficient is set to be calculated using the Schiller-Naumann drag law.

Setup → **Models** → **Multiphase** → **Phases** → **Phases Interactions** **Edit...**



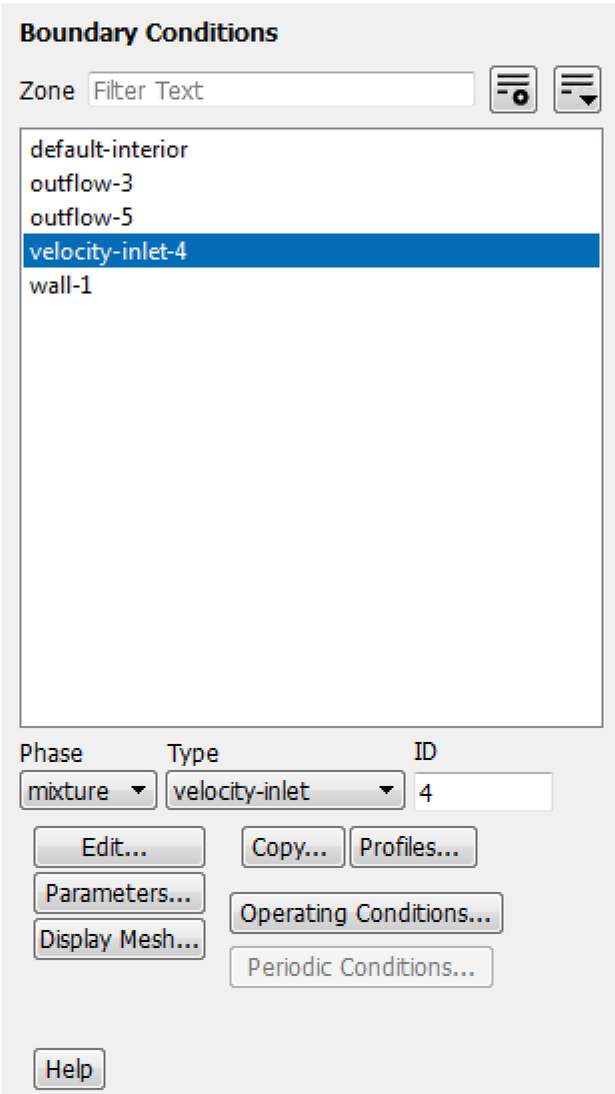
a. Click the **Drag** tab.

b. Retain the default selection of **schiller-naumann** from the **Drag Coefficient** drop-down list.

The Schiller-Naumann drag law describes the drag between the spherical particle and the surrounding liquid for a wide range of conditions provided the bubbles remain approximately spherical. In this case, the bubbles have a diameter of 1 mm which is within the spherical regime.

c. Click **OK** to close the **Phase Interaction** dialog box.

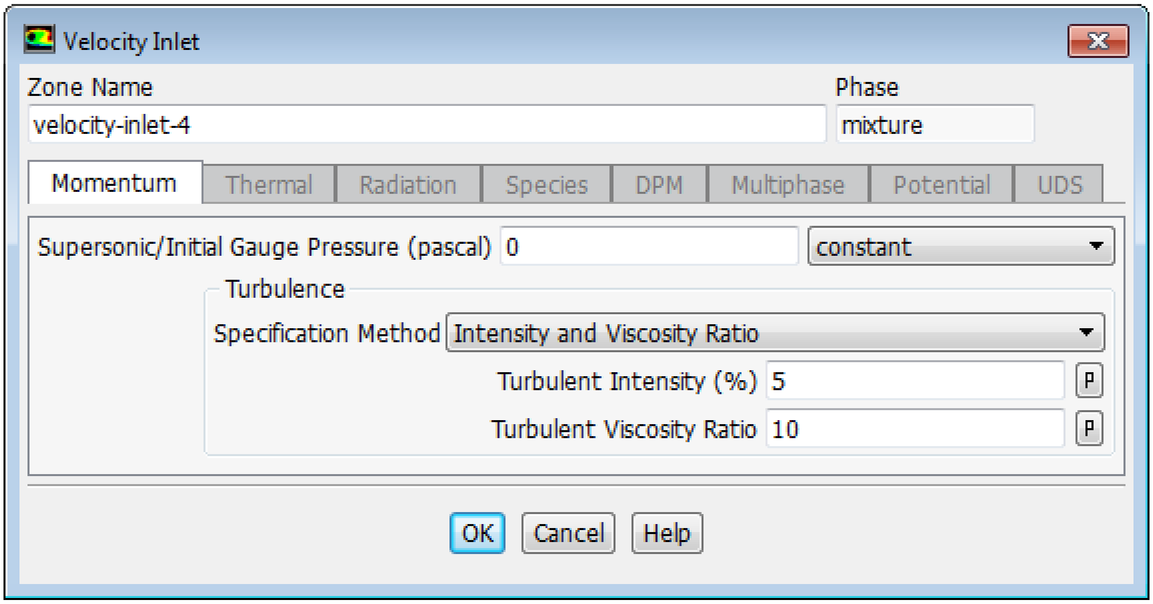
19.4.7. Boundary Conditions



For this problem, you need to set the boundary conditions for three boundaries: the velocity inlet and the two outflows. Since this is a mixture multiphase model, you will set the conditions at the velocity inlet that are specific for the mixture (conditions that apply to all phases) and also conditions that are specific to the primary and secondary phases.

1. Set the boundary conditions at the velocity inlet (**velocity-inlet-4**) for the mixture.

Setup → **Boundary Conditions** → **velocity-inlet-4** **Edit...**



a. Retain the default selection of **Intensity and Viscosity Ratio** as the turbulence **Specification Method**.

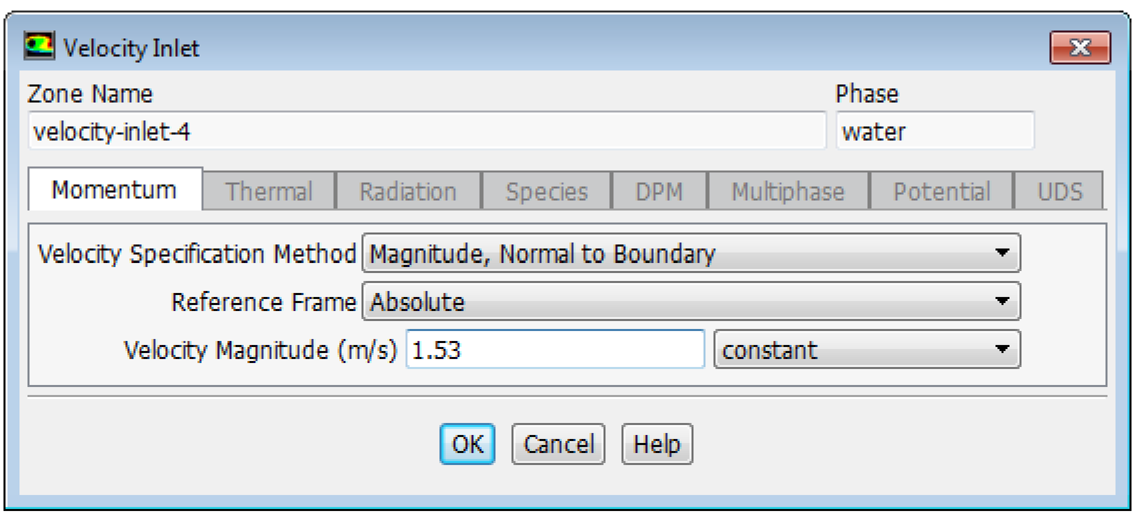
b. Retain the default of `5` % for **Turbulent Intensity**.

c. Retain the default of `10` for **Turbulent Viscosity Ratio**.

d. Click **OK** to close the **Velocity Inlet** dialog box.

2. Set the boundary conditions at the velocity inlet (**velocity-inlet-4**) for the primary phase (water).

Setup → **Boundary Conditions** → **velocity-inlet-4** → **water** **Edit...**



a. Retain the default selection of **Magnitude, Normal to Boundary** from the **Velocity Specification Method** drop-down list.

b. Retain the default selection of **Absolute** from the **Reference Frame** drop-down list.

c. Enter `1.53` m/s for **Velocity Magnitude**.

d. Click **OK** to close the **Velocity Inlet** dialog box.

Note

Note that you can also open the **Velocity Inlet** dialog box by double-clicking the **Setup/Boundary Conditions/velocity-inlet-4/water** tree item.

3. Set the boundary conditions at the velocity inlet (**velocity-inlet-4**) for the secondary phase (air).

Setup → **Boundary Conditions** → **velocity-inlet-4** → **air** **Edit...**

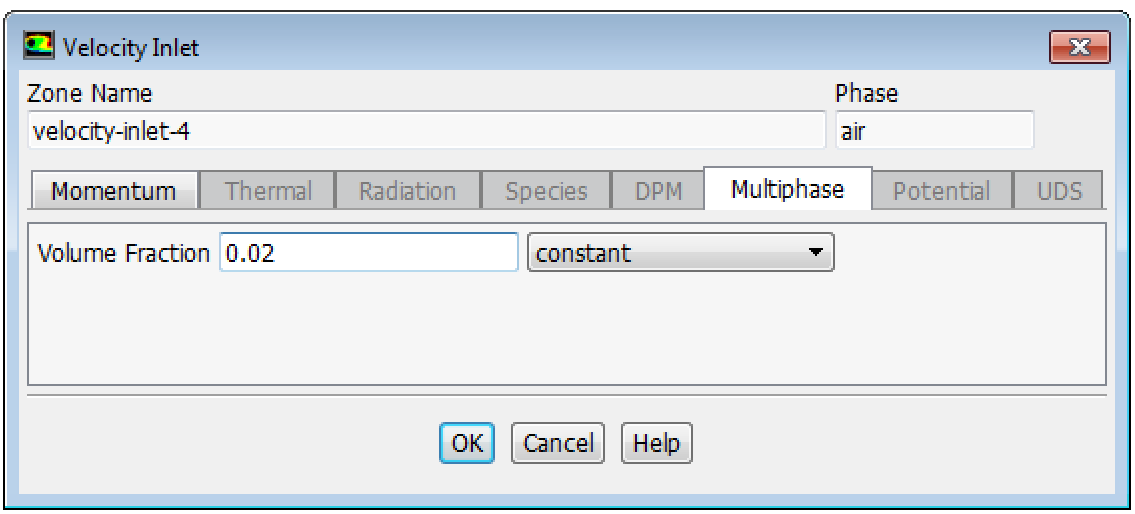
a. Retain the default selection of **Magnitude, Normal to Boundary** from the **Velocity Specification Method** drop-down list.

b. Retain the default selection of **Absolute** from the **Reference Frame** drop-down list.

c. Enter `1.53` m/s for **Velocity Magnitude**.

In multiphase flows, the volume rate of each phase is usually known. Volume rate divided by the inlet area gives the superficial velocity, which is the product of the inlet physical velocity and the volume fraction. When you have two phases, you must enter two physical velocities and the volume fraction of the secondary phase. Here it is assumed that bubbles at the inlet are moving at the same physical speed as the water.

d. Click the **Multiphase** tab and enter `0.02` for **Volume Fraction**.



e. Click **OK** to close the **Velocity Inlet** dialog box.

Note

Note that you can also open the **Velocity Inlet** dialog box by double-clicking the **Setup/Boundary Conditions/velocity-inlet-4/air** tree item.

4. Set the boundary conditions at **outflow-5** for the mixture.

Setup → **Boundary Conditions** → **outflow-5** **Edit...**

a. Click **Edit...** to open the **Outflow** dialog box.

i. Enter `0.62` for **Flow Rate Weighting**.

ii. Click **OK** to close the **Outflow** dialog box.

5. Set the boundary conditions at **outflow-3** for the mixture.

Setup → **Boundary Conditions** → **outflow-3** **Edit...**

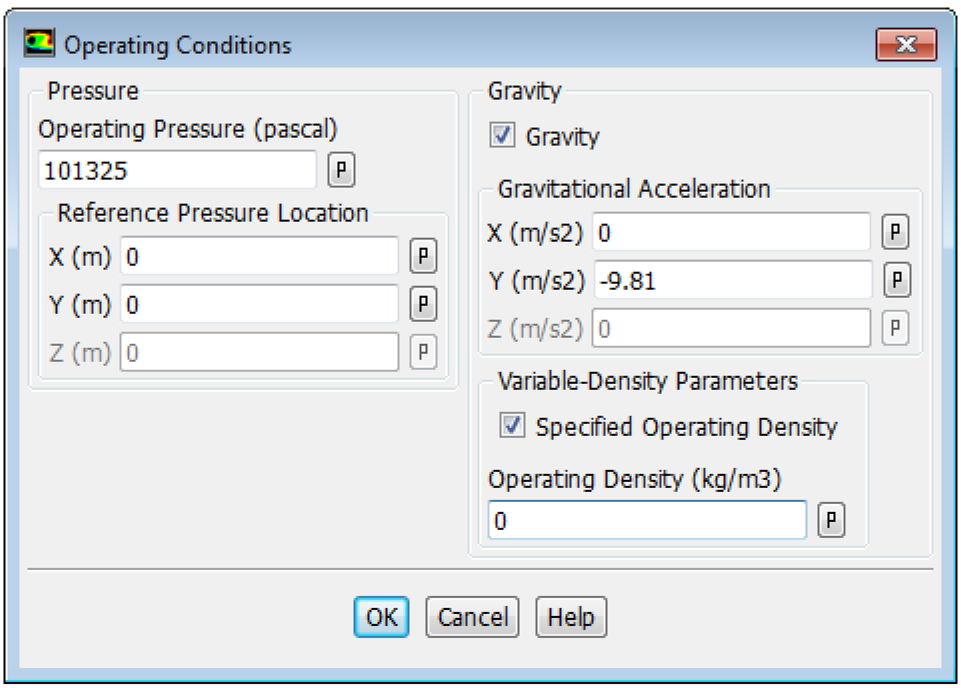
a. Enter `0.38` for **Flow Rate Weighting**.

b. Click **OK** to close the **Outflow** dialog box.

19.4.8. Operating Conditions

1. Set the gravitational acceleration.

Setup → **Boundary Conditions** → **Operating Conditions...**



a. Enable **Gravity**.

*The **Operating Conditions** dialog box will expand to show additional inputs.*

b. Enter `-9.81` m/s^2 for **Y** in the **Gravitational Acceleration** group box.

c. Enable **Specified Operating Density**.

d. Enter `0` kg/m^3 for **Operating Density**.

ANSYS Fluent redefines the fluid pressure by removing the hydrostatic component based on an average density in the domain or a user-specified operating density. By setting the operating density to `0` you force the hydrostatic pressure to appear explicitly in the postprocessed results. For more information, refer to the Fluent User's Guide.

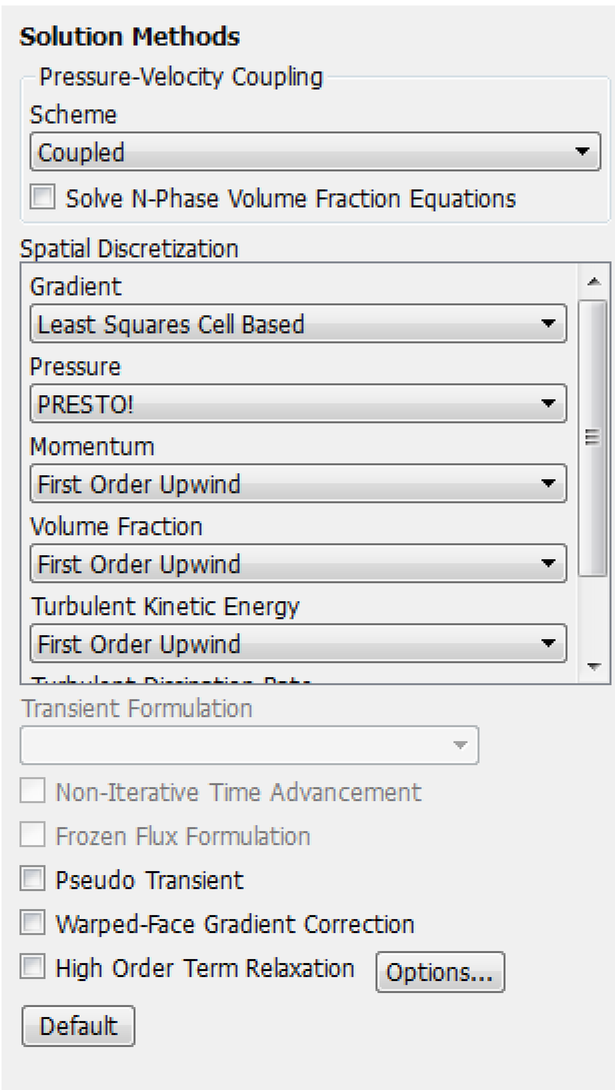
e. Click **OK** to close the **Operating Conditions** dialog box.

19.4.9. Solution Using the Mixture Model

You will begin by calculating a preliminary solution using first-order discretization for momentum, volume fraction and turbulence quantities. You will then change to higher-order methods to refine the solution.

1. Set the solution parameters.

Solving → **Solution** → **Methods...**



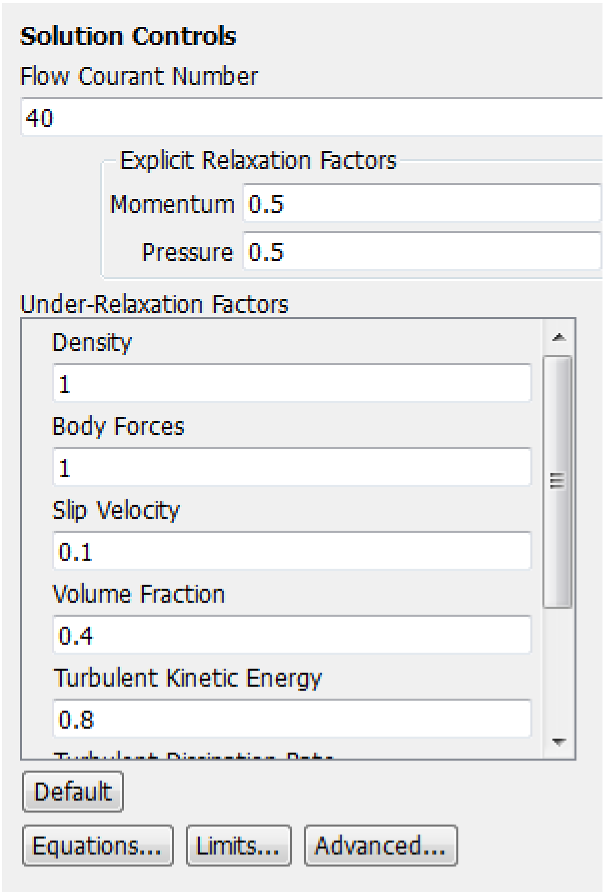
a. Select **Coupled** from the **Scheme** drop-down list.

b. Confirm that **PRESTO!** is selected from the **Pressure** drop-down list.

The PRESTO! method for pressure is a good choice when buoyancy and inertial forces are present.

2. Set the solution controls.

   Solving → **Solution** → **Controls...**

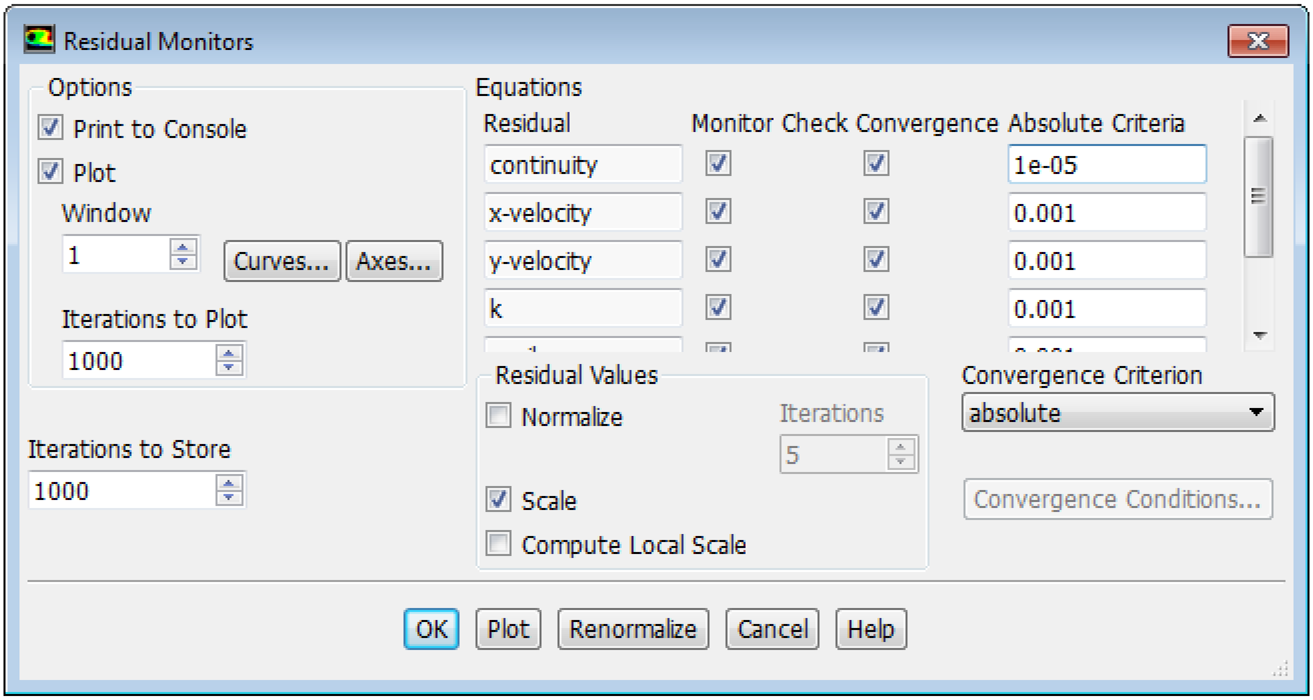


   a. Enter `40` for **Flow Courant Number**.

   b. Enter `0.4` for **Volume Fraction** in the **Under-Relaxation Factors** group box.

3. Enable the plotting of residuals during the calculation.

   Solving → **Reports** → **Residuals...**



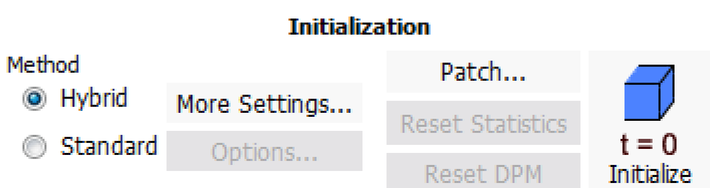
a. Ensure that **Plot** is enabled in the **Options** group box.

b. Enter `1e-05` for **Absolute Criteria** for **continuity**.

c. Click **OK** to close the **Residual Monitors** dialog box.

4. Initialize the solution.

Solving → Initialization → Hybrid



a. Select **Hybrid** as the initialization **Method** (**Initialization** group).

b. Click **Initialize**.

Note

For flows in complex topologies, hybrid initialization will provide better initial velocity and pressure fields than standard initialization. In general, this will help in improving the convergence behavior of the solver.

5. Save the case file (`tee_1a.cas.gz`).

File → Write → Case...

6. Start the calculation by requesting 1400 iterations.

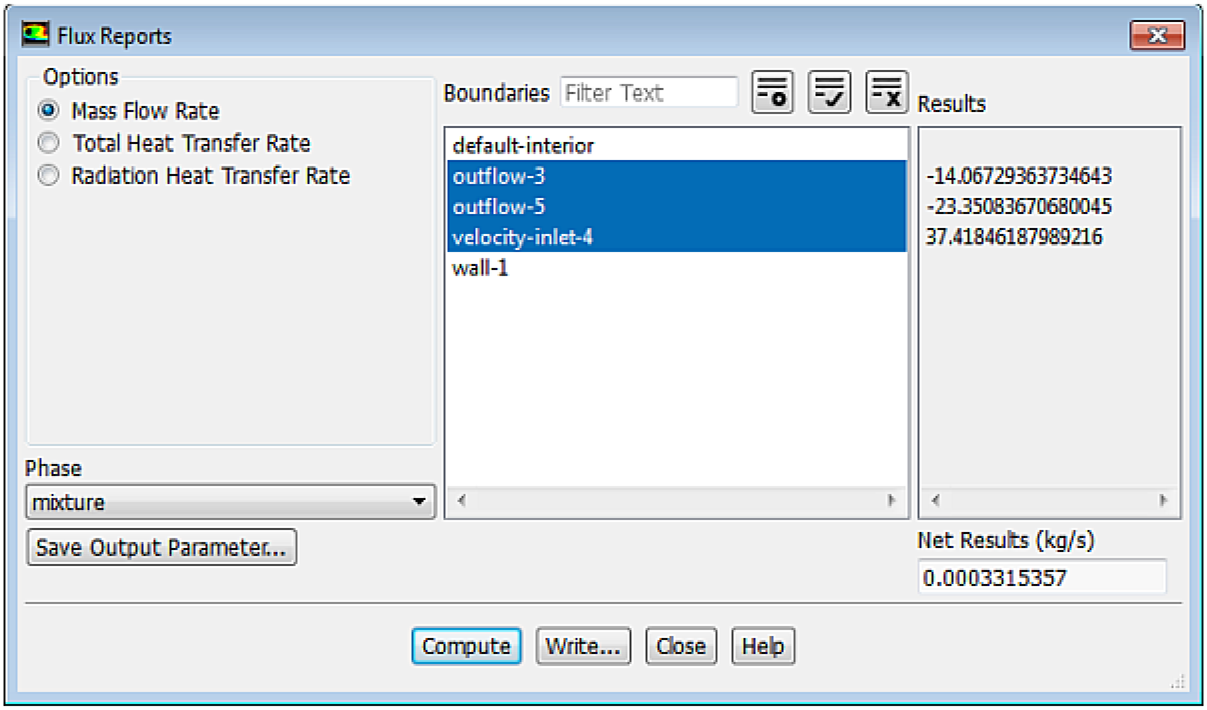
Solving → Run Calculation

7. Save the case and data files (`tee_1a.cas.gz` and `tee_1a.dat.gz`).

   File → Write → Case & Data...

8. Check the total mass flow rate for each phase.

   Postprocessing → Reports → Fluxes...



   a. Retain the default selection of **Mass Flow Rate** in the **Options** list.

   b. Select **water** from the **Phase** drop-down list.

   c. Select **outflow-3**, **outflow-5**, and **velocity-inlet-4** from the **Boundaries** selection list.

   d. Click **Compute**.

      Note that the net mass flow rate of water is a small fraction of the inlet and outlet flow rates (<0.1%), indicating that mass is conserved.

   e. Select **air** from the **Phase** drop-down list and click **Compute** again.

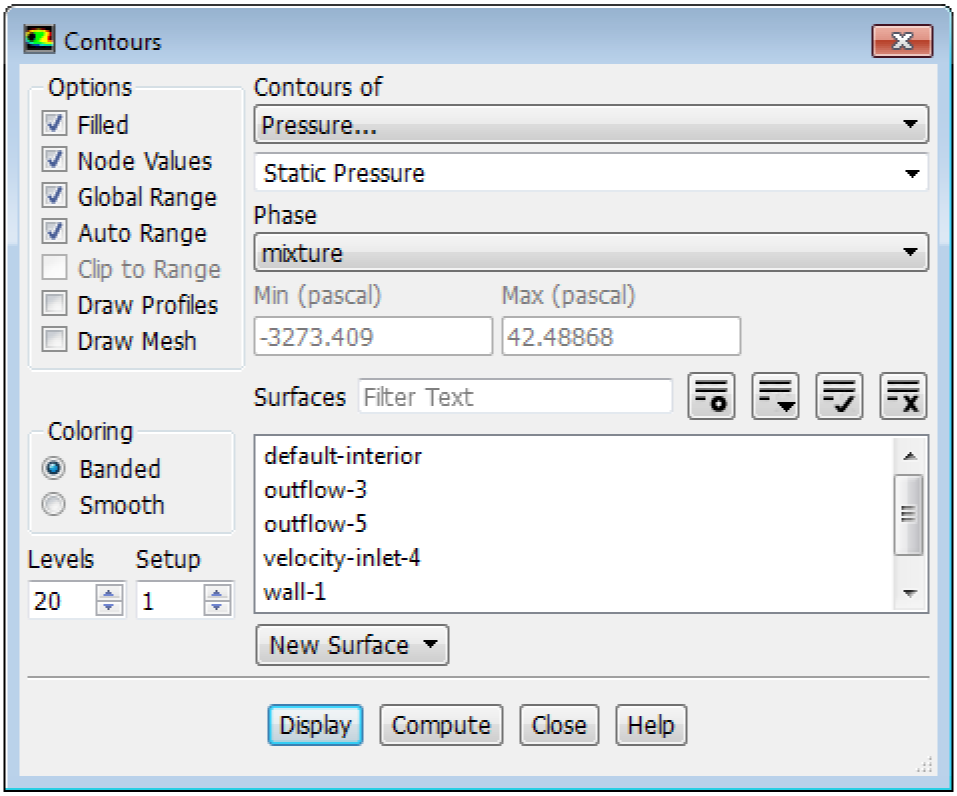
      Again, note that the net mass flow rate of air is small compared to the inlet and outlet flow rates.

   f. Close the **Flux Reports** dialog box.

19.4.10. Postprocessing for the Mixture Solution

1. Display the static pressure field in the tee (Figure 19.3: Contours of Static Pressure (p. 796)).

   Postprocessing → Graphics → Contours → Edit...

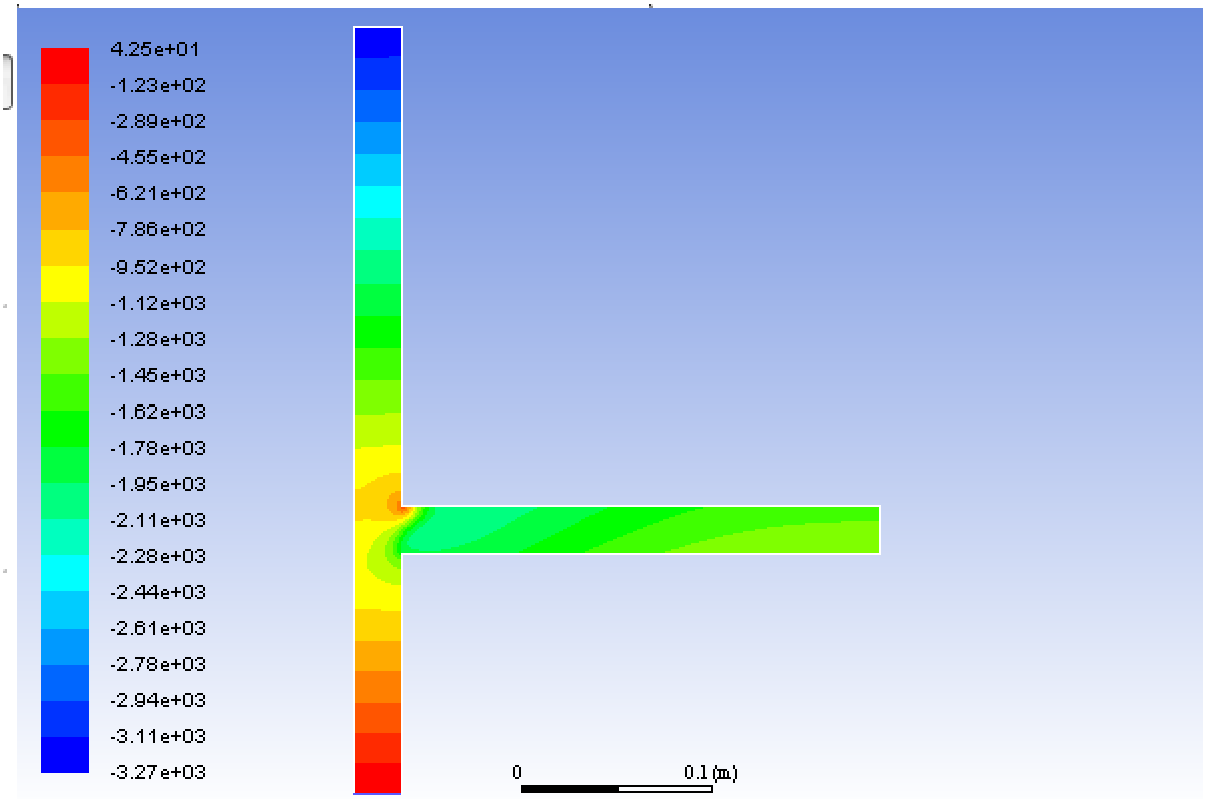


a. Enable **Filled** in the **Options** group box.

b. Retain the default selection of **Pressure...** and **Static Pressure** from the **Contours of** drop-down lists.

c. Click **Display**.

Figure 19.3: Contours of Static Pressure



In Figure 19.3: Contours of Static Pressure (p. 796) the hydrostatic pressure gradient is readily apparent in the vertical arm — a result of setting the Operating Density to `0`.

2. Display contours of velocity magnitude (Figure 19.4: Contours of Velocity Magnitude (p. 797)).

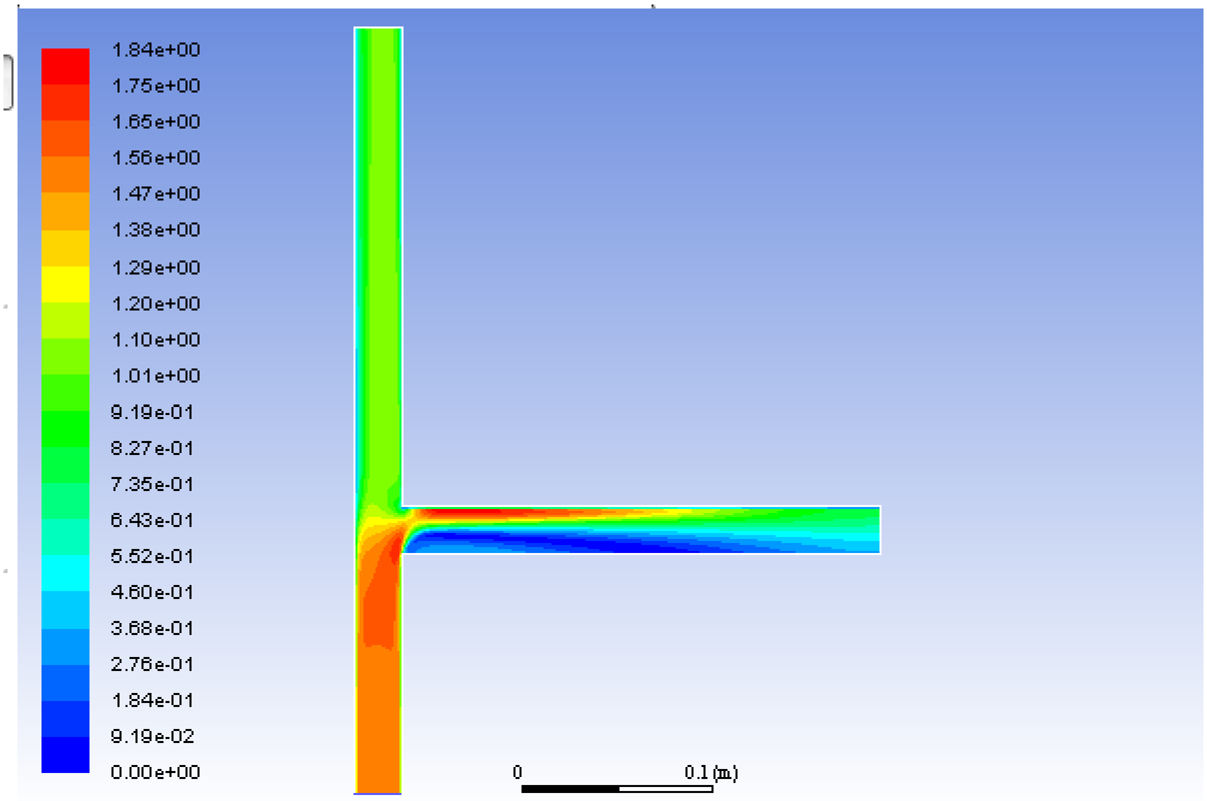
   Postprocessing → **Graphics** → **Contours** → **Edit...**

   a. Select **Velocity...** and **Velocity Magnitude** from the **Contours of** drop-down lists.

   b. Select **water** from the **Phase** drop-down list.

   c. Click **Display**.

Figure 19.4: Contours of Velocity Magnitude



3. Display the volume fraction of air (Figure 19.5: Contours of Air Volume Fraction (p. 798)).

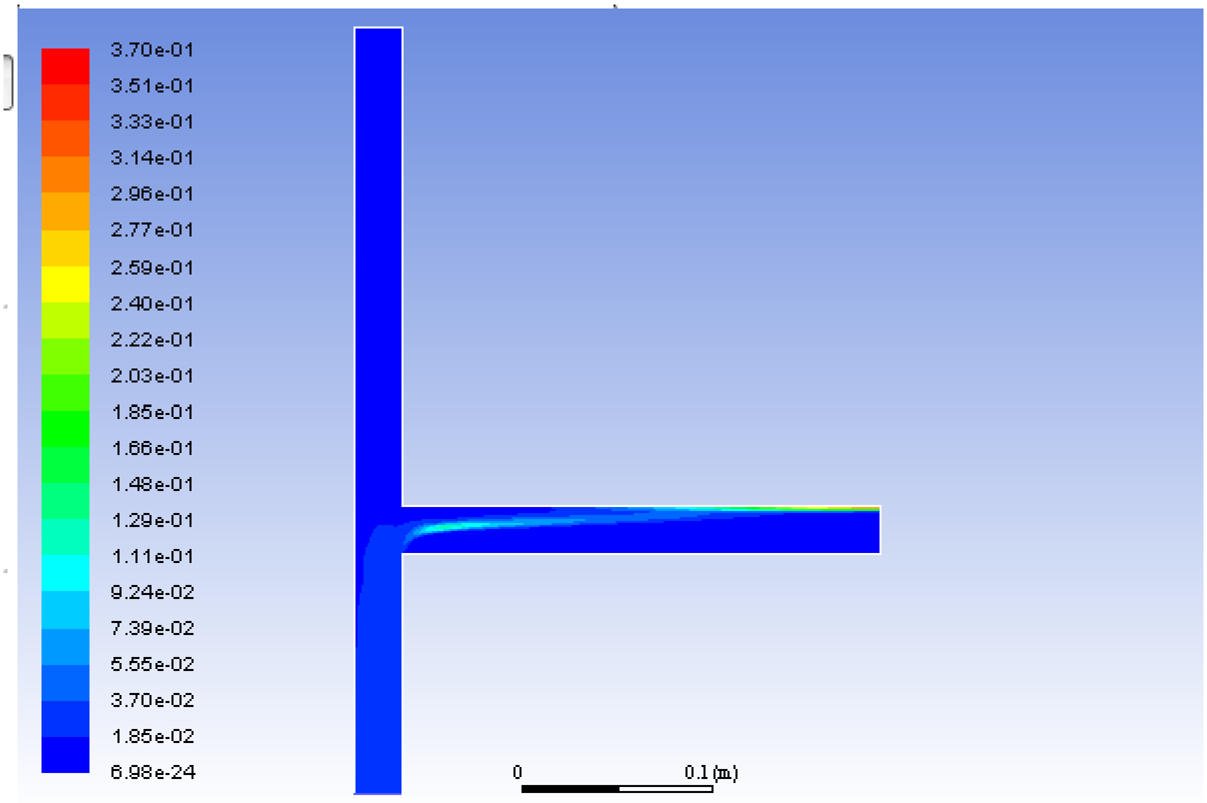
   Postprocessing → **Graphics** → **Contours** → **Edit...**

   a. Select **Phases...** and **Volume fraction** from the **Contours of** drop-down lists.

   b. Select **air** from the **Phase** drop-down list.

   c. Click **Display** and close the **Contours** dialog box.

Figure 19.5: Contours of Air Volume Fraction



When gravity acts downwards, it induces stratification in the side arm of the tee junction. In *Figure 19.5: Contours of Air Volume Fraction (p. 798)**, you can see that the gas (air) tends to concentrate on the upper part of the side arm. In this case, gravity acts against inertia that tends to concentrate gas on the low pressure side, thereby creating gas pockets. In the vertical arm, both the gas and the water have velocities in the same direction, and therefore there is no separation. The outflow split modifies the relation between inertia forces and gravity to a large extent, and has an important role in flow distribution and on the gas concentration.*

19.4.11. Higher Order Solution using the Mixture Model

In this step you will change to higher order discretization schemes and continue the calculation to refine the solution.

1. Revisit the **Solution Methods** task page and make the following selections:

Group	Setting	Value
Spatial Discretization	**Pressure**	**PRESTO!**
	Momentum	**Third-Order MUSCL**
	Volume Fraction	**QUICK**
	Turbulent Kinetic Energy	**Third-Order MUSCL**
	Turbulent Dissipation Rate	**Third-Order MUSCL**

2. Run the calculation for an additional 1400 iterations.

3. Save the case and data files as `tee_1b.cas.gz` and `tee_1b.dat.gz`

File → Write → Case & Data...

4. Plot the contours of air volume fraction using the higher order method on the same scale as in Figure 19.5: Contours of Air Volume Fraction (p. 798).

   Postprocessing → Graphics → Contours → Edit...

   a. Select **Phases...** and **Volume fraction** from the **Contours of** drop-down lists.

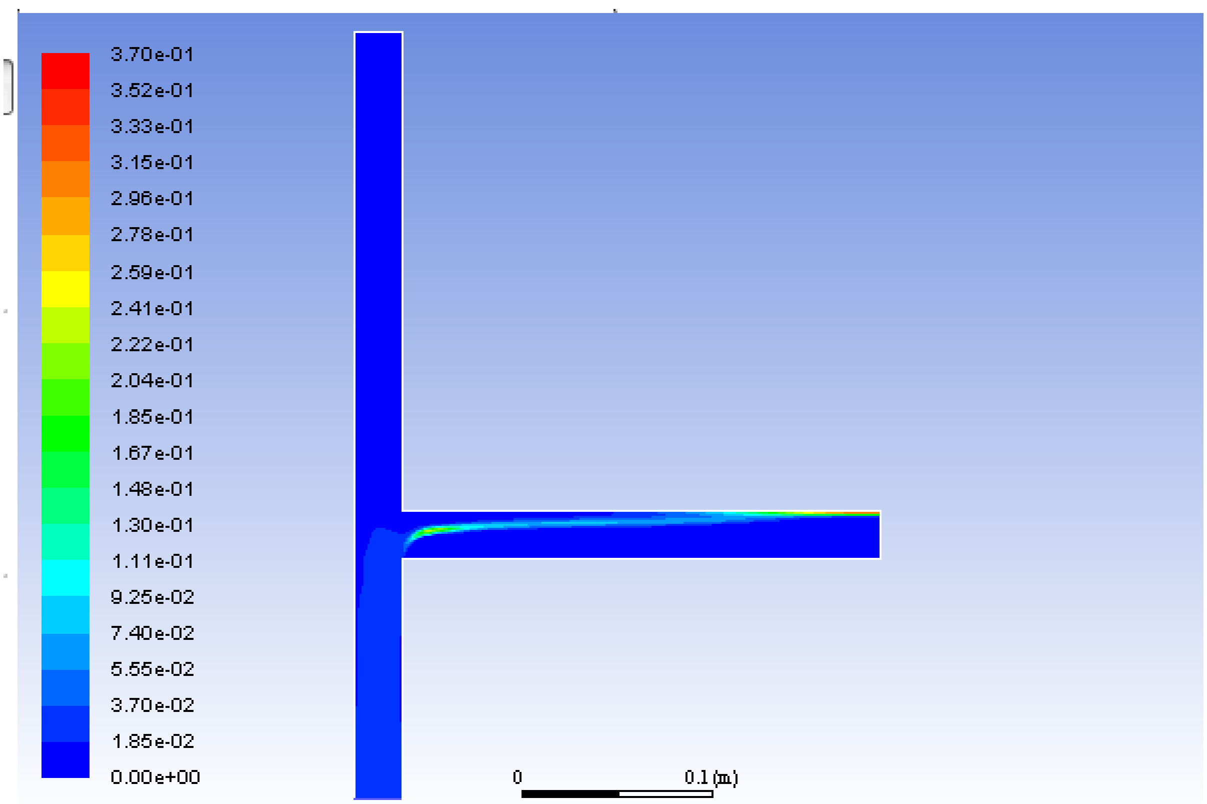
   b. Select **air** from the **Phase** drop-down list.

   c. Disable **Auto Range** and **Clip to Range**.

   d. Enter `0` and `3.70e-1` for **Min** and **Max**, respectively.

   e. Click **Display** and close the **Contours** dialog box.

Figure 19.6: Contours of Air Volume Fraction — Higher Order Solution

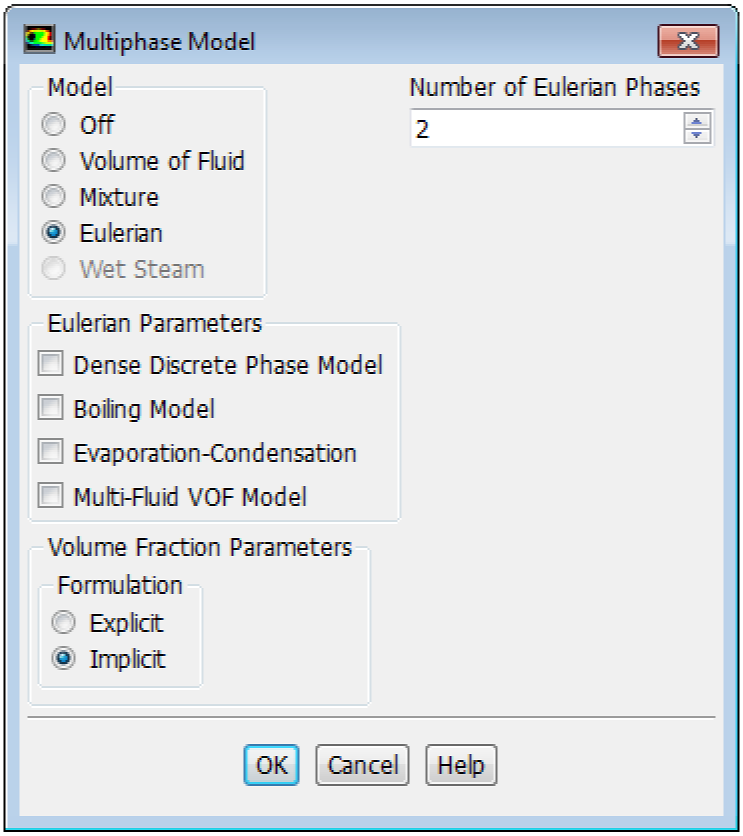


19.4.12. Setup and Solution for the Eulerian Model

The mixture model is a simplification of the Eulerian model and is valid only when bubble inertia can be neglected. This assumption can be violated in the recirculation pattern. The Eulerian model also offers models for various non-drag forces that are not available when using the mixture model. As a result, the Eulerian model is expected to make a more realistic prediction in this case. You will use the solution obtained using the mixture model as an initial condition for the calculation using the Eulerian model. Because you have already computed a reasonable initial solution, you will continue with the higher order discretization methods.

1. Select the Eulerian multiphase model.

   Setting Up Physics → Models → Multiphase...



   a. Select **Eulerian** in the **Model** list.

   b. Click **OK** to close the **Multiphase Model** dialog box.

2. Specify the drag and lift laws to be used for computing the interphase momentum transfer.

   Setup → Models → Multiphase → Phases → Phases Interactions **Edit...**

   a. In the **Drag** tab, retain the default selection of **schiller-naumann** from the **Drag Coefficient** drop-down list.

   b. In the **Lift** tab, select **legendre-magnaudet** from the **Lift Coefficient** drop-down list.

   Lift forces can arise when the gradient of the primary phase velocity field has a component normal to the bubble flow.

   c. Click **OK** to close the **Phase Interaction** dialog box.

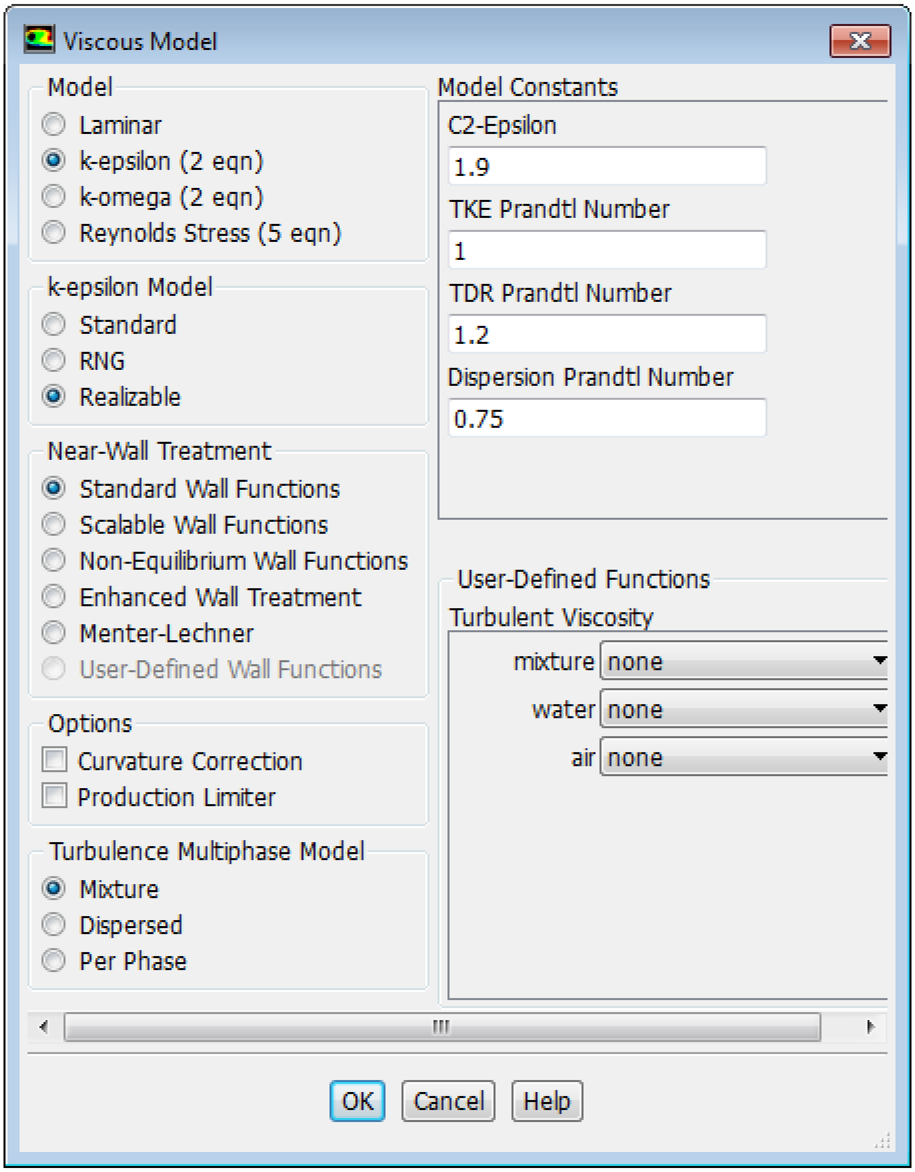
Note

For this problem, there are no parameters to be set for the individual phases other than those that you specified when you set up the phases for the mixture model calculation. If you use the Eulerian model for a flow involving a granular secondary phase, you will need to set additional parameters. There are also other options in the **Phase Interaction** dialog box that may be relevant for other applications.

For details on setting up an Eulerian multiphase calculation, see the Fluent User's Guide.

3. Select the multiphase turbulence model.

   Setting Up Physics → Models → Viscous...



   a. Retain the default selection of **Mixture** in the **Turbulence Multiphase Model** list.

      In this case the dispersed phase volume concentration is relatively small so the mixture turbulence model is sufficient to capture the important features of the turbulent flow.

   b. Click **OK** to close the **Viscous Model** dialog box.

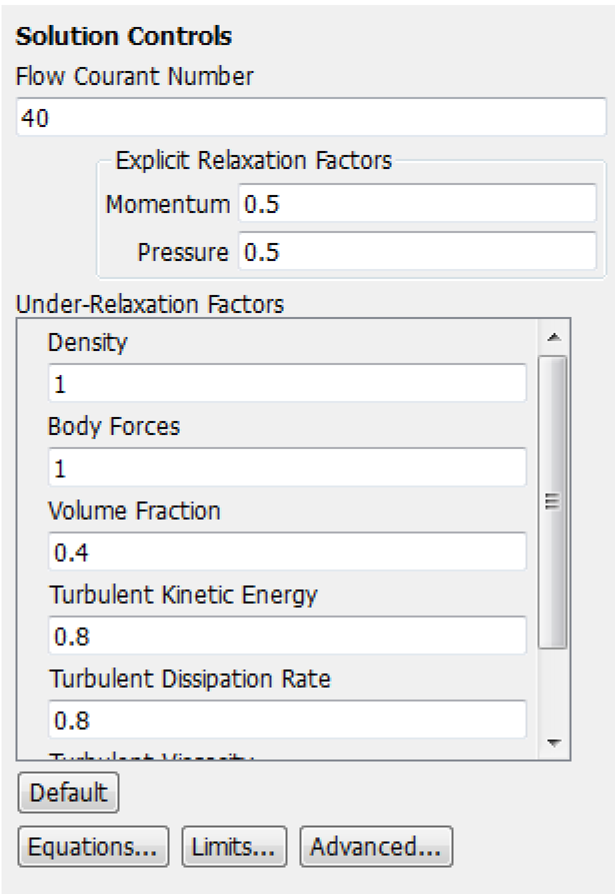
4. Confirm that the solution parameters are set to use the higher-order discretization schemes.

Revisit the **Solution Methods** task page and verify that the settings are as follows:

Group	Setting	Value
Pressure-Velocity Coupling	**Scheme**	**Coupled**
Spatial Discretization	**Momentum**	**Third-Order MUSCL**
	Volume Fraction	**QUICK**
	Turbulent Kinetic Energy	**Third-Order MUSCL**
	Turbulent Dissipation Rate	**Third-Order MUSCL**

5. Set the solution controls

   Solution Controls



   a. Enter `40` for **Flow Courant Number**.

   b. Enter `0.5` for **Momentum** and for **Pressure** in the **Explicit Relaxation Factors** group box.

   c. Confirm that **Volume Fraction** is set to `0.4` in the **Under-Relaxation Factors** group box.

6. Continue the solution by requesting 1400 additional iterations.

   Solving → **Run Calculation** → **Calculate**

7. Check that the mass imbalance is small (less than about 0.2 %) using the **Flux Reports** dialog box as for the Mixture model solution.

   Postprocessing → **Reports** → **Fluxes...**

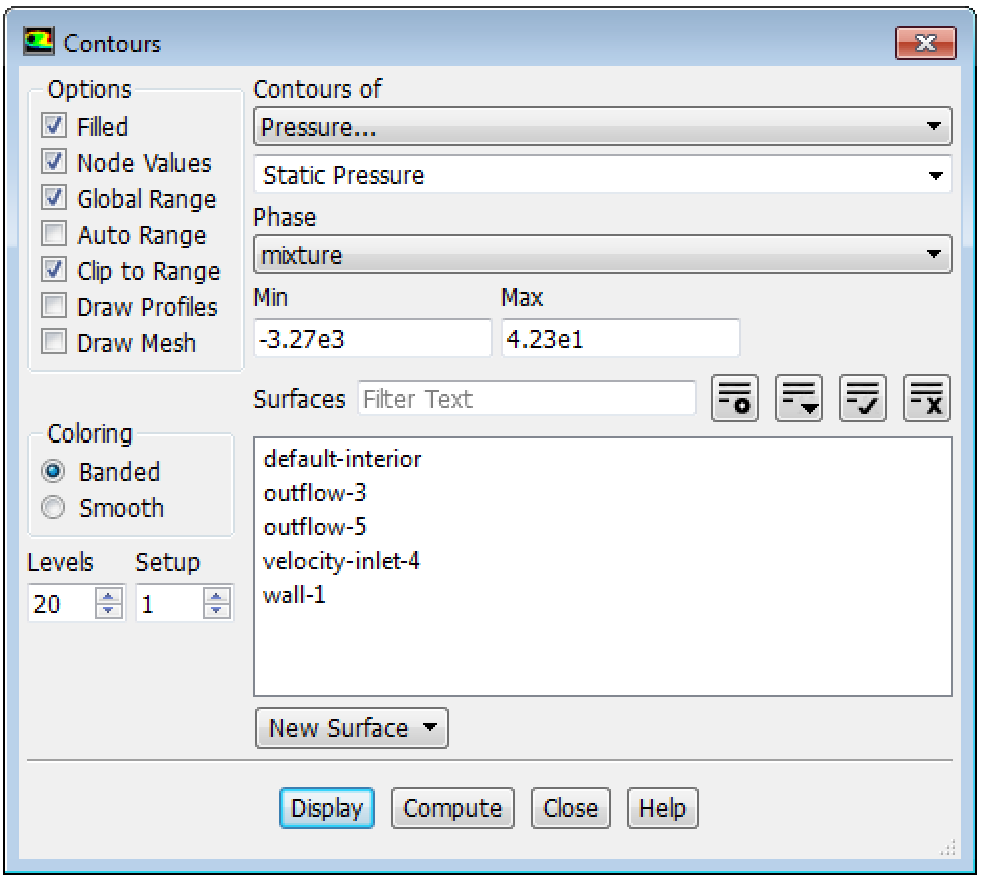
8. Save the case and data files (`tee_2.cas.gz` and `tee_2.dat.gz`).

   File → **Write** → **Case & Data...**

19.4.13. Postprocessing for the Eulerian Model

1. Display the static pressure field in the tee for the mixture (Figure 19.7: Contours of Static Pressure — Eulerian Model (p. 804)).

   Postprocessing → **Graphics** → **Contours** → **Edit...**



   a. Select **Pressure...** from the **Contours of** drop-down list.

      *By default, **Dynamic Pressure** will be displayed in the lower **Contours of** drop-down list. This will automatically change to **Static Pressure** after you select the appropriate phase in the next step.*

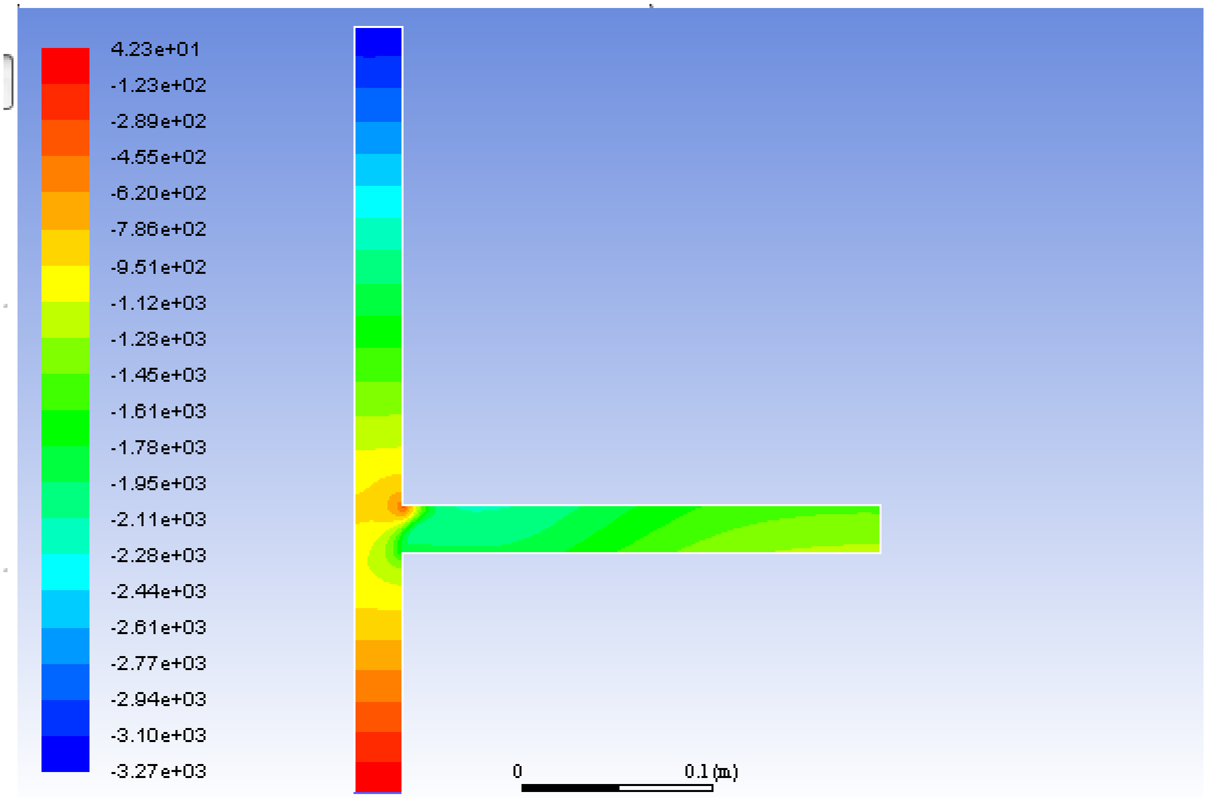
   b. Select **mixture** from the **Phase** drop-down list.

      *The lower **Contours of** drop-down list will now display **Static Pressure**.*

   c. As before, disable **Auto Range** (**Clip to Range** will be enabled) and set the **Min** and **Max** values to match those in Figure 19.3: Contours of Static Pressure (p. 796).

d. Click **Display**.

Figure 19.7: Contours of Static Pressure — Eulerian Model



2. Display contours of velocity magnitude for water (Figure 19.8: Contours of Water Velocity Magnitude — Eulerian Model (p. 805)).

   Postprocessing → **Graphics** → **Contours** → **Edit...**

   a. Select **Velocity...** and **Velocity Magnitude** from the **Contours of** drop-down lists.

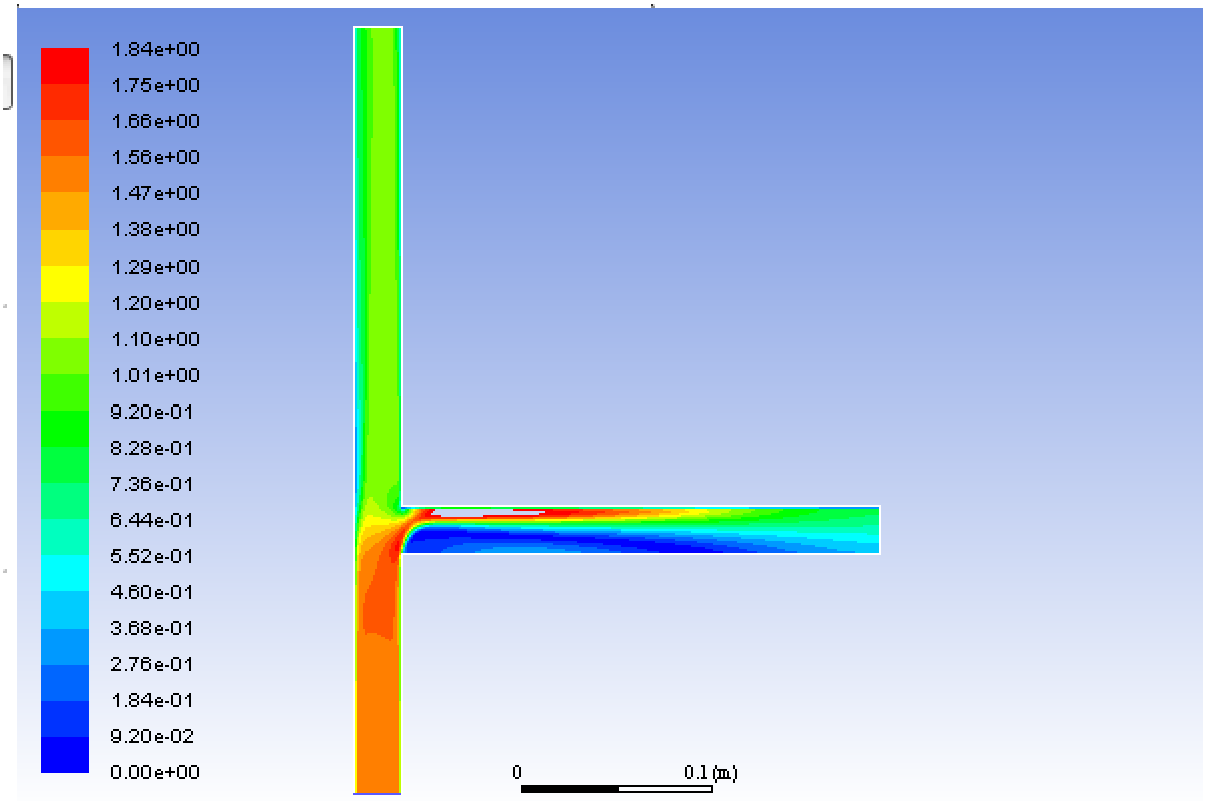
   b. Retain the selection of **water** from the **Phase** drop-down list.

      Since the Eulerian model solves individual momentum equations for each phase, you can choose the phase for which solution data is plotted.

   c. Set the range to match that in Figure 19.4: Contours of Velocity Magnitude (p. 797).

   d. Click **Display**.

Figure 19.8: Contours of Water Velocity Magnitude — Eulerian Model



3. Display the volume fraction of air (Figure 19.9: Contours of Air Volume Fraction — Eulerian model (p. 806)).

   Postprocessing → **Graphics** → **Contours** → **Edit...**

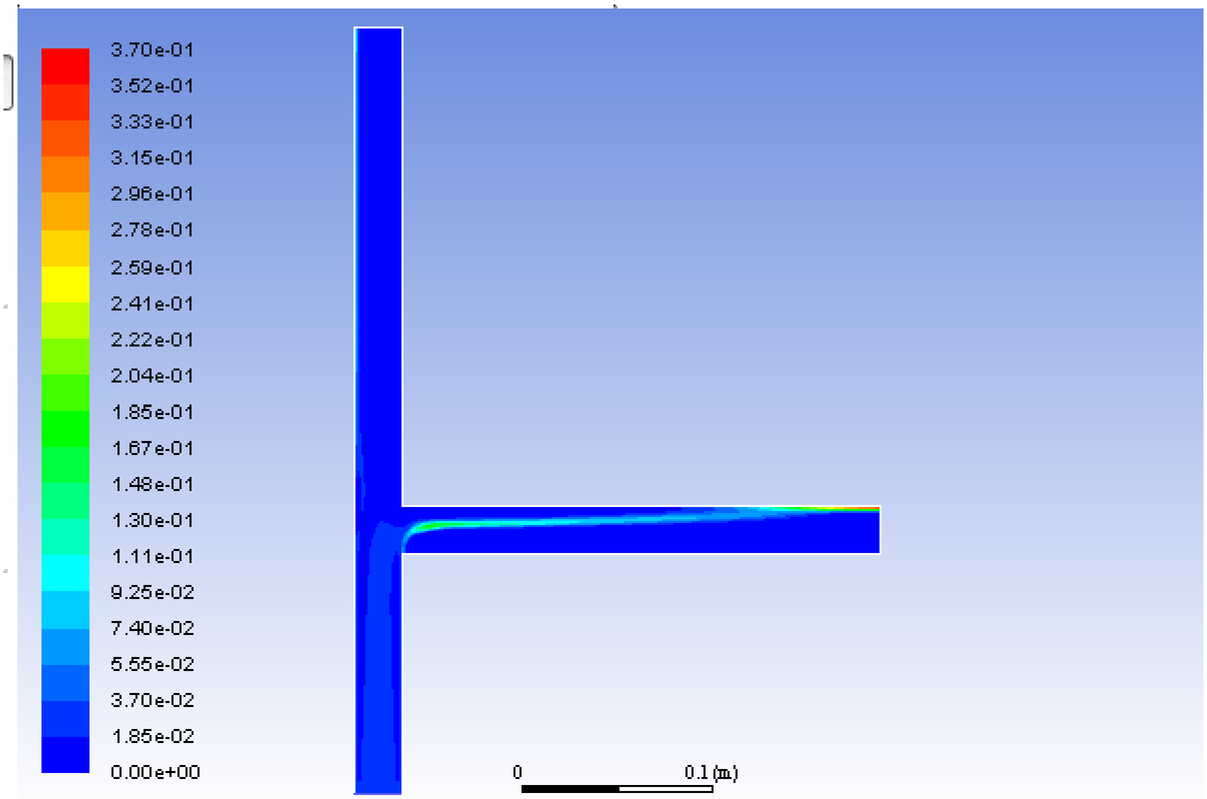
   a. Select **Phases...** and **Volume fraction** from the **Contours of** drop-down lists.

   b. Select **air** from the **Phase** drop-down list.

   c. Set the range to match that in Figure 19.5: Contours of Air Volume Fraction (p. 798).

   d. Click **Display** and close the **Contours** dialog box.

Figure 19.9: Contours of Air Volume Fraction — Eulerian model



Compare the volume fraction plot in Figure 19.9: Contours of Air Volume Fraction — Eulerian model (p. 806) with the volume fraction plot using the mixture model in Figure 19.6: Contours of Air Volume Fraction — Higher Order Solution (p. 799). Notice that the path of the concentrated air stream in the side arm extends farther into the side arm before drifting to the top surface. As is apparent from the velocity plots, there is a substantial velocity gradient across the side arm as a result of the recirculation near the lower corner of the tee junction. As the dispersed phase bubbles travel along the side arm with the flow, this velocity gradient induces a lift force which tends to oppose the buoyancy force thereby delaying the accumulation of the air concentration along the top surface of the side arm.

19.5. Summary

This tutorial demonstrated how to set up and solve a multiphase problem using the mixture model and the Eulerian model. You learned how to set boundary conditions for the mixture and both phases. The solution obtained with the mixture model was used as a starting point for the calculation with the Eulerian model. After completing calculations for each model, you displayed the results to allow for a comparison of the two approaches. For more information about the mixture and Eulerian models, see the Fluent User's Guide.

19.6. Further Improvements

This tutorial guides you through the steps to reach an initial set of solutions. You may be able to obtain a more accurate solution by adapting the mesh. Mesh adaption can also ensure that the solution is independent of the mesh. These steps are demonstrated in Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121).

Chapter 20: Modeling Solidification

This tutorial is divided into the following sections:

20.1. Introduction

This tutorial illustrates how to set up and solve a problem involving solidification and will demonstrate how to do the following:

- Define a solidification problem.
- Define pull velocities for simulation of continuous casting.
- Define a surface tension gradient for Marangoni convection.
- Solve a solidification problem.

20.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

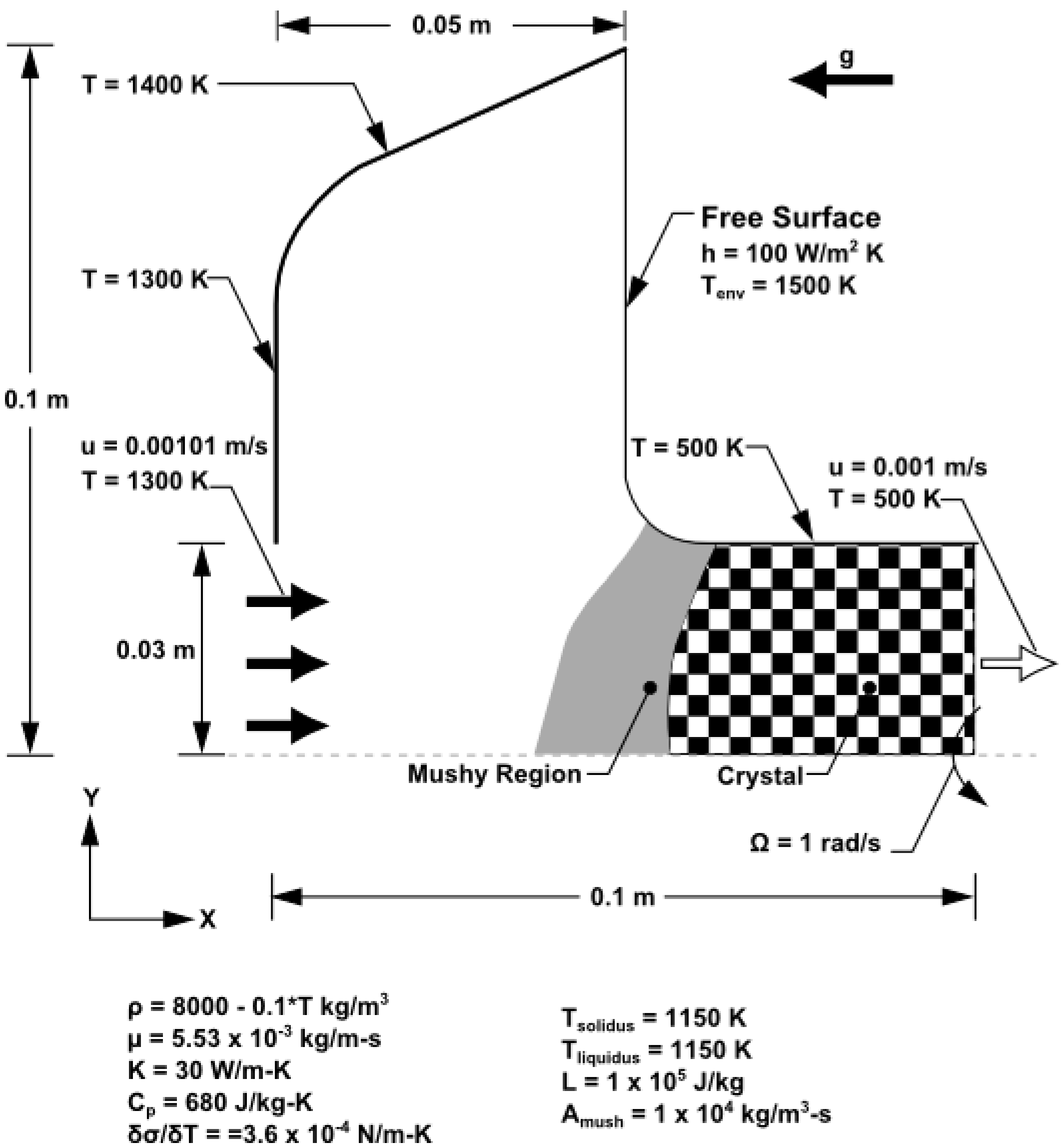
and that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

20.3. Problem Description

This tutorial demonstrates the setup and solution procedure for a fluid flow and heat transfer problem involving solidification, namely the Czochralski growth process. The geometry considered is a 2D axisymmetric bowl (shown in Figure 20.1: Solidification in Czochralski Model (p. 808)), containing liquid metal. The bottom and sides of the bowl are heated above the liquidus temperature, as is the free surface of the liquid. The liquid is solidified by heat loss from the crystal and the solid is pulled out of the domain at a rate of 0.001 m/s and a temperature of 500 K. There is a steady injection of liquid at the bottom of the bowl with a velocity of $1.01 \times 10^{-3} m/s$ and a temperature of 1300 K. Material properties are listed in Figure 20.1: Solidification in Czochralski Model (p. 808).

Starting with an existing 2D mesh, the details regarding the setup and solution procedure for the solidification problem are presented. The steady conduction solution for this problem is computed as an initial condition. Then, the fluid flow is enabled to investigate the effect of natural and Marangoni convection in a transient fashion.

Figure 20.1: Solidification in Czochralski Model



In the above figure, A_{mush} is the mushy zone constant. For details on modeling the solidification/melting process, refer to momentum equations in the Fluent Theory Guide.

20.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

20.4.1. Preparation

To prepare for running this tutorial:

1. Set up a working folder on the computer you will be using.

2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

   Note

   If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

3. Enter the name of this tutorial into the search bar.

4. Narrow the results by using the filter on the left side of the page.

   a. Click **ANSYS Fluent** under **Product**.

   b. Click **18.0** under **Version**.

5. Select this tutorial from the list.

6. Click the **solidification_R180.zip** link to download the input files.

7. Unzip the `solidification_R180.zip` file you downloaded to your working folder.

   The file `solid.msh` can be found in the `solidification` directory created after unzipping the file.

8. Use Fluent Launcher to start the **2D** single precision (disable **Double Precision**) version of ANSYS Fluent.

   Fluent Launcher displays your **Display Options** preferences from the previous session.

   For more information about the Fluent Launcher, see starting ANSYS Fluent using the Fluent Launcher in the Fluent Getting Started Guide.

9. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.

10. Ensure that the **Serial** processing option is selected.

20.4.2. Reading and Checking the Mesh

1. Read the mesh file `solid.msh`.

File → Read → Mesh...

As the mesh is read by ANSYS Fluent, messages will appear in the console reporting the progress of the reading.

A warning about the use of axis boundary conditions is displayed in the console. You are asked to consider making changes to the zone type or change the problem definition to axisymmetric. You will change the problem to axisymmetric swirl later in this tutorial.

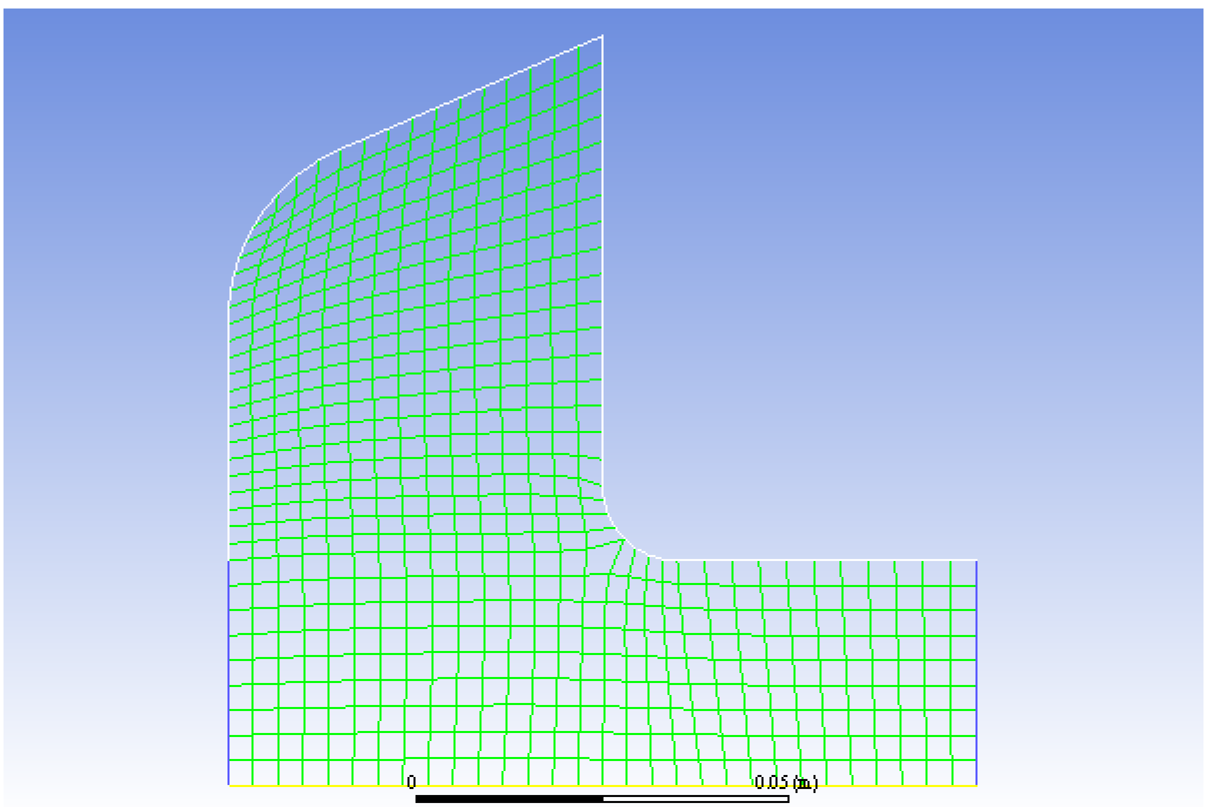
2. Check the mesh.

   Setting Up Domain → Mesh → Check

   ANSYS Fluent will perform various checks on the mesh and will report the progress in the console. Make sure that the minimum volume is a positive number.

3. Examine the mesh (Figure 20.2: Mesh Display (p. 810)).

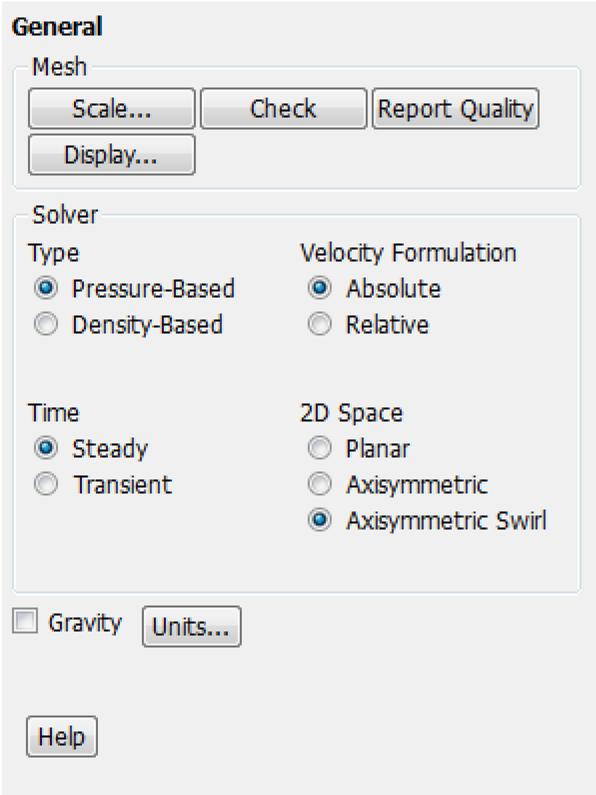
Figure 20.2: Mesh Display



20.4.3. Specifying Solver and Analysis Type

1. Select **Axisymmetric Swirl** from the **2D Space** list.

   Setup → General

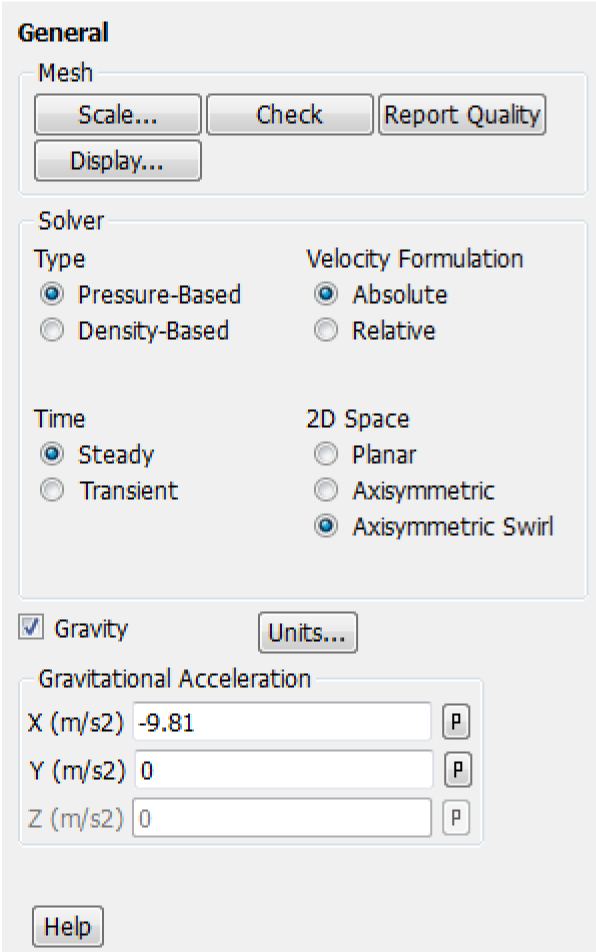


The geometry comprises an axisymmetric bowl. Furthermore, swirling flows are considered in this problem, so the selection of ***Axisymmetric Swirl*** *best defines this geometry.*

Also, note that the rotation axis is the X axis. Hence, the X direction is the axial direction and the Y direction is the radial direction. When modeling axisymmetric swirl, the swirl direction is the tangential direction.

2. Add the effect of gravity on the model.

Setup → **General** → **Gravity**



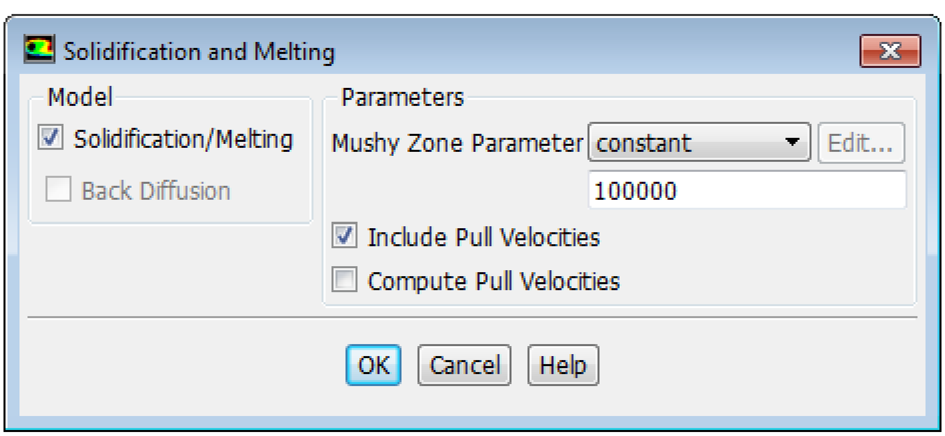
a. Enable **Gravity**.

b. Enter $-9.81\ m/s^2$ for **X** in the **Gravitational Acceleration** group box.

20.4.4. Specifying the Models

1. Define the solidification model.

Setup → **Models** → **Solidification & Melting** **Edit...**



a. Enable the **Solidification/Melting** option in the **Solidification and Melting** dialog box.

*The **Solidification and Melting** dialog box will expand to show the related parameters.*

b. Retain the default value of `100000` for the **Mushy Zone Constant**.

This default value is acceptable for most cases.

c. Enable the **Include Pull Velocities** option.

By including the pull velocities, you will account for the movement of the solidified material as it is continuously withdrawn from the domain in the continuous casting process.

*When you enable this option, the **Solidification and Melting** dialog box will expand to show the **Compute Pull Velocities** option. If you were to enable this additional option, ANSYS Fluent would compute the pull velocities during the calculation. This approach is computationally expensive and is recommended only if the pull velocities are strongly dependent on the location of the liquid-solid interface. In this tutorial, you will patch values for the pull velocities instead of having ANSYS Fluent compute them.*

For more information about computing the pull velocities, see the Fluent User's Guide.

d. Click **OK** to close the **Solidification and Melting** dialog box.

*An **Information** dialog box opens, telling you that available material properties have changed for the solidification model. You will set the material properties later, so you can click **OK** in the dialog box to acknowledge this information.*

Note

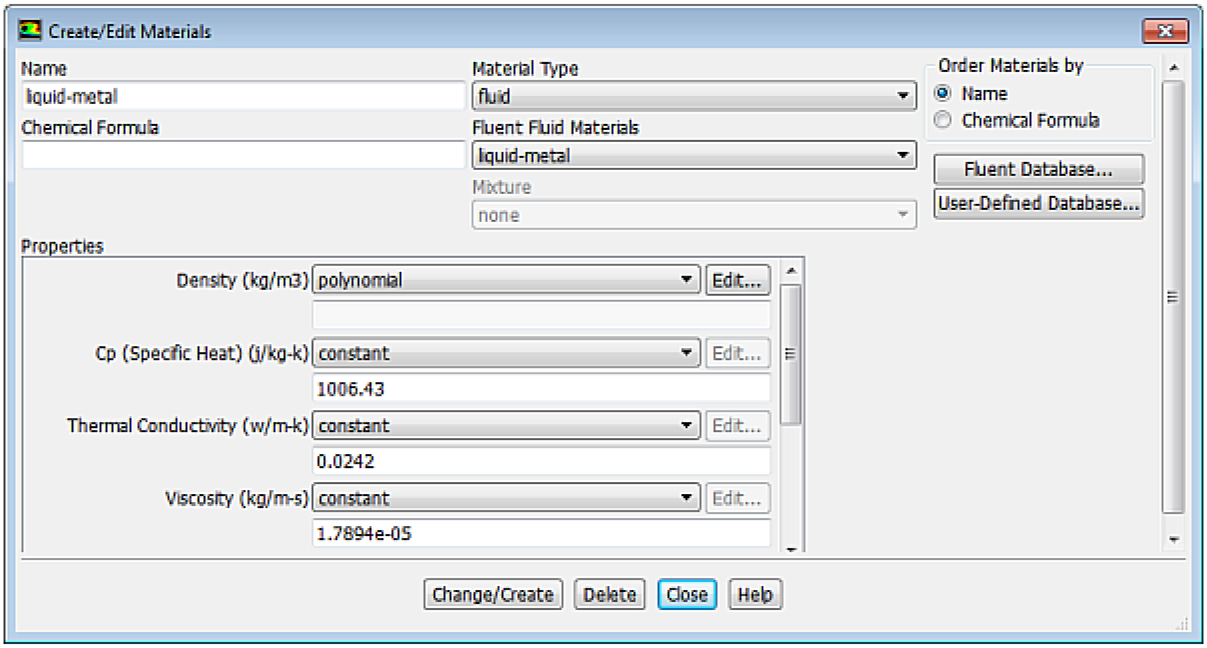
ANSYS Fluent will automatically enable the energy calculation when you enable the solidification model, so you need not visit the **Energy** dialog box.

20.4.5. Defining Materials

In this step, you will create a new material and specify its properties, including the melting heat, solidus temperature, and liquidus temperature.

1. Define a new material.

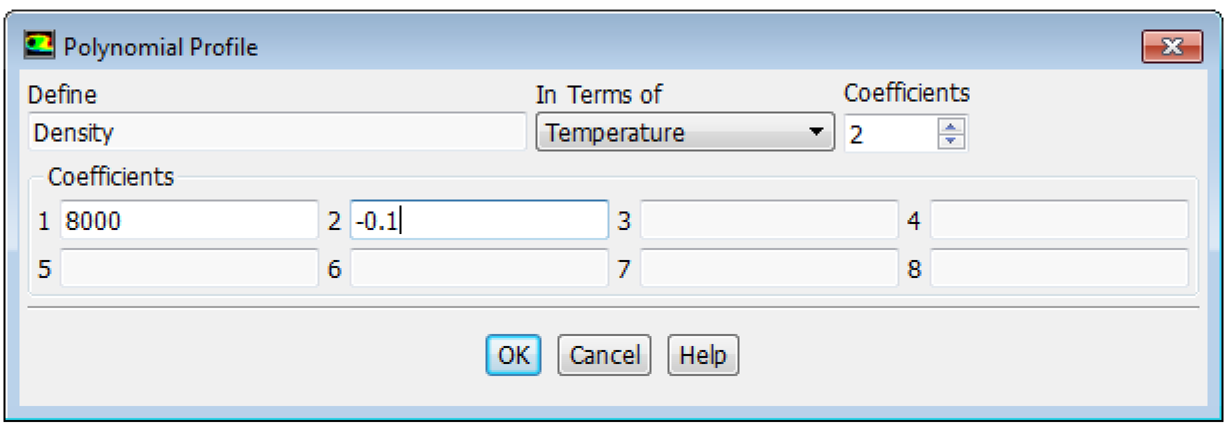
Setup → **Materials** → **Fluid** → **air** → **Edit...**



a. Enter `liquid-metal` for **Name**.

b. Select **polynomial** from the **Density** drop-down list in the **Properties** group box.

c. Configure the following settings In the **Polynomial Profile** dialog box:



i. Set **Coefficients** to 2.

ii. In the **Coefficients** group box, enter `8000` for **1** and `-0.1` for **2**.

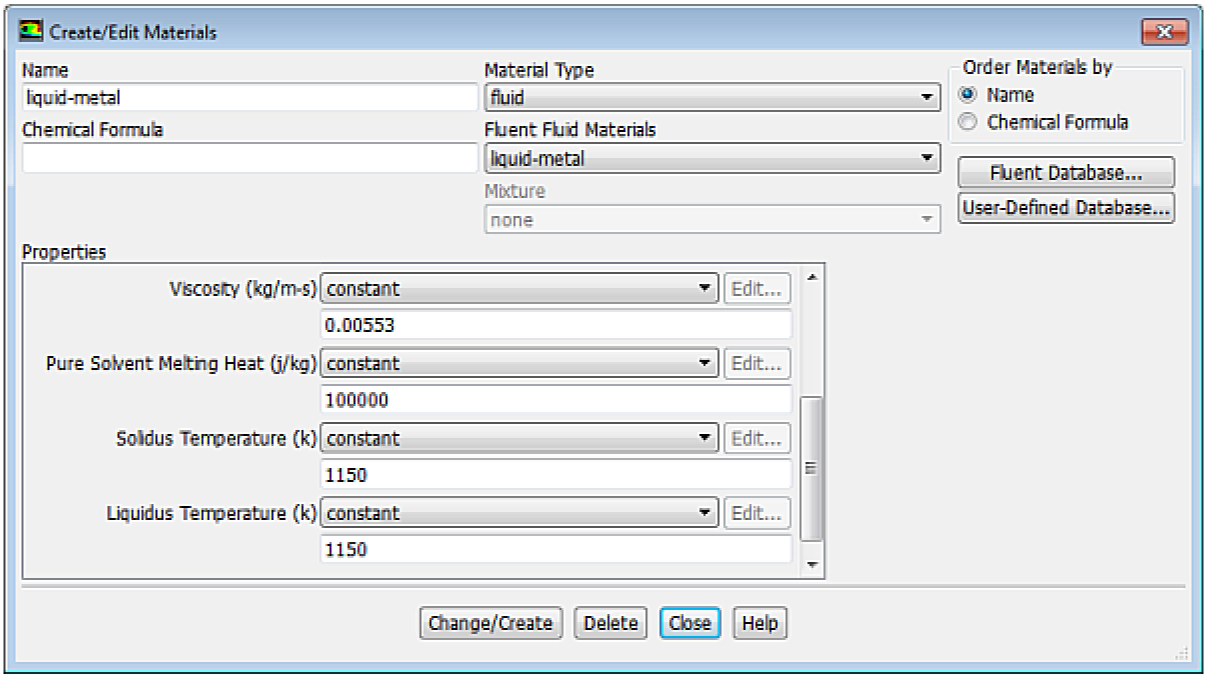
As shown in Figure 20.1: Solidification in Czochralski Model (p. 808), the density of the material is defined by a polynomial function: $\rho = 8000 - 0.1T$.

iii. Click **OK** to close the **Polynomial Profile** dialog box.

d. In the **Question** dialog box, click **Yes** to overwrite **fluid-1** and add the new material (**liquid-metal**) to the **Fluent Fluid Materials** drop-down list.

e. Click **OK** in the information dialog box.

f. Select **liquid-metal** from the **Fluent Fluid Materials** drop-down list to set the other material properties.



g. Enter `680` $j/ kg{-}k$ for **Cp (Specific Heat)**.

h. Enter `30` $w/ m{-}k$ for **Thermal Conductivity**.

i. Enter `0.00553` $kg/ m{-}s$ for **Viscosity**.

j. Enter `100000` j/ kg for **Pure Solvent Melting Heat**.

   *Scroll down the group box to find **Pure Solvent Melting Heat** and the properties that follow.*

k. Enter `1150` K for **Solidus Temperature**.

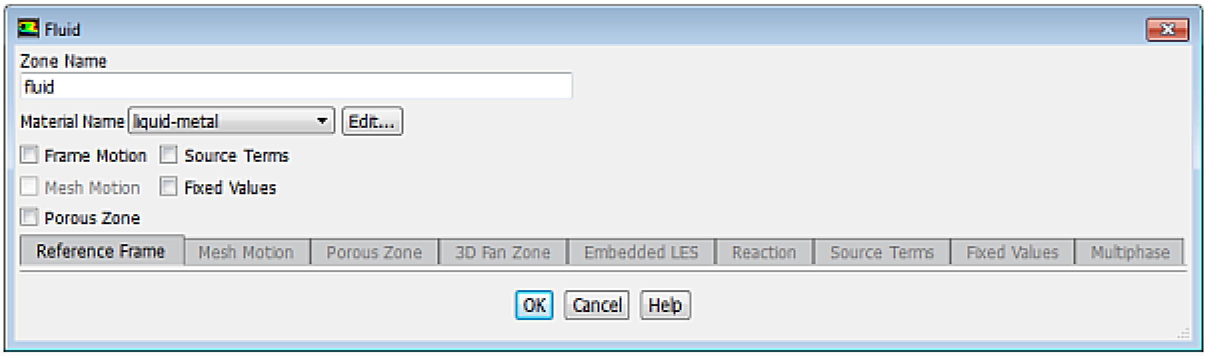
l. Enter `1150` K for **Liquidus Temperature**.

m. Click **Change/Create** and close the **Create/Edit Materials** dialog box.

20.4.6. Setting the Cell Zone Conditions

1. Set the cell zone conditions for the fluid (**fluid**).

   Setup → **Cell Zone Conditions** → **fluid** → **Edit...**



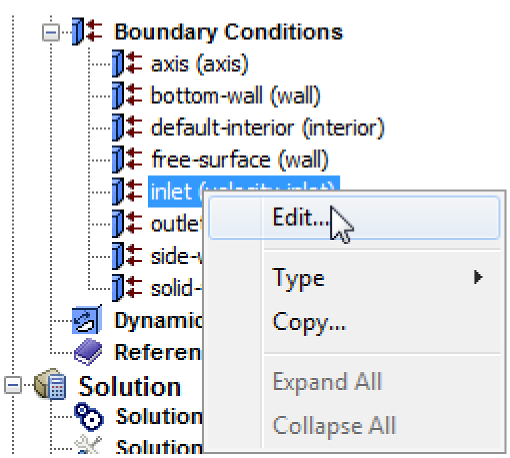
a. Select **liquid-metal** from the **Material Name** drop-down list.

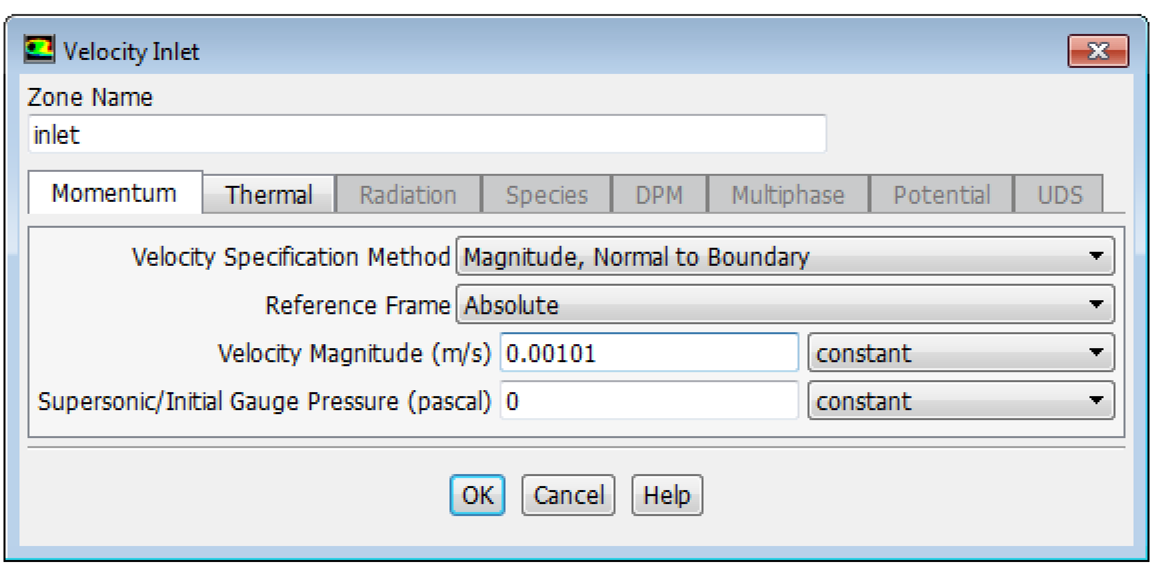
b. Click **OK** to close the **Fluid** dialog box.

20.4.7. Setting the Boundary Conditions

1. Set the boundary conditions for the inlet (**inlet**).

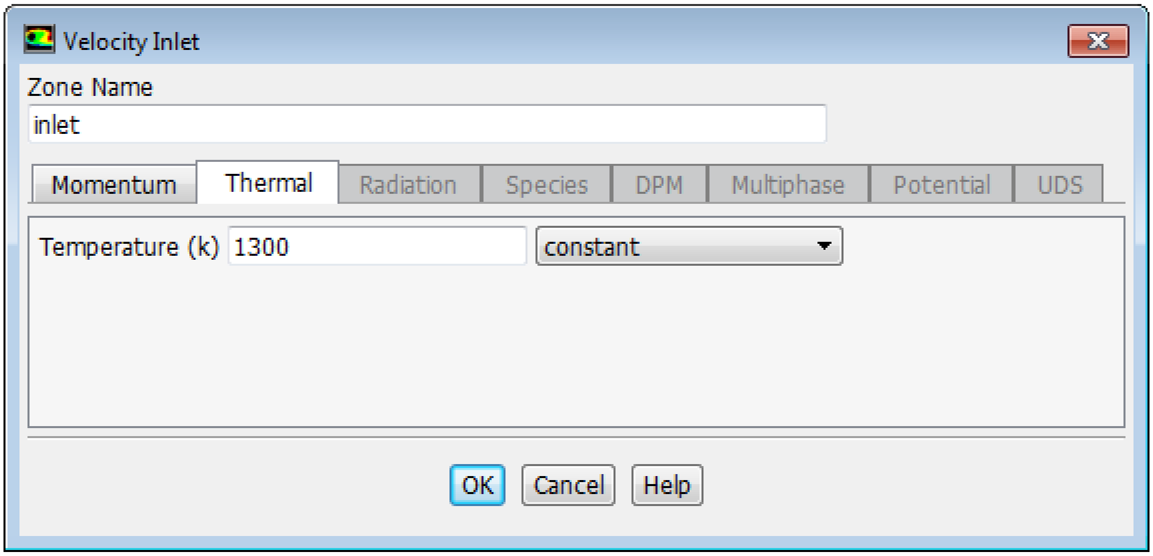
Setup → **Boundary Conditions** → **inlet** **Edit...**





a. Enter `0.00101` m/s for **Velocity Magnitude**.

b. Click the **Thermal** tab and enter `1300` K for **Temperature**.

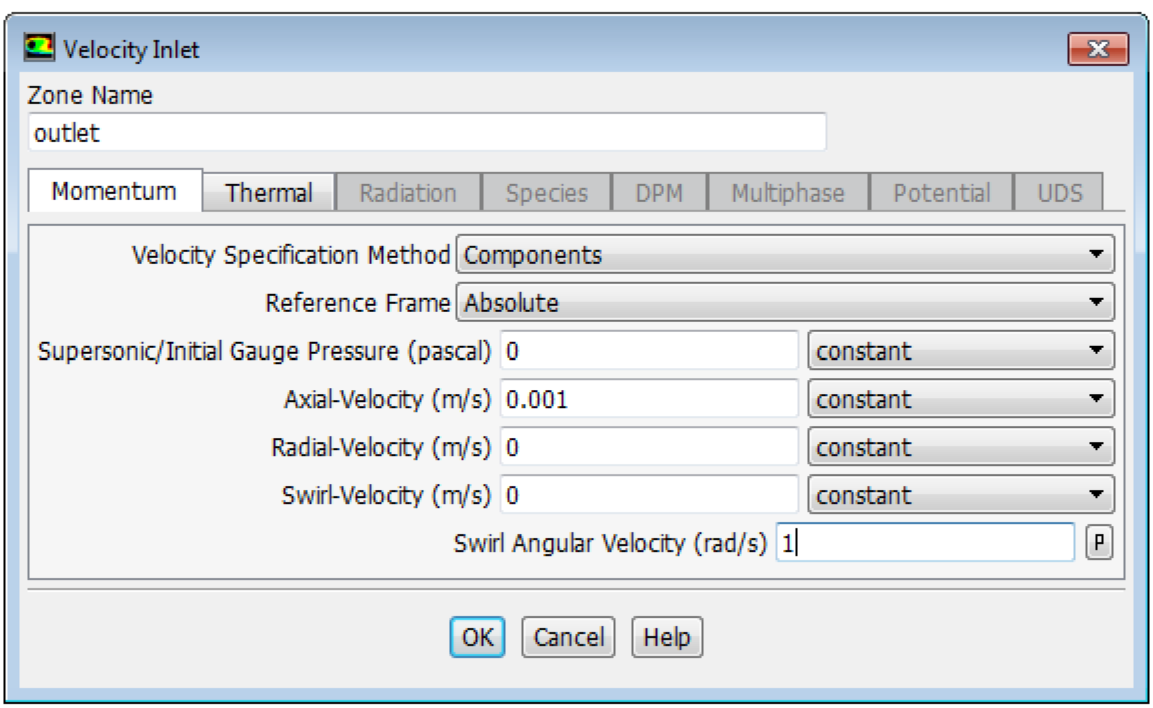


c. Click **OK** to close the **Velocity Inlet** dialog box.

2. Set the boundary conditions for the outlet (**outlet**).

Setup → **Boundary Conditions** → **outlet** **Edit...**

Here, the solid is pulled out with a specified velocity, so a velocity inlet boundary condition is used with a positive axial velocity component.



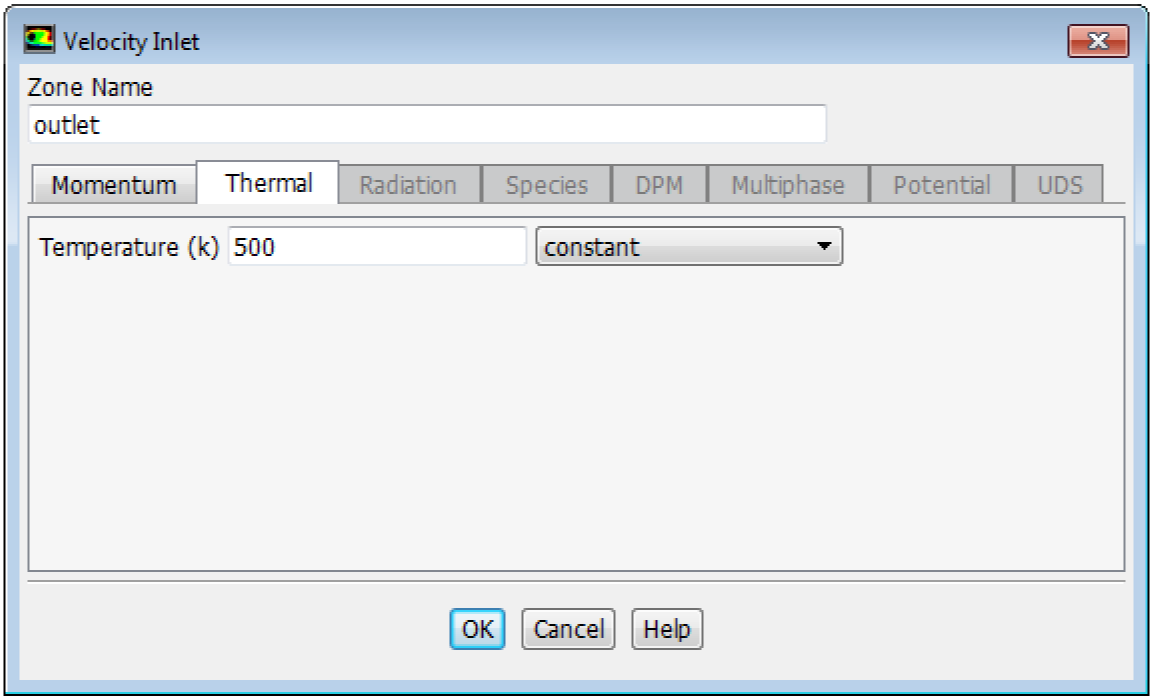
a. From the **Velocity Specification Method** drop-down list, select **Components**.

*The **Velocity Inlet** dialog box will change to show related inputs.*

b. Enter `0.001` m/s for **Axial-Velocity**.

c. Enter `1` rad/s for **Swirl Angular Velocity**.

d. Click the **Thermal** tab and enter `500` K for **Temperature**.

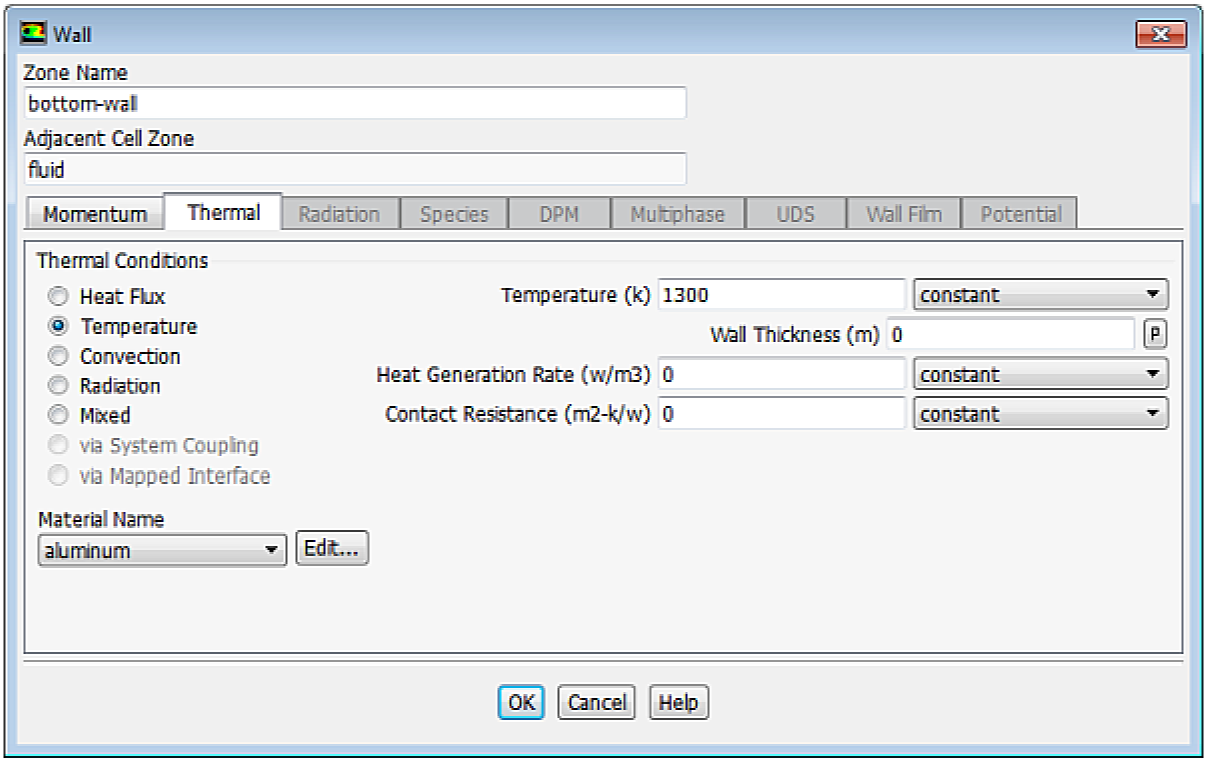


e. Click **OK** to close the **Velocity Inlet** dialog box.

3. Set the boundary conditions for the bottom wall (**bottom-wall**).

Setup → **Boundary Conditions** → **bottom-wall** **Edit...**

a. Click the **Thermal** tab.



i. Select **Temperature** in the **Thermal Conditions** group box.

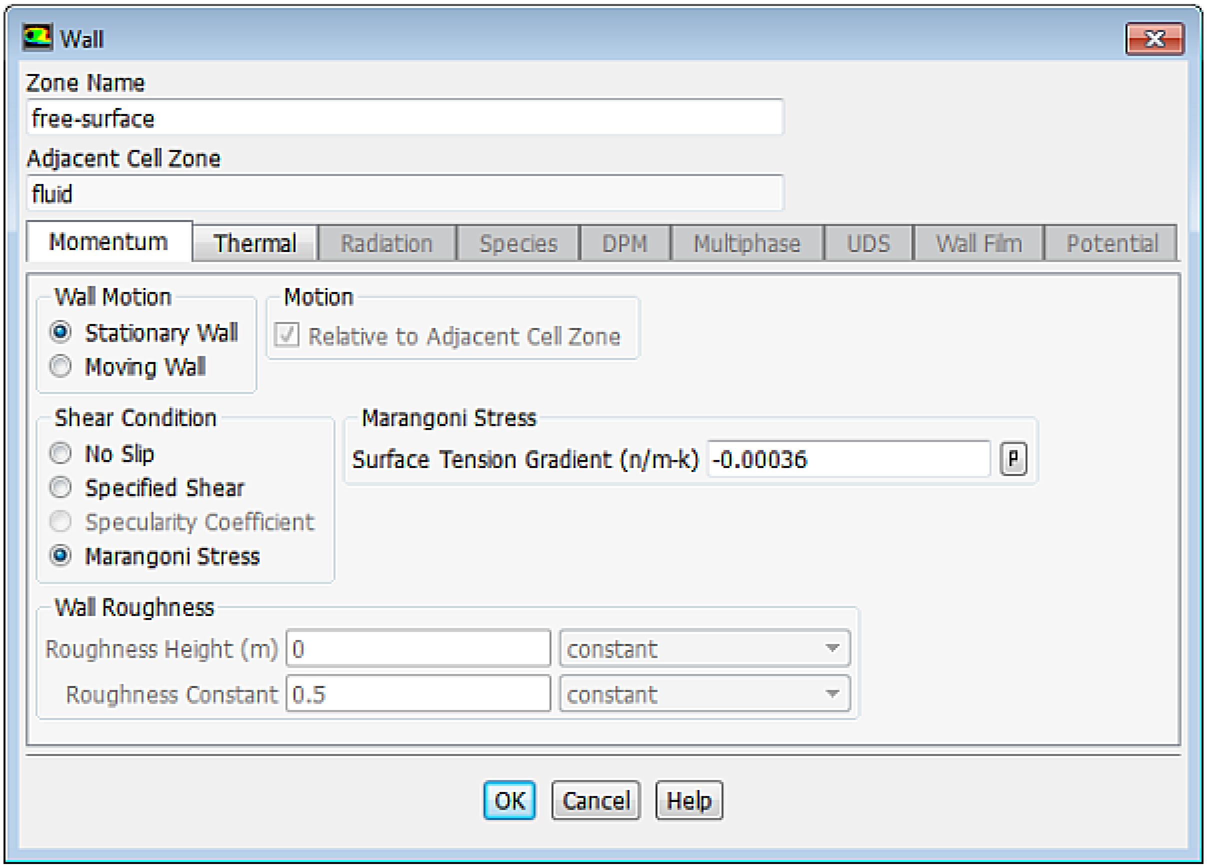
ii. Enter 1300 K for **Temperature**.

b. Click **OK** to close the **Wall** dialog box.

4. Set the boundary conditions for the free surface (**free-surface**).

Setup → **Boundary Conditions** → **free-surface** **Edit...**

The specified shear and Marangoni stress boundary conditions are useful in modeling situations in which the shear stress (rather than the motion of the fluid) is known. A free surface condition is an example of such a situation. In this case, the convection is driven by the Marangoni stress and the shear stress is dependent on the surface tension, which is a function of temperature.

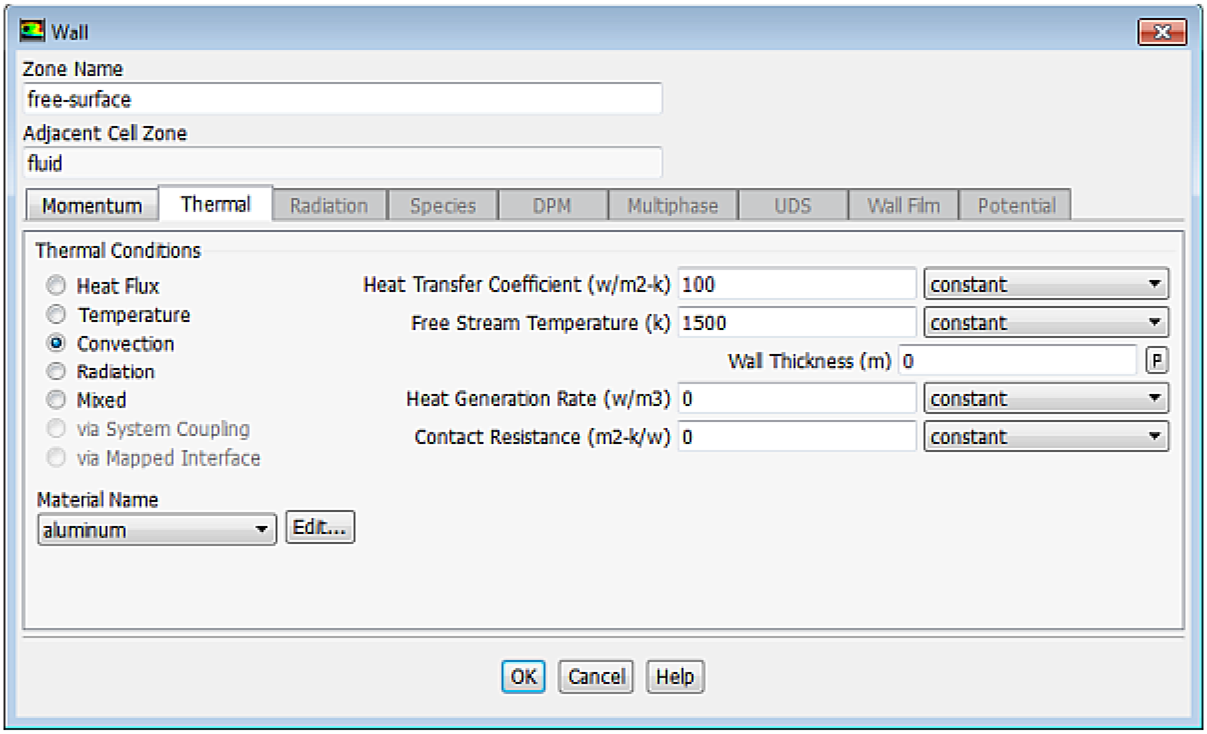


a. Select **Marangoni Stress** in the **Shear Condition** group box.

   *The **Marangoni Stress** condition allows you to specify the gradient of the surface tension with respect to temperature at a wall boundary.*

b. Enter `-0.00036` $n/\ m\text{−}k$ for **Surface Tension Gradient**.

c. Click the **Thermal** tab to specify the thermal conditions.



i. Select **Convection** from the **Thermal Conditions** group box.

ii. Enter `100` $w/\ m^2{-}k$ for **Heat Transfer Coefficient**.

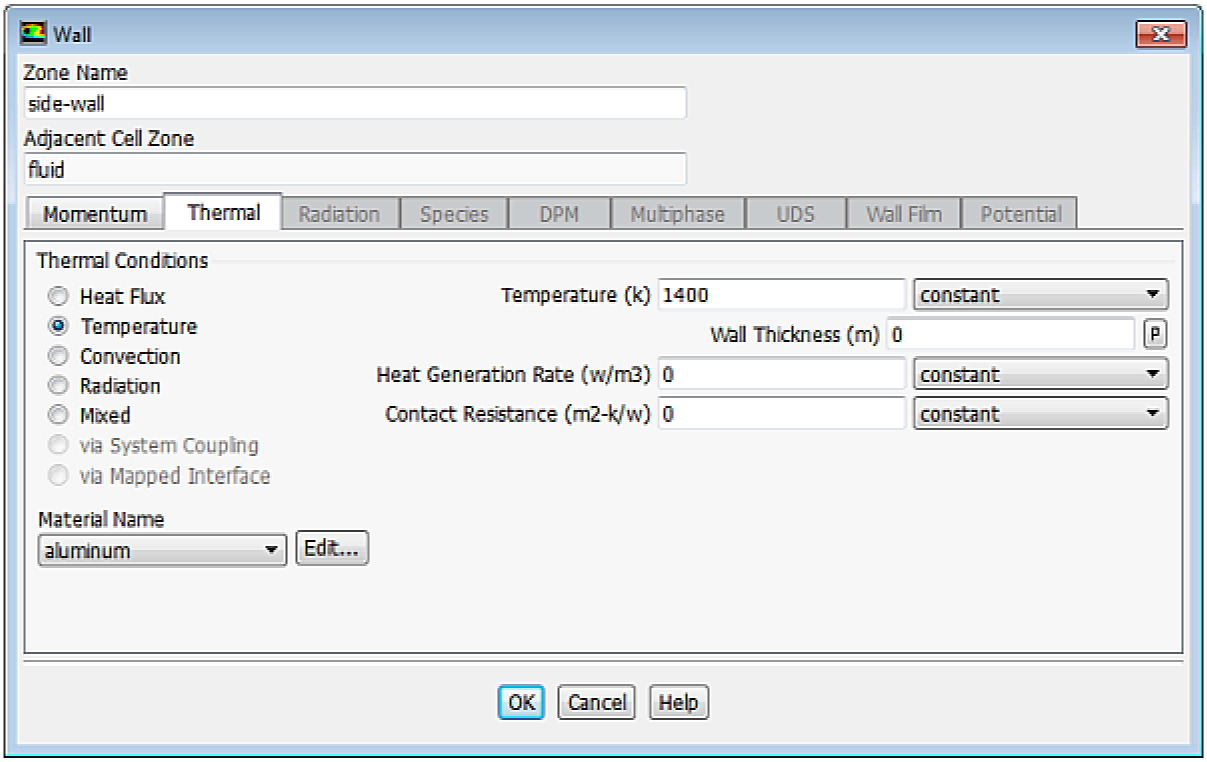
iii. Enter `1500` K for **Free Stream Temperature**.

d. Click **OK** to close the **Wall** dialog box.

5. Set the boundary conditions for the side wall (**side-wall**).

Setup → **Boundary Conditions** → **side-wall** **Edit...**

a. Click the **Thermal** tab.



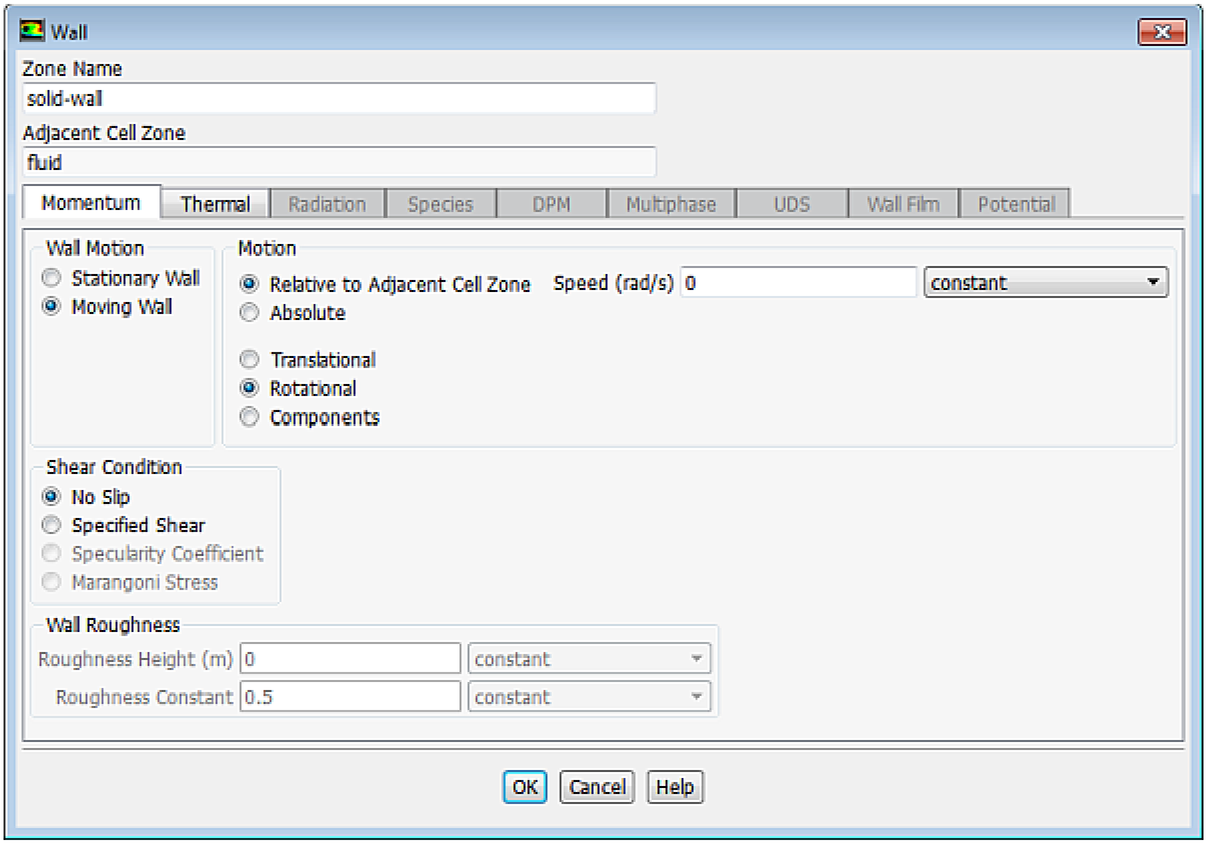
i. Select **Temperature** from the **Thermal Conditions** group box.

ii. Enter `1400` K for the **Temperature**.

b. Click **OK** to close the **Wall** dialog box.

6. Set the boundary conditions for the solid wall (**solid-wall**).

Setup → **Boundary Conditions** → **solid-wall** **Edit...**



a. From the **Wall Motion** group box, select **Moving Wall**.

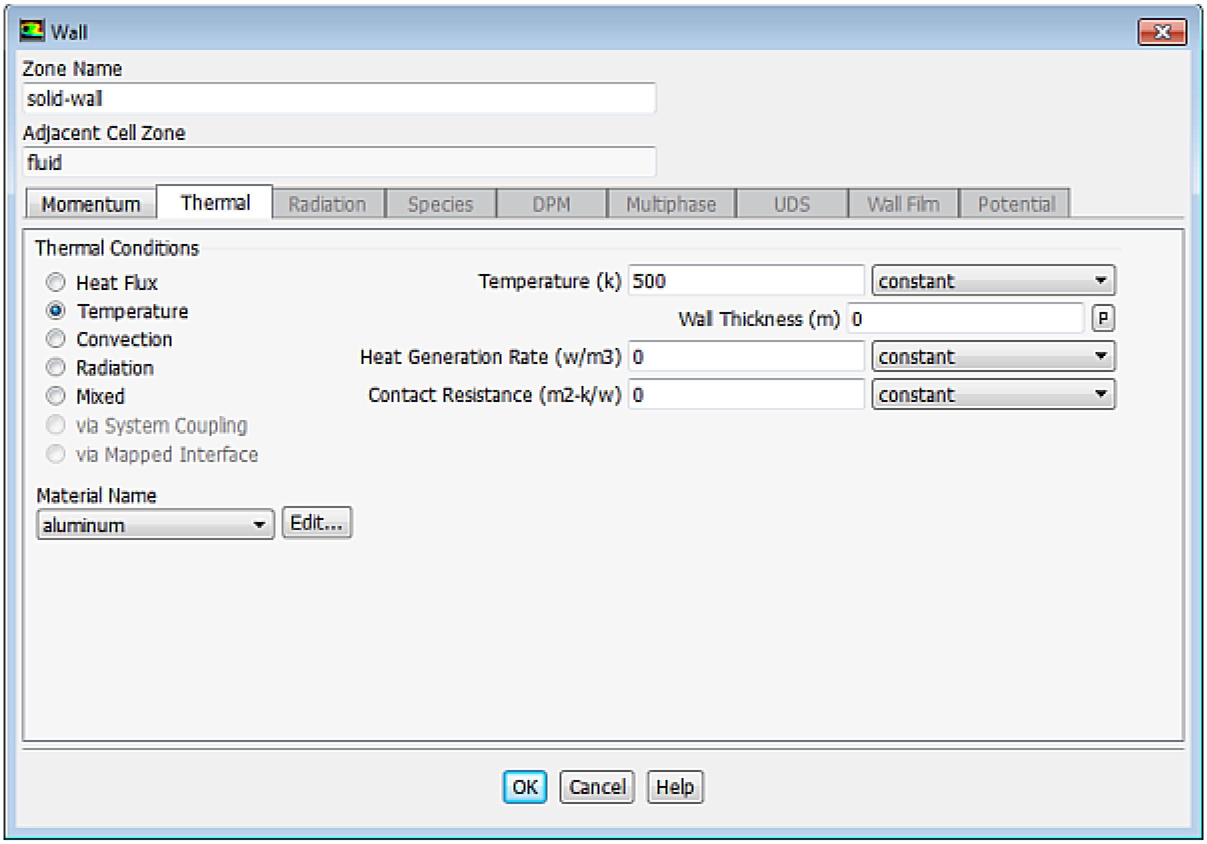
*The **Wall** dialog box is expanded to show additional parameters.*

b. in the **Motion** group box, in the lower box, select **Rotational**.

*The **Wall** dialog box is changed to show the rotational speed.*

c. Enter `1.0` rad/s for **Speed**.

d. Click the **Thermal** tab to specify the thermal conditions.



i. Select **Temperature** from the **Thermal Conditions** selection list.

ii. Enter 500 K for **Temperature**.

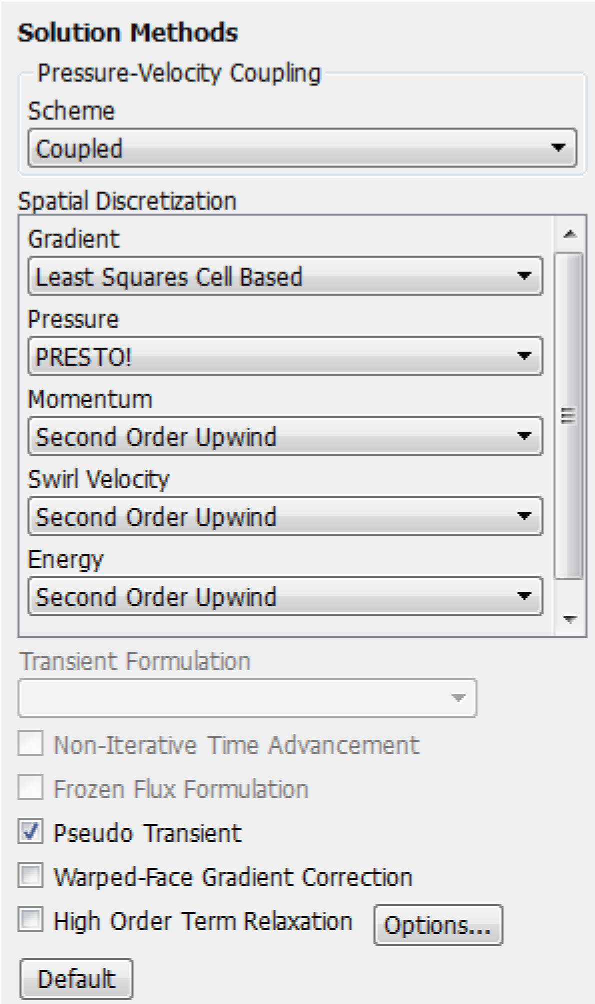
e. Click **OK** to close the **Wall** dialog box.

20.4.8. Solution: Steady Conduction

In this step, you will specify the discretization schemes to be used and temporarily disable the calculation of the flow and swirl velocity equations, so that only conduction is calculated. This steady-state solution will be used as the initial condition for the time-dependent fluid flow and heat transfer calculation.

1. Set the solution parameters.

Solving → **Solution** → **Methods...**



a. Select **Coupled** from the **Scheme** drop-down list in the **Pressure-Velocity Coupling** group box.

b. Select **PRESTO!** from the **Pressure** drop-down list in the **Spatial Discretization** group box.

   *The **PRESTO!** scheme is well suited for rotating flows with steep pressure gradients.*

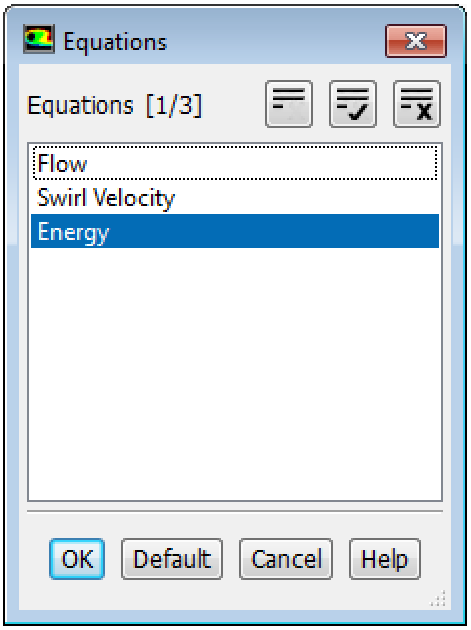
c. Retain the default selection of **Second Order Upwind** from the **Momentum**, **Swirl Velocity**, and **Energy** drop-down lists.

d. Enable **Pseudo Transient**.

   The Pseudo Transient option enables the pseudo transient algorithm in the coupled pressure-based solver. This algorithm effectively adds an unsteady term to the solution equations in order to improve stability and convergence behavior. Use of this option is recommended for general fluid flow problems.

2. Enable the calculation for energy.

   Solving → Controls → Equations...

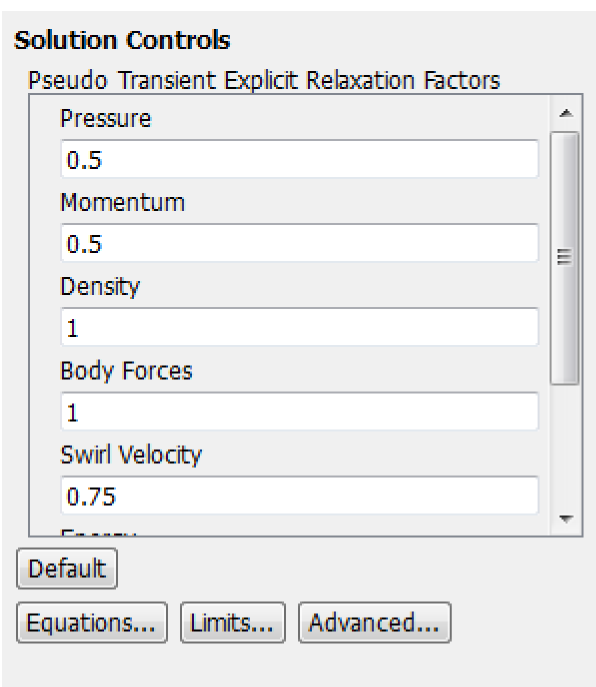


a. Deselect **Flow** and **Swirl Velocity** from the **Equations** selection list to disable the calculation of flow and swirl velocity equations.

b. Click **OK** to close the **Equations** dialog box.

3. Confirm the **Relaxation Factors**.

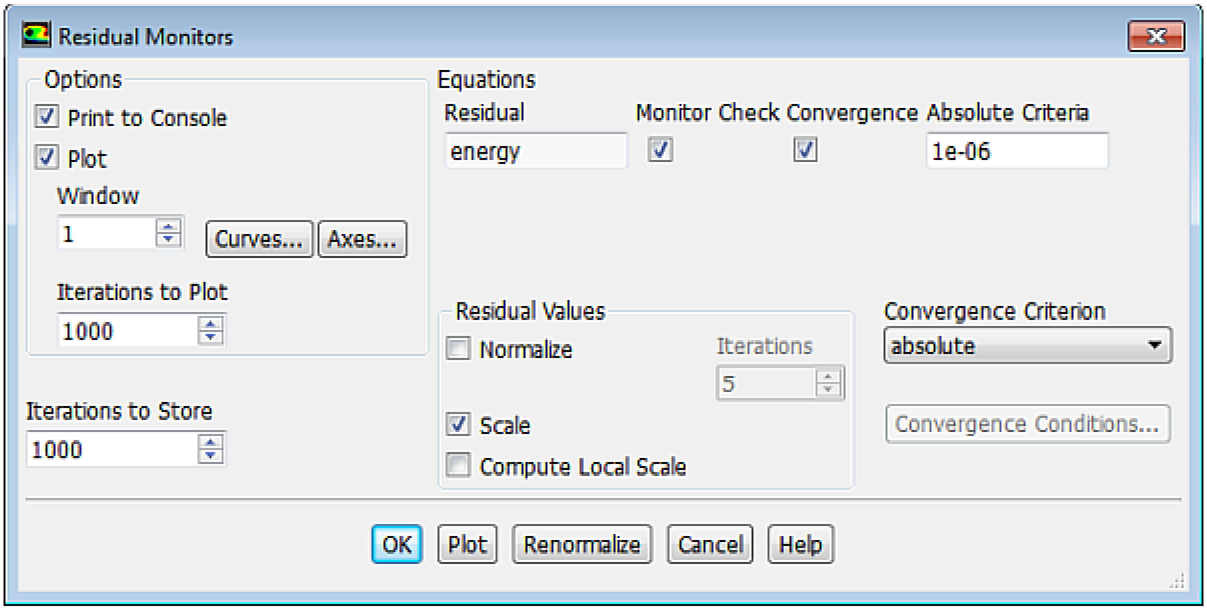
Solving → **Controls** → **Controls...**



Retain the default values.

4. Enable the plotting of residuals during the calculation.

Solving → Reports → Residuals...

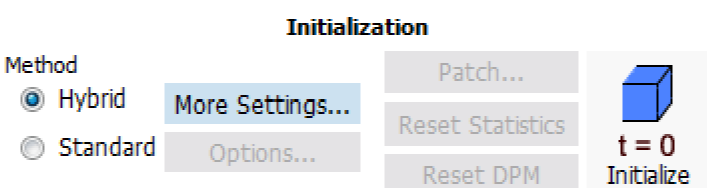


a. Ensure **Plot** is enabled in the **Options** group box.

b. Click **OK** to accept the remaining default settings and close the **Residual Monitors** dialog box.

5. Initialize the solution.

Solving → Initialization



a. Retain the **Method** at the default of**Hybrid** in the **Initialization** group.

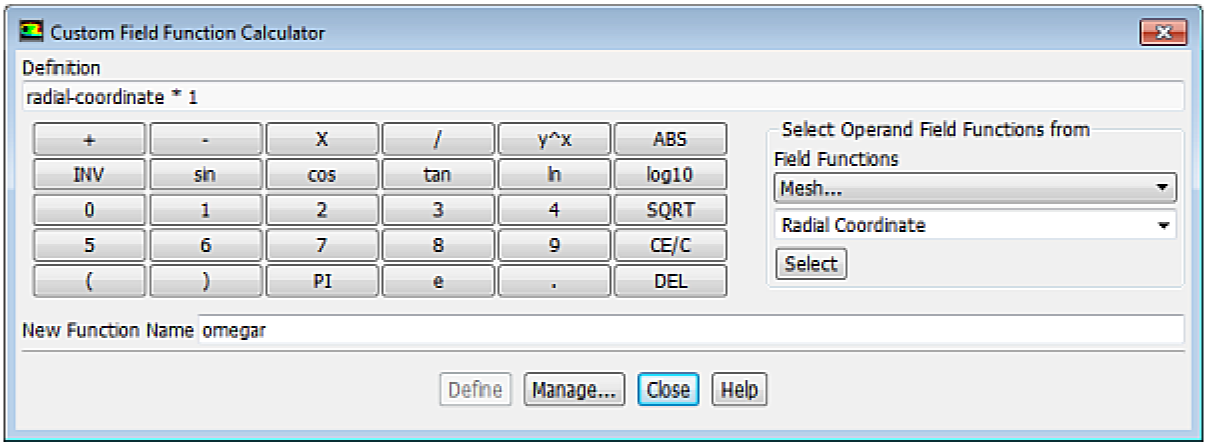
For flows in complex topologies, hybrid initialization will provide better initial velocity and pressure field than standard initialization. This in general will help in improving the convergence behavior of the solver.

b. Click **Initialize**.

6. Define a custom field function for the swirl pull velocity.

Parameters & Customization → Custom Field Functions **New...**

In this step, you will define a field function to be used to patch a variable value for the swirl pull velocity in the next step. The swirl pull velocity is equal to $\Omega \cdot r$, where Ω is the angular velocity, and r is the radial coordinate. Since Ω = 1 rad/s, you can simplify the equation to simply r. In this example, the value of Ω is included for demonstration purposes.



a. From the **Field Functions** drop-down lists, select **Mesh...** and **Radial Coordinate**.

b. Click the **Select** button to add `radial-coordinate` in the **Definition** field.

If you make a mistake, click the ***DEL*** *button on the calculator pad to delete the last item you added to the function definition.*

c. Click the × button on the calculator pad.

d. Click the **1** button.

e. Enter `omegar` for **New Function Name**.

f. Click **Define**.

The `omegar` item appears under the **Parameters & Customisation/Parameters** tree branch.

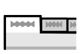
Note

To check the function definition or delete the custom field function, click **Manage...**. Then in the **Field Function Definitions** dialog box, from the **Field Functions** selection list, select **omegar** to view the function definition.

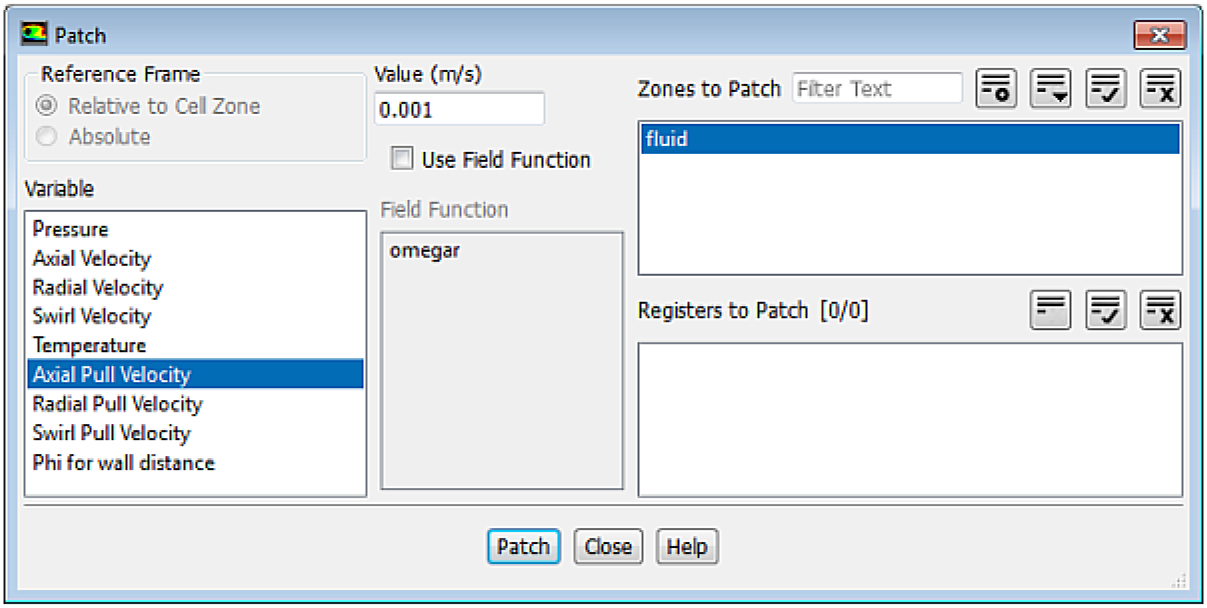
---

g. Close the **Custom Field Function Calculator** dialog box.

7. Patch the pull velocities.

 **Solving** → **Initialization** → **Patch...**

As noted earlier, you will patch values for the pull velocities, rather than having ANSYS Fluent compute them. Since the radial pull velocity is zero, you will patch just the axial and swirl pull velocities.



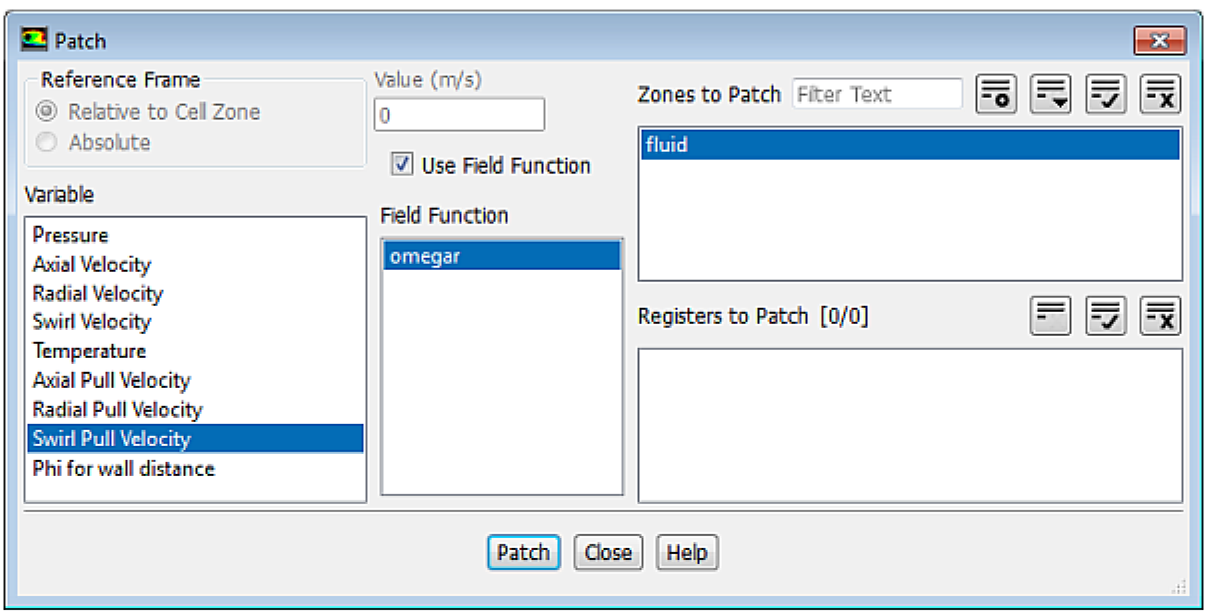
a. From the **Variable** selection list, select **Axial Pull Velocity**.

b. Enter `0.001` *m/ s* for **Value**.

c. From the **Zones to Patch** selection list, select **fluid**.

d. Click **Patch**.

You have just patched the axial pull velocity. Next you will patch the swirl pull velocity.



e. From the **Variable** selection list, select **Swirl Pull Velocity**.

*Scroll down the list to find **Swirl Pull Velocity**.*

f. Enable the **Use Field Function** option.

g. Select **omegar** from the **Field Function** selection list.

h. Ensure that **fluid** is selected from the **Zones to Patch** selection list.

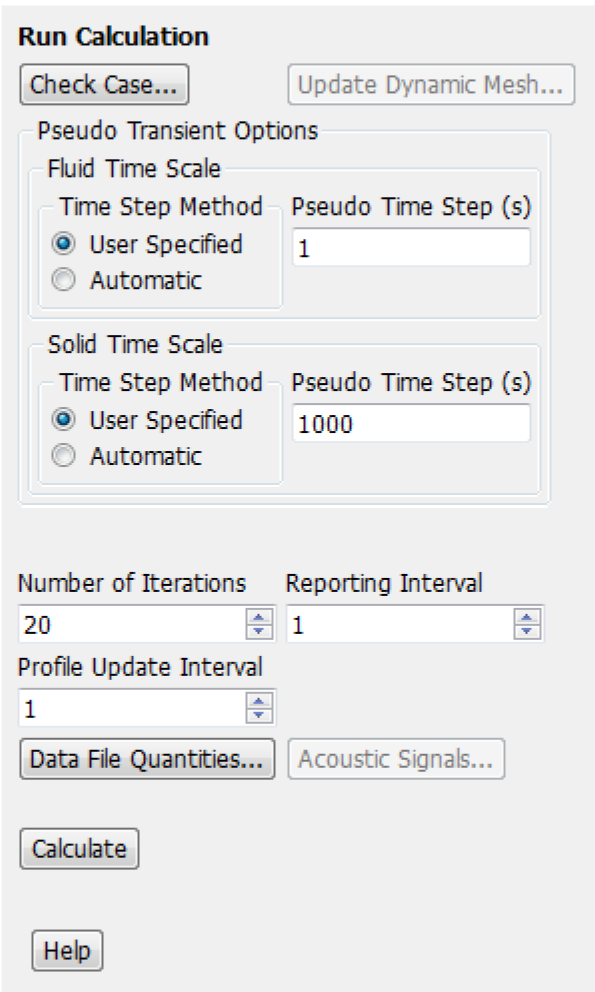
i. Click **Patch** and close the **Patch** dialog box.

8. Save the initial case and data files (`solid0.cas.gz` and `solid0.dat.gz`).

   File → Write → Case & Data...

9. Start the calculation by requesting 20 iterations.

   Solving → Run Calculation → Advanced...



a. In the **Run Calculation** task page, select **User Specified** for the **Time Step Method** in both the **Fluid Time Scale** and the **Solid Time Scale** group boxes.

b. Retain the default values of `1` and `1000` for the **Pseudo Time Step (s)** in the **Fluid Time Scale** and the **Solid Time Scale** group boxes, respectively.

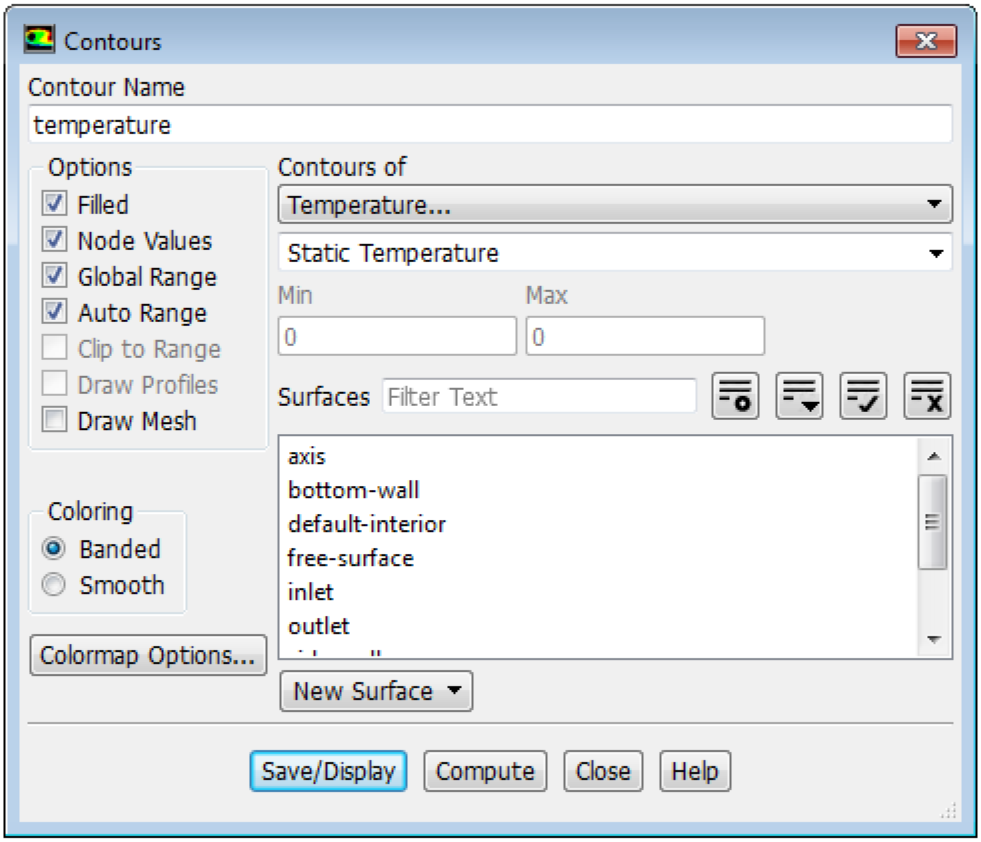
c. Enter `20` for **Number of Iterations**.

d. Click **Calculate**.

The solution will converge in approximately 12 iterations.

10. Create and display the definition of filled temperature contours (Figure 20.3: Contours of Temperature for the Steady Conduction Solution (p. 832)).

    Postprocessing → **Graphics** → **Contours** → **New...**



    a. Enter `temperature` for **Contour Name**.

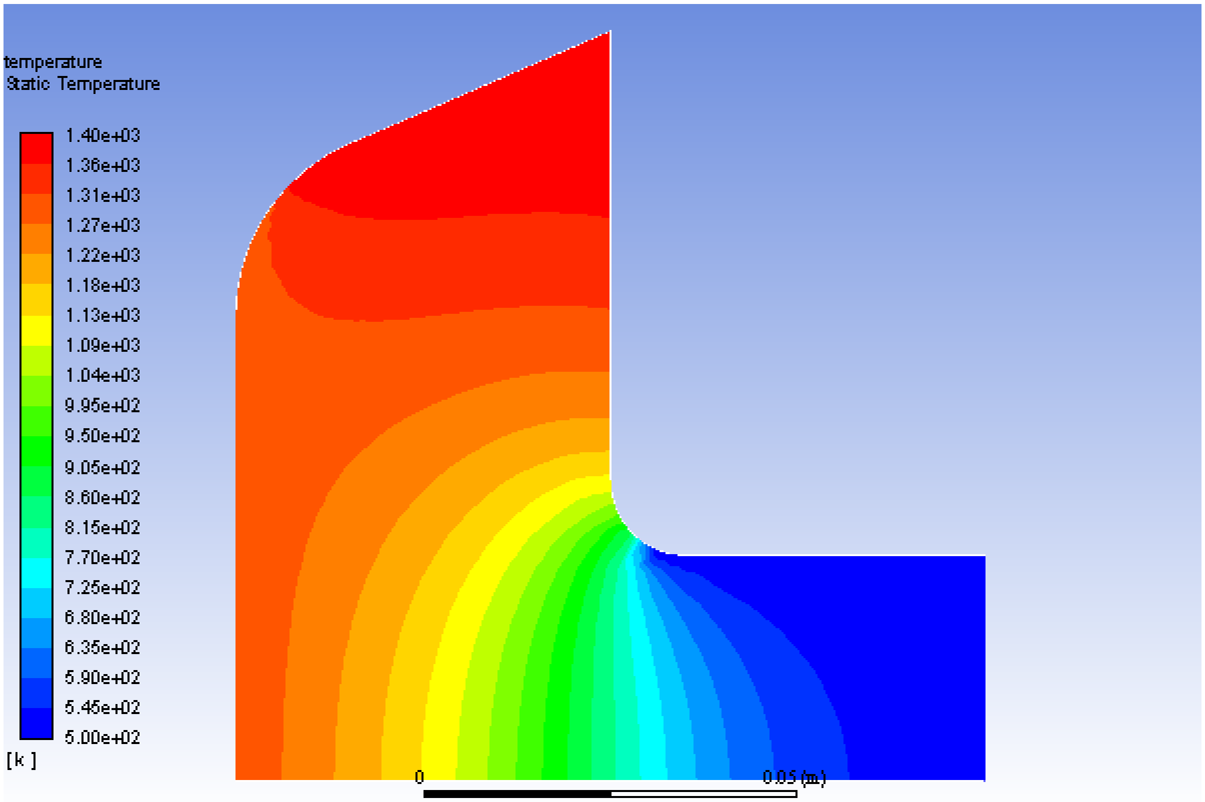
    b. Enable the **Filled** option.

    c. Select **Temperature...** and **Static Temperature** from the **Contours of** drop-down lists.

    d. Click **Save/Display** (Figure 20.3: Contours of Temperature for the Steady Conduction Solution (p. 832)).

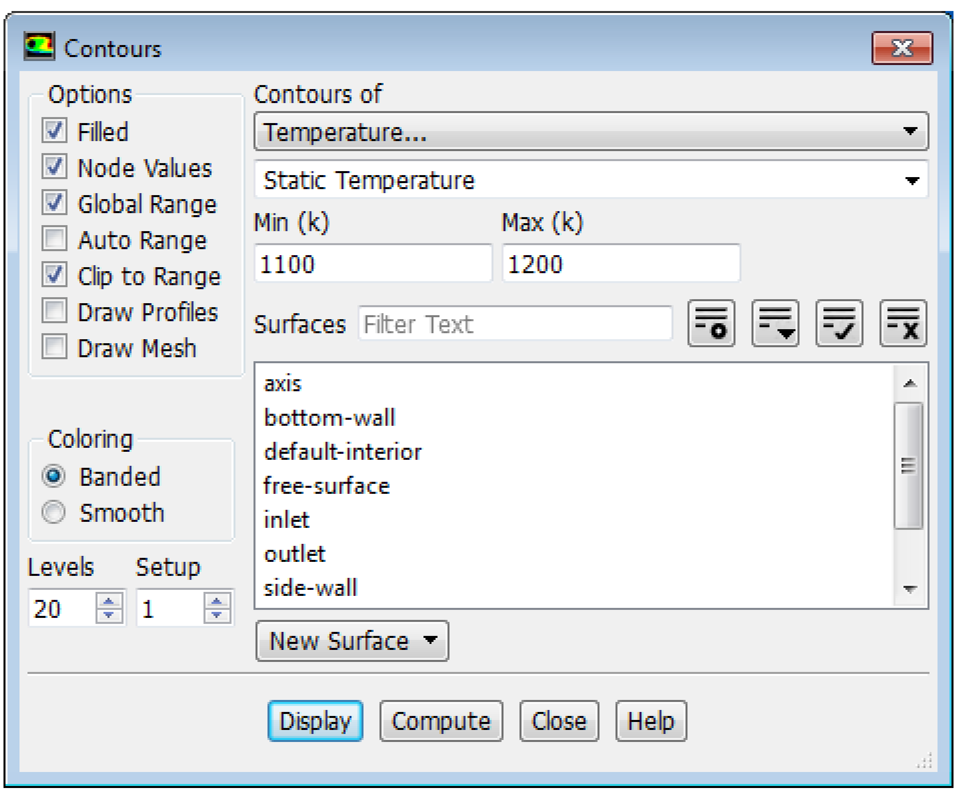
       The **temperature** contour definition appear under the **Results/Graphics/Contours** tree branch.

Figure 20.3: Contours of Temperature for the Steady Conduction Solution



11. Display filled contours of temperature to determine the thickness of mushy zone.

Postprocessing → **Graphics** → **Contours** → **Edit...**



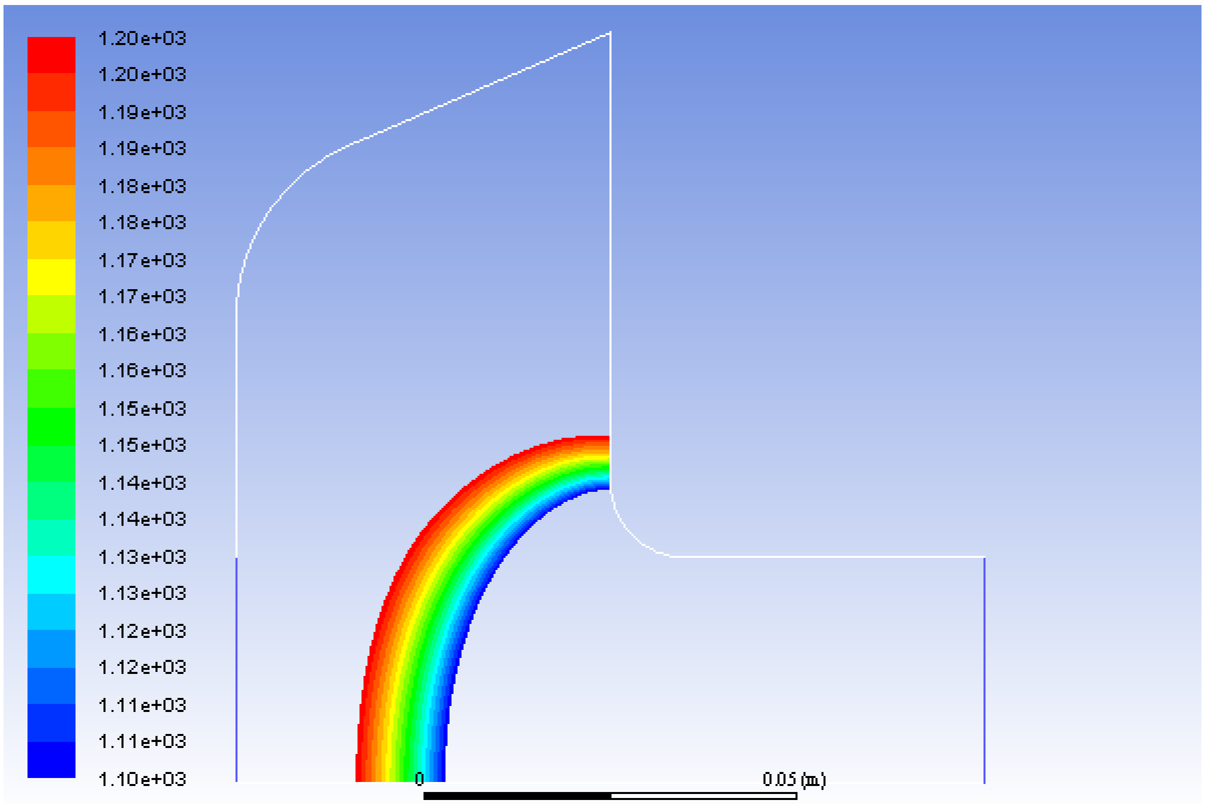
a. Disable **Auto Range** in the **Options** group box.

   *The **Clip to Range** option is automatically enabled.*

b. Enter `1100` for **Min** and `1200` for **Max**.

c. Click **Display** (See Figure 20.4: Contours of Temperature (Mushy Zone) for the Steady Conduction Solution (p. 833)) and close the **Contours** dialog box.

Figure 20.4: Contours of Temperature (Mushy Zone) for the Steady Conduction Solution



12. Save the case and data files for the steady conduction solution (`solid.cas.gz` and `solid.dat.gz`).

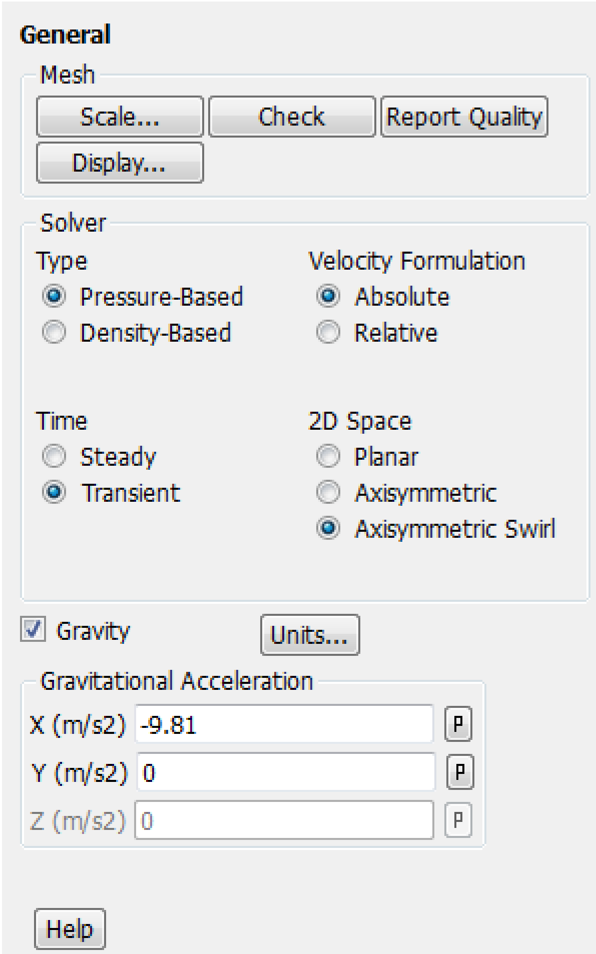
    File → **Write** → **Case & Data...**

20.4.9. Solution: Transient Flow and Heat Transfer

In this step, you will turn on time dependence and include the flow and swirl velocity equations in the calculation. You will then solve the transient problem using the steady conduction solution as the initial condition.

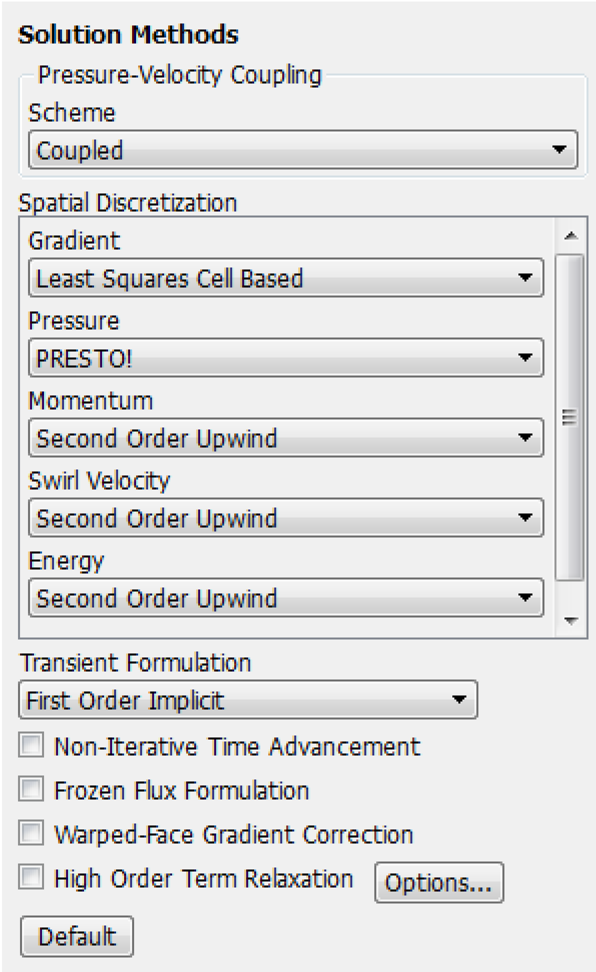
1. Enable a time-dependent solution by selecting **Transient** from the **Time** list.

   Setup → **General**



2. Set the solution parameters.

 **Solving → Solution → Methods...**

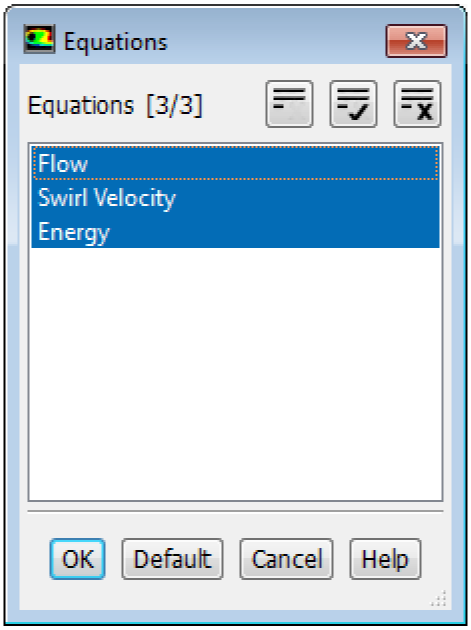


a. Retain the default selection of **First Order Implicit** from the **Transient Formulation** drop-down list.

b. Ensure that **PRESTO!** is selected from the **Pressure** drop-down list in the **Spatial Discretization** group box.

3. Enable calculations for flow and swirl velocity.

Solving → **Controls** → **Equations...**

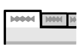


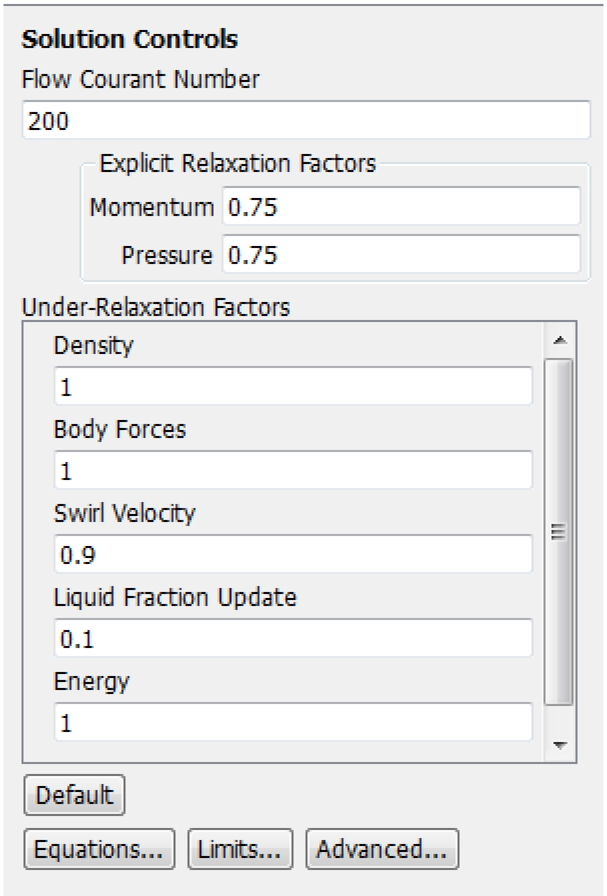
a. Select **Flow** and **Swirl Velocity** and ensure that **Energy** is selected from the **Equations** selection list.

   *Now all three items in the **Equations** selection list will be selected.*

b. Click **OK** to close the **Equations** dialog box.

4. Set the **Under-Relaxation Factors.**

 **Solving** → **Controls** → **Controls...**



a. Enter `0.1` for **Liquid Fraction Update**.

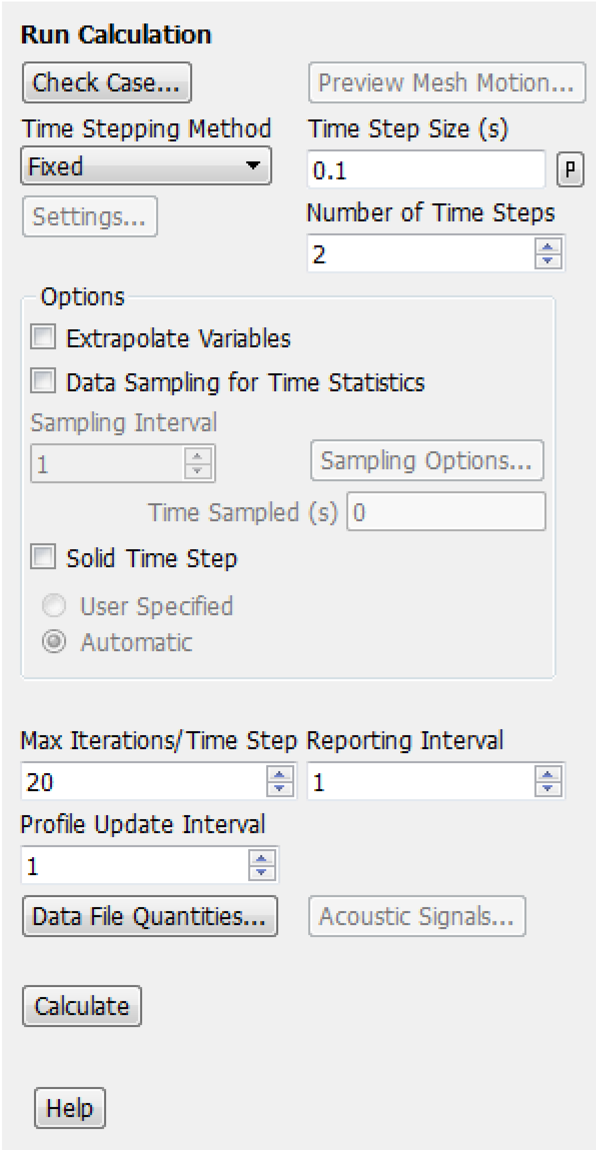
b. Retain the default values for other **Under-Relaxation Factors.**

5. Save the initial case and data files (`solid01.cas.gz` and `solid01.dat.gz`).

   File → **Write** → **Case & Data...**

6. Run the calculation for 2 time steps.

   Solving → **Run Calculation** → **Advanced...**



a. Enter `0.1` s for **Time Step Size**.

b. Set the **Number of Time Steps** to 2.

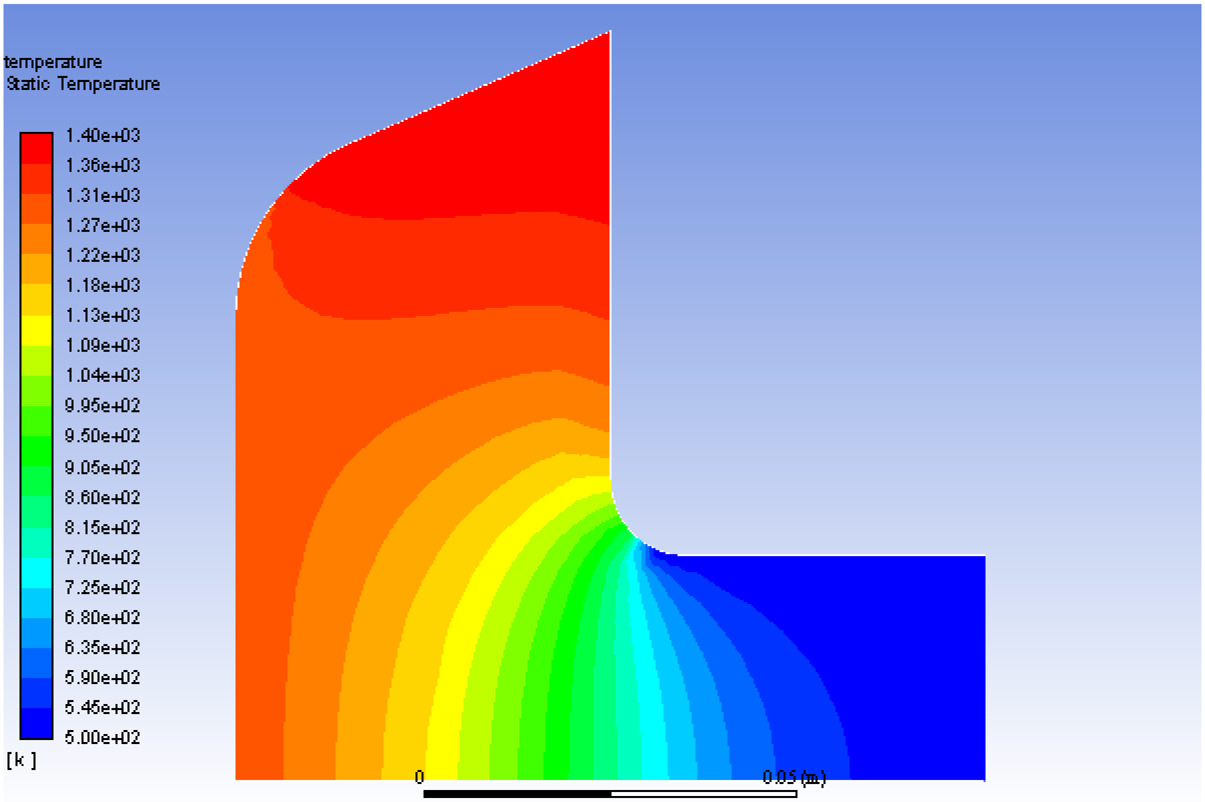
c. Retain the default value of `20` for **Max Iterations/Time Step**.

d. Click **Calculate**.

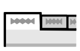
7. Display filled contours of the temperature after 0.2 seconds using the **temperature** contours definition that you created earlier.

Results → **Graphics** → **Contours** → **temperature** **Display**

Figure 20.5: Contours of Temperature at t=0.2 s



8. Create and display the definition of stream function contours (Figure 20.6: Contours of Stream Function at t=0.2 s (p. 840)).

 **Postprocessing** → **Graphics** → **Contours** → **New...**

a. Enter **`stream-function`** for **Contour Name**.

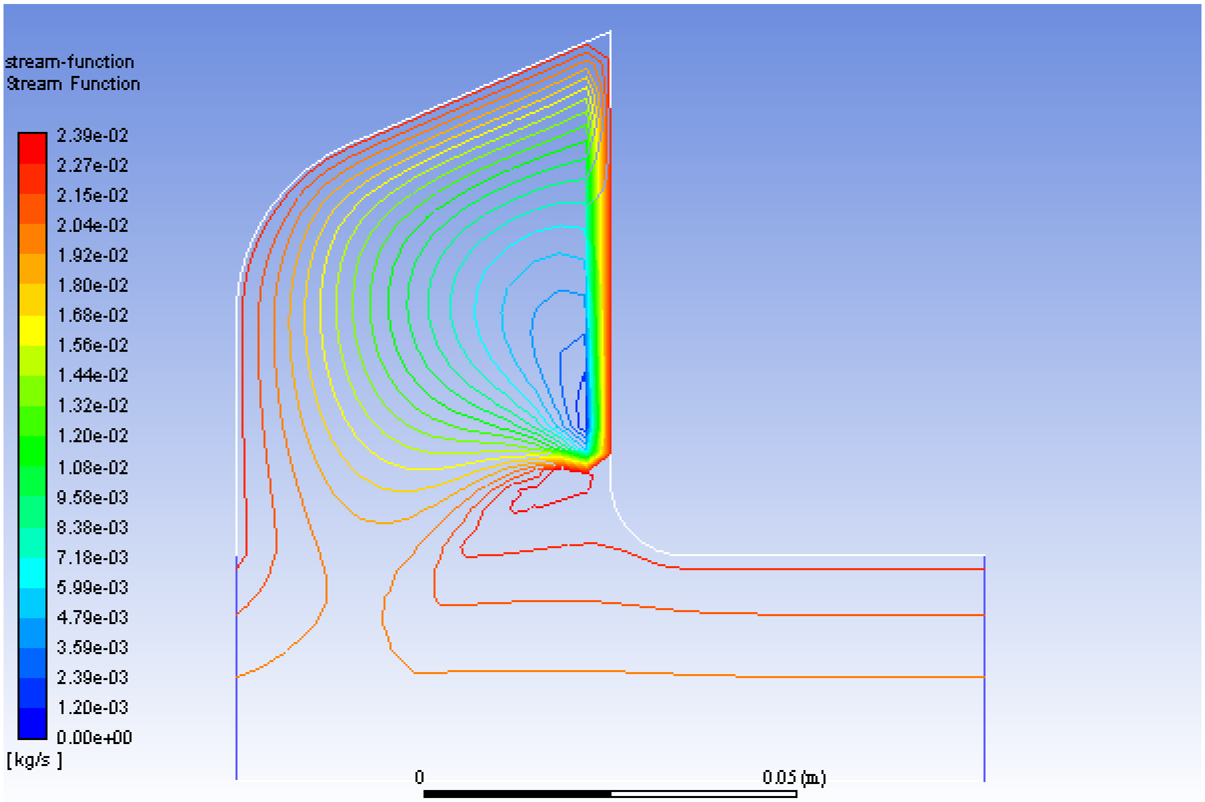
b. Disable **Filled** in the **Options** group box.

c. Select **Velocity...** and **Stream Function** from the **Contours of** drop-down lists.

d. Click **Save/Display**.

The **stream-function** contour definition appear under the **Results/Graphics/Contours** tree branch.

Figure 20.6: Contours of Stream Function at t=0.2 s



As shown in Figure 20.6: Contours of Stream Function at t=0.2 s (p. 840), the liquid is beginning to circulate in a large eddy, driven by natural convection and Marangoni convection on the free surface.

9. Create and display the definition of liquid fraction contours by modifying the **stream-function** contour definition (Figure 20.7: Contours of Liquid Fraction at t=0.2 s (p. 841)).

   Postprocessing → **Graphics** → **Contours** → **New...**

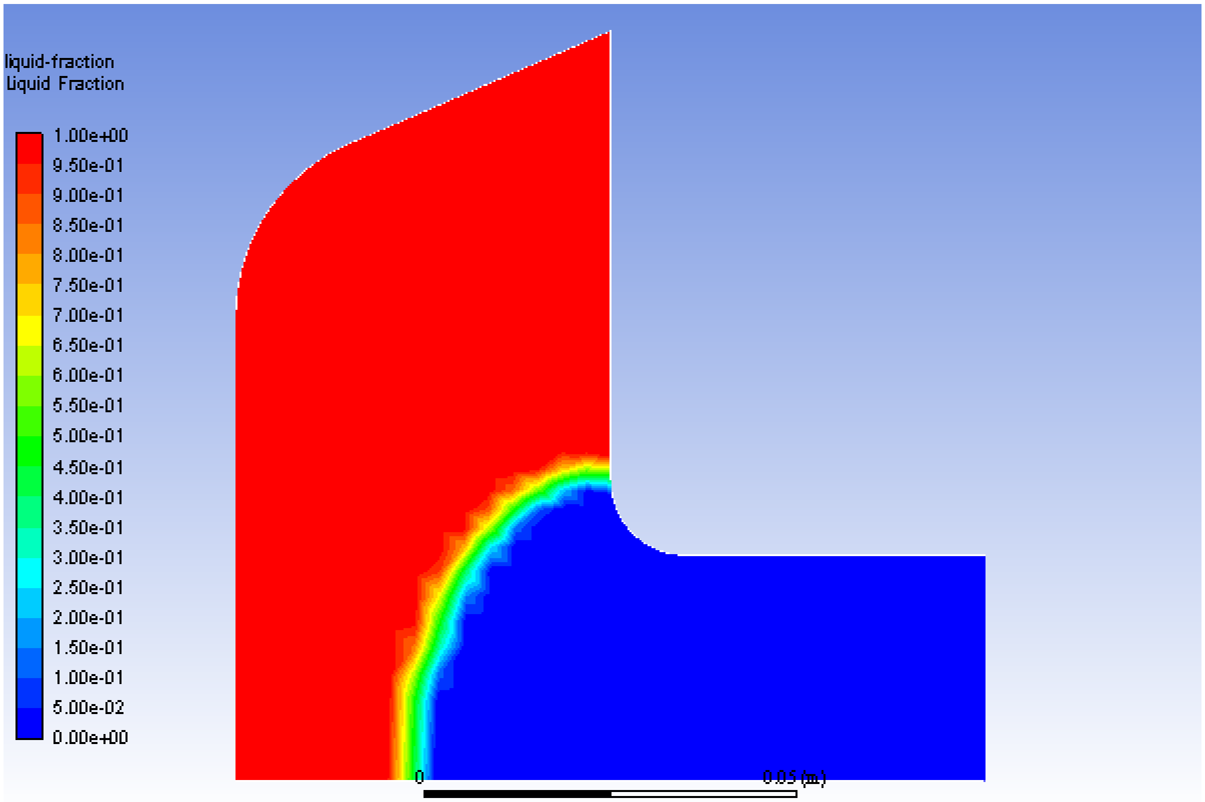
   a. Enter **`liquid-fraction`** for **Contour Name**.

   b. Enable **Filled** in the **Options** group box.

   c. Select **Solidification/Melting...** and **Liquid Fraction** from the **Contours of** drop-down lists.

   d. Click **Save/Display** and close the **Contours** dialog box.

Figure 20.7: Contours of Liquid Fraction at t=0.2 s



The liquid fraction contours show the current position of the melt front. Note that in Figure 20.7: Contours of Liquid Fraction at t=0.2 s (p. 841), the mushy zone divides the liquid and solid regions roughly in half.

10. Continue the calculation for 48 additional time steps.

    Solving → **Run Calculation** → **Advanced...**

    a. Enter 48 for **Number of Time Steps**.

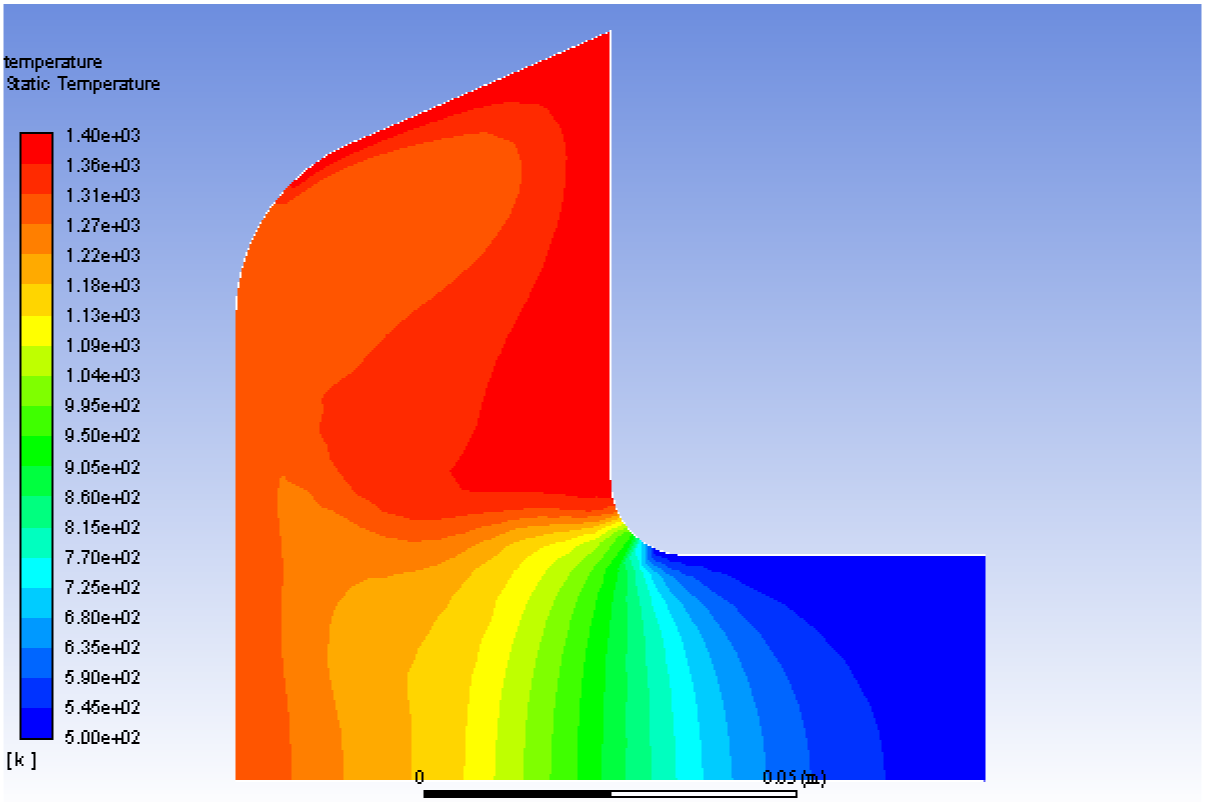
    b. Click **Calculate**.

    After a total of 50 time steps have been completed, the elapsed time will be 5 seconds.

11. Display filled contours of the temperature after 5 seconds using the contour definition created earlier (Figure 20.8: Contours of Temperature at t=5 s (p. 842)).

    Results → **Graphics** → **Contours** → **temperature** **Display**

Figure 20.8: Contours of Temperature at t=5 s



As shown in Figure 20.8: Contours of Temperature at t=5 s (p. 842), the temperature contours are fairly uniform through the melt front and solid material. The distortion of the temperature field due to the re-circulating liquid is also clearly evident.

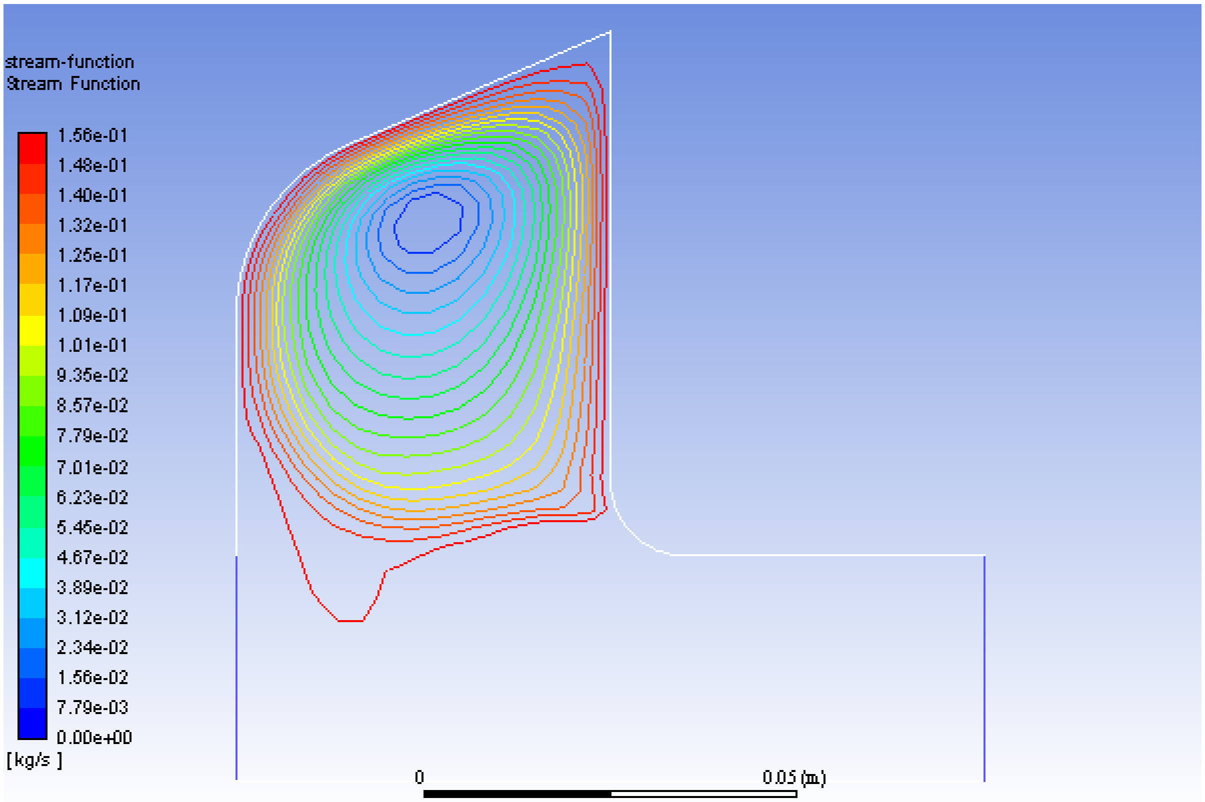
In a continuous casting process, it is important to pull out the solidified material at the proper time. If the material is pulled out too soon, it will not have solidified (that is, it will still be in a mushy state). If it is pulled out too late, it solidifies in the casting pool and cannot be pulled out in the required shape. The optimal rate of pull can be determined from the contours of liquidus temperature and solidus temperature.

12. Display contours of stream function (Figure 20.9: Contours of Stream Function at t=5 s (p. 843)).

Results → Graphics → Contours → stream-function Display

As shown in Figure 20.9: Contours of Stream Function at t=5 s (p. 843), the flow has developed more fully by 5 seconds, as compared with Figure 20.6: Contours of Stream Function at t=0.2 s (p. 840) after 0.2 seconds. The main eddy, driven by natural convection and Marangoni stress, dominates the flow.

Figure 20.9: Contours of Stream Function at t=5 s



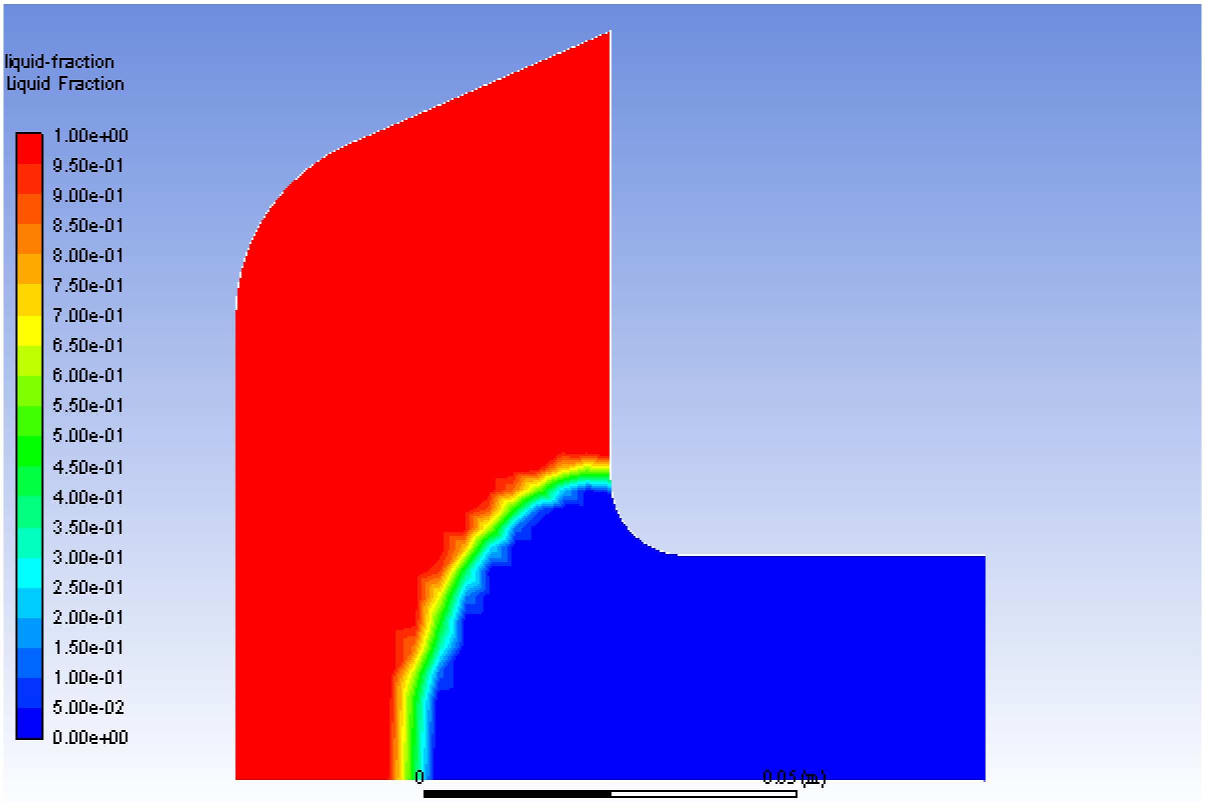
To examine the position of the melt front and the extent of the mushy zone, you will plot the contours of liquid fraction.

13. Display filled contours of liquid fraction (Figure 20.10: Contours of Liquid Fraction at t=5 s (p. 844)).

Results → **Graphics** → **Contours** → **liquid-fraction** **Display**

The introduction of liquid material at the left of the domain is balanced by the pulling of the solidified material from the right. After 5 seconds, the equilibrium position of the melt front is beginning to be established (Figure 20.10: Contours of Liquid Fraction at t=5 s (p. 844)).

Figure 20.10: Contours of Liquid Fraction at t=5 s



14. Save the case and data files for the solution at 5 seconds (`solid5.cas.gz` and `solid5.dat.gz`).

File → Write → Case & Data...

20.5. Summary

In this tutorial, you studied the setup and solution for a fluid flow problem involving solidification for the Czochralski growth process.

The solidification model in ANSYS Fluent can be used to model the continuous casting process where a solid material is continuously pulled out from the casting domain. In this tutorial, you patched a constant value and a custom field function for the pull velocities instead of computing them. This approach is used for cases where the pull velocity is not changing over the domain, as it is computationally less expensive than having ANSYS Fluent compute the pull velocities during the calculation.

For more information about the solidification/melting model, see the Fluent User's Guide.

20.6. Further Improvements

This tutorial guides you through the steps to reach an initial set of solutions. You may be able to obtain a more accurate solution by using an appropriate higher-order discretization scheme and by adapting the mesh. Mesh adaption can also ensure that the solution is independent of the mesh. These steps are demonstrated in Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121).

Chapter 21: Using the Eulerian Granular Multiphase Model with Heat Transfer

This tutorial is divided into the following sections:

21.1. Introduction

This tutorial examines the flow of air and a granular solid phase consisting of glass beads in a hot gas fluidized bed, under uniform minimum fluidization conditions. The results obtained for the local wall-to-bed heat transfer coefficient in ANSYS Fluent can be compared with analytical results [1].

This tutorial demonstrates how to do the following:

- Use the Eulerian granular model.
- Set boundary conditions for internal flow.
- Compile a User-Defined Function (UDF) for the gas and solid phase thermal conductivities.
- Calculate a solution using the pressure-based solver.

21.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

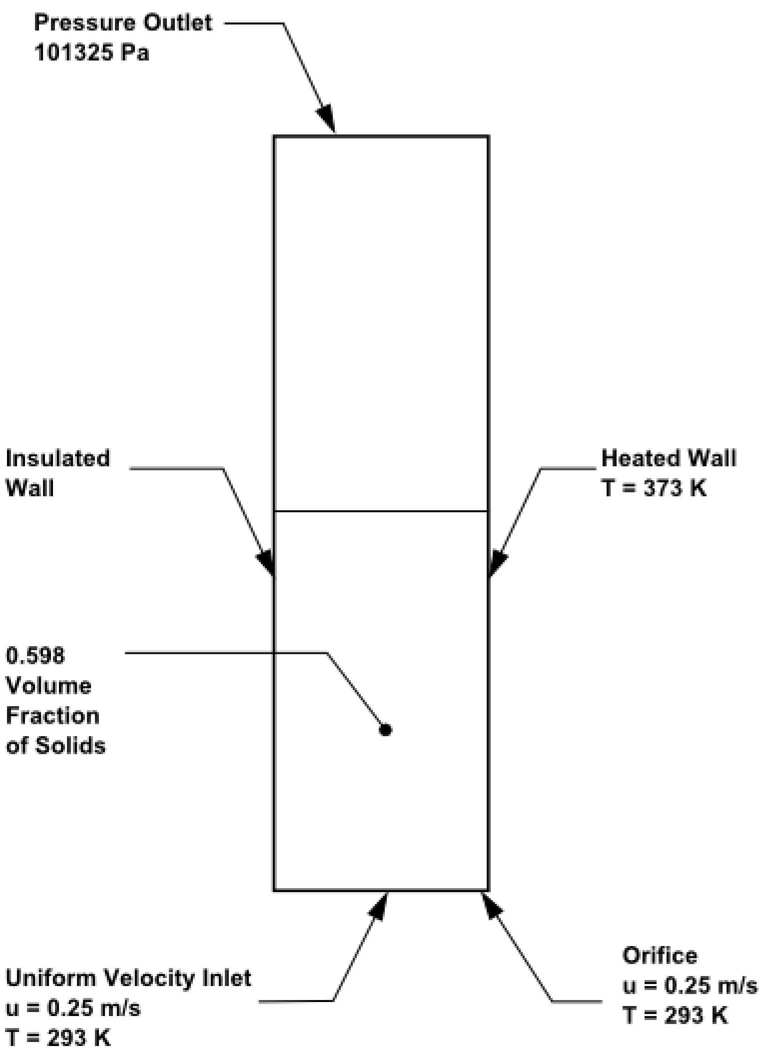
and that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

In order to complete the steps to compile the UDF, you will need to have a working C compiler installed on your machine.

21.3. Problem Description

This problem considers a hot gas fluidized bed in which air flows upwards through the bottom of the domain and through an additional small orifice next to a heated wall. A uniformly fluidized bed is examined, which you can then compare with analytical results [1]. The geometry and data for the problem are shown in Figure 21.1: Problem Schematic (p. 846).

Figure 21.1: Problem Schematic



21.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

21.4.1. Preparation

To prepare for running this tutorial:

1. Set up a working folder on the computer you will be using.

2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

   Note

   If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

3. Enter the name of this tutorial into the search bar.

4. Narrow the results by using the filter on the left side of the page.

   a. Click **ANSYS Fluent** under **Product**.

   b. Click **18.0** under **Version**.

5. Select this tutorial from the list.

6. Click the **eulerian_granular_heat_R180.zip** link to download the input files.

7. Unzip `eulerian_granular_heat_R180.zip` to your working folder.

   The files, `fluid-bed.msh` and `conduct.c`, can be found in the `eulerian_granular_heat` folder created after unzipping the file.

8. Use Fluent Launcher to enable **Double Precision** and start the **2D** version of ANSYS Fluent.

   Fluent Launcher displays your **Display Options** preferences from the previous session.

   For more information about the Fluent Launcher, see starting ANSYS Fluent using the Fluent Launcher in the Fluent Getting Started Guide.

9. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.

10. Run in **Serial** by selecting **Serial** under **Processing Options**.

11. Ensure that **Set up Compilation Environment for UDF** is enabled in the **Environment** tab of the Fluent Launcher window. This will allow you to compile the UDF.

Note

The double precision solver is recommended for modeling multiphase flow simulations.

21.4.2. Mesh

1. Read the mesh file `fluid-bed.msh`.

   File → **Read** → **Mesh...**

   As ANSYS Fluent reads the mesh file, it will report the progress in the console.

2. Check the mesh.

   Setting Up Domain → **Mesh** → **Check**

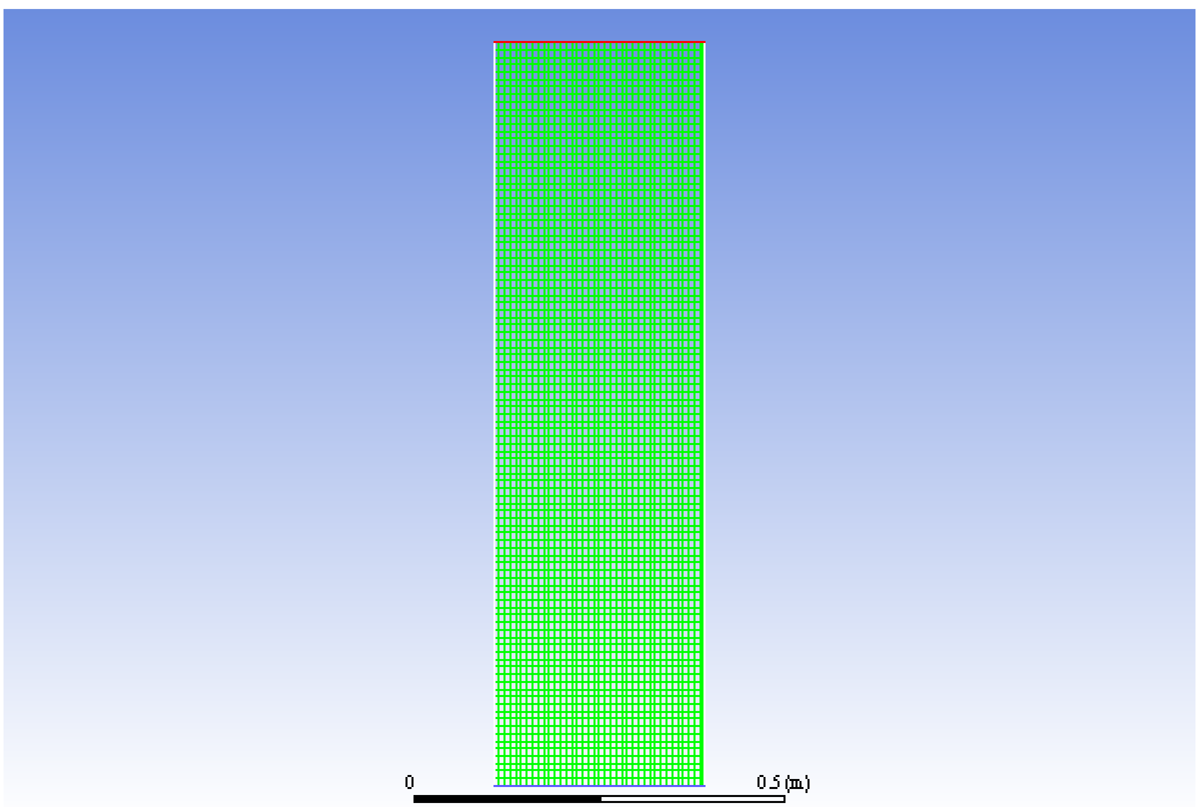
   ANSYS Fluent will perform various checks on the mesh and will report the progress in the console. Make sure that the reported minimum volume is a positive number.

3. Examine the mesh (Figure 21.2: Mesh Display of the Fluidized Bed (p. 848)).

Extra

You can use the right mouse button to check which zone number corresponds to each boundary. If you click the right mouse button on one of the boundaries in the graphics window, its zone number, name, and type will be printed in the ANSYS Fluent console. This feature is especially useful when you have several zones of the same type and you want to distinguish between them quickly.

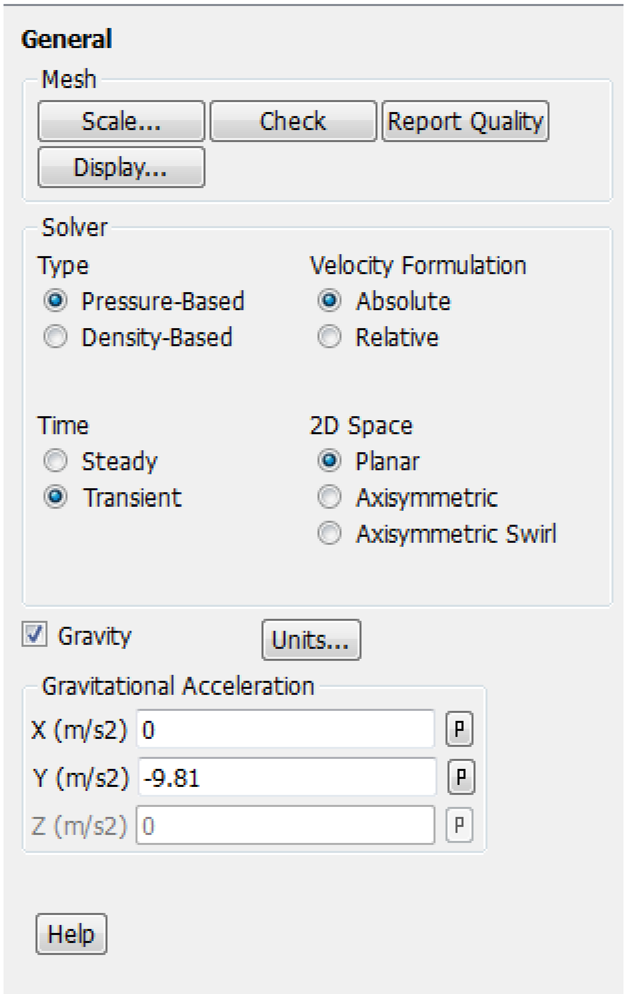
Figure 21.2: Mesh Display of the Fluidized Bed



21.4.3. Solver Settings

1. Enable the pressure-based transient solver.

   Setup → **General**

   a. Retain the default selection of **Pressure-Based** from the **Type** list.

      The pressure-based solver must be used for multiphase calculations.

   b. Select **Transient** from the **Time** list.

2. Set the gravitational acceleration.

   Setup → **General** → **Gravity**

   a. Enter `-9.81` m/s^2 for the **Gravitational Acceleration** in the **Y** direction.

21.4.4. Models

1. Enable the Eulerian multiphase model for two phases.

*You will use the default settings for the Eulerian model, so you can enable it directly from the tree by right-clicking the **Multiphase** node and choosing **Eulerian** from the context menu.*

Setup → **Models** → **Multiphase** **Eulerian**

2. Enable heat transfer by enabling the energy equation.

   Setup → **Models** → **Energy** **On**

   An **Information** dialog box appears reminding you to confirm the property values. Click **OK** in the **Information** dialog box to continue.

3. Retain the default laminar viscous model.

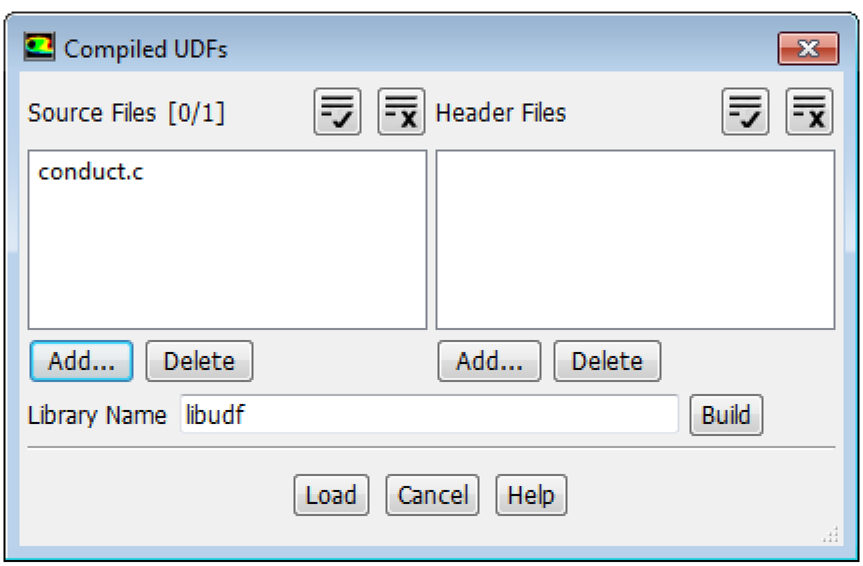
   The decision to use the laminar model should be based on the Stokes number for the particles suspended in the fluid flow.

   Setup → **Models** → **Viscous** **Model** → **Laminar**

21.4.5. UDF

1. Compile the user-defined function, `conduct.c`, that will be used to define the thermal conductivity for the gas and solid phase.

   User Defined → **User Defined** → **Functions** → **Compiled...**



   a. Click the **Add...** button below the **Source Files** option to open the **Select File** dialog box.

      i. Select the file **conduct.c** and click **OK** in the **Select File** dialog box.

   b. Click **Build**.

      ANSYS Fluent will create a `libudf` *folder and compile the UDF. Also, a **Warning** dialog box will open asking you to make sure that UDF source file and case/data files are in the same folder.*

c. Click **OK** to close the **Warning** dialog box.

d. Click **Load** to load the UDF.

Extra

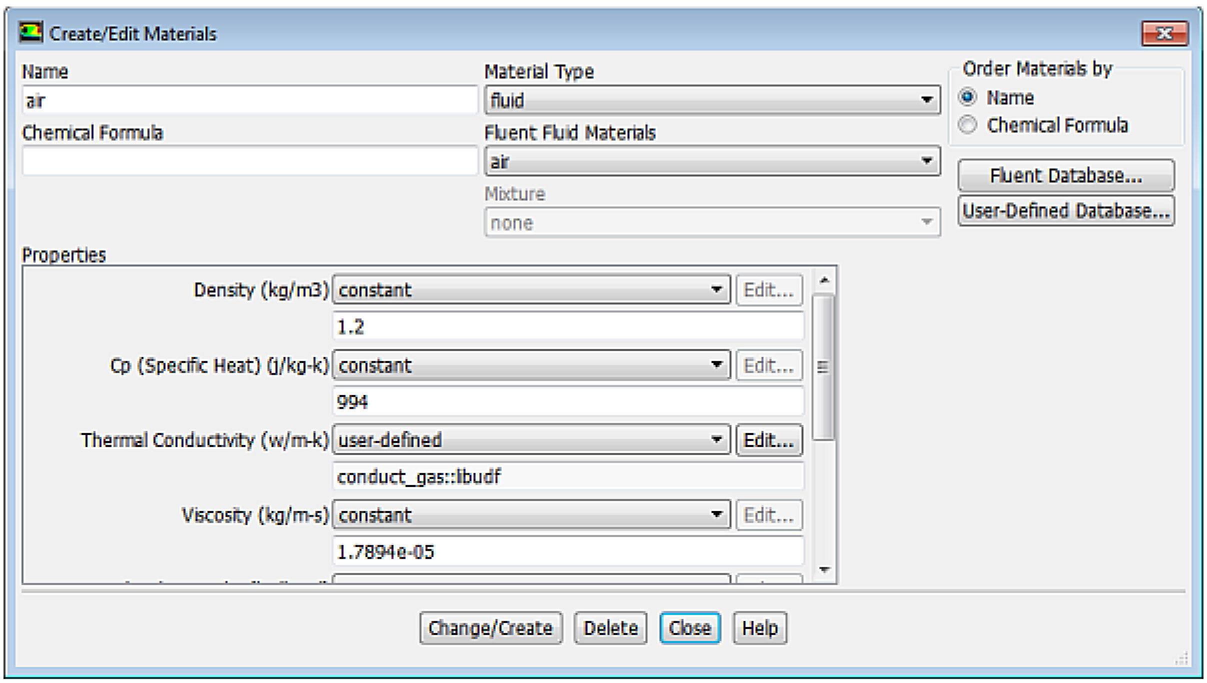
If you decide to read in the case file that is provided for this tutorial on the Customer Portal, you will need to compile the UDF associated with this tutorial in your working folder. This is necessary because ANSYS Fluent will expect to find the correct UDF libraries in your working folder when reading the case file.

21.4.6. Materials

1. Modify the properties for air, which will be used for the primary phase.

Setup → **Materials** → **air** → **Create/Edit...**

The properties used for air are modified to match data used by Kuipers et al. [1]



a. Enter `1.2` kg/m^3 for **Density**.

b. Enter 994 J/kg-K for **Cp**.

c. Select **user-defined** from the **Thermal Conductivity** drop-down list to open the **User Defined Functions** dialog box.

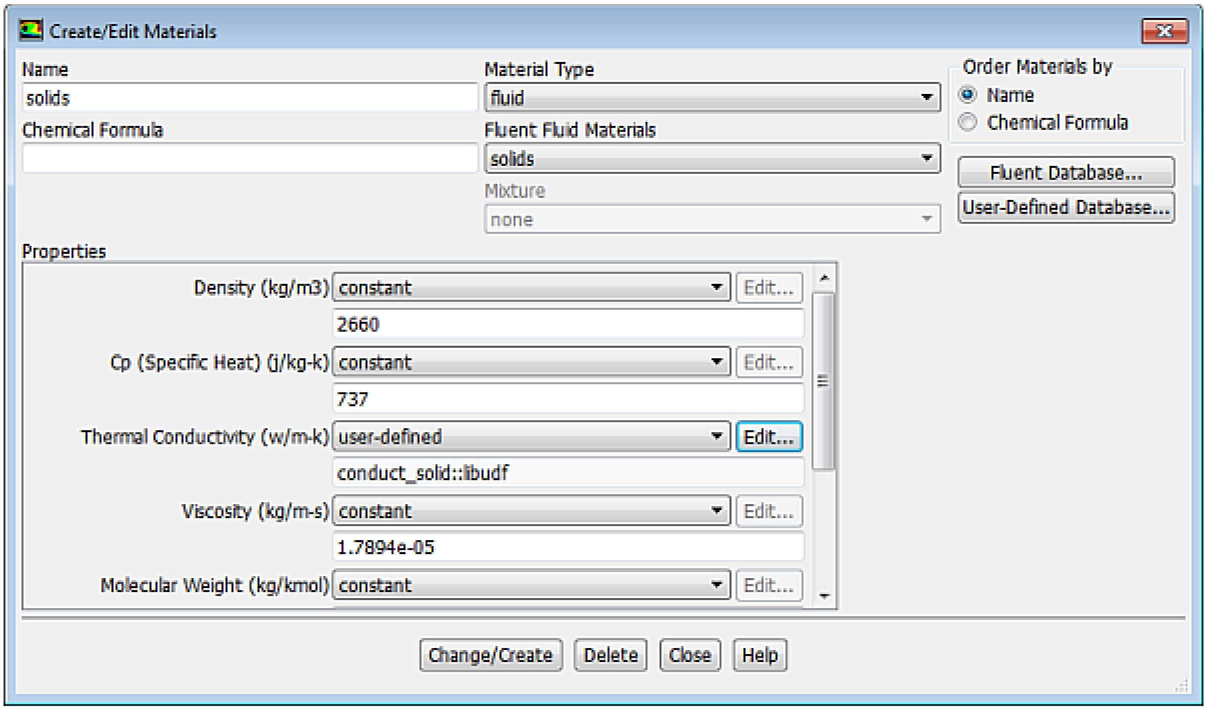
   i. Select `conduct_gas::libudf` from the available list.

   ii. Click **OK** to close the **User Defined Functions** dialog box.

d. Click **Change/Create**.

2. Define a new fluid material for the granular phase (the glass beads).

   Setup → **Materials** → **air** → **Create/Edit...**



   a. Enter `solids` for **Name**.

   b. Enter `2660` kg/m^3 for **Density**.

   c. Enter `737` J/kg-K for **Cp**.

   d. Retain the selection of **user-defined** from the **Thermal Conductivity** drop-down list.

   e. Click the **Edit...** button to open the **User Defined Functions** dialog box.

      i. Select `conduct_solid::libudf` in the **User Defined Functions** dialog box and click **OK**.

         *A **Question** dialog box will open asking if you want to overwrite **air**.*

      ii. Click **No** in the **Question** dialog box.

   f. Select **solids** from the **Fluent Fluid Materials** drop-down list.

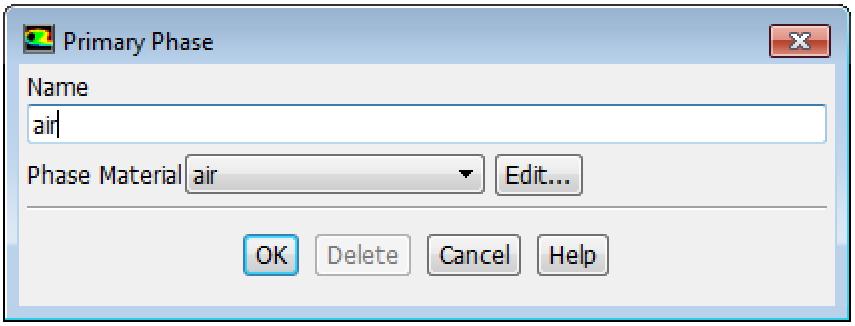
   g. Click **Change/Create** and close the **Materials** dialog box.

21.4.7. Phases

You will now configure the phases.

1. Define air as the primary phase.

   Setup → **Multiphase** → **Phases** → **phase-1 - Primary Phase** → **Edit...**



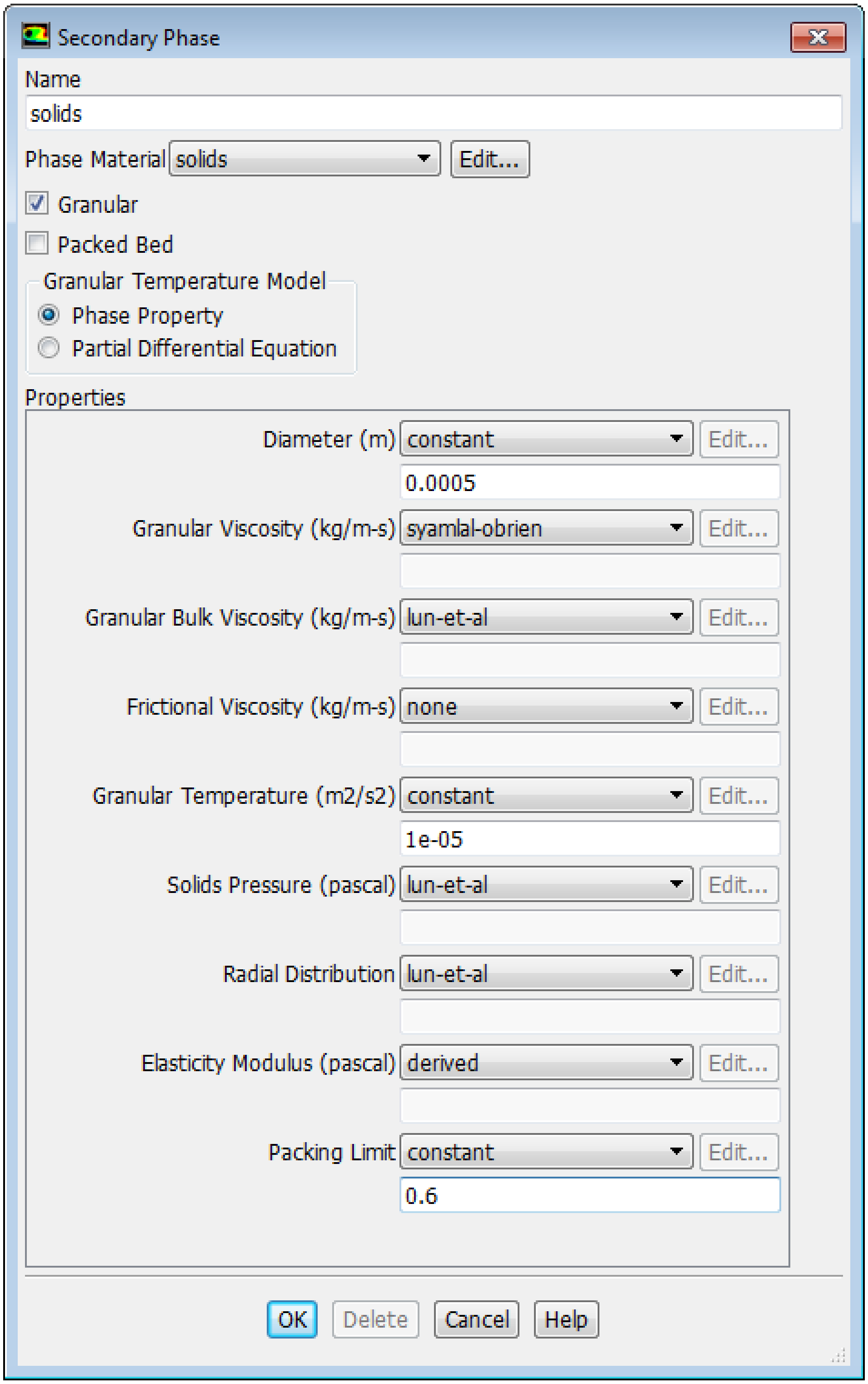
a. Enter `air` for **Name**.

b. Ensure that **air** is selected from the **Phase Material** drop-down list.

c. Click **OK** to close the **Primary Phase** dialog box.

2. Define **solids** (glass beads) as the secondary phase.

Setup → Multiphase → Phases → phase-2 - Secondary Phase → Edit...



a. Enter `solids` for **Name**.

b. Select **solids** from the **Phase Material** drop-down list.

c. Enable **Granular**.

d. Retain the default selection of **Phase Property** in the **Granular Temperature Model** group box.

e. Enter `0.0005` m for **Diameter**.

f. Select **syamlal-obrien** from the **Granular Viscosity** drop-down list.

g. Select **lun-et-al** from the **Granular Bulk Viscosity** drop-down list.

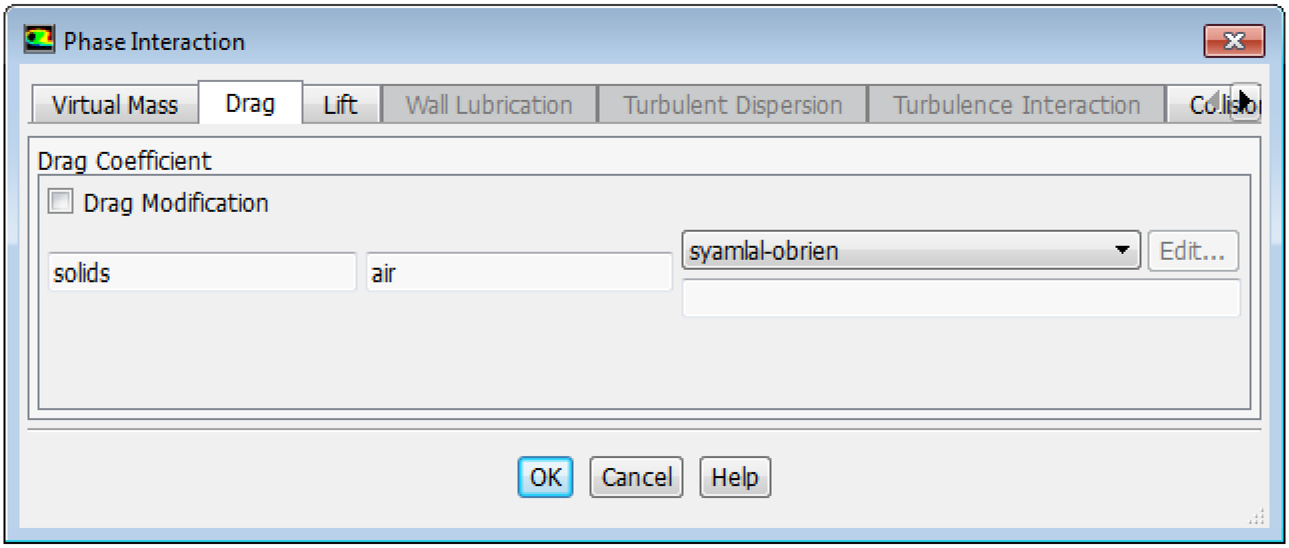
h. Select **constant** from the **Granular Temperature** drop-down list and enter `1e-05`.

i. Enter `0.6` for the **Packing Limit**.

j. Click **OK** to close the **Secondary Phase** dialog box.

3. Define the interphase interactions formulations to be used.

Setup → **Multiphase** → **Phase Interactions** **Edit...**



a. In the **Drag** tab, select **syamlal-obrien** from the **Drag Coefficient** drop-down list.

b. In the **Heat** tab, select **gunn** from the **Heat Transfer Coefficient** drop-down list.

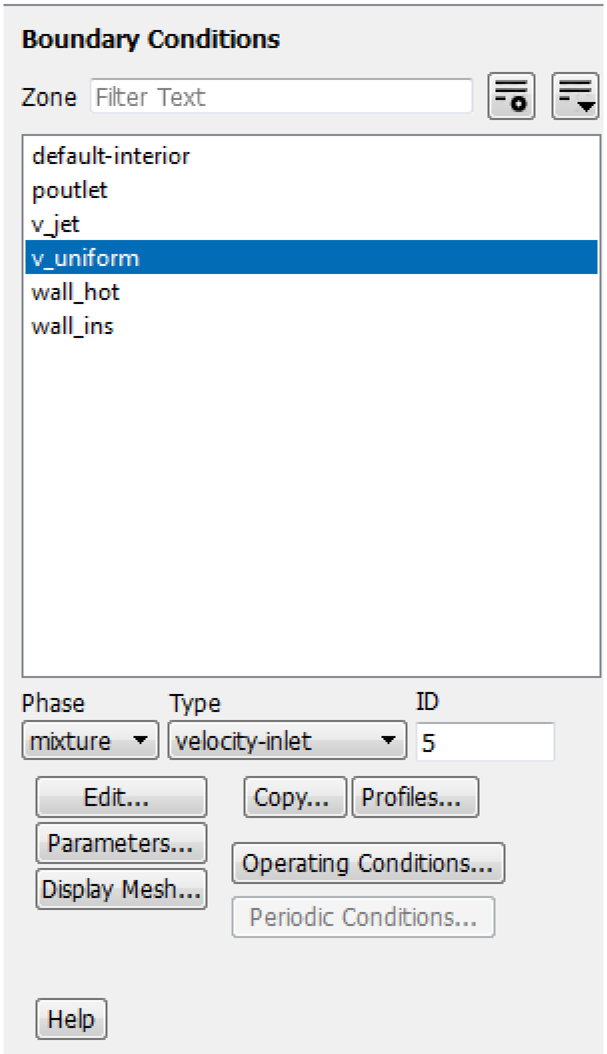
The interphase heat exchange is simulated, using a drag coefficient, the default restitution coefficient for granular collisions of 0.9, and a heat transfer coefficient. Granular phase lift is not very relevant in this problem, and in fact is rarely used.

c. Click **OK** to close the **Phase Interaction** dialog box.

21.4.8. Boundary Conditions

For this problem, you need to set the boundary conditions for all boundaries.

Setup → **Boundary Conditions**



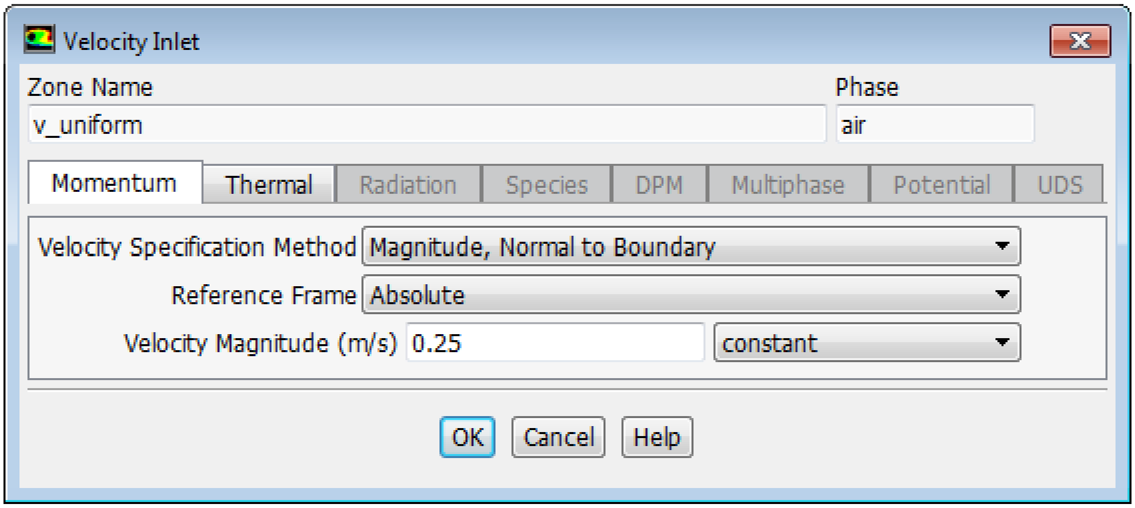
1. Set the boundary conditions for the lower velocity inlet (**v_uniform**) for the primary phase.

   Setup → **Boundary Conditions** → **v_uniform**

   For the Eulerian multiphase model, you will specify conditions at a velocity inlet that are specific to the primary and secondary phases.

   a. Select **air** from the **Phase** drop-down list.

   b. Click the **Edit...** button to open the **Velocity Inlet** dialog box.



i. Retain the default selection of **Magnitude, Normal to Boundary** from the **Velocity Specification Method** drop-down list.

ii. Enter `0.25` m/s for the **Velocity Magnitude**.

iii. Click the **Thermal** tab and enter 293 K for **Temperature**.

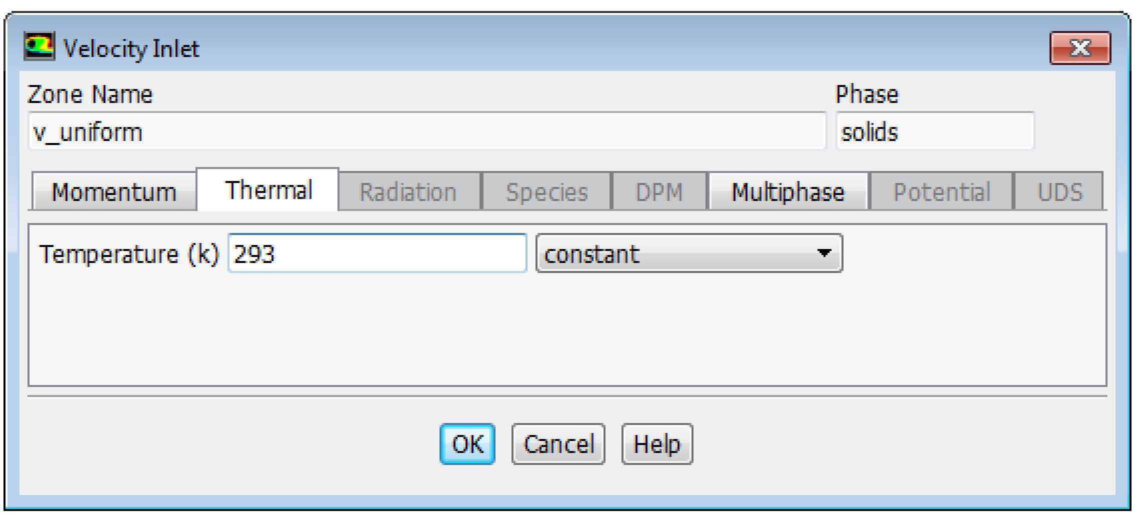
iv. Click **OK** to close the **Velocity Inlet** dialog box.

2. Set the boundary conditions for the lower velocity inlet (**v_uniform**) for the secondary phase.

Setup → **Boundary Conditions** → **v_uniform**

a. Select **solids** from the **Phase** drop-down list.

b. Click the **Edit...** button to open the **Velocity Inlet** dialog box.



i. Retain the default **Velocity Specification Method** and **Reference Frame**.

ii. Retain the default value of `0` m/s for the **Velocity Magnitude**.

iii. Click the **Thermal** tab and enter 293 K for **Temperature**.

iv. Click the **Multiphase** tab and retain the default value of `0` for **Volume Fraction**.

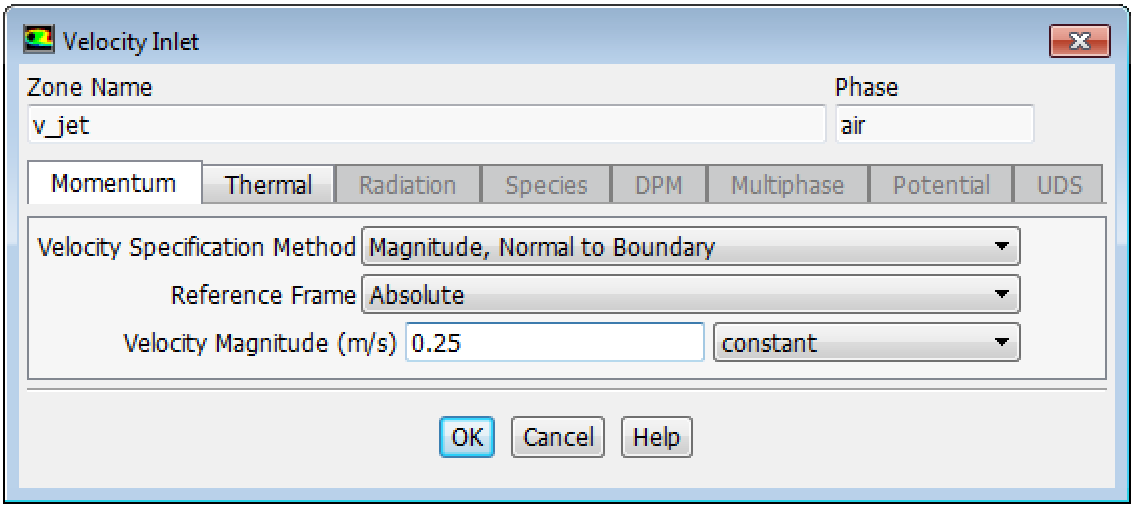
v. Click **OK** to close the **Velocity Inlet** dialog box.

3. Set the boundary conditions for the orifice velocity inlet (**v_jet**) for the primary phase.

Setup → **Boundary Conditions** → **v_jet**

a. Select **air** from the **Phase** drop-down list.

b. Click the **Edit...** button to open the **Velocity Inlet** dialog box.



i. Retain the default **Velocity Specification Method** and **Reference Frame**.

ii. Enter `0.25` m/s for the **Velocity Magnitude**.

In order for a comparison with analytical results [1] to be meaningful, in this simulation you will use a uniform value for the air velocity equal to the minimum fluidization velocity at both inlets on the bottom of the bed.

iii. Click the **Thermal** tab and enter `293` K for **Temperature**.

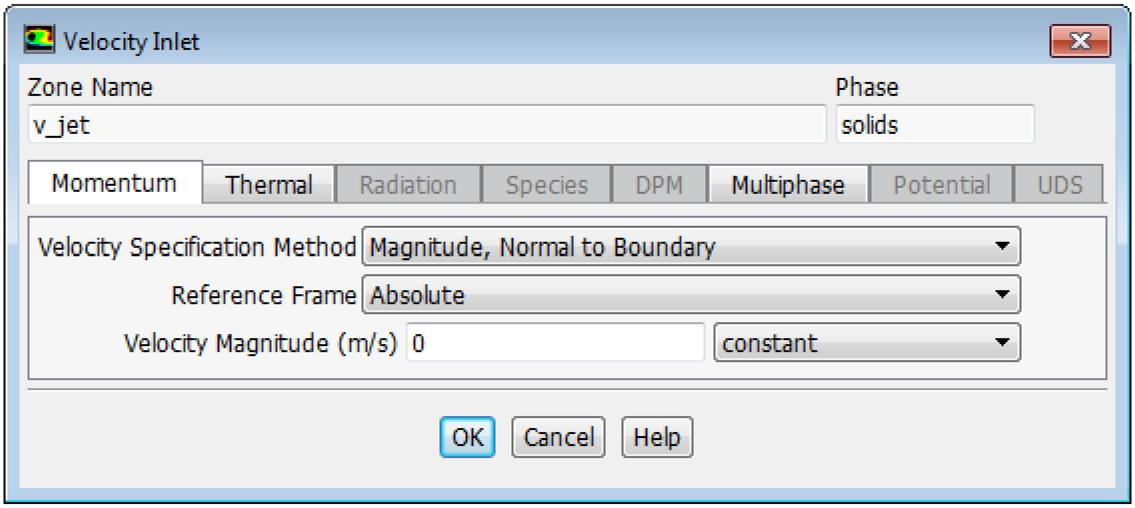
iv. Click **OK** to close the **Velocity Inlet** dialog box.

4. Set the boundary conditions for the orifice velocity inlet (**v_jet**) for the secondary phase.

Setup → **Boundary Conditions** → **v_jet**

a. Select **solids** from the **Phase** drop-down list.

b. Click the **Edit...** button to open the **Velocity Inlet** dialog box.



i. Retain the default **Velocity Specification Method** and **Reference Frame**.

ii. Retain the default value of `0` m/s for the **Velocity Magnitude**.

iii. Click the **Thermal** tab and enter `293` K for **Temperature**.

iv. Click the **Multiphase** tab and retain the default value of `0` for the **Volume Fraction**.

v. Click **OK** to close the **Velocity Inlet** dialog box.

5. Set the boundary conditions for the pressure outlet (**poutlet**) for the mixture phase.

   Setup → **Boundary Conditions** → **poutlet**

   For the Eulerian granular model, you will specify conditions at a pressure outlet for the mixture and for both phases.

   The thermal conditions at the pressure outlet will be used only if flow enters the domain through this boundary. You can set them equal to the inlet values, as no flow reversal is expected at the pressure outlet. In general, however, it is important to set reasonable values for these downstream scalar values, in case flow reversal occurs at some point during the calculation.

   a. Select **mixture** from the **Phase** drop-down list.

   b. Click the **Edit...** button to open the **Pressure Outlet** dialog box.

      i. Retain the default value of `0` Pascal for **Gauge Pressure**.

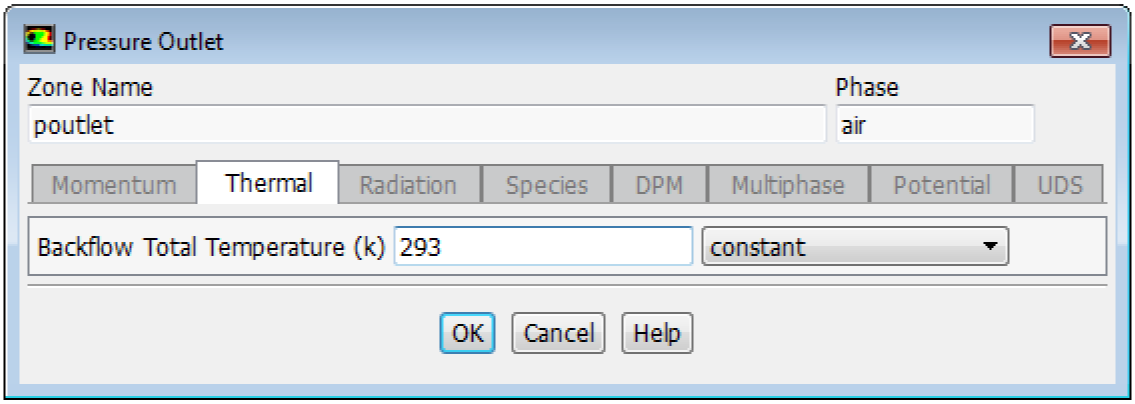
      ii. Click **OK** to close the **Pressure Outlet** dialog box.

6. Set the boundary conditions for the pressure outlet (**poutlet**) for the primary phase.

   Setup → **Boundary Conditions** → **poutlet**

   a. Select **air** from the **Phase** drop-down list.

   b. Click the **Edit...** button to open the **Pressure Outlet** dialog box.



i. In the **Thermal** tab, enter 293 K for **Backflow Total Temperature**.

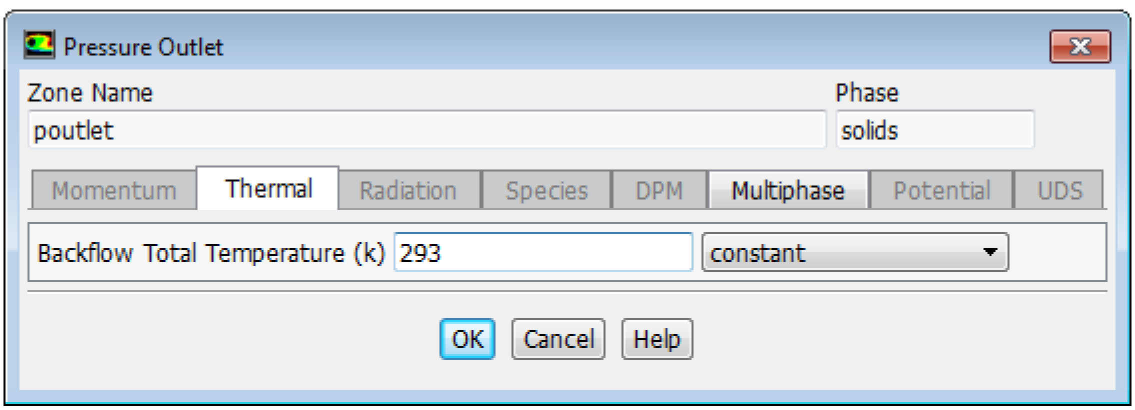
ii. Click **OK** to close the **Pressure Outlet** dialog box.

7. Set the boundary conditions for the pressure outlet (**poutlet**) for the secondary phase.

Setup → **Boundary Conditions** → **poutlet**

a. Select **solids** from the **Phase** drop-down list.

b. Click the **Edit...** button to open the **Pressure Outlet** dialog box.



i. In the **Thermal** tab, enter 293 K for the **Backflow Total Temperature**.

ii. In the **Multiphase** tab, retain default settings.

iii. Click **OK** to close the **Pressure Outlet** dialog box.

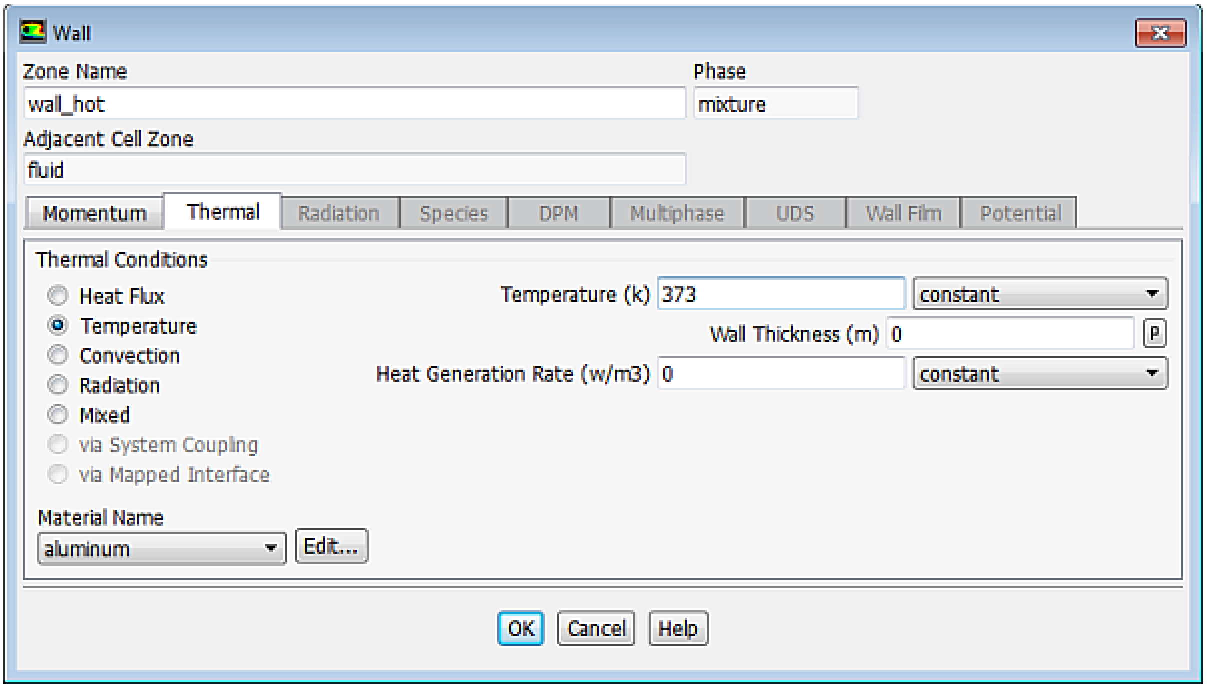
8. Set the boundary conditions for the heated wall (**wall_hot**) for the mixture.

Setup → **Boundary Conditions** → **wall_hot**

For the heated wall, you will set thermal conditions for the mixture, and momentum conditions (zero shear) for both phases.

a. Select **mixture** from the **Phase** drop-down list.

b. Click the **Edit...** button to open the **Wall** dialog box.



i. In the **Thermal** tab, select **Temperature** from the **Thermal Conditions** list.

ii. Enter 373 K for **Temperature**.

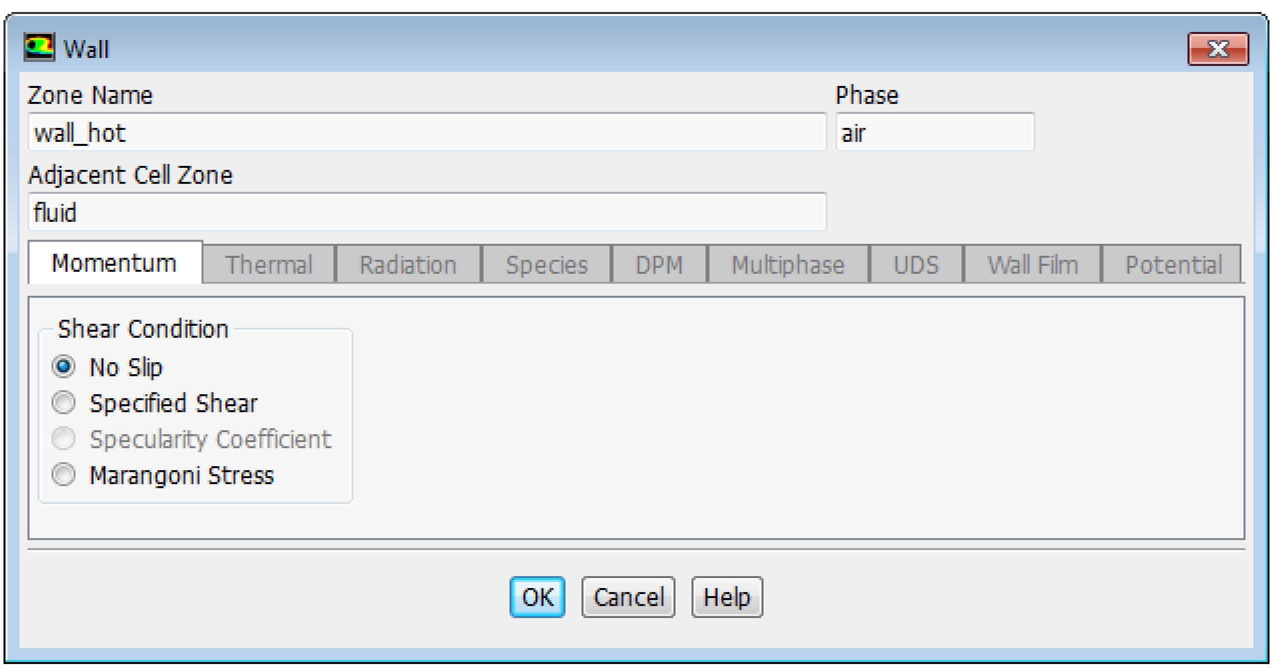
iii. Click **OK** to close the **Wall** dialog box.

9. Set the boundary conditions for the heated wall (**wall_hot**) for the primary phase.

Setup → **Boundary Conditions** → **wall_hot**

a. Select **air** from the **Phase** drop-down list.

b. Click the **Edit...** button to open the **Wall** dialog box.



c. Retain the default **No Slip** condition and click **OK** to close the **Wall** dialog box.

10. Set the boundary conditions for the heated wall (**wall_hot**) for the secondary phase (**solids**) same as that of the primary phase.

Setup → **Boundary Conditions** → **wall_hot**

For the secondary phase, you will retain the default no slip condition as for the primary phase.

11. Set the boundary conditions for the adiabatic wall (**wall_ins**).

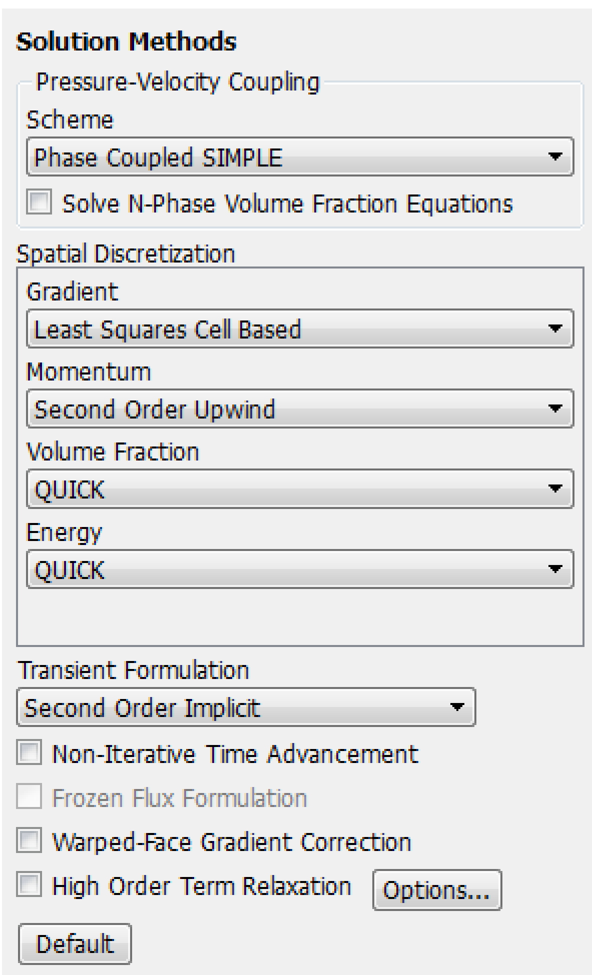
Setup → **Boundary Conditions** → **wall_ins**

For the adiabatic wall, retain the default thermal conditions for the mixture (zero heat flux), and the default momentum conditions (no slip) for both phases.

21.4.9. Solution

1. Select the second order implicit transient formulation and higher-order spatial discretization schemes.

Solving → **Solution** → **Methods...**



a. Select **Second Order Implicit** from the **Transient Formulation** drop-down list.

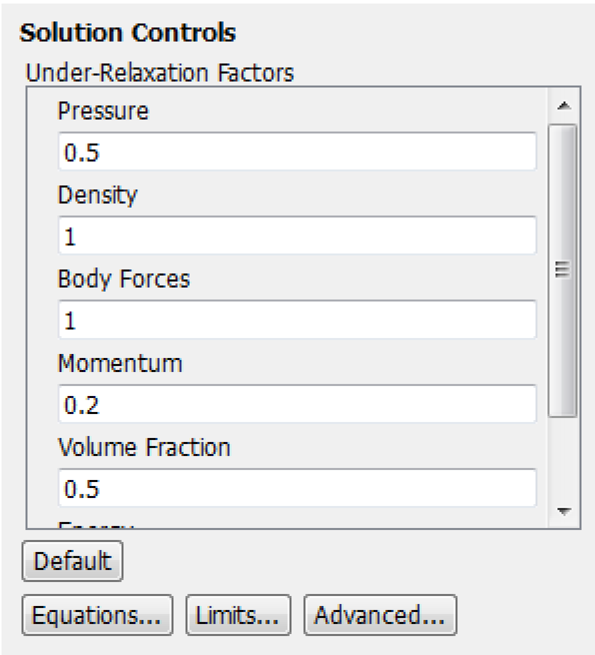
b. Modify the discretization methods in the **Spacial Discretization** group box.

i. Select **Second Order Upwind** for **Momentum**.

ii. Select **Quick** for **Volume Fraction** and **Energy**.

2. Set the solution parameters.

Solving → **Controls** → **Controls...**



a. Enter `0.5` for **Pressure**.

b. Enter `0.2` for **Momentum**.

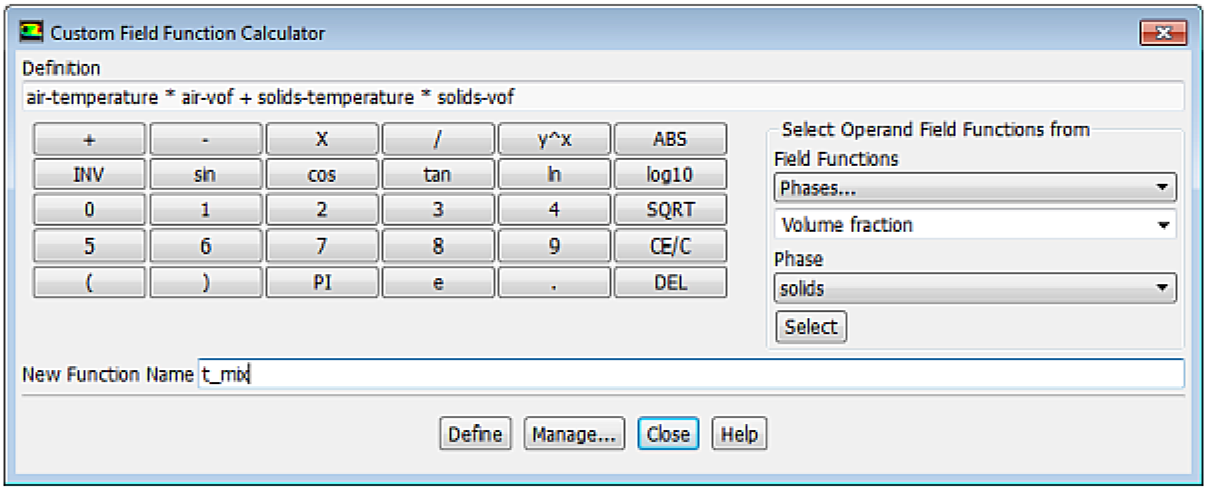
3. Ensure that the plotting of residuals is enabled during the calculation.

Solving → **Reports** → **Residuals...**

4. Define a custom field function for the heat transfer coefficient.

User Defined → **Field Functions** → **Custom...**

Initially, you will define functions for the mixture temperature, and thermal conductivity, then you will use these to define a function for the heat transfer coefficient.



a. Define the function `t_mix`.

   i. Select **Temperature...** and **Static Temperature** from the **Field Functions** drop-down lists.

   ii. Ensure that **air** is selected from the **Phase** drop-down list and click **Select**.

   iii. Click the multiplication symbol in the calculator pad.

   iv. Select **Phases...** and **Volume fraction** from the **Field Functions** drop-down list.

   v. Ensure that **air** is selected from the **Phase** drop-down list and click **Select**.

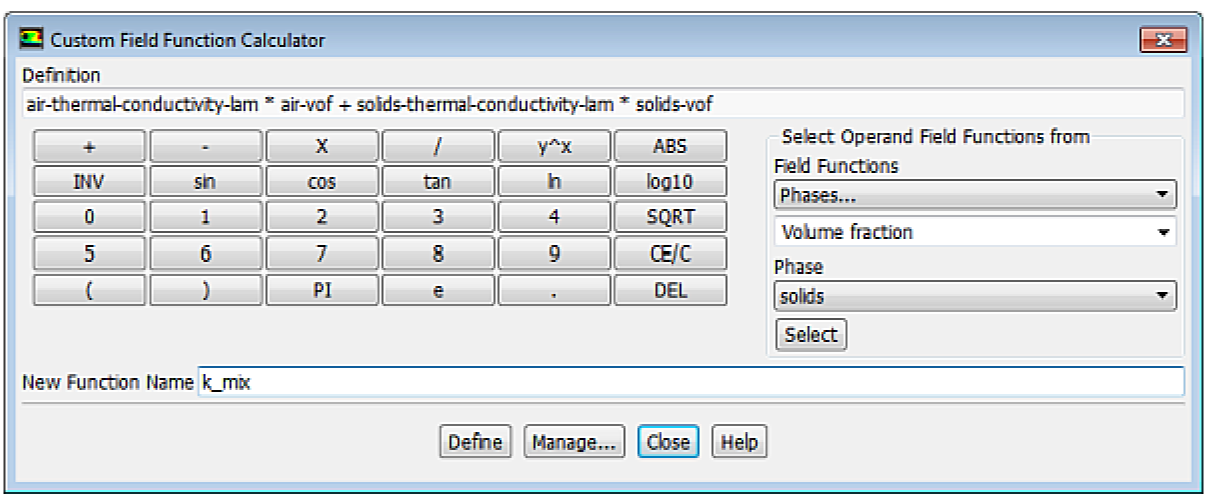
   vi. Click the addition symbol in the calculator pad.

   vii. Similarly, add the term `solids-temperature * solids-vof`.

   viii. Enter `t_mix` for **New Function Name**.

   ix. Click **Define**.

b. Define the function `k_mix`.



   i. Select **Properties...** and **Thermal Conductivity** from the **Field Functions** drop-down lists.

ii. Select **air** from the **Phase** drop-down list and click **Select**.

iii. Click the multiplication symbol in the calculator pad.

iv. Select **Phases...** and **Volume fraction** from the **Field Functions** drop-down lists.

v. Ensure that **air** is selected from the **Phase** drop-down list and click **Select**.

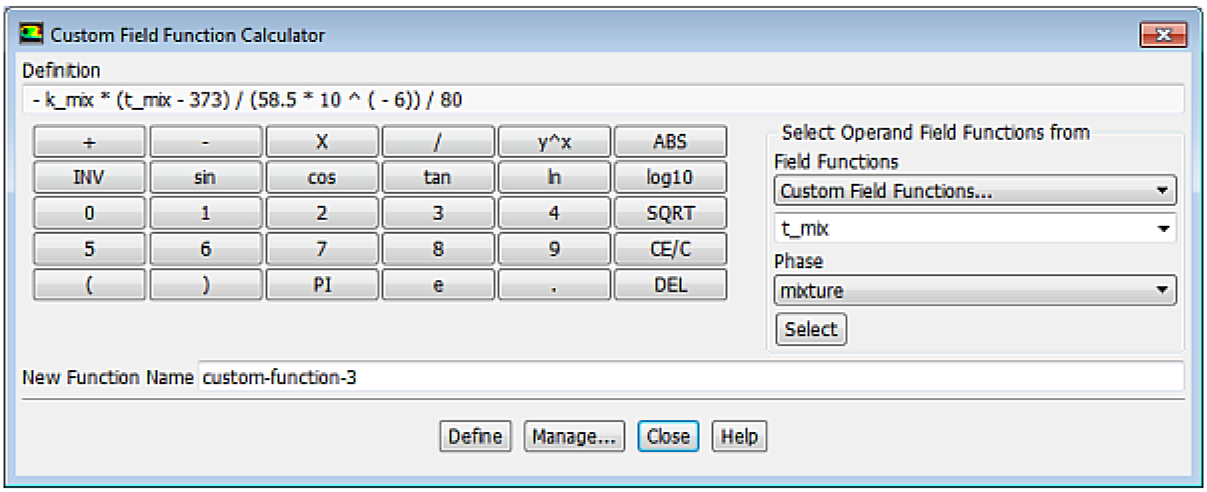
vi. Click the addition symbol in the calculator pad.

vii. Similarly, add the term `solids-thermal-conductivity-lam * solids-vof`.

viii. Enter `k_mix` for **New Function Name**.

ix. Click **Define**.

c. Define the function `ave_htc`.



i. Click the subtraction symbol in the calculator pad.

ii. From the **Field Functions** drop-down lists, select **Custom Field Functions...** and **k_mix** and click **Select**.

iii. Use the calculator pad and the **Field Functions** lists to complete the definition of the function.

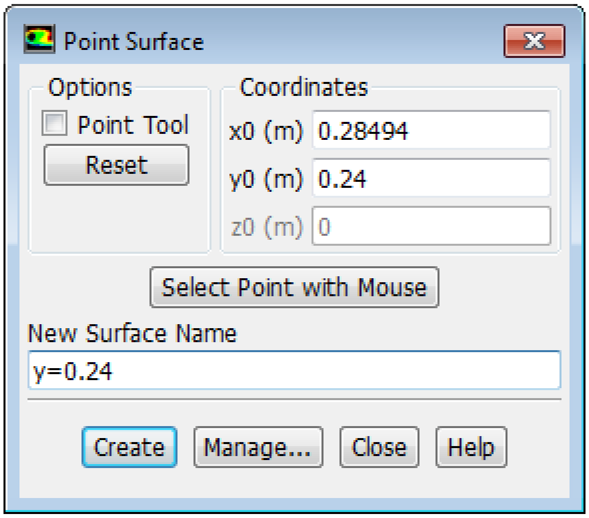
$$-k\_mix \times (t\_mix - 373) / (58.5 \times 10^{(-6)}) / 80$$

iv. Enter `ave_htc` for **New Function Name**.

v. Click **Define** and close the **Custom Field Function Calculator** dialog box.

5. Define the point surface in the cell next to the wall on the plane $y=0.24$.

Setting Up Domain → **Surface** → **Create** → **Point...**



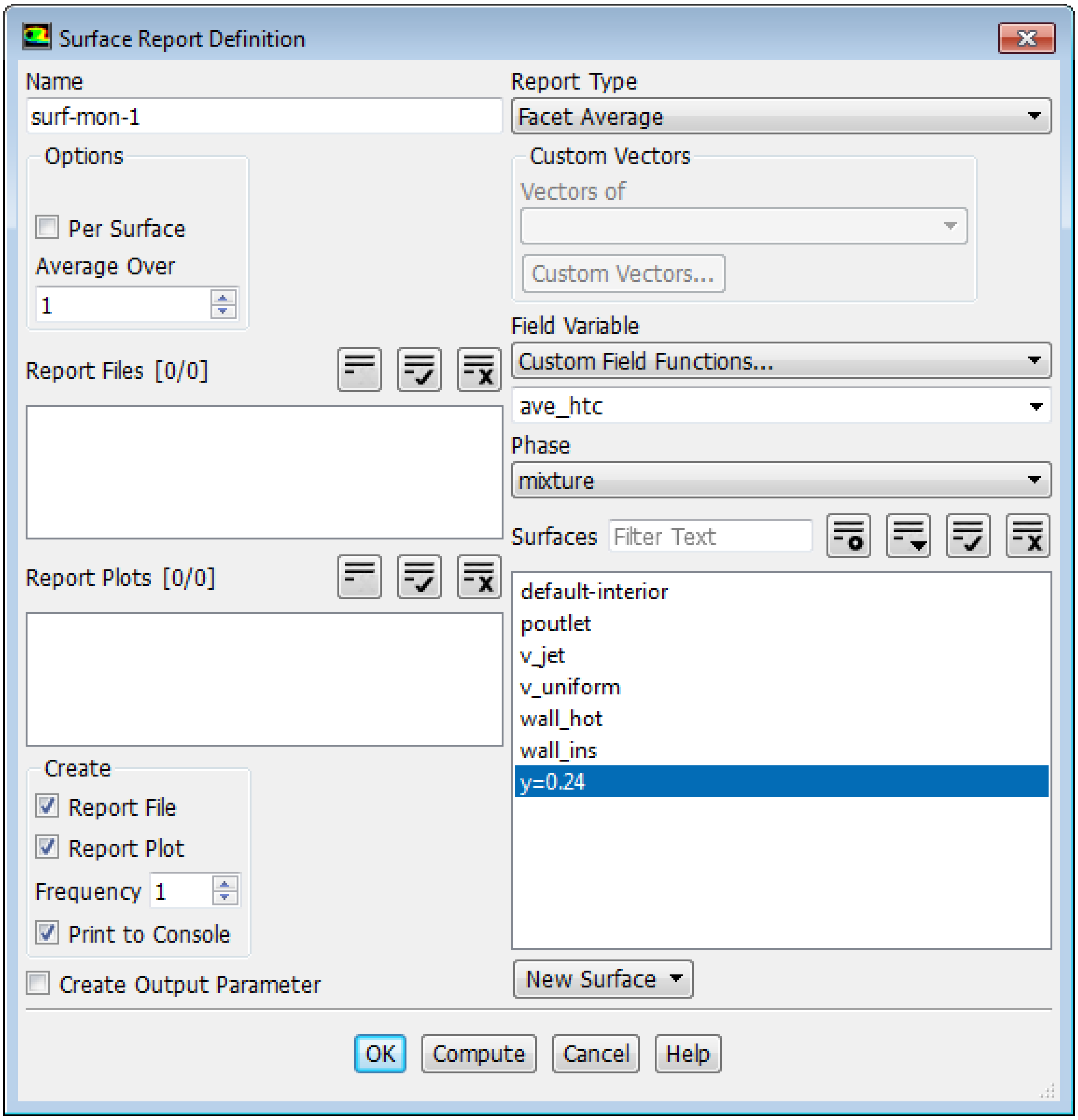
a. Enter `0.28494` m for **x0** and `0.24` m for **y0** in the **Coordinates** group box.

b. Enter `y=0.24` for **New Surface Name**.

c. Click **Create** and close the **Point Surface** dialog box.

6. Create a surface report definition for the heat transfer coefficient.

Solving → Reports → Definitions → New → Surface Report → Facet Average



a. Enter **surf-mon-1** for **Name** of the surface report definition.

b. In the **Create** group box, enable **Report File**, **Report Plot** and **Print to Console**.

c. Select **Custom Field Functions...** and **ave_htc** from the **Field Variable** drop-down lists.

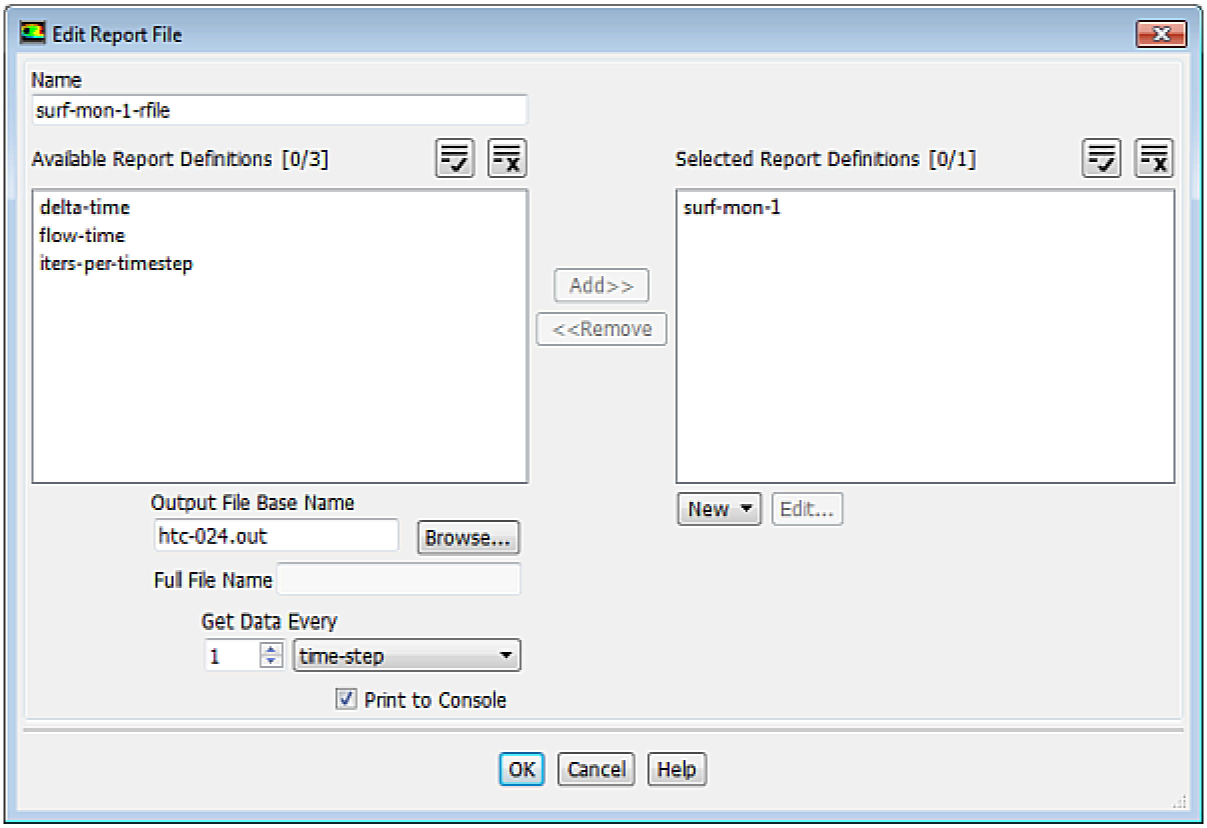
d. Select **y=0.24** from the **Surfaces** selection list.

e. Click **OK** to save the surface report definition settings and close the **Surface Report Definition** dialog box.

   surf-mon-1-rplot and **surf-mon-1-rfile** that are automatically generated by Fluent appear in the tree (under **Solution/Monitors/Report Plots** and **Solution/Monitors/Report Files**, respectively).

f. Rename the report output file.

   Solution → **Monitors** → **Report Files** → **surf-mon-1-rfile** **Edit...**

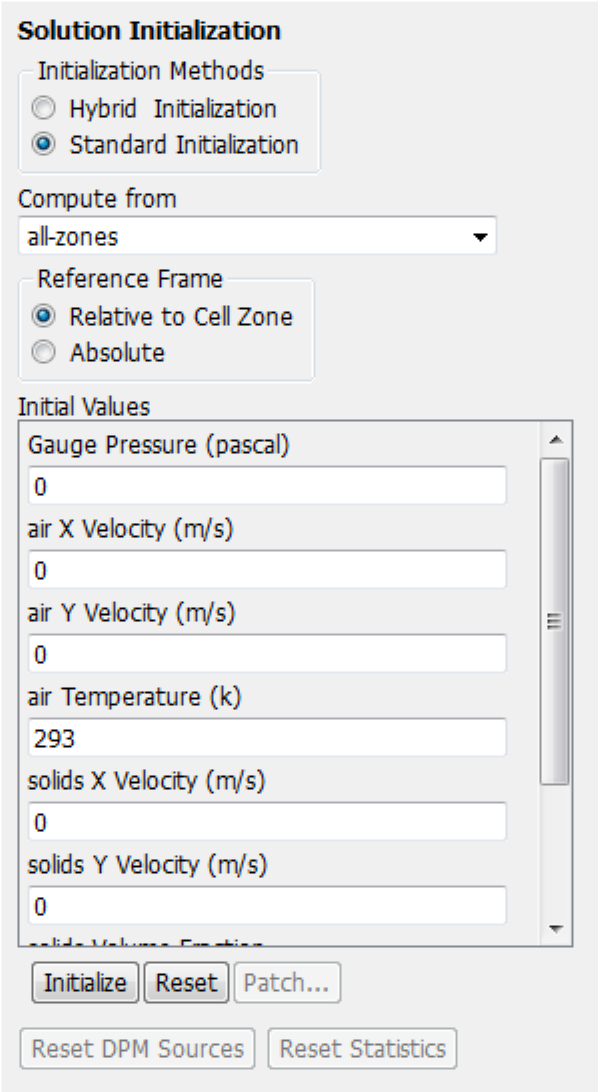


i. Enter `htc-024.out` for **Output File Base Name**.

ii. Click **OK** to close the **Edit Report File** dialog box.

7. Initialize the solution.

Solving → Initialization → Options...



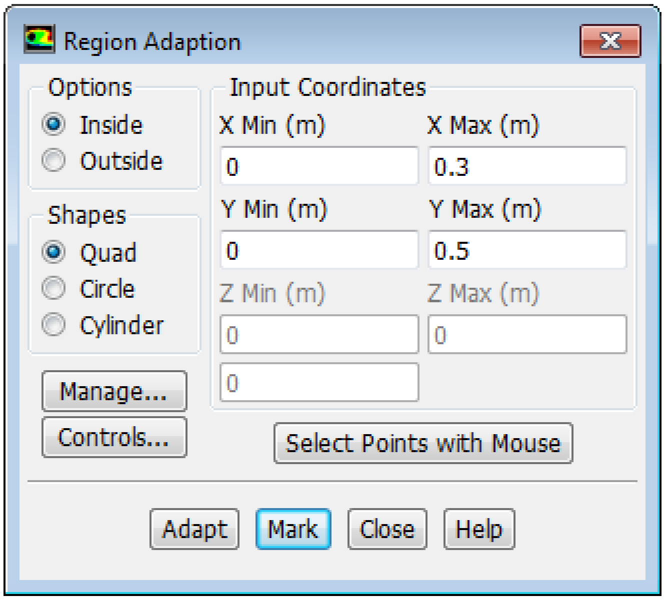
a. Select **all-zones** from the **Compute from** drop-down list.

b. Retain the default values and click **Initialize**.

8. Define an adaption register for the lower half of the fluidized bed.

Setting Up Domain → Adapt → Mark/Adapt Cells → Region...

This register is used to patch the initial volume fraction of solids in the next step.



a. Enter `0.3` m for **Xmax** and `0.5` m for **Ymax** in the **Input Coordinates** group box.

b. Click **Mark**.

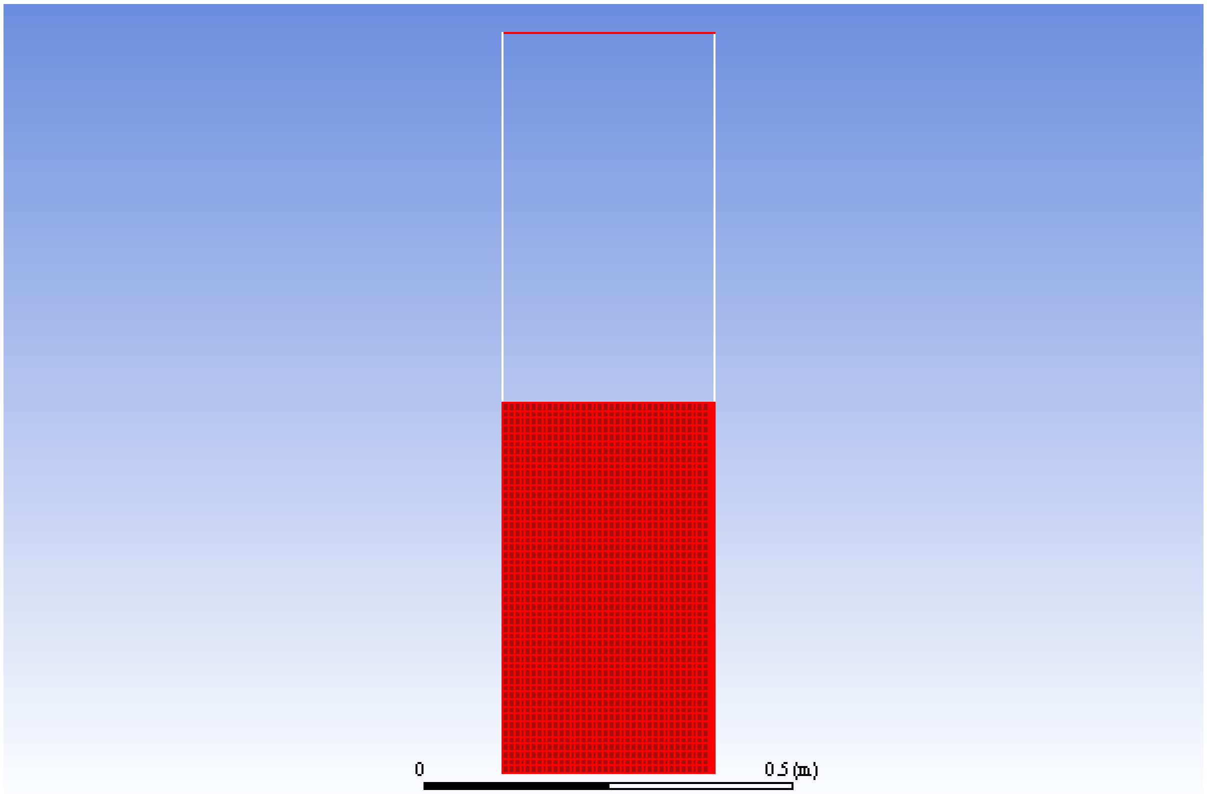
c. Click the **Manage...** button to open the **Manage Adaption Registers** dialog box.

   i. Ensure that **hexahedron-r0** is selected from the **Registers** selection list.

   ii. Click **Display** and close the **Manage Adaption Registers** dialog box.

   After you define a region for adaption, it is a good practice to display it to visually verify that it encompasses the intended area.

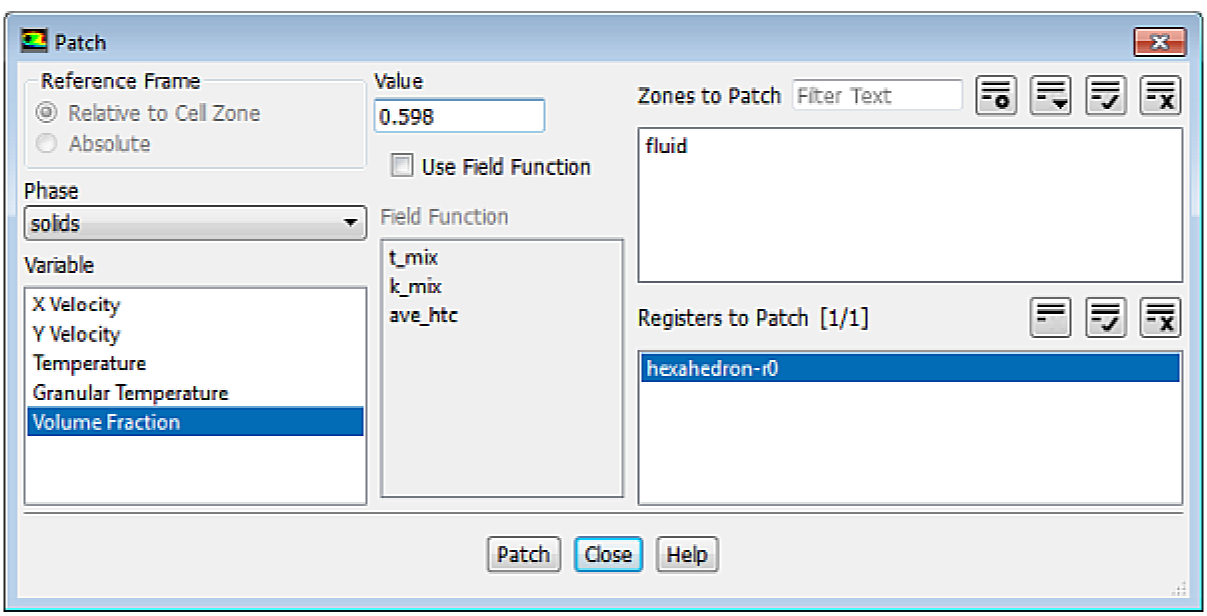
Figure 21.3: Region Marked for Patching



d. Close the **Region Adaption** dialog box.

9. Patch the initial volume fraction of solids in the lower half of the fluidized bed.

Solving → Initialization → Patch...



a. Select **solids** from the **Phase** drop-down list.

b. Select **Volume Fraction** from the **Variable** selection list.

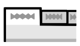
c. Enter `0.598` for **Value**.

d. Select **hexahedron-r0** from the **Registers to Patch** selection list.

e. Click **Patch** and close the **Patch** dialog box.

At this point, it is a good practice to display contours of the variable you just patched, to ensure that the desired field was obtained.

10. Display contours of **Volume Fraction** of solids (Figure 21.4: Initial Volume Fraction of Granular Phase (solids) (p. 872)).

 **Postprocessing** → **Graphics** → **Contours** → **Edit...**

a. Enable **Filled** in the **Options** group box.

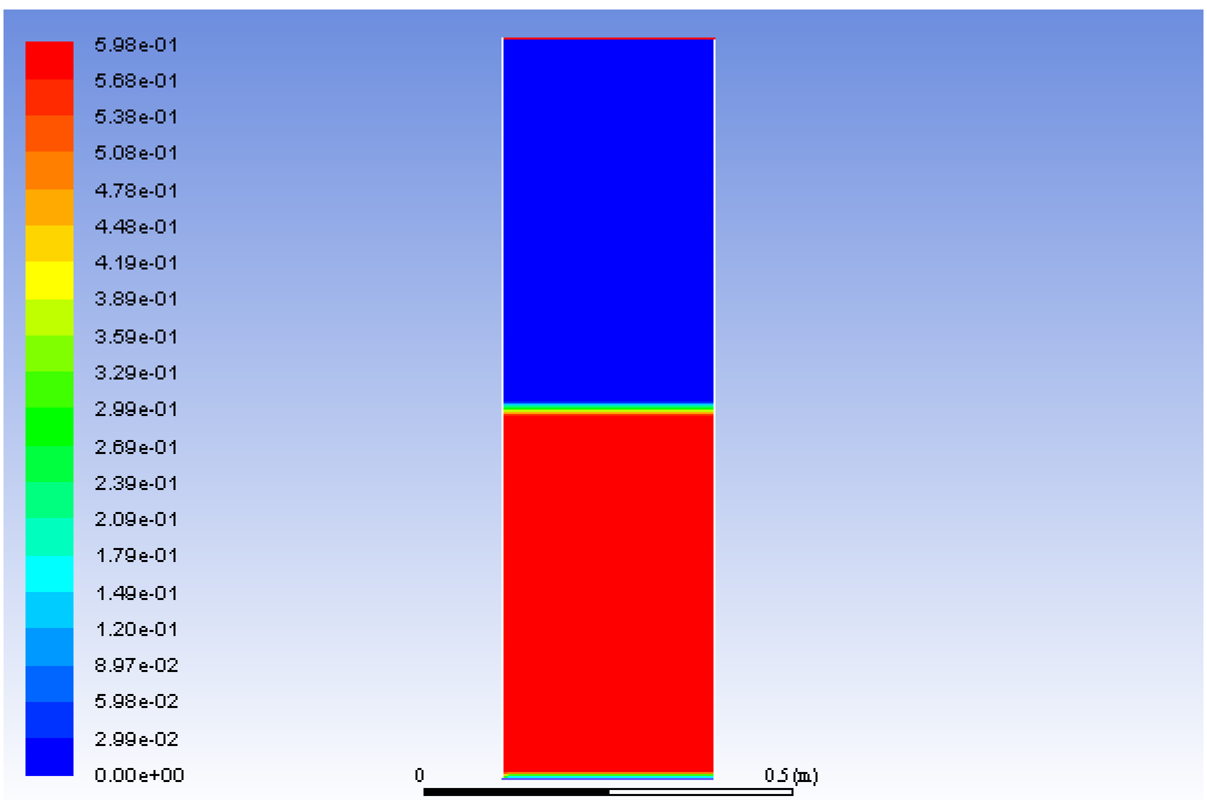
b. Select **Phases...** from the upper **Contours of** drop-down list.

c. Select **solids** from the **Phase** drop-down list.

d. Ensure that **Volume fraction** is selected from the lower **Contours of** drop-down list.

e. Click **Display** and close the **Contours** dialog box.

Figure 21.4: Initial Volume Fraction of Granular Phase (solids)

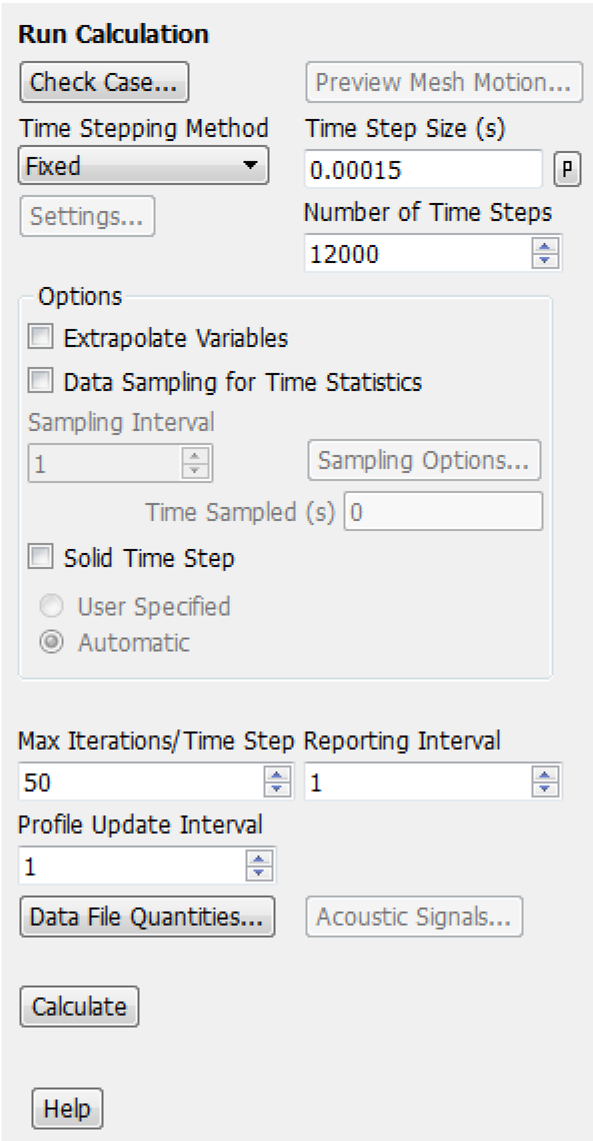


11. Save the case file (`fluid-bed.cas.gz`).

File → **Write** → **Case...**

12. Start calculation.

Solving → **Run Calculation** → **Advanced...**



a. Set `0.00015` for **Time Step Size(s)**.

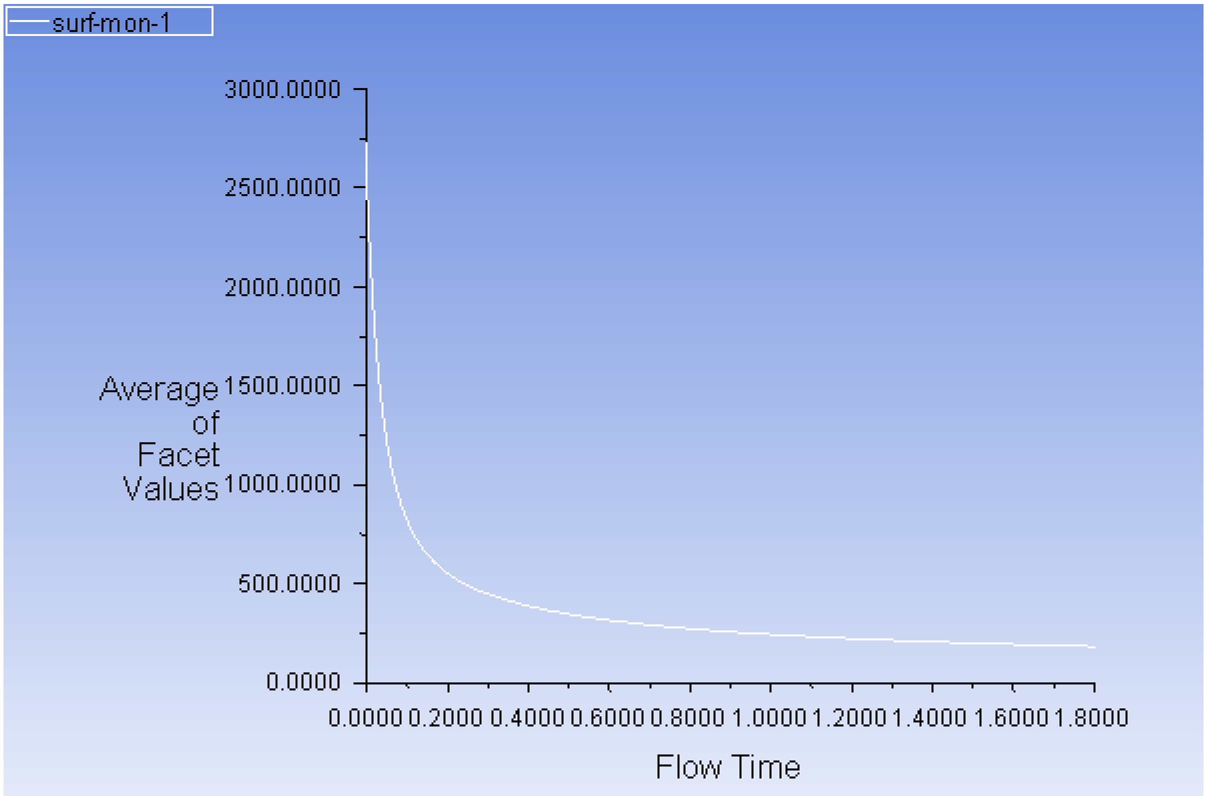
b. Set `12000` for **Number of Time Steps**.

c. Enter `50` for **Max Iterations/Time Step**.

d. Click **Calculate**.

The plot of the value of the mixture-averaged heat transfer coefficient in the cell next to the heated wall versus time is in excellent agreement with results published for the same case [1].

Figure 21.5: Plot of Mixture-Averaged Heat Transfer Coefficient in the Cell Next to the Heated Wall Versus Time



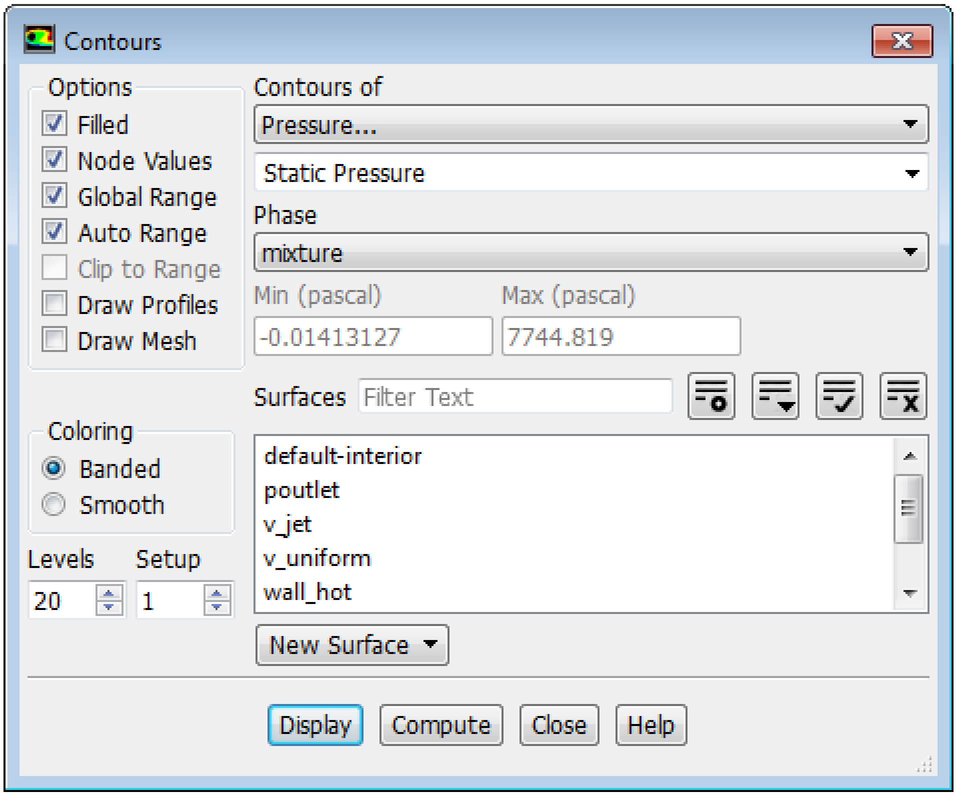
13. Save the case and data files (`fluid-bed.cas.gz` and `fluid-bed.dat.gz`).

    File → Write → Case & Data...

21.4.10. Postprocessing

1. Display the pressure field in the fluidized bed (Figure 21.6: Contours of Static Pressure (p. 876)).

    Postprocessing → Graphics → Contours → Edit...

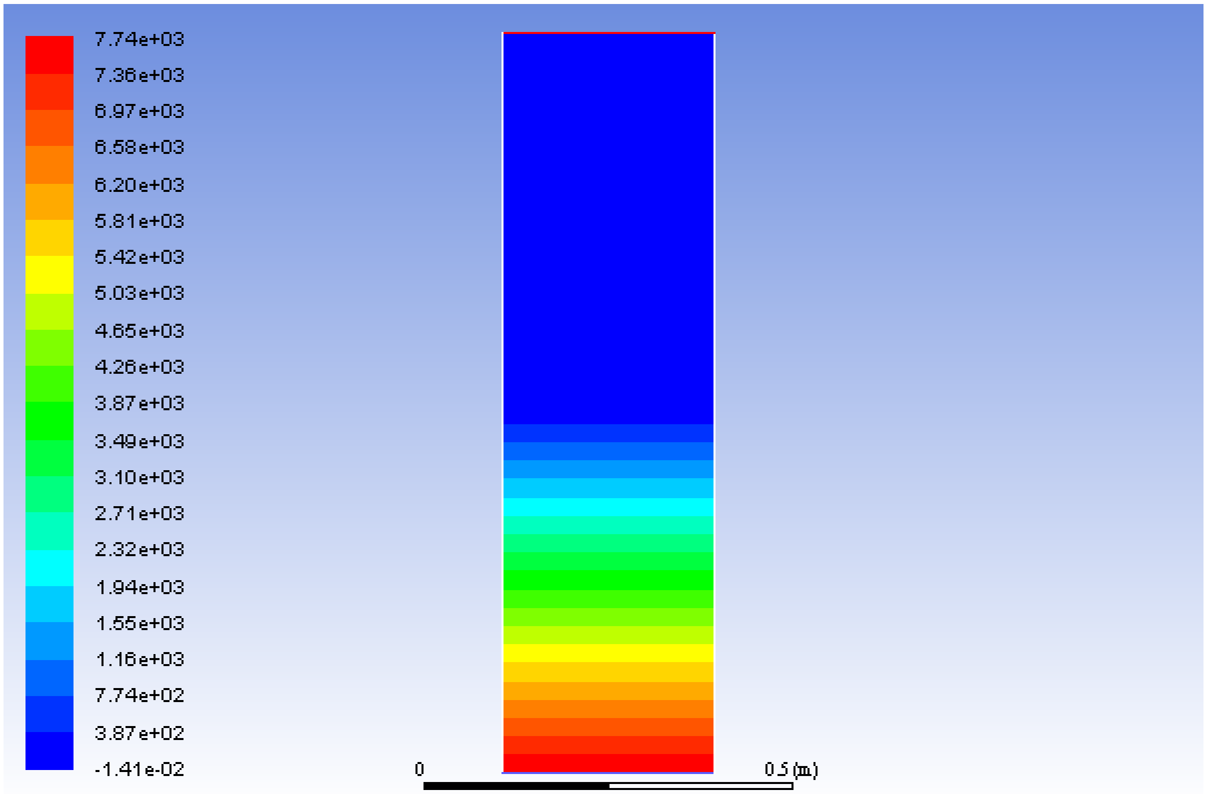


a. Select **mixture** from **Phase** drop-down list.

b. Select **Pressure...** and **Static Pressure** from the **Contours of** drop-down lists.

c. Click **Display**.

Figure 21.6: Contours of Static Pressure



Note the build-up of static pressure in the granular phase.

2. Display the volume fraction of solids (Figure 21.7: Contours of Volume Fraction of Solids (p. 877)).

   a. Select **solids** from the **Phase** drop-down list.

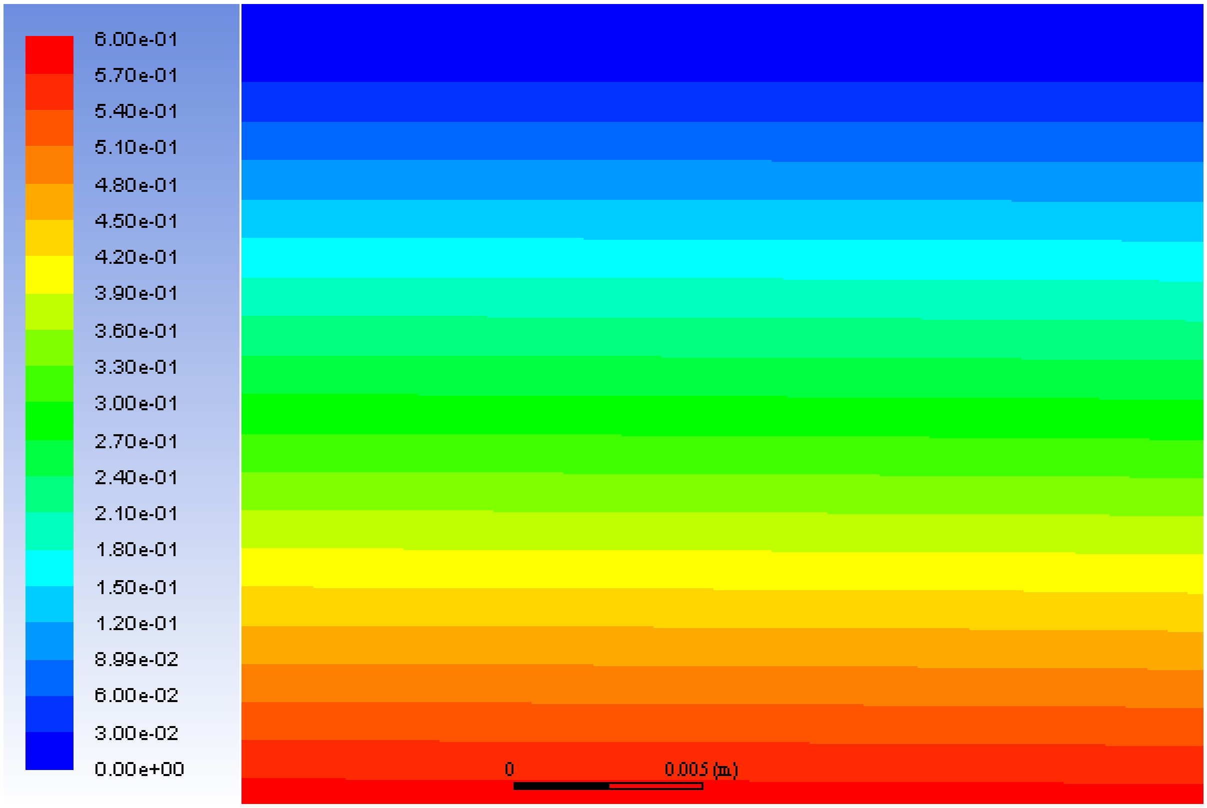
   b. Select **Phases...** and **Volume fraction** from the **Contours of** drop-down lists.

   c. Click **Display** and close the **Contours** dialog box.

   d. Zoom in to show the contours close to the region where the change in volume fraction is the greatest.

   Note that the region occupied by the granular phase has expanded slightly, as a result of fluidization.

Figure 21.7: Contours of Volume Fraction of Solids



21.5. Summary

This tutorial demonstrated how to set up and solve a granular multiphase problem with heat transfer, using the Eulerian model. You learned how to set boundary conditions for the mixture and both phases. The solution obtained is in excellent agreement with analytical results from Kuipers et al. [1].

21.6. Further Improvements

This tutorial guides you through the steps to reach an initial solution. You may be able to obtain a more accurate solution by using an appropriate higher-order discretization scheme and by adapting the mesh further. Mesh adaption can also ensure that the solution is independent of the mesh. These steps are demonstrated in Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121).

21.7. References

1. J. A. M. Kuipers, W. Prins, and W. P. M. Van Swaaij "Numerical Calculation of Wall-to-Bed Heat Transfer Coefficients in Gas-Fluidized Beds", Department of Chemical Engineering, Twente University of Technology, in AIChE Journal, July 1992, Vol. 38, No. 7.

Chapter 22: Postprocessing

This tutorial is divided into the following sections:

22.1. Introduction

This tutorial demonstrates the postprocessing capabilities of Fluent using a 3D model of a flat circuit board with a heat generating electronic chip mounted on it. The flow over the chip is laminar and involves conjugate heat transfer.

The heat transfer involves conduction in the chip and conduction and convection in the surrounding fluid. The physics of conjugate heat transfer such as this are common in many engineering applications, including the design and cooling of electronic components.

In this tutorial, you will read the case and data files (without doing the calculation) and perform a number of postprocessing exercises.

This tutorial demonstrates how to do the following:

- Add lights to the display at multiple locations.
- Create surfaces for the display of 3D data.
- Display filled contours of temperature on several surfaces.
- Display velocity vectors.
- Mirror a display about a symmetry plane.
- Create animations.
- Create a scene.
- Display results on successive slices of the domain.
- Display pathlines.
- Plot quantitative results.
- Overlay and "explode" a display.
- Annotate the display.

22.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

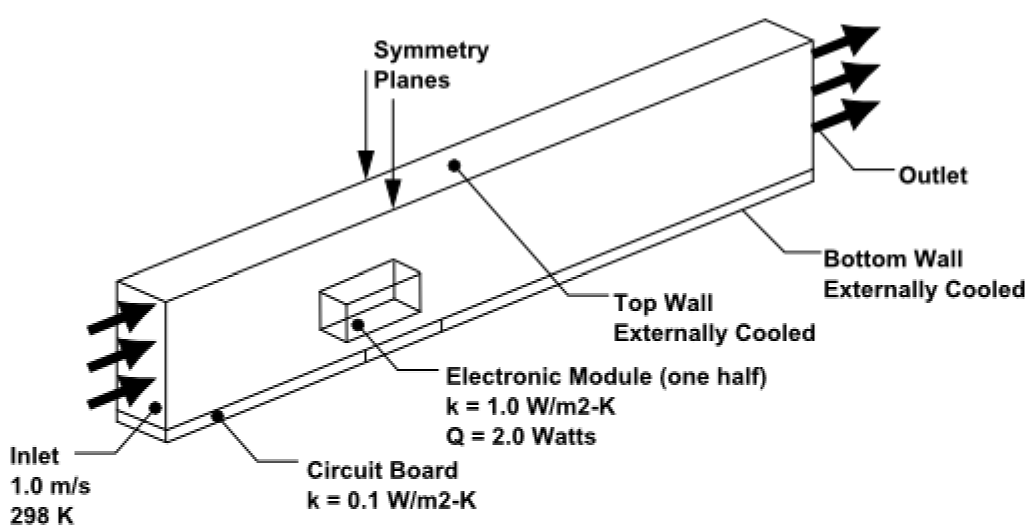
and that you are familiar with the Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

22.3. Problem Description

The problem considered is shown schematically in Figure 22.1: Problem Specification (p. 880). The configuration consists of a series of side-by-side electronic chips, or modules, mounted on a circuit board. Air flow, confined between the circuit board and an upper wall, cools the modules. To take advantage of the symmetry present in the problem, the model will extend from the middle of one module to the plane of symmetry between it and the next module.

As shown in the figure, each half-module is assumed to generate 2.0 Watts and to have a bulk conductivity of 1.0 $\mathrm{W/m^2}$-K. The circuit board conductivity is assumed to be one order of magnitude lower: 0.1 $\mathrm{W/m^2}$-K. The air flow enters the system at 298 K with a velocity of 1 m/s. The Reynolds number of the flow, based on the module height, is about 600. The flow is therefore treated as laminar.

Figure 22.1: Problem Specification



22.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

22.4.1. Preparation

To prepare for running this tutorial:

1. Set up a working folder on the computer you will be using.

2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

   Note

   If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

3. Enter the name of this tutorial into the search bar.

4. Narrow the results by using the filter on the left side of the page.

   a. Click **ANSYS Fluent** under **Product**.

   b. Click **18.0** under **Version**.

5. Select this tutorial from the list.

6. Click the **postprocess_R180.zip** link to download the input files.

7. Unzip the `postprocess_R180.zip` file you downloaded to your working folder.

   The files `chip.cas.gz` and `chip.dat.gz` can be found in the `postprocess` folder created after unzipping the file.

8. Use Fluent Launcher to start the **3D** version of Fluent.

   Fluent Launcher displays your **Display Options** preferences from the previous session.

   For more information about Fluent Launcher, see starting ANSYS Fluent using the Fluent Launcher in the Fluent Getting Started Guide.

9. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.

10. Ensure that the **Serial** processing option is selected.

11. Ensure that the **Double Precision** option is disabled.

22.4.2. Reading the Mesh

1. Read in the case and data files `chip.cas.gz` and `chip.dat.gz`.

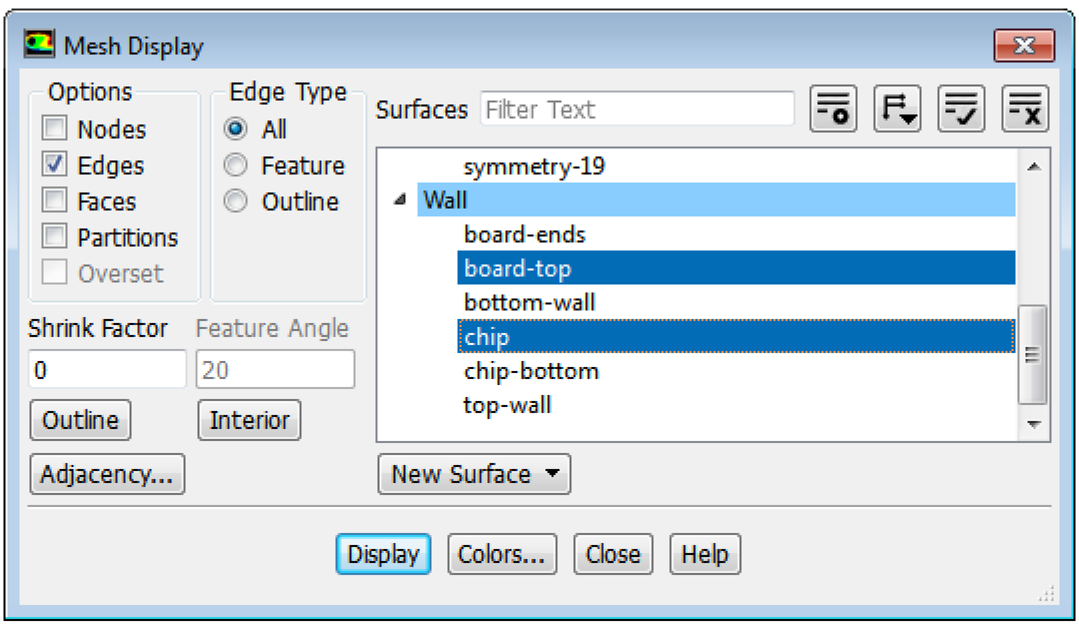
   File → Read → Case & Data...

   When you select the case file, Fluent will read the data file automatically.

22.4.3. Manipulating the Mesh in the Viewer

1. Display the mesh surfaces **board-top** and **chip**.

   Setting Up Domain → Mesh → Display...



   a. Retain the default enabling of the **Edges** option and disable the **Faces** option in the **Options** group box.

   b. Deselect all surfaces and select **board-top** and **chip** from the **Surfaces** selection list.

      *Click to deselect all surfaces. Click and select **Surface Type** under **Group By** to list the surfaces by type, as shown above.*

   c. Click the **Colors...** button to open the **Mesh Colors** dialog box.

      i. Select **Color by ID** in the **Options** group box.

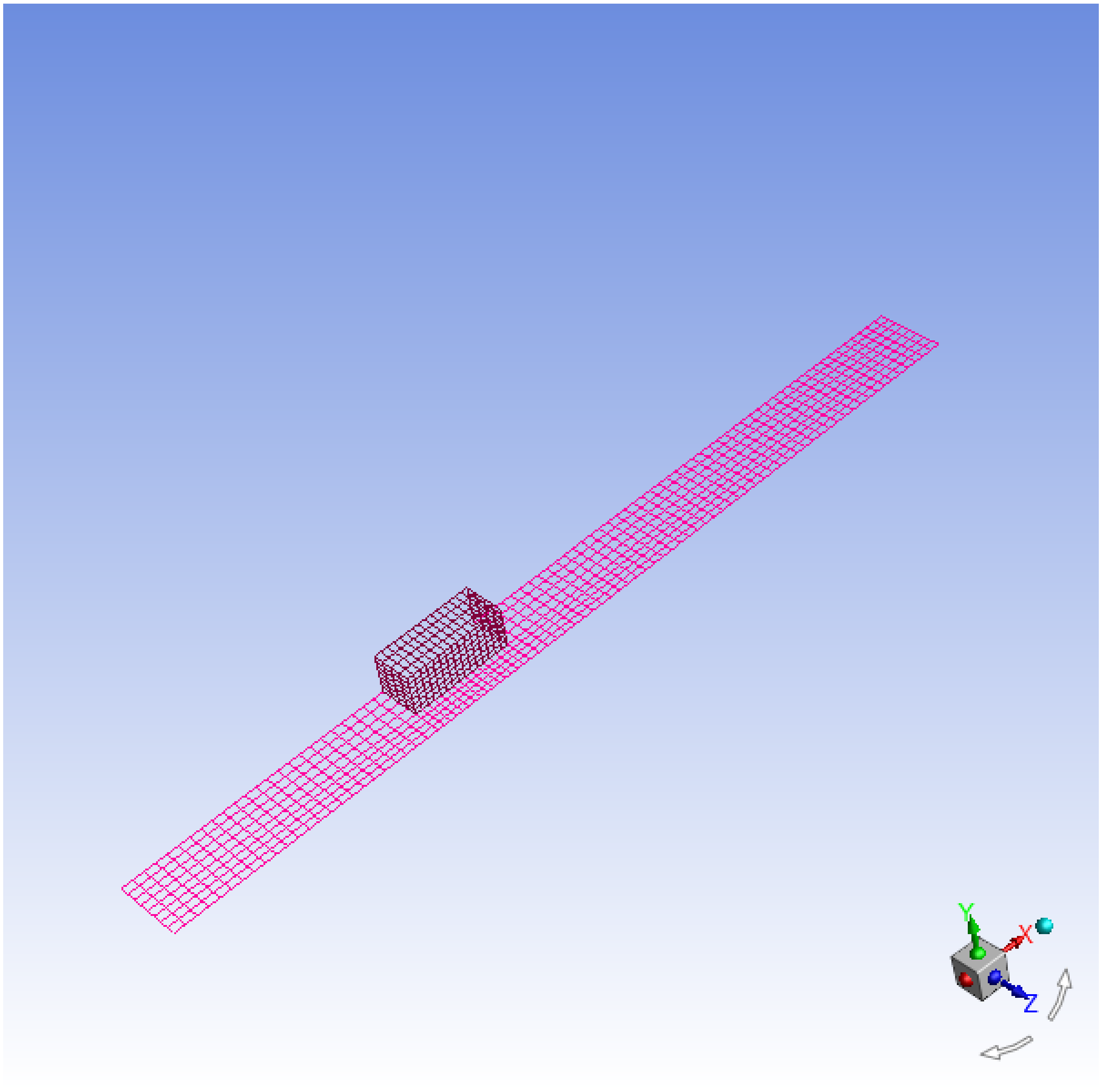
      ii. Click **Reset Colors** to reset the mesh colors to the default settings and close the **Mesh Colors** dialog box.

d. Click **Display**.

2. Rotate and adjust the magnification of the view.

   Use the left mouse button to rotate the view. Use the middle mouse button to adjust the magnification until you obtain an enlarged display of the circuit board in the region of the chip, as shown in *Figure 22.2: Mesh Display of the Chip and Board Surfaces (p. 883)**.*

Figure 22.2: Mesh Display of the Chip and Board Surfaces



Extra

You can click the right mouse button on one of the mesh boundaries displayed in the graphics window and its surface group, ID, and name will be displayed in the console. This feature is especially useful when you have several zones of the same type and you want to distinguish between them.

3. Display the mesh faces.

   Setting Up Domain → Mesh → Display...

   a. Disable **Edges** and enable **Faces** in the **Options** group box.

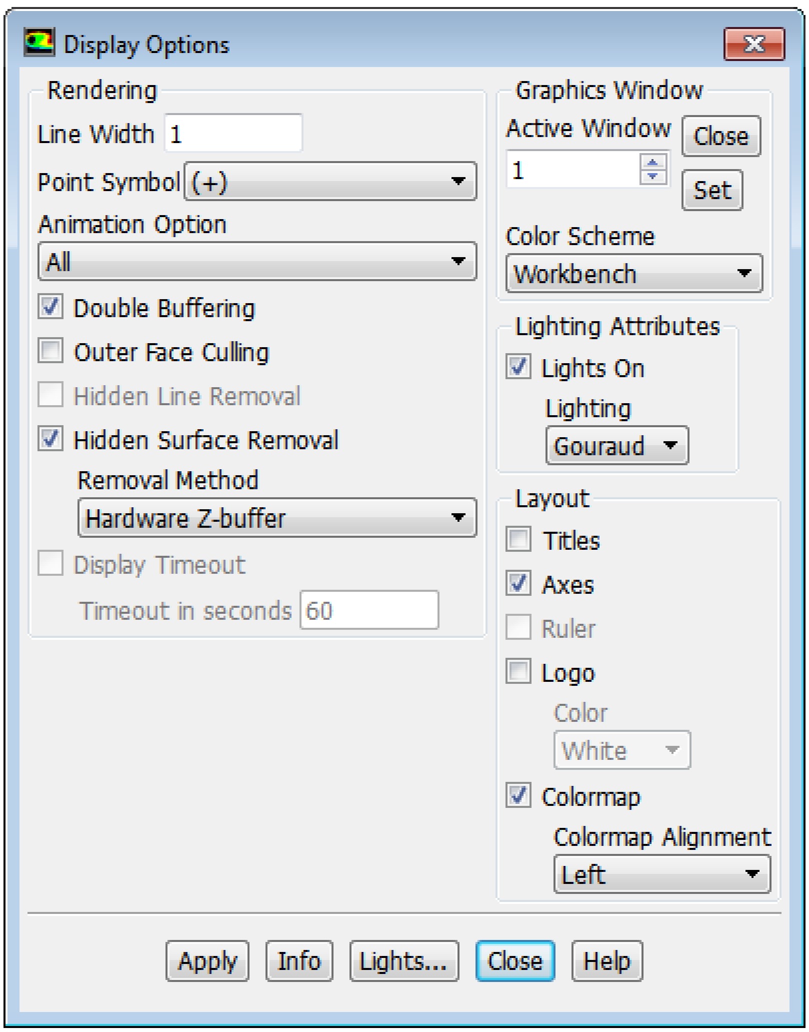
   b. Click **Display** and close the **Mesh Display** dialog box.

22.4.4. Adding Lights

1. Add lighting effects.

   *The default light settings add a white light at the position (1,1,1). The default light is defined in the **Lights** dialog box by the **Light ID** 0 with **Direction** vectors (**X**, **Y**, **Z**) as (1, 1, 1).*

   Viewing → Display → Options...



   a. Make sure that the **Lights On** option is enabled in the **Lighting Attributes** group box.

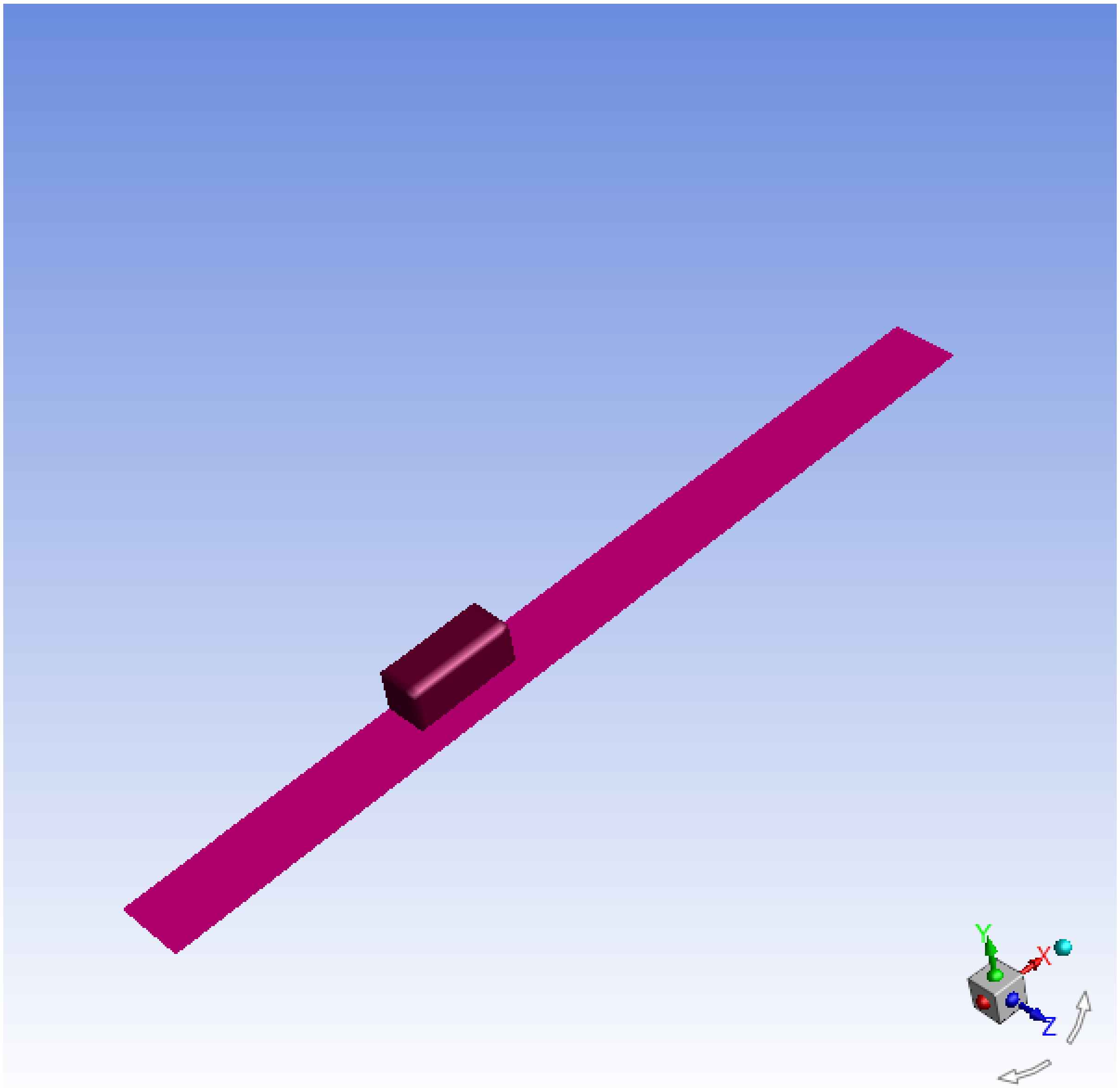
   b. Retain **Gouraud** as the selected **Lighting** method.

      ***Flat** is the most basic lighting whereas **Gouraud** gives better color gradation. Note that **Gouraud** rounds off corners, and so should be used with caution on highly angular geometries.*

c. Click **Apply** and close the **Display Options** dialog box.

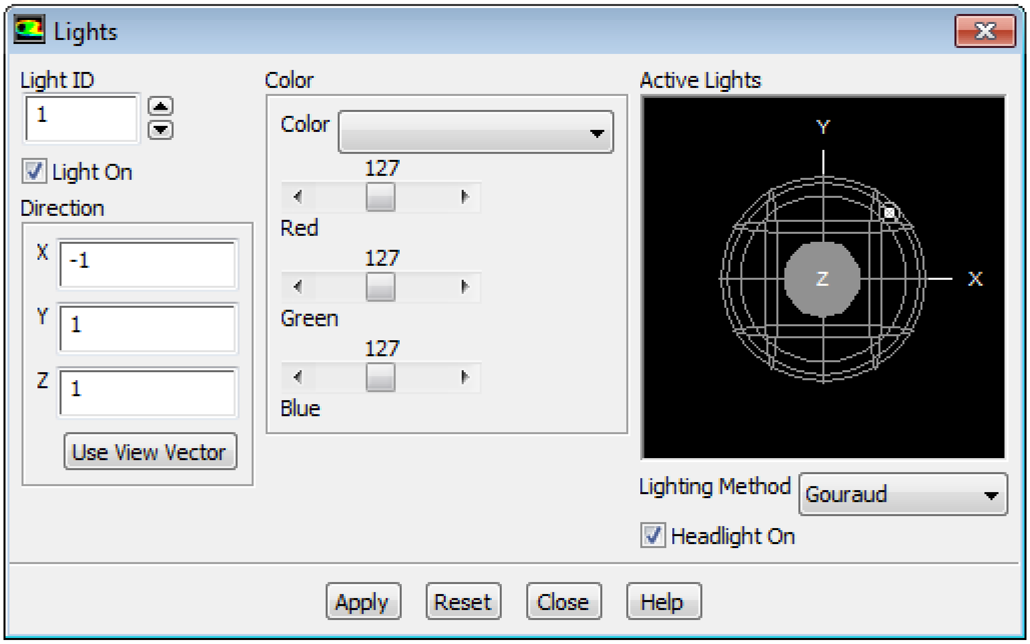
Shading will be added to the surface mesh display (Figure 22.3: Graphics Window with Default Lighting (p. 885)).

Figure 22.3: Graphics Window with Default Lighting



2. Add lights in two directions, (-1, 1, 1) and (-1, 1, -1).

Viewing → Graphics → Lights...



You can also open the ***Lights*** *dialog box by clicking the* ***Lights...*** *button in the* ***Display Options*** *dialog box.*

a. Set **Light ID** to 1.

b. Enable **Light On**.

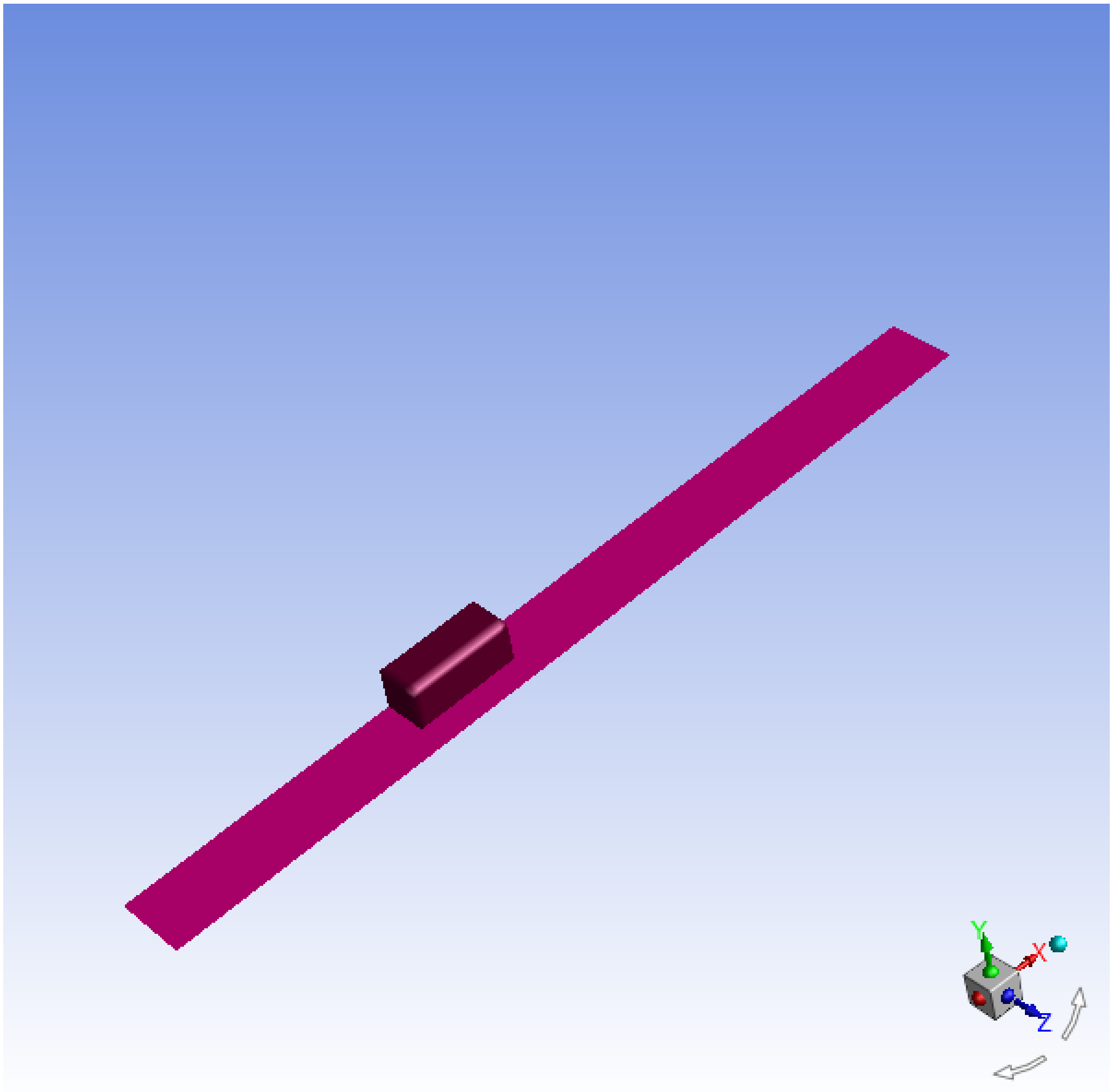
c. Enter -1, 1, and 1 for **X**, **Y**, and **Z** respectively in the **Direction** group box.

d. Retain the selection of **Gouraud** in the **Lighting Method** drop-down list.

e. Make sure that the **Headlight On** option is enabled.

The ***Headlight On*** *option provides constant lighting effect from a light source directly in front of the model, in the direction of the view. You can turn off the headlight by disabling the* ***Headlight On*** *option (the results of this action are shown in* *Figure 22.4: Display with Additional Lighting: - Headlight Off (p. 887)*).

Figure 22.4: Display with Additional Lighting: - Headlight Off

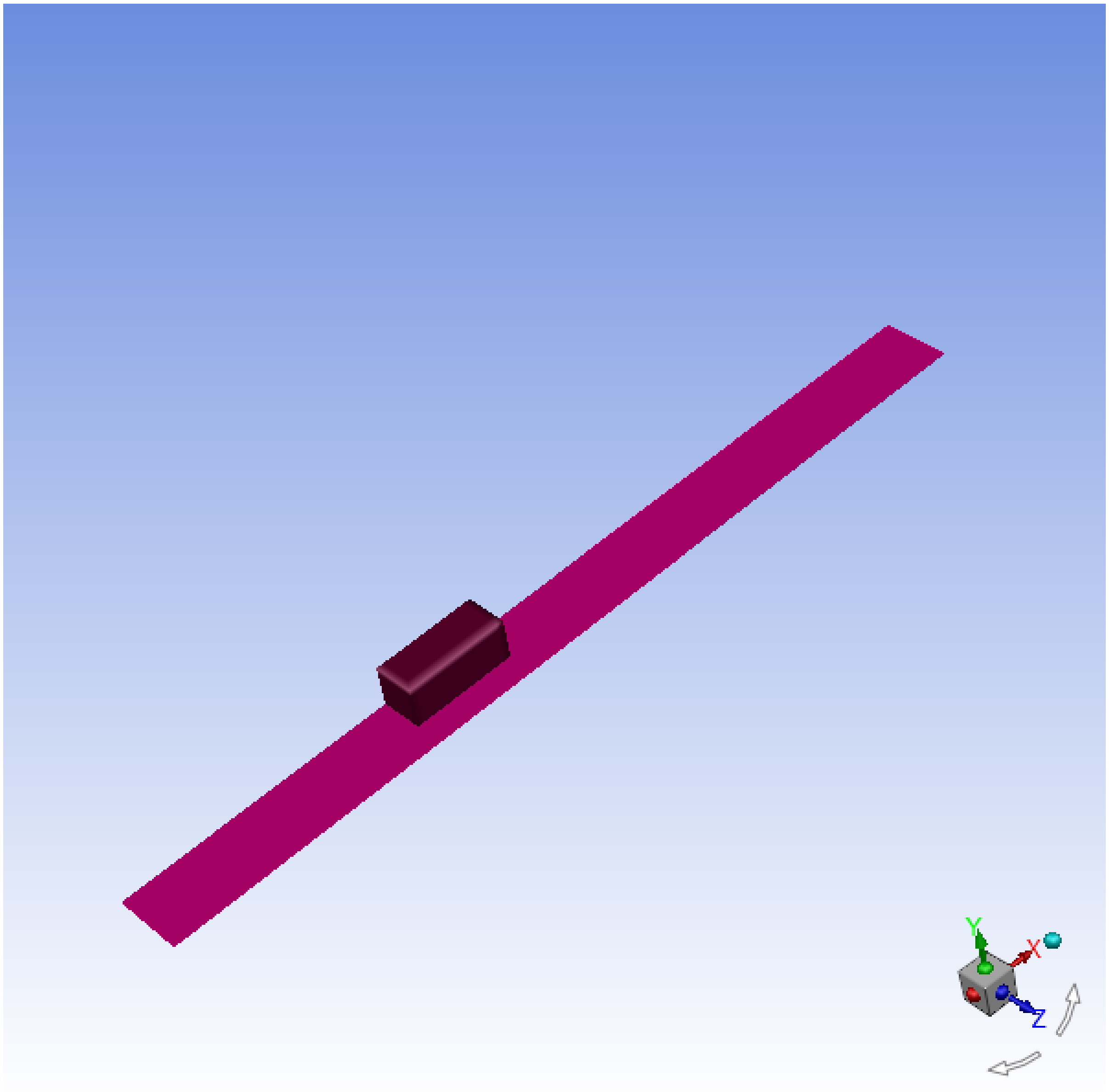


f. Click **Apply**.

g. Similarly, add a second light (**Light ID**= 2 with **Light On** enabled) with a **Direction** of (`-1`, `1`, `-1`).

*The result will be more softly shaded display (**Figure 22.5: Display with Additional Lighting (p. 888)**).*

Figure 22.5: Display with Additional Lighting



h. Close the **Lights** dialog box.

Extra

You can use the left mouse button to rotate the ball in the **Active Lights** window to gain a perspective view on the relative locations of the lights that are currently active, and see the shading effect on the ball at the center.

You can also change the color of one or more of the lights by selecting the color from the **Color** drop-down list or by moving the **Red**, **Green**, and **Blue** sliders.

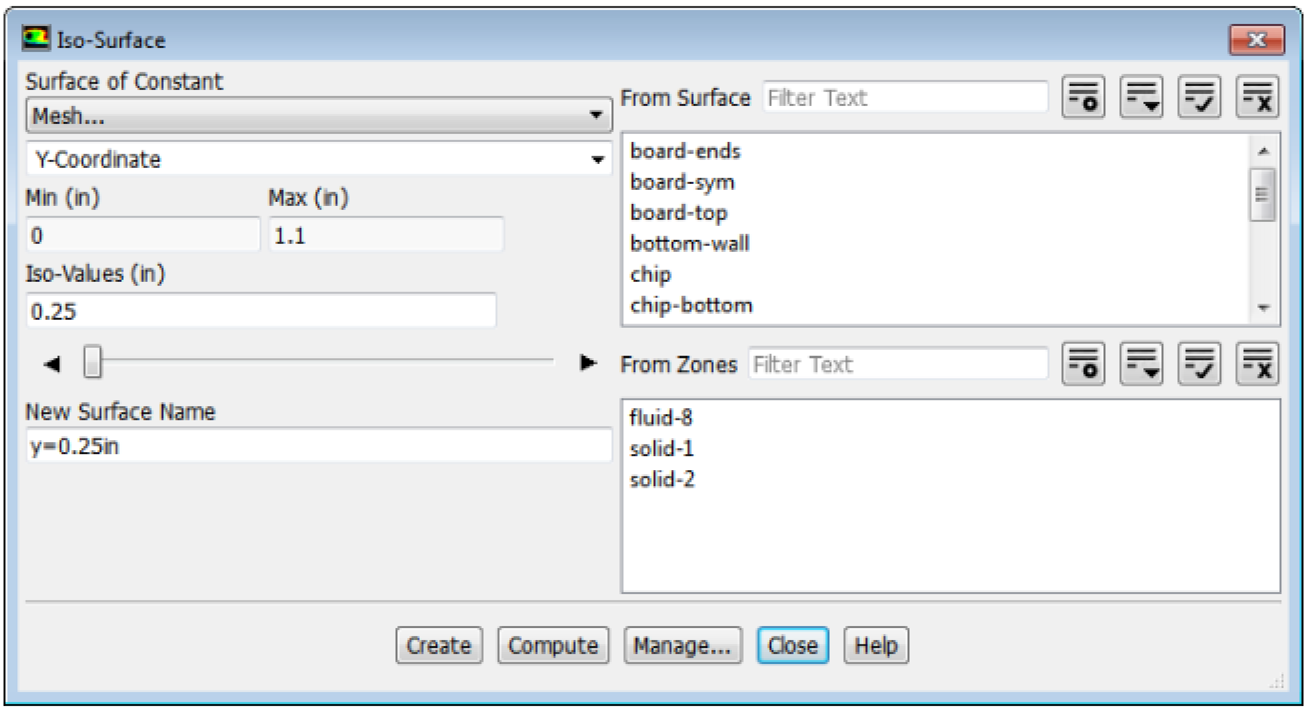
22.4.5. Creating Isosurfaces

*To display results in a 3D model, you will need surfaces on which the data can be displayed. Fluent creates surfaces for all boundary zones automatically. Several surfaces have been renamed after reading the case file. Examples are **board-sym** and **board-ends**, which correspond to the side and end faces of the circuit board.*

You can define additional surfaces for viewing the results, such as a plane in Cartesian space. In this exercise, you will create a horizontal plane cutting through the middle of the module with a Y value of 0.25 inches. You can use this surface to display the temperature and velocity fields.

1. Create a surface of constant Y coordinate.

   Postprocessing → **Surface** → **Create** → **Iso-Surface...**



   a. Select **Mesh...** and **Y-Coordinate** from the **Surface of Constant** drop-down lists.

   b. Click **Compute**.

      *The **Min** and **Max** fields display the Y extents of the domain.*

   c. Enter `0.25` for **Iso-Values**.

   d. Enter `y=0.25in` for **New Surface Name**.

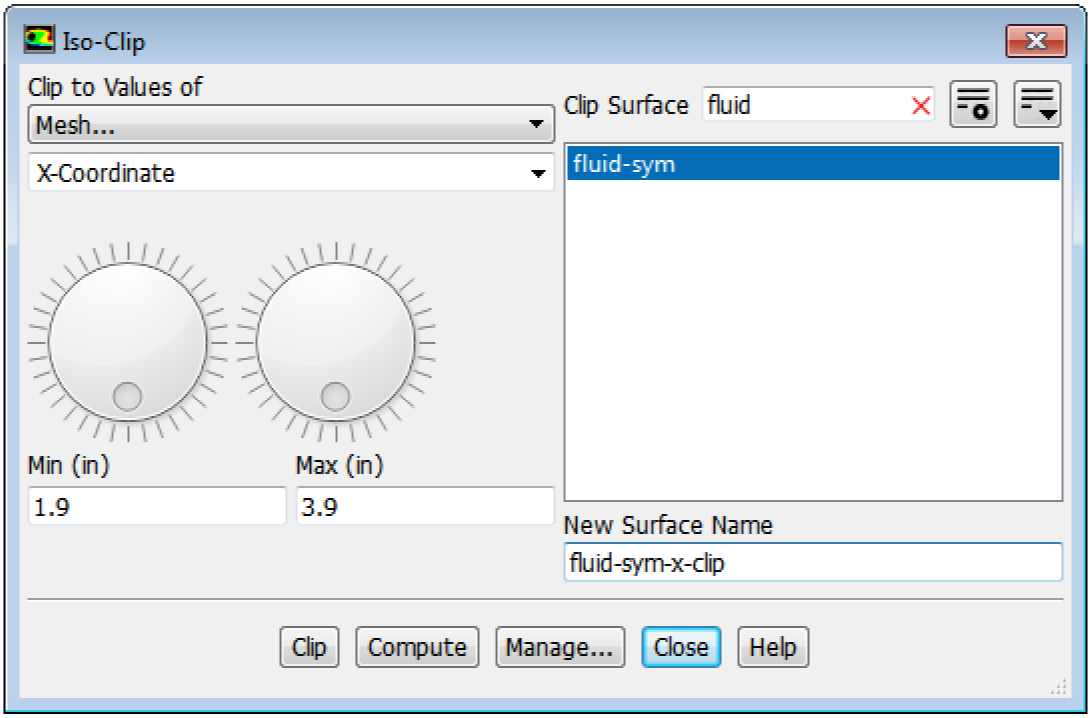
   Tip

   When you are creating multiple postprocessing surfaces, it can be helpful to group surfaces by type for viewing in lists (Click and select **Surface Type** under **Group By**). All iso-surfaces will be grouped together.

e. Click **Create** and close the **Iso-Surface** dialog box.

2. Create a clipped surface for the X coordinate of the fluid (**fluid-sym**).

Setting Up Domain → **Surface** → **Create** → **Iso-Clip...**



a. Select **Mesh...** and **X-Coordinate** from the **Clip to Values of** drop-down lists.

b. Select **fluid-sym** from the **Clip Surface** selection list. *You can type* `fluid` *into the **Filter Text** box to quickly find this surface.*

c. Click **Compute**.

*The **Min** and **Max** fields display the X extents of the domain.*

d. Enter `1.9` and `3.9` for **Min** and **Max** respectively.

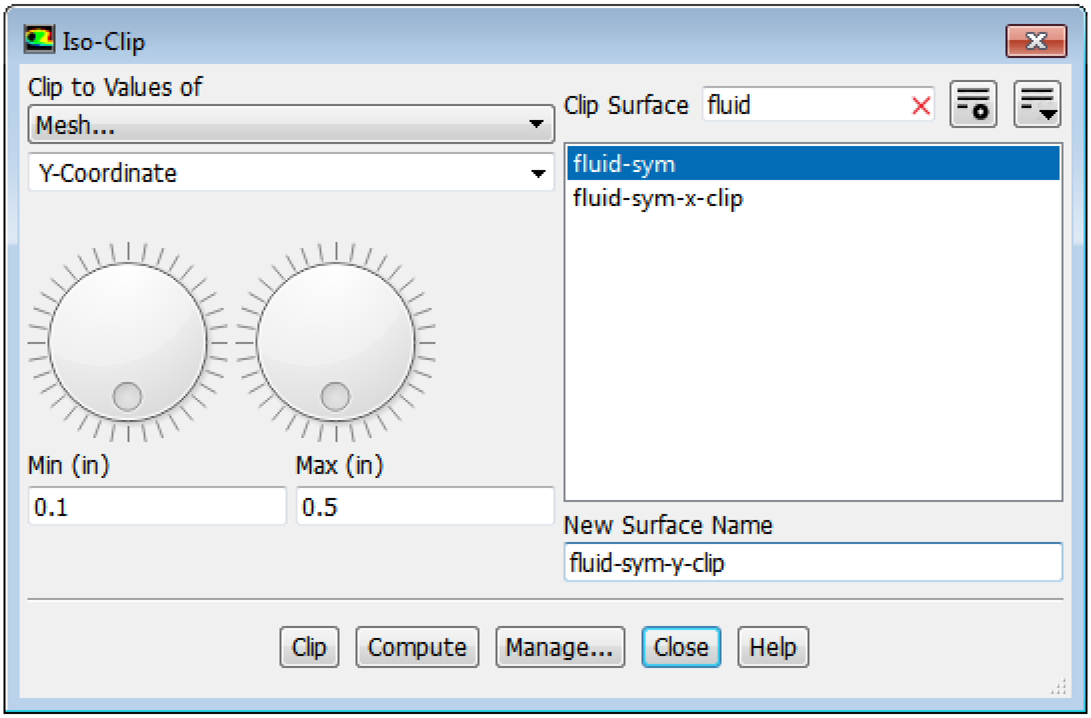
This will isolate the area around the chip.

e. Enter `fluid-sym-x-clip` for **New Surface Name**.

f. Click **Clip**.

3. Create a clipped surface for the Y coordinate of the fluid (**fluid-sym**).

Setting Up Domain → **Surface** → **Create** → **Iso-Clip...**



a. Select **Mesh...** and **Y-Coordinate** from the **Clip to Values of** drop-down lists.

b. Retain the selection of **fluid-sym** from the **Clip Surface** selection list.

c. Click **Compute**.

   *The **Min** and **Max** fields display the Y extents of the domain.*

d. Enter `0.1` and `0.5` for **Min** and **Max** respectively.

   This will isolate the area around the chip.

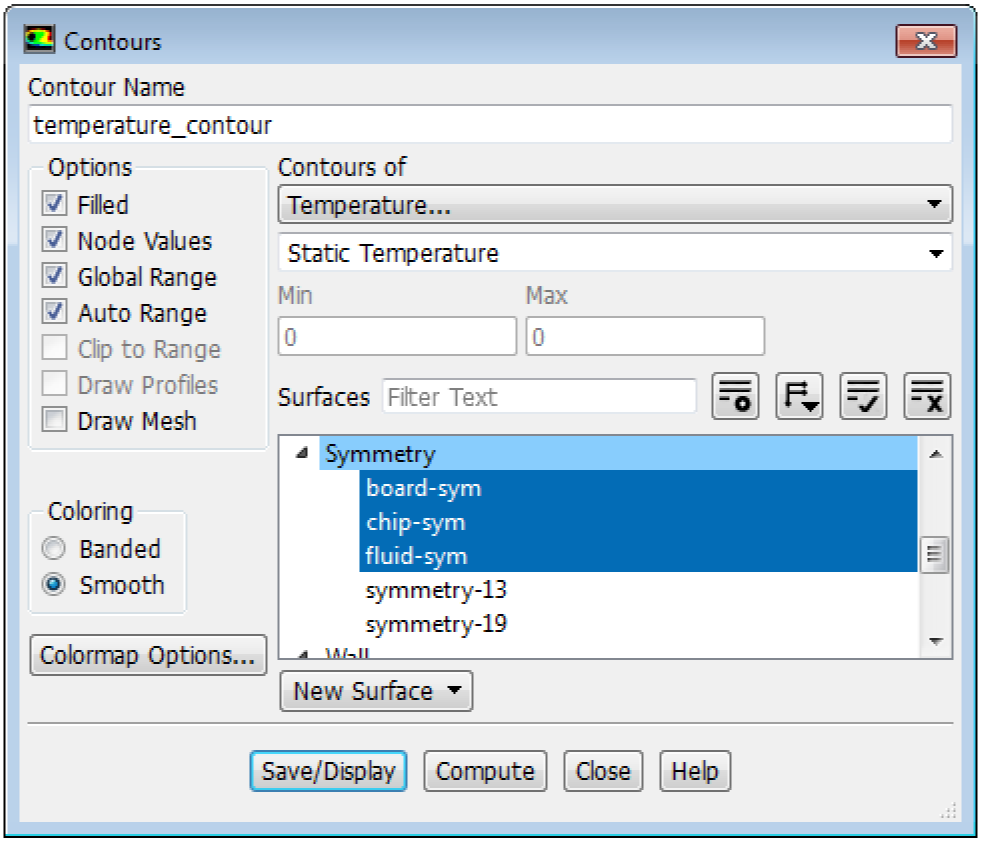
e. Enter `fluid-sym-y-clip` for **New Surface Name**.

f. Click **Clip** and close the **Iso-Clip** dialog box.

22.4.6. Generating Contours

1. Display filled contours of temperature on the symmetry plane (Figure 22.6: Filled Contours of Temperature on the Symmetry Surfaces (p. 893)).

   Postprocessing → **Graphics** → **Contours** → **New...**



a. Enter `temperature_contour` for **Contour Name**.

b. Ensure **Filled**, **Node Values**, **Global Range**, and **Auto Range** are enabled in the **Options** group box.

c. Select **Smooth** for **Coloring**.

d. Select **Temperature...** and **Static Temperature** from the **Contours of** drop-down lists.

e. Click  and select **Surface Type** under **Group By** (if surfaces are not already grouped by type).

f. Select **board-sym**, **chip-sym**, and **fluid-sym** (under **Symmetry** in the **Surfaces** selection list.)

g. Click **Save/Display**.

h. Rotate and adjust the magnification of the view using the left and middle mouse buttons, respectively, to obtain the view as shown in Figure 22.6: Filled Contours of Temperature on the Symmetry Surfaces (p. 893).

Tip

If the model disappears from the graphics window at any time, or if you are having difficulty manipulating it with the mouse, do one of the following:

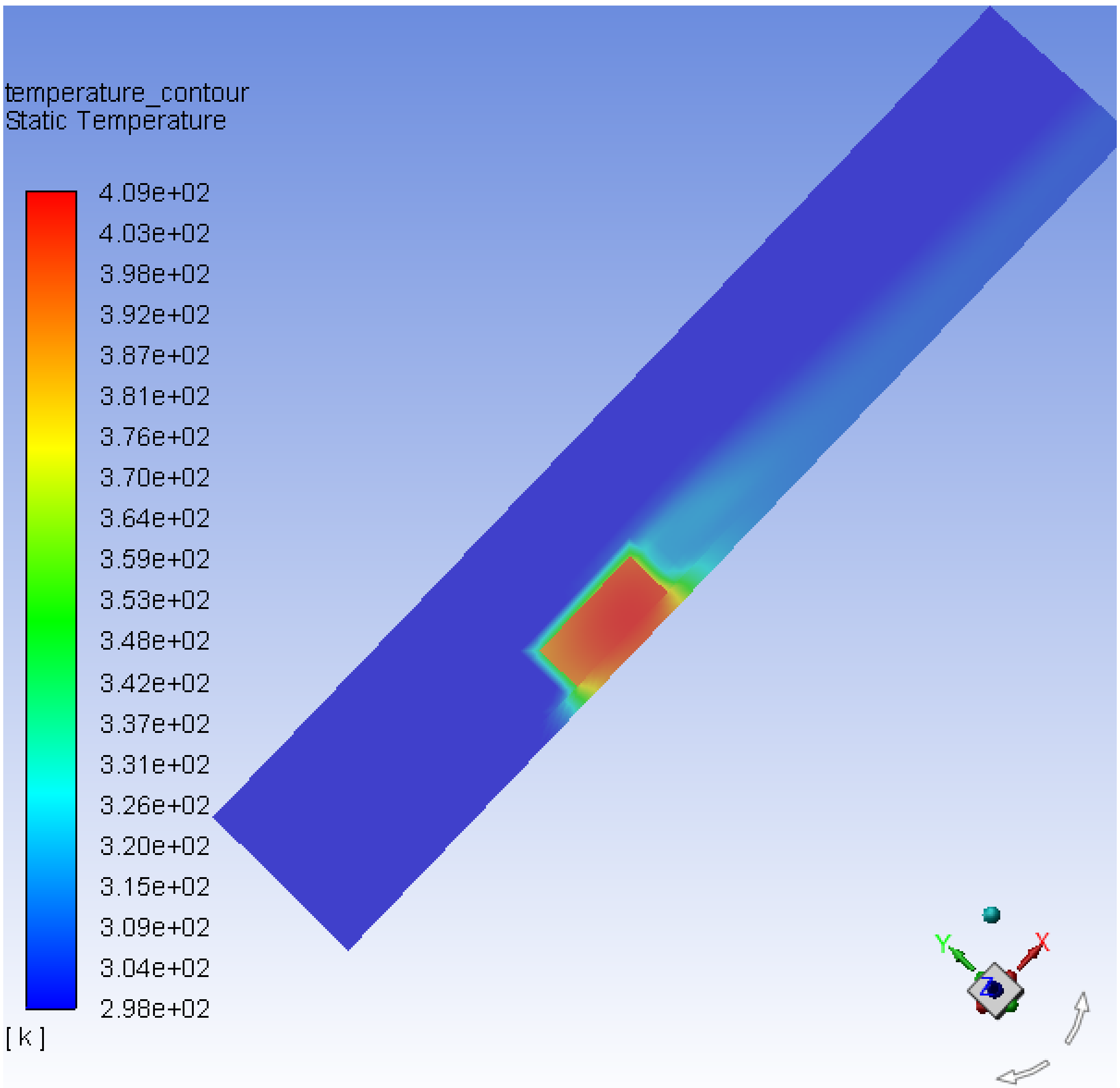
- Click the **Fit to Window** button in the graphics toolbar.
- Open the **Views** dialog box by right-clicking **Graphics** in the tree (under **Results**) and selecting **Views...** from the menu that opens, and then use the **Default** button

to reset the view. You could also click **Camera...** in this dialog box to open the **Camera Parameters** dialog box, where you could select **orthographic** from the **Projection** drop-down list to reduce the likelihood of zooming through the geometry.

- Press the **Ctrl** + **L** to revert to a previous view.

The peak temperatures in the chip appear where the heat is generated, along with the higher temperatures in the wake where the flow is recirculating.

Figure 22.6: Filled Contours of Temperature on the Symmetry Surfaces



2. Display filled contours of temperature for the clipped surface (Figure 22.7: Filled Contours of Temperature on the Clipped Surface (p. 894)).

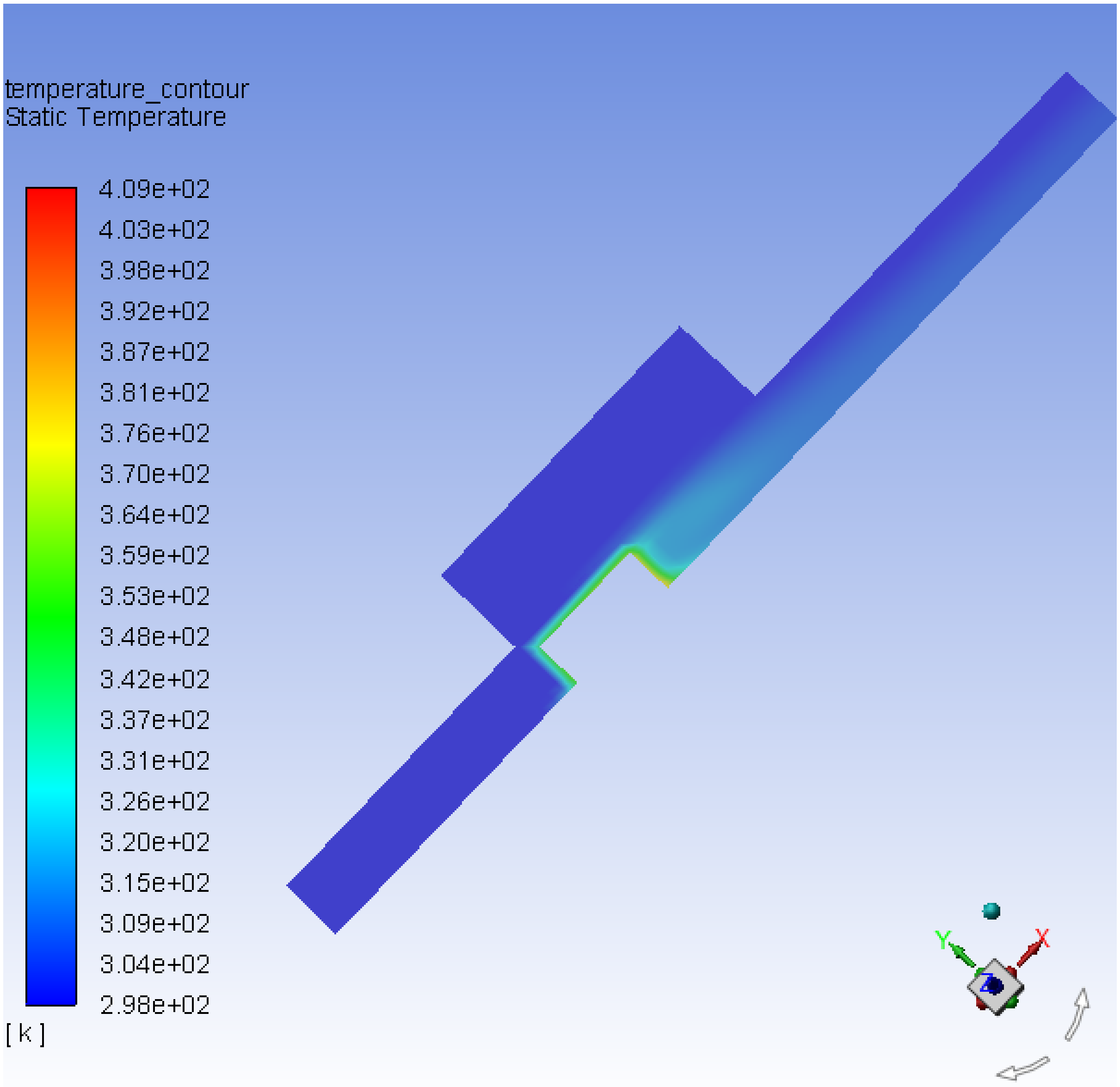
Results → **Graphics** → **Contours** → **temperature_contour** **Edit...**

a. Click to deselect all surfaces from the **Surfaces** selection list and then select **fluid-sym-x-clip** and **fluid-sym-y-clip**.

b. Click **Save/Display**.

A clipped surface appears, colored by temperature (Figure 22.7: Filled Contours of Temperature on the Clipped Surface (p. 894)).

Figure 22.7: Filled Contours of Temperature on the Clipped Surface



3. Display filled contours of temperature on the plane, **y=0.25in** (Figure 22.8: Temperature Contours on the Surface, Y= 0.25 in. (p. 895)).

Results → Graphics → Contours → temperature_contour Edit...

a. Click to deselect all surfaces from the **Surfaces** selection list and then select **y=0.25in**.

b. Click **Save/Display** and close the **Contours** dialog box.

   The filled temperature contours will be displayed on the ***y=0.25in*** *plane.*

c. Orient the view to display the contours.

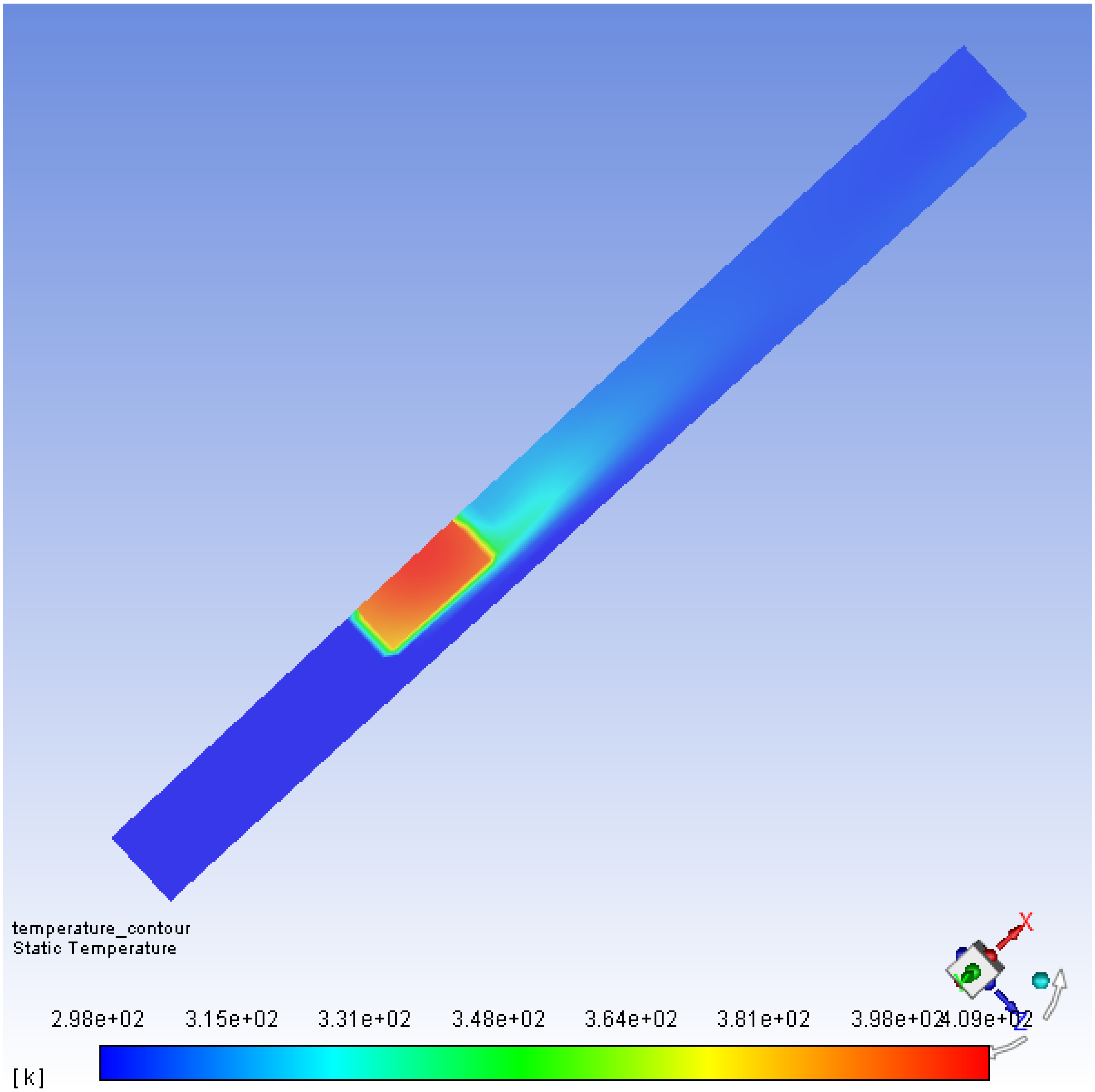
4. Change the location of the colormap in the graphics window.

   Left-click the colormap in the graphics window and drag it to the bottom of the graphics window. *This can also be accomplished using the* ***Display Options*** *dialog box.*

   Tip

   You can increase/decrease the size of the colormap by dragging the corners of the box that appears when you hover over the colormap.

Figure 22.8: Temperature Contours on the Surface, Y= 0.25 in.



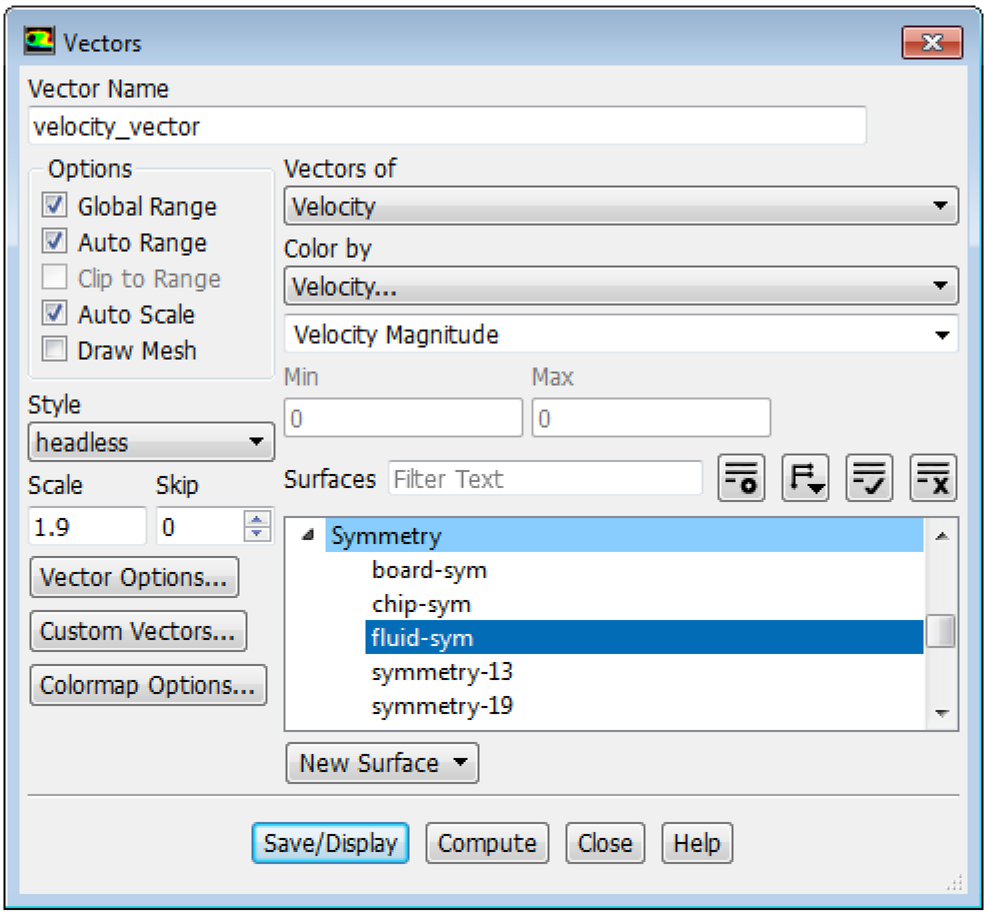
*In Figure 22.8: Temperature Contours on the Surface, Y= 0.25 in. (p. 895), the high temperatures in the wake of the module are clearly visible. You can also display other quantities such as velocity magnitude or pressure using the **Contours** dialog box.*

22.4.7. Generating Velocity Vectors

Velocity vectors provide an excellent visualization of the flow around the module, depicting details of the wake structure.

1. Display velocity vectors on the symmetry plane through the module centerline (Figure 22.9: Velocity Vectors in the Module Symmetry Plane (p. 898)).

   Postprocessing → **Graphics** → **Vectors** → **New...**



   a. Enter `velocity_vector` for **Vector Name**.

   b. Confirm that **Velocity** is selected under **Vectors of** and that **Color by** is set to **Velocity...** and **Velocity Magnitude**.

   c. Ensure **Global Range**, **Auto Range**, and **Auto Scale** are the only enabled **Options**.

   d. Enter `1.9` for **Scale**.

   e. Click the deselect-all button to deselect all surfaces from the **Surfaces** selection list and then select **fluid-sym**.

   f. Click **Save/Display** and close the **Vectors** dialog box.

g. Orient the view to display the vectors.

Extra

You can display velocity vectors for the clipped surfaces.

Results → **Graphics** → **Vectors** → **velocity_vector** **Edit...**

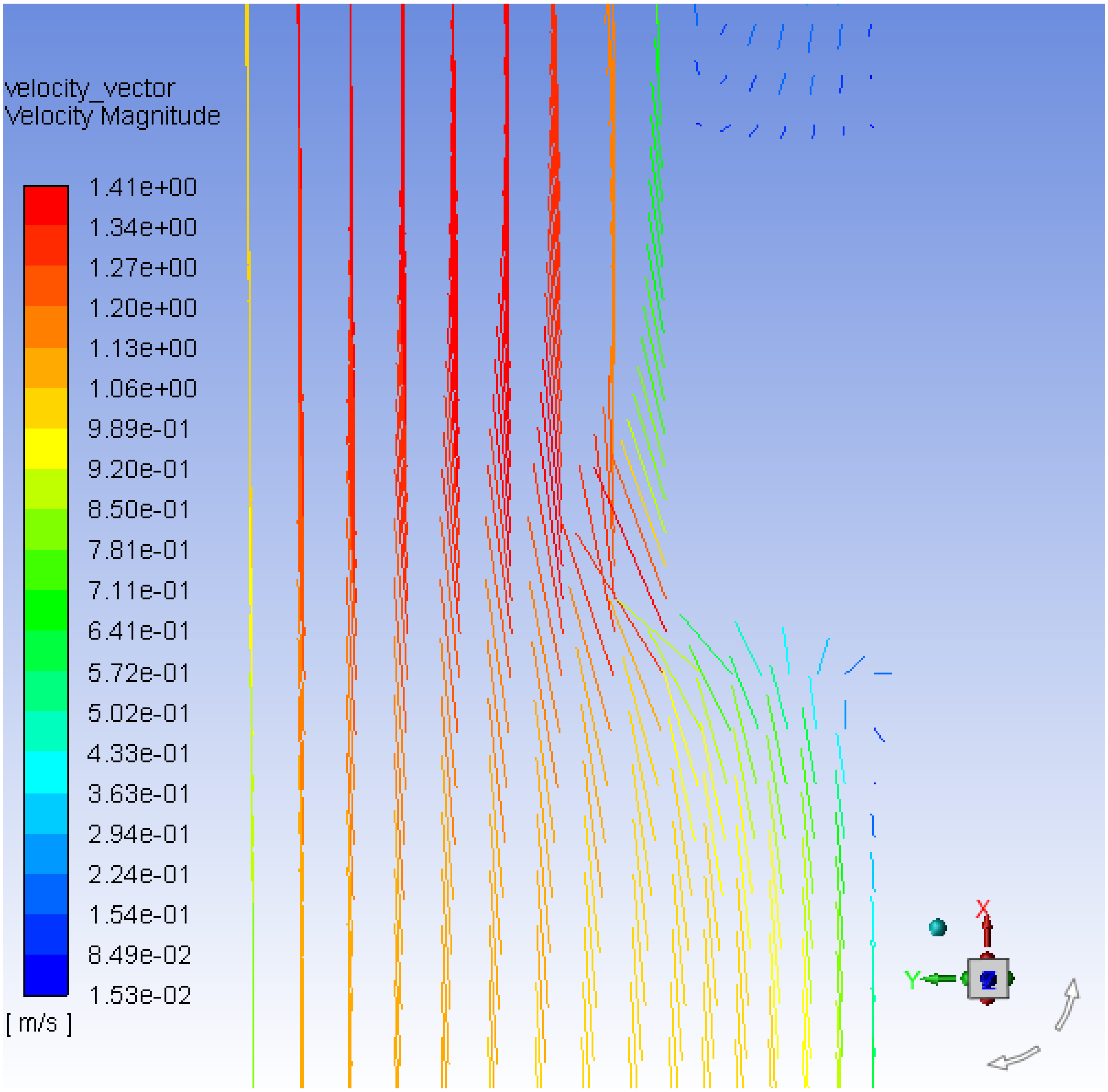
a. Deselect all surfaces from the **Surfaces** selection list and then select **fluid-sym-x-clip** and **fluid-sym-y-clip**.

b. Click **Save/Display**.

---

2. Rotate and adjust the magnification of the view to observe the vortex near the stagnation point and in the wake of the module (Figure 22.9: Velocity Vectors in the Module Symmetry Plane (p. 898)).

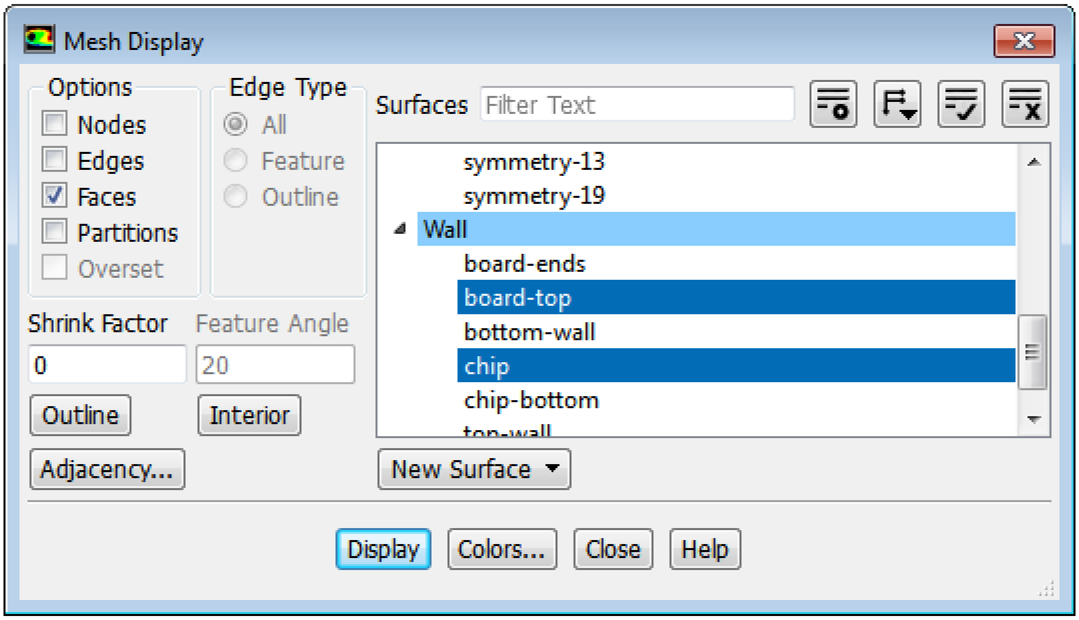
Figure 22.9: Velocity Vectors in the Module Symmetry Plane



3. Plot velocity vectors in the horizontal plane intersecting the module (Figure 22.10: Velocity Vectors Intersecting the Surface (p. 900)).

   Postprocessing → **Graphics** → **Vectors** → **Edit...**

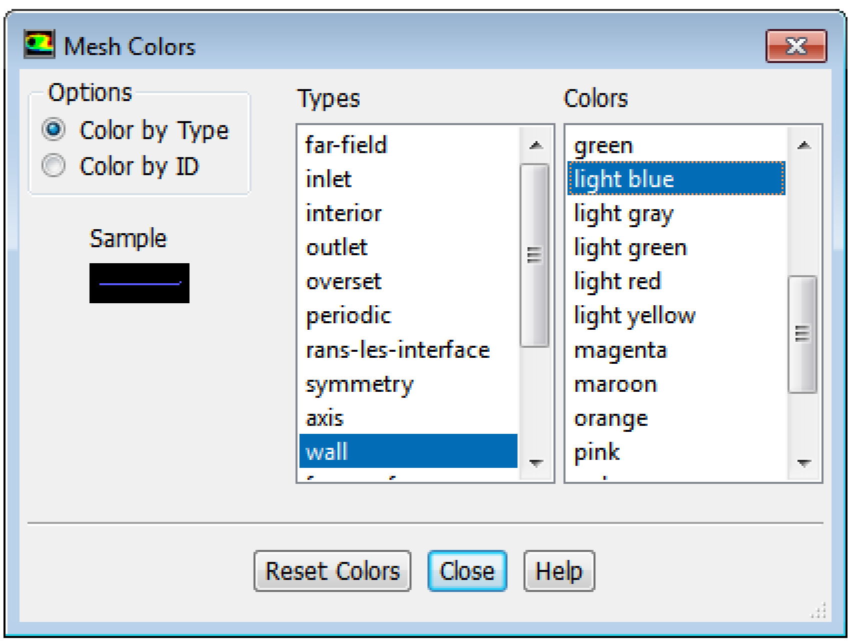
   a. Enable **Draw Mesh** in the **Options** group box to open the **Mesh Display** dialog box.



i. Ensure that **Faces** is enabled in the **Options** group box.

ii. Retain the selection of **board-top** and **chip** from the **Surfaces** selection list.

iii. Click the **Colors...** button to open the **Mesh Colors** dialog box.



A. Select **Color by Type** in the **Options** group box.

B. Select **wall** from the **Types** selection list.

C. Select **light blue** from the **Colors** selection list and close the **Mesh Colors** dialog box.

iv. Click **Display** and close the **Mesh Display** dialog box.

b. Enter 3.8 for **Scale**.

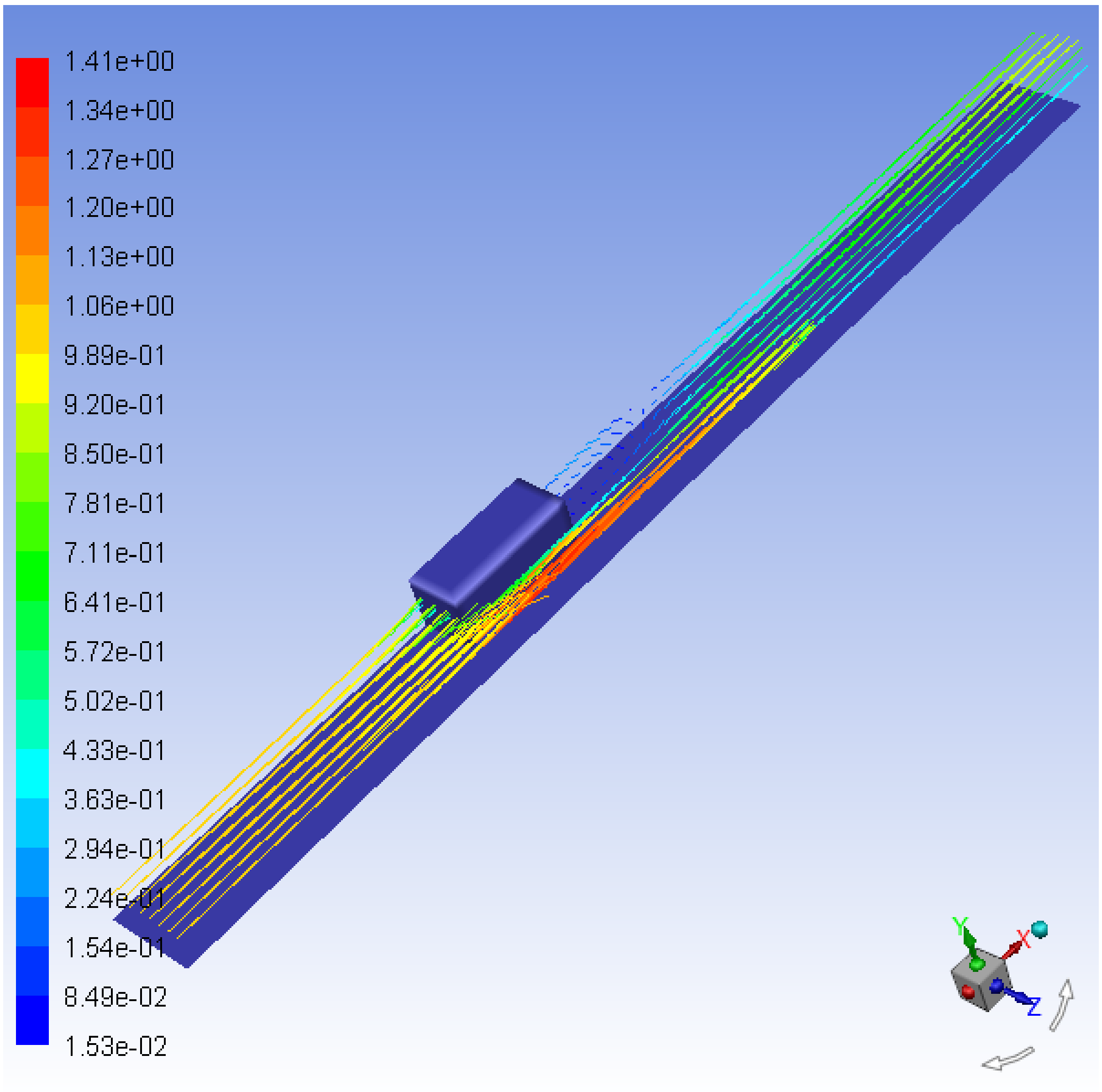
c. Deselect all surfaces.

d. Select **y=0.25in** from the **Surfaces** selection list.

e. Click **Display** and close the **Vectors** dialog box.

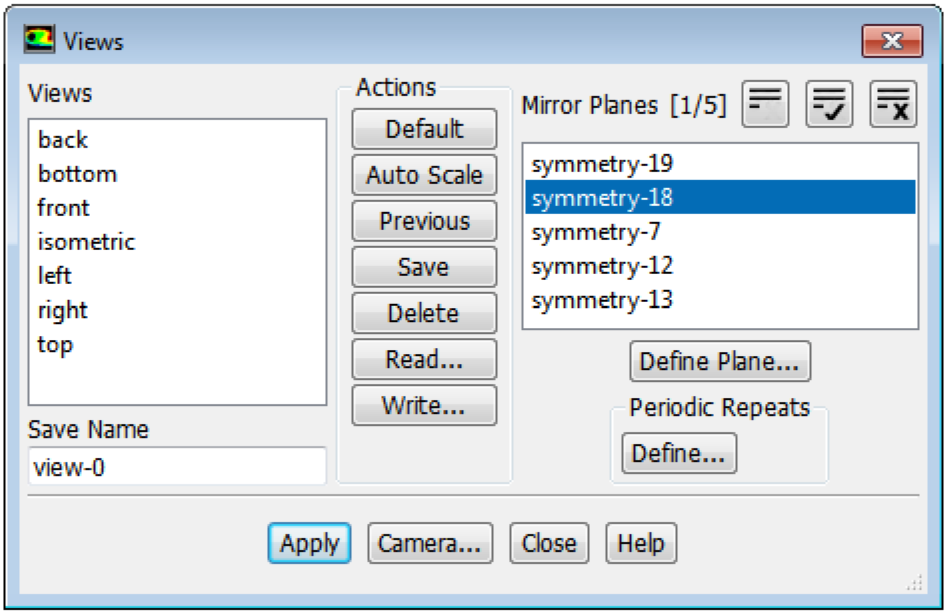
f. Rotate the view and reduce the magnification to obtain the view as shown in Figure 22.10: Velocity Vectors Intersecting the Surface (p. 900).

Figure 22.10: Velocity Vectors Intersecting the Surface



4. Mirror the image about the chip symmetry plane (Figure 22.11: Velocity Vectors After Mirroring (p. 902)).

Viewing → Display → Views...



a. Select **symmetry-18** from the **Mirror Planes** selection list.

Note

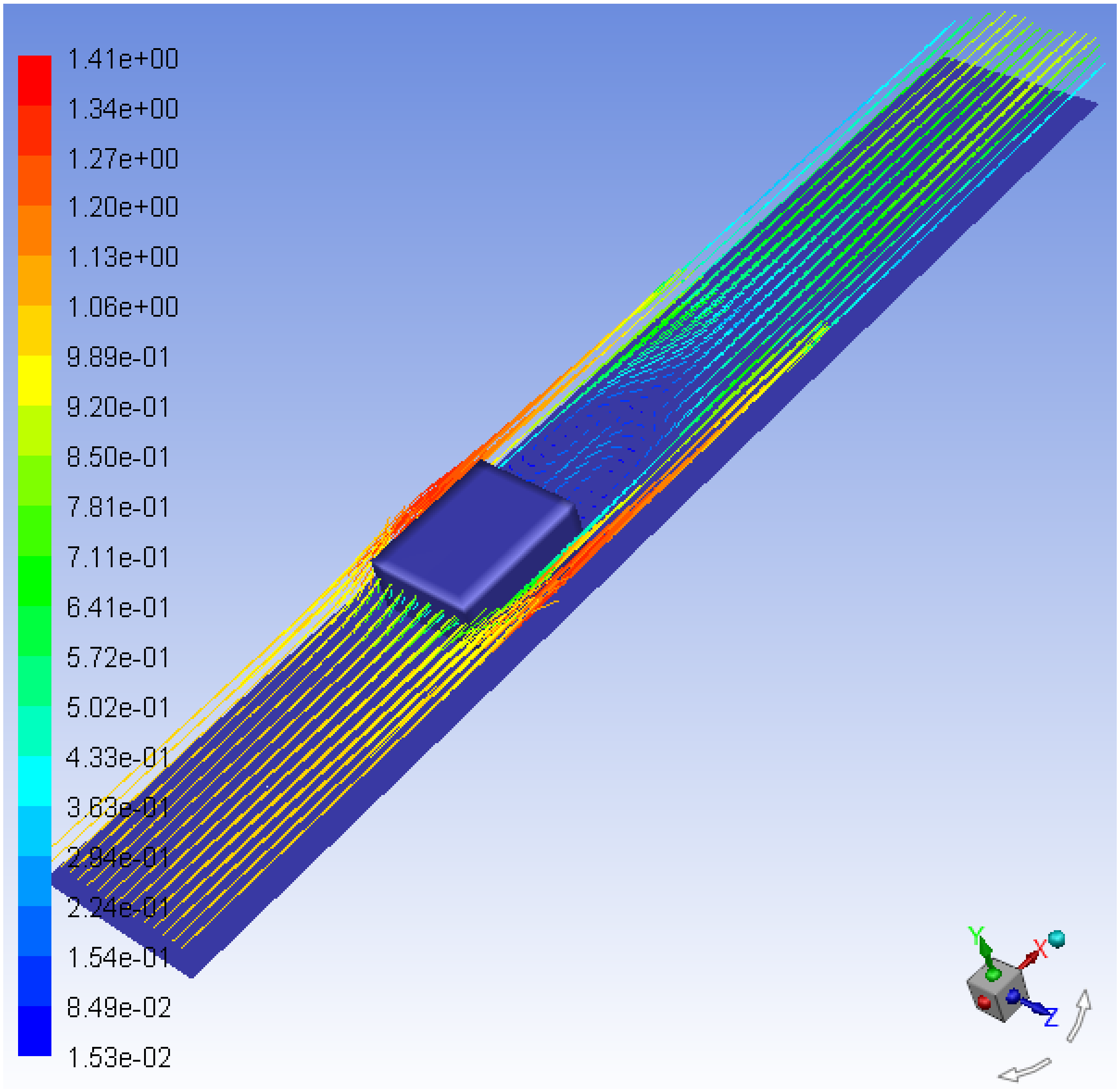
This zone is the centerline plane of the module and its selection will create a mirror of the entire display about the centerline plane.

---

b. Click **Apply** and close the **Views** dialog box.

The display will be updated in the graphics window (Figure 22.11: Velocity Vectors After Mirroring (p. 902)).

Figure 22.11: Velocity Vectors After Mirroring



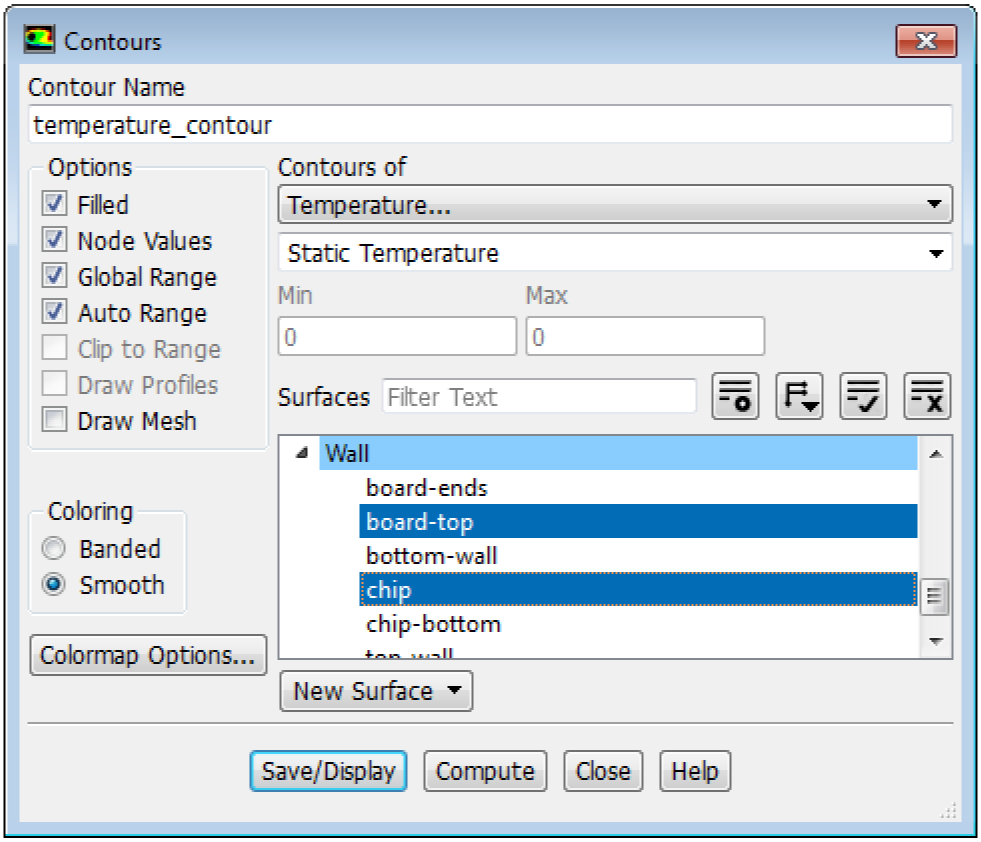
22.4.8. Creating an Animation

Using Fluent, you can animate the solution. For information on animating the solution, see *Using Dynamic Meshes (p. 501)**. In this tutorial, you will animate between two static views of the graphics window.*

You will display the surface temperature distribution on the module and the circuit board by selecting the corresponding boundaries. You will also create the key frames and view the transition between the key frames, dynamically, using the animation feature.

1. Display filled contours of surface temperature on the board-top and chip surfaces. (Figure 22.12: Filled Temperature Contours on the Chip and Board Top Surfaces (p. 904)).

   Results → Graphics → Contours → temperature_contour **Edit...**



a. Ensure that **Filled** is enabled in the **Options** group box.

b. Retain the selection of **Temperature...** and **Static Temperature** from the **Contours of** drop-down lists.

c. Deselect all surfaces.

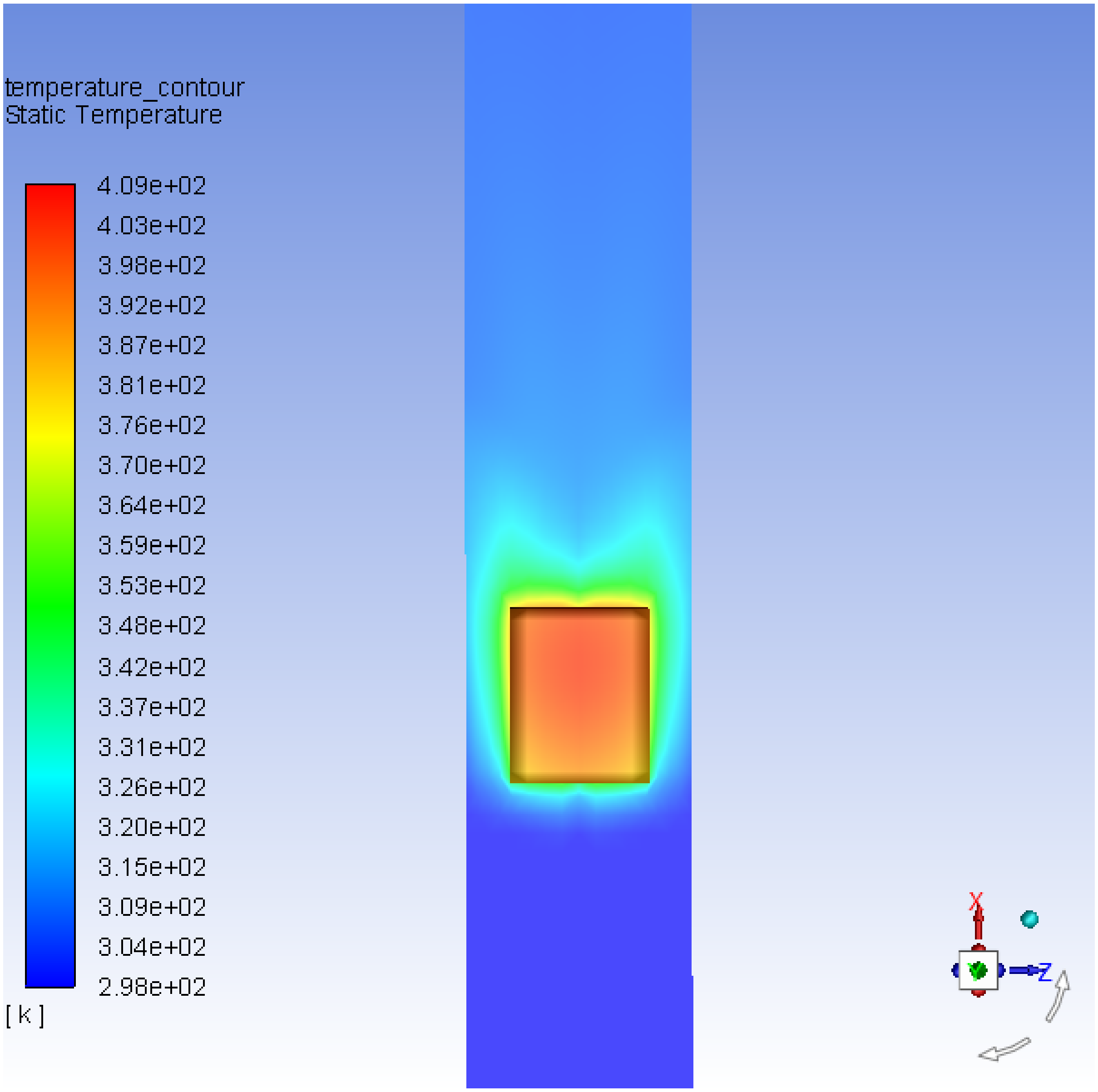
d. Select **board-top** and **chip** from the **Surfaces** selection list.

e. Click **Save/Display** and close the **Contours** dialog box.

f. Reorient the display as needed to obtain the view shown in Figure 22.12: Filled Temperature Contours on the Chip and Board Top Surfaces (p. 904).

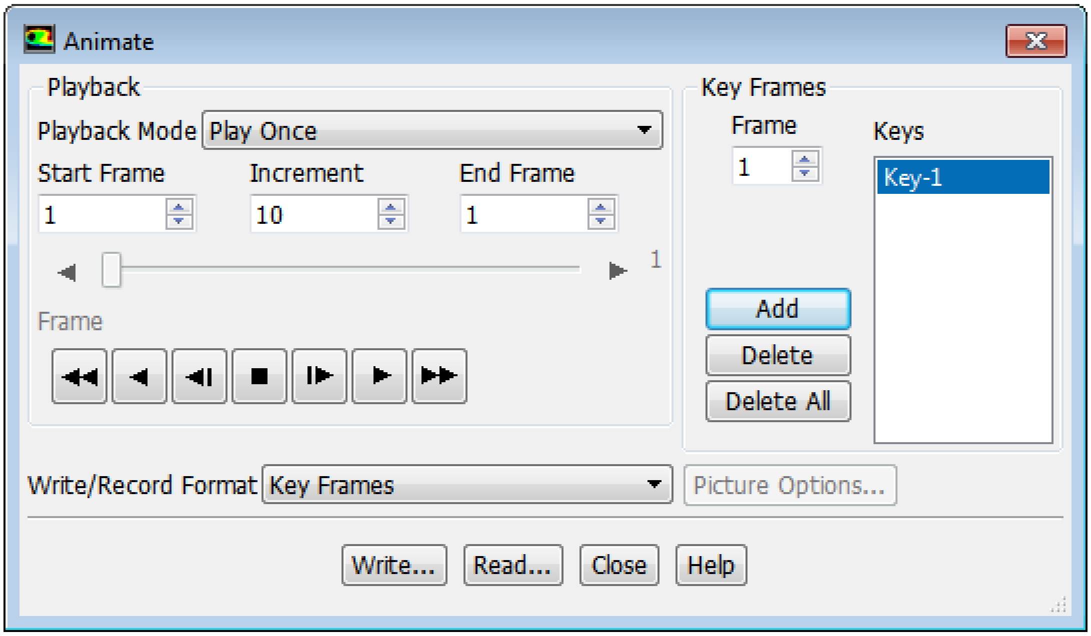
   Figure 22.12: Filled Temperature Contours on the Chip and Board Top Surfaces (p. 904) shows the high temperatures on the downstream portions of the module and relatively localized heating of the circuit board around the module.

Figure 22.12: Filled Temperature Contours on the Chip and Board Top Surfaces



2. Create the key frames by changing the point of view.

Postprocessing → **Animation** → **Scene Animation...**



*You will use the current display (Figure 22.12: Filled Temperature Contours on the Chip and Board Top Surfaces (p. 904)) as the starting view for the animation (**Frame** = 1).*

a. Click **Add** in the **Key Frames** group box to create the first frame for your animation.

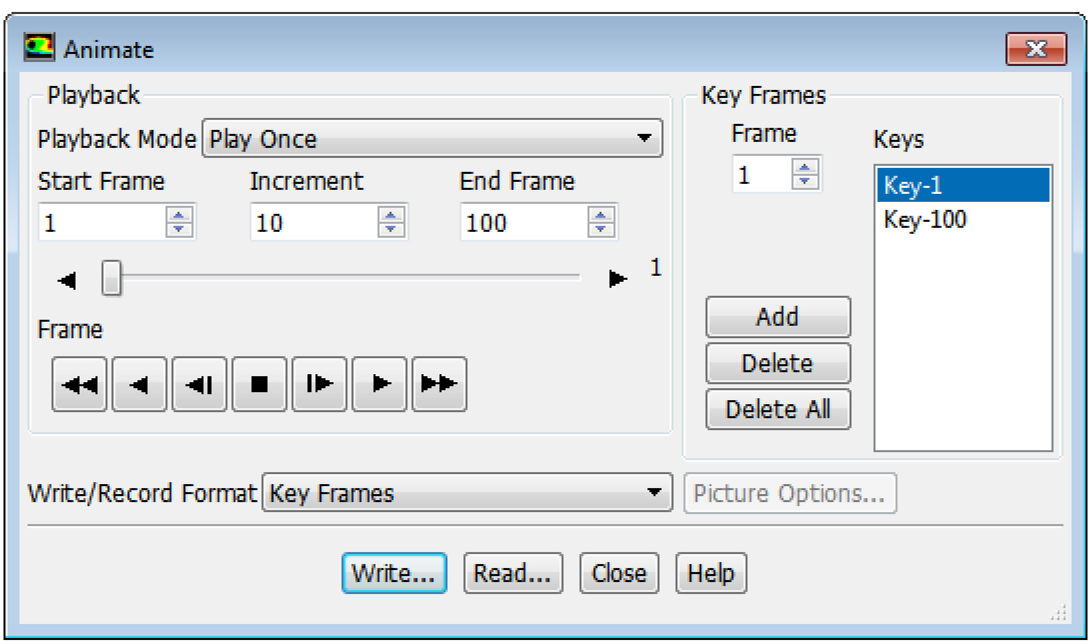
   *This will store the current display as **Key-1**.*

b. Magnify the view to focus on the module region.

c. Enter `100` for **Frame** in the **Key Frames** group box.

d. Click **Add** to create the tenth frame for your animation.

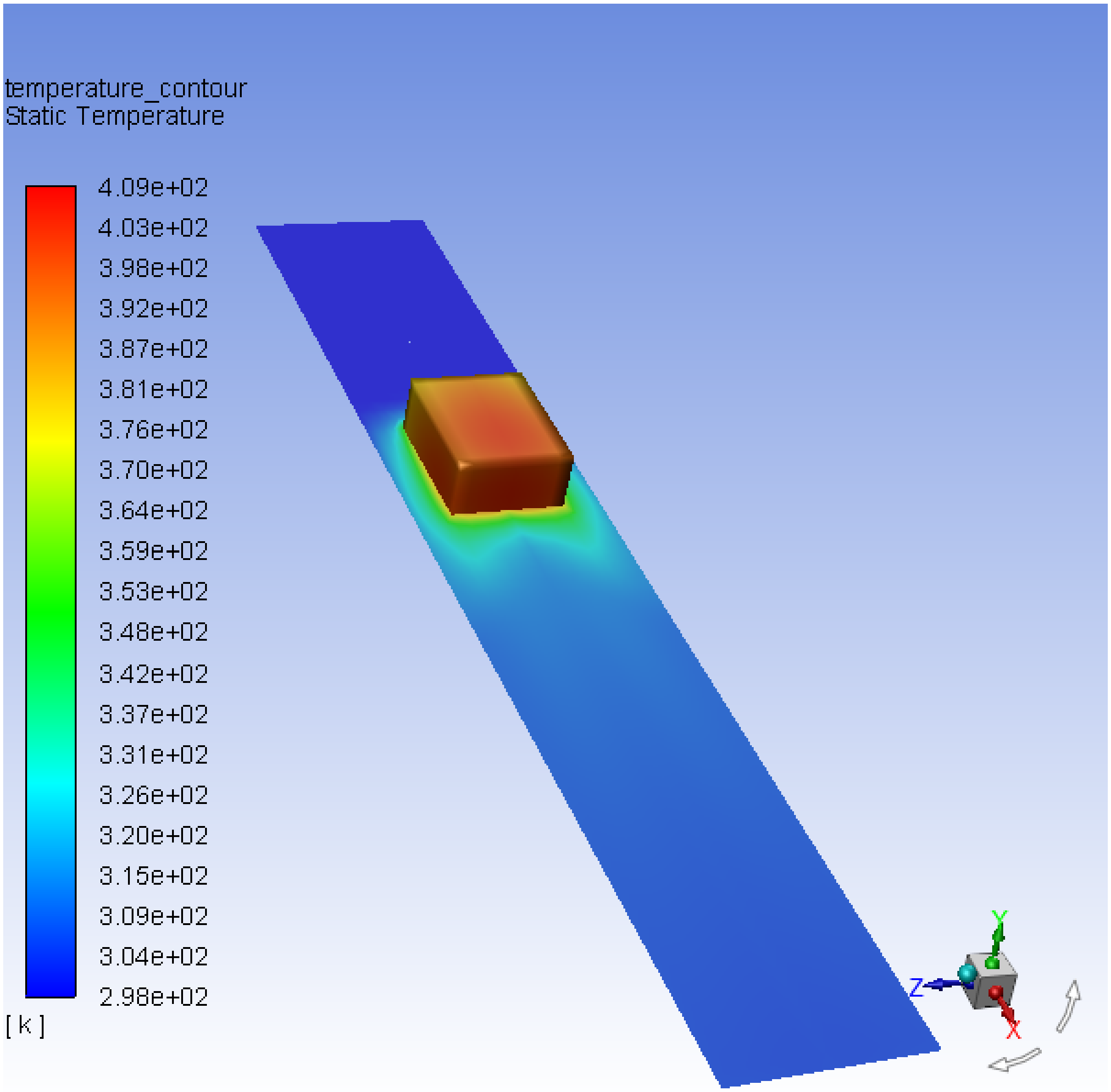
   *This will store the new display as **Key-100**.*



The magnified view will be the one-hundredth key frame of the animation, with intermediate displays (2 through 99) to be filled in during the animation.

e. Rotate the view and adjust the magnification so that the downstream side of the module is in the foreground (Figure 22.13: Filled Temperature Contours on the Chip and Board Top Surfaces (p. 906)).

Figure 22.13: Filled Temperature Contours on the Chip and Board Top Surfaces



f. Enter `200` for **Frame**.

g. Click **Add** to create the two-hundredth frame for your animation.

*This will store the new display as **Key-200**.*

Note

You can check the display view of any of your saved key frames by selecting it in the **Keys** list.

---

3. View the scene animation by clicking the "play" button ( ) in the **Playback** group box.

   Note

   If your computer is having difficulty displaying the animation with 200 frames, you can increase the increment to 50.

   ---

   *While effective animation is best conducted on "high-end" graphics workstations, you can view scene animations on any workstation. If the graphics display speed is slow, the animation playback will take some time and will appear choppy, with the redrawing very obvious. On fast graphics workstations, the animation will appear smooth and continuous and will provide an excellent visualization of the display from a variety of spatial orientations. On many machines, you can improve the smoothness of the animation by enabling the **Double Buffering** option in the **Display Options** dialog box.*

   To produce a slower animation, increase the number of frames between the key frames. The more sparsely you place your key frames, the more transition frames Fluent creates between the key frames and therefore stretching out your animation.

   Note

   You can also make use of animation tools of Fluent for transient cases as demonstrated in Modeling Transient Compressible Flow (p. 267).

   Extra

   You can change the **Playback Mode** if you want to "auto repeat" or "auto reverse" the animation. When you are in either of these modes, you can click the "stop" button ( ) to stop the continuous animation.

   ---

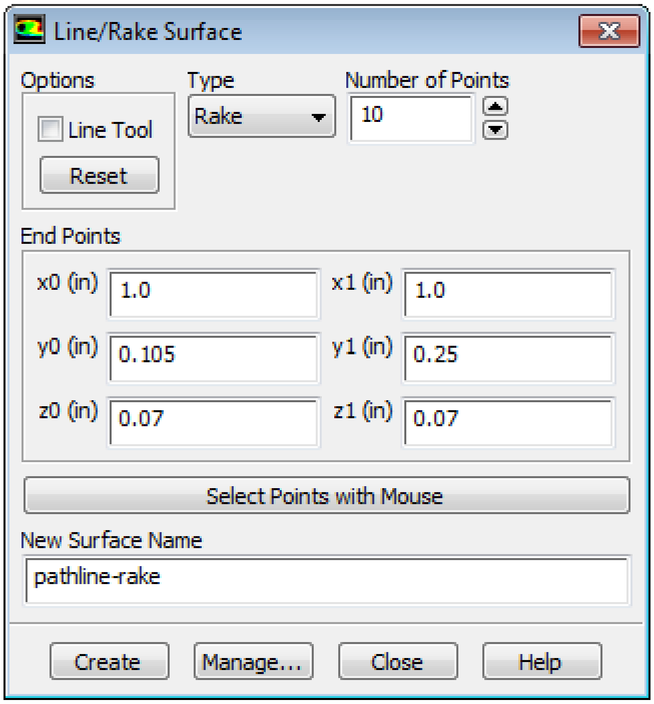
4. Close the **Animate** dialog box.

22.4.9. Displaying Pathlines

Pathlines are the lines traveled by neutrally buoyant particles in equilibrium with the fluid motion. Pathlines are an excellent tool for visualization of complex three-dimensional flows. In this example, you will use pathlines to examine the flow around and in the wake of the module.

1. Create a rake from which the pathlines will emanate.

   Postprocessing → **Surface** → **Create** → **Line/Rake...**



a. Select **Rake** from the **Type** drop-down list.

   *A rake surface consists of a specified number of points equally spaced between two specified endpoints. A line surface (the other option in the **Type** drop-down list) is a line that includes the specified endpoints and extends through the domain; data points on a line surface will not be equally spaced.*

b. Retain the default value of `10` for **Number of Points**.

   This will generate 10 pathlines.

c. Enter a starting coordinate of (`1.0, 0.105, 0.07`) and an ending coordinate of (`1.0, 0.25, 0.07`) in the **End Points** group box.

   This will define a vertical line in front of the module, about halfway between the centerline and edge.

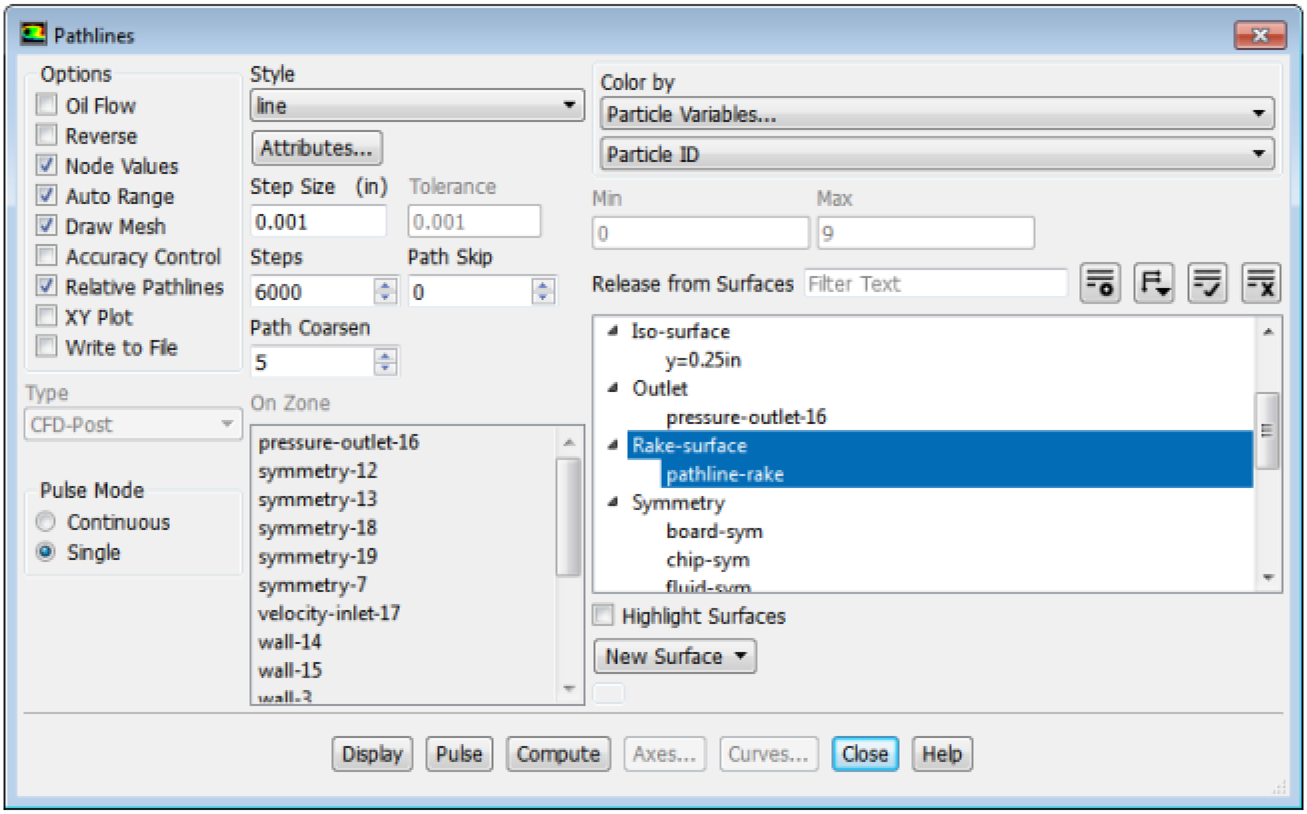
d. Enter `pathline-rake` for **New Surface Name**.

   You will refer to the rake by this name when you plot the pathlines.

e. Click **Create** and close the **Line/Rake Surface** dialog box.

2. Draw the pathlines (Figure 22.14: Pathlines Display (p. 910)).

   Postprocessing → **Graphics** → **Pathlines...** → **Edit...**



a. Enable **Draw Mesh** in the **Options** group box to open the **Mesh Display** dialog box.

   i. Ensure that **Faces** is enabled in the **Options** group box.

   ii. Retain the selection of **board-top** and **chip** from the **Surfaces** selection list.

   These surfaces should already be selected from the earlier exercise where the mesh was displayed with velocity vectors.

   iii. Close the **Mesh Display** dialog box.

b. Enter `0.001` inch for **Step Size**.

c. Enter `6000` for **Steps**.

Note

As a general guideline to follow when you are setting these two parameters is that if you want the particles to advance through a domain of length L, the **Step Size** times the number of **Steps** should be approximately equal to L.

---

d. Set **Path Coarsen** to 5.

*Coarsening the pathline simplifies the plot and reduces the plotting time. The coarsening factor specified for **Path Coarsen** indicates the interval at which the points are plotted for a given pathline in any cell.*

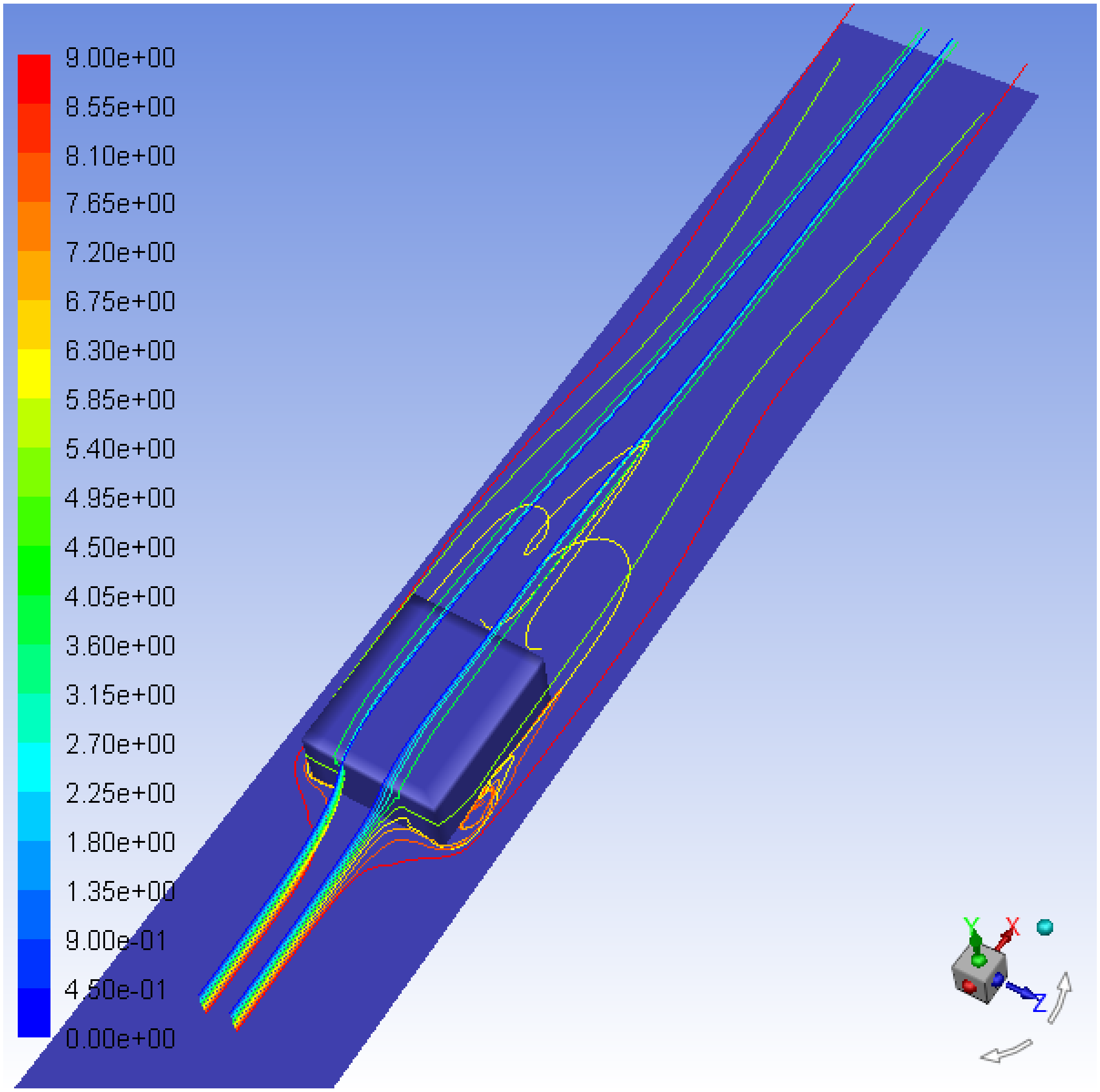
e. Select **pathline-rake** from the **Release from Surfaces** selection list.

f. Click **Display**.

The pathlines will be drawn on the surface.

g. Rotate and magnify the view so that the flow field is in front and the wake of the chip is visible as shown in Figure 22.14: Pathlines Display (p. 910).

Figure 22.14: Pathlines Display



3. Write the pathlines to a file.

a. Enable **Write to File** in the **Options** group box.

*The **Display** button changes to a **Write...** button.*

b. Select **Fieldview** from the **Type** drop-down list.

c. Click the **Write...** button to open the **Select File** dialog box.

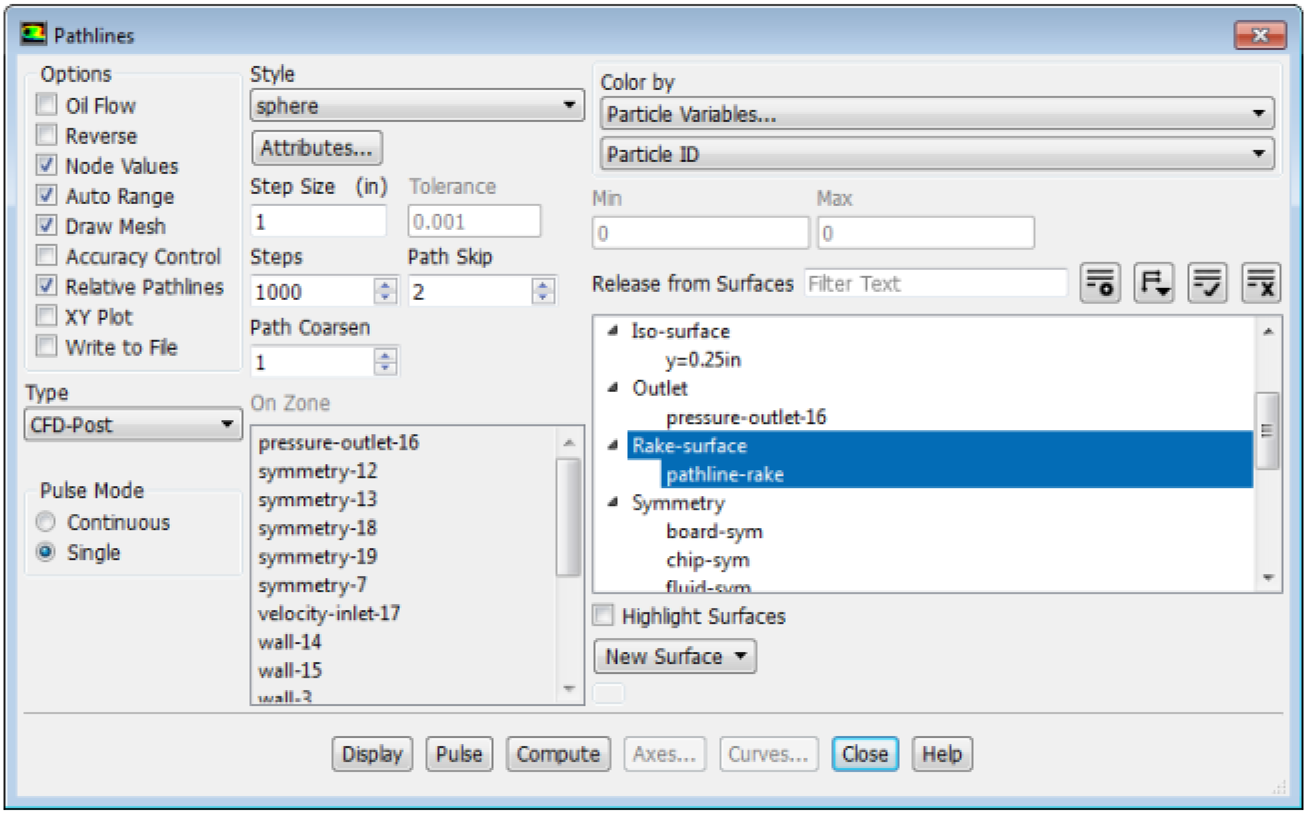
i. Enter `chip-pathline` for **Fieldview File**.

ii. Click **OK** to close the **Select File** dialog box.

Fluent will save the file in the Fieldview format with a .fvp extension.

4. Display pathlines as spheres.

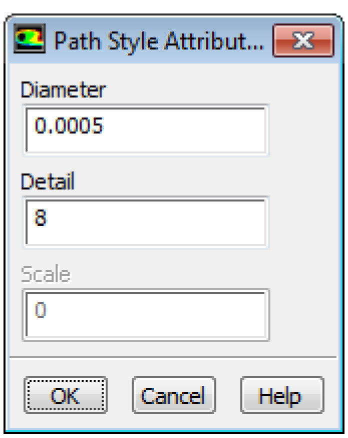
Postprocessing → **Graphics** → **Pathlines...** → **Edit...**



a. Disable **Write to File** in the **Options** group box.

b. Select **sphere** from the **Style** drop-down list.

c. Click the **Attributes...** button to open the **Path Style Attributes** dialog box.



i. Enter `0.0005` for **Diameter**.

ii. Click **OK** to close the **Path Style Attributes** dialog box.

d. Enter `1` inch for **Step Size** and `1000` for **Steps** respectively.

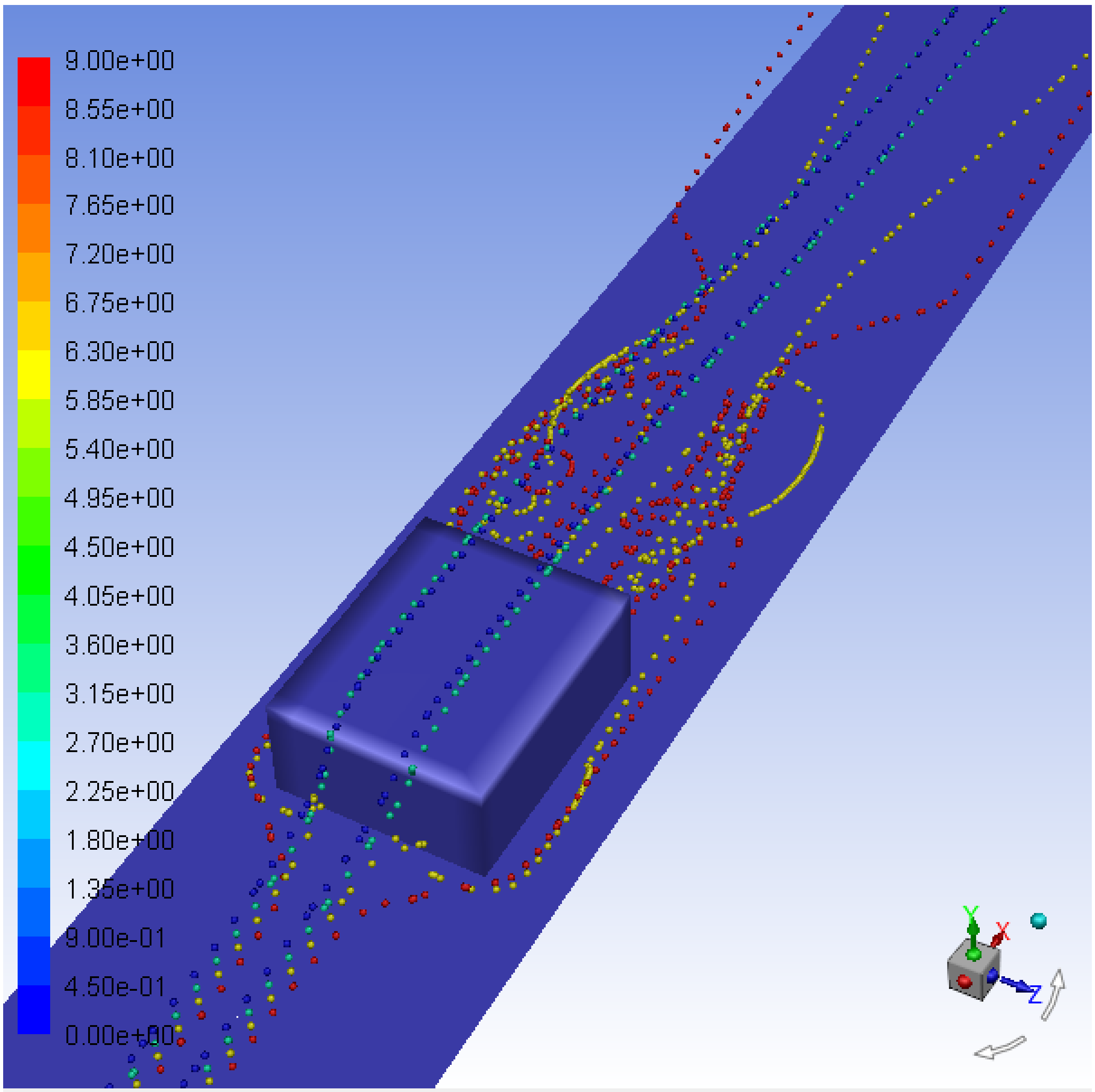
e. Set **Path Skip** to `2` and **Path Coarsen** to `1`.

f. Retain the selection of **pathline-rake** in the **Release from Surfaces** selection list.

g. Click **Display**.

The spherical pathlines will be drawn along the surface.

Figure 22.15: Sphere Pathlines Display

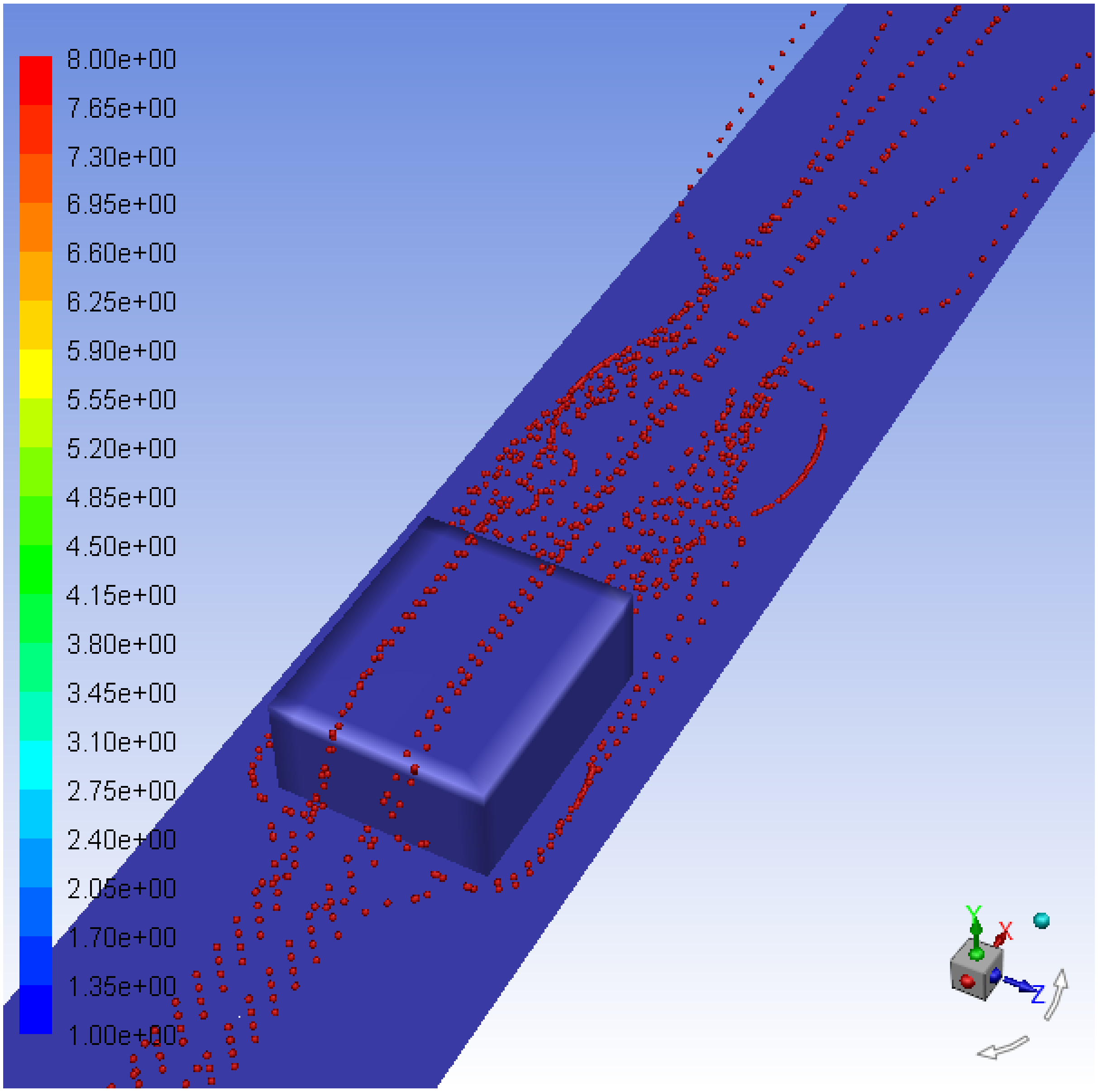


h. Select **Surface ID** from the lower **Color by** drop-down list.

i. Click **Display** and close the **Pathlines** dialog box.

This will color the pathlines by the surface they are released from (Figure 22.16: Sphere Pathlines Colored by Surface ID (p. 913)).

Figure 22.16: Sphere Pathlines Colored by Surface ID



Note

As an optional exercise, you can create solution animations for pathlines using the **Animation Sequence** dialog box.

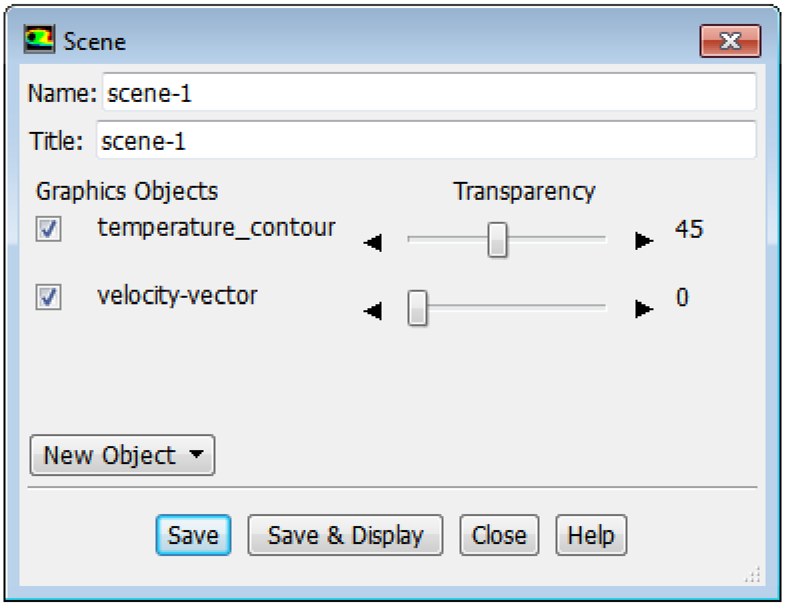
Solution → **Calculation Activities** → **Solution Animations** → **Edit...**

22.4.10. Creating a Scene With Vectors and Contours

Scenes allow you to display multiple graphics plots in single window.

1. Open the **Scene** dialog box.

   Results → **Scene** **New...**



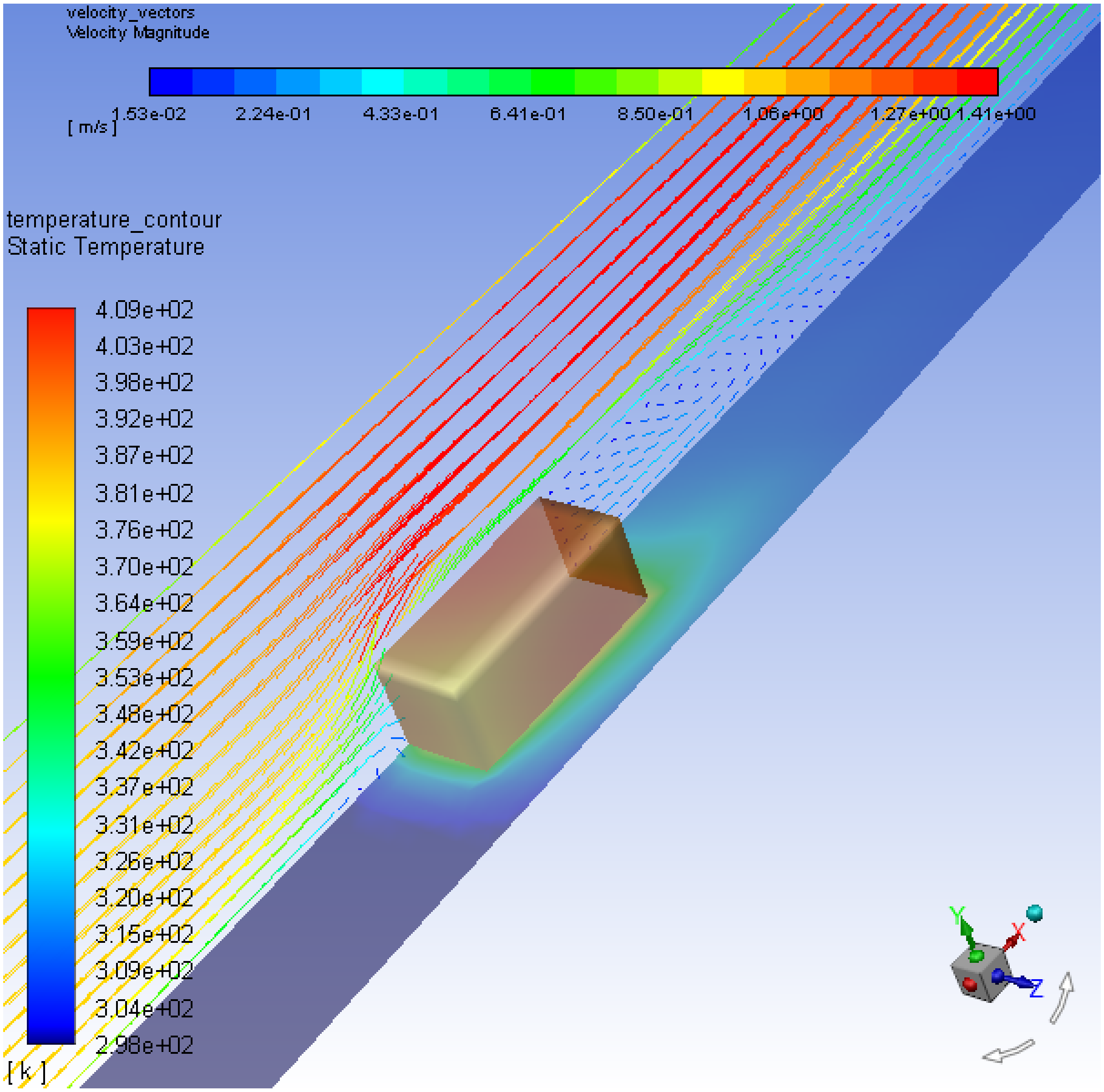
2. Enable **temperature_contour** and **velocity_vector**.

3. Increase the transparency of **temperature_contour** using the **Transparency** slider.

4. Click **Save & Display** to create the scene and display it in the graphics window.

5. Drag the velocity vector colormap to the top of the graphics window and modify the orientation and zoom of the scene to match Figure 22.17: Temperature Contours and Velocity Vectors Scene (p. 915).

6. **Figure 22.17: Temperature Contours and Velocity Vectors Scene**



Note

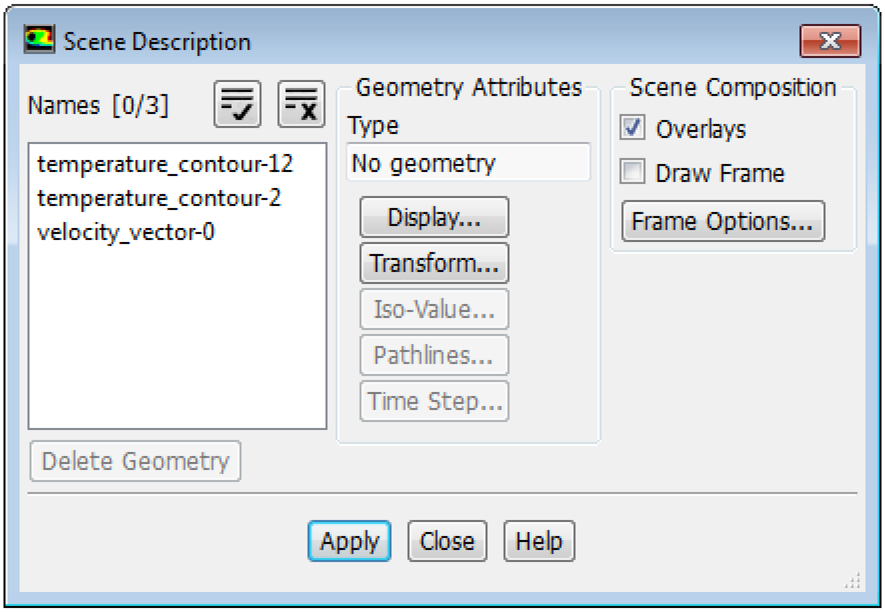
As an optional exercise, you can add a mesh graphics object to the scene as well, by selecting **Mesh...** from the **New Object** drop-down in the **Scene** dialog box. Once the mesh graphics object is saved/created, select it from the **Graphics Objects** list and click **Save & Display** in the **Scene** dialog box.

22.4.11. Advanced Overlay of Pathlines on a Scene

*The advanced overlay capability, provided in the **Scene Description** dialog box, allows you to display more results on top of an existing display, including those not supported by the scene created in the previous section. You can demonstrate this capability by adding pathlines to the scene that you just plotted.*

1. Enable the overlays feature.

Viewing → Graphics → Compose...

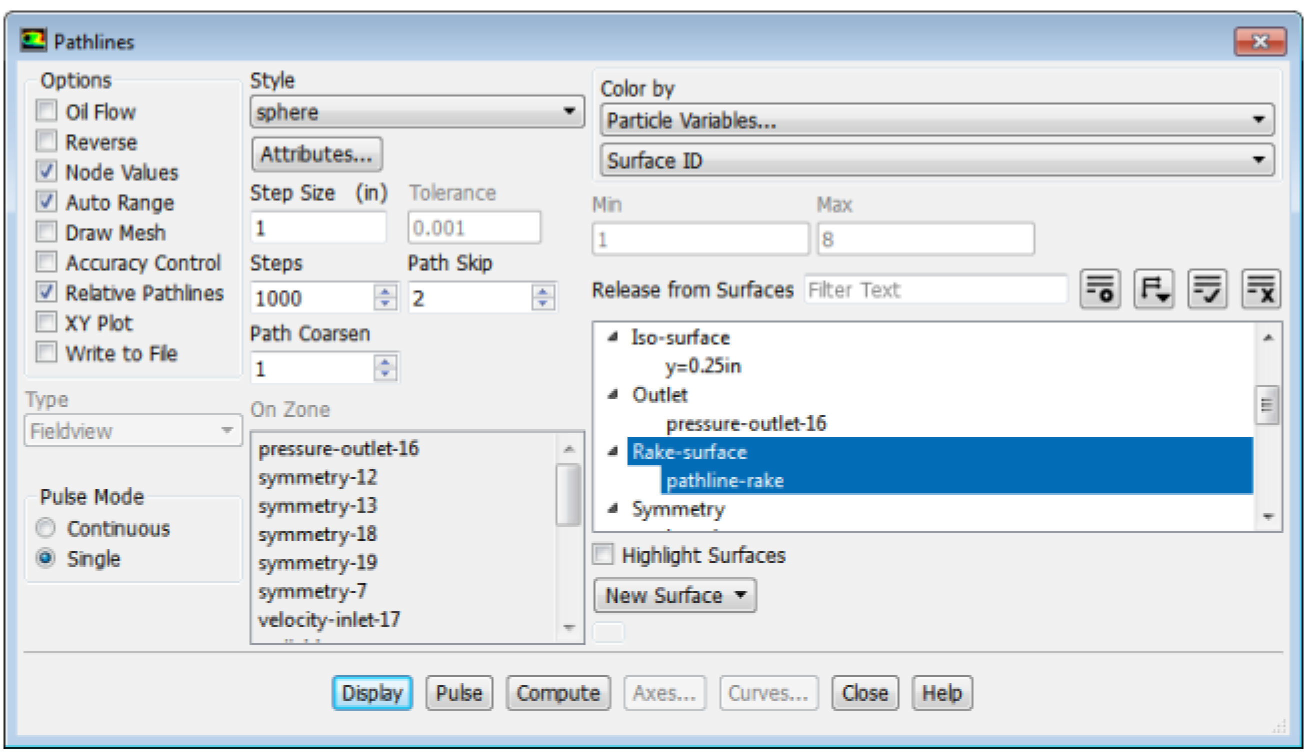


a. Enable **Overlays** in the **Scene Composition** group box.

b. Click **Apply** and close the **Scene Description** dialog box.

2. Add a plot of particle surface ID pathlines to the velocity vector and temperature scene.

Postprocessing → Graphics → Pathlines → Edit...

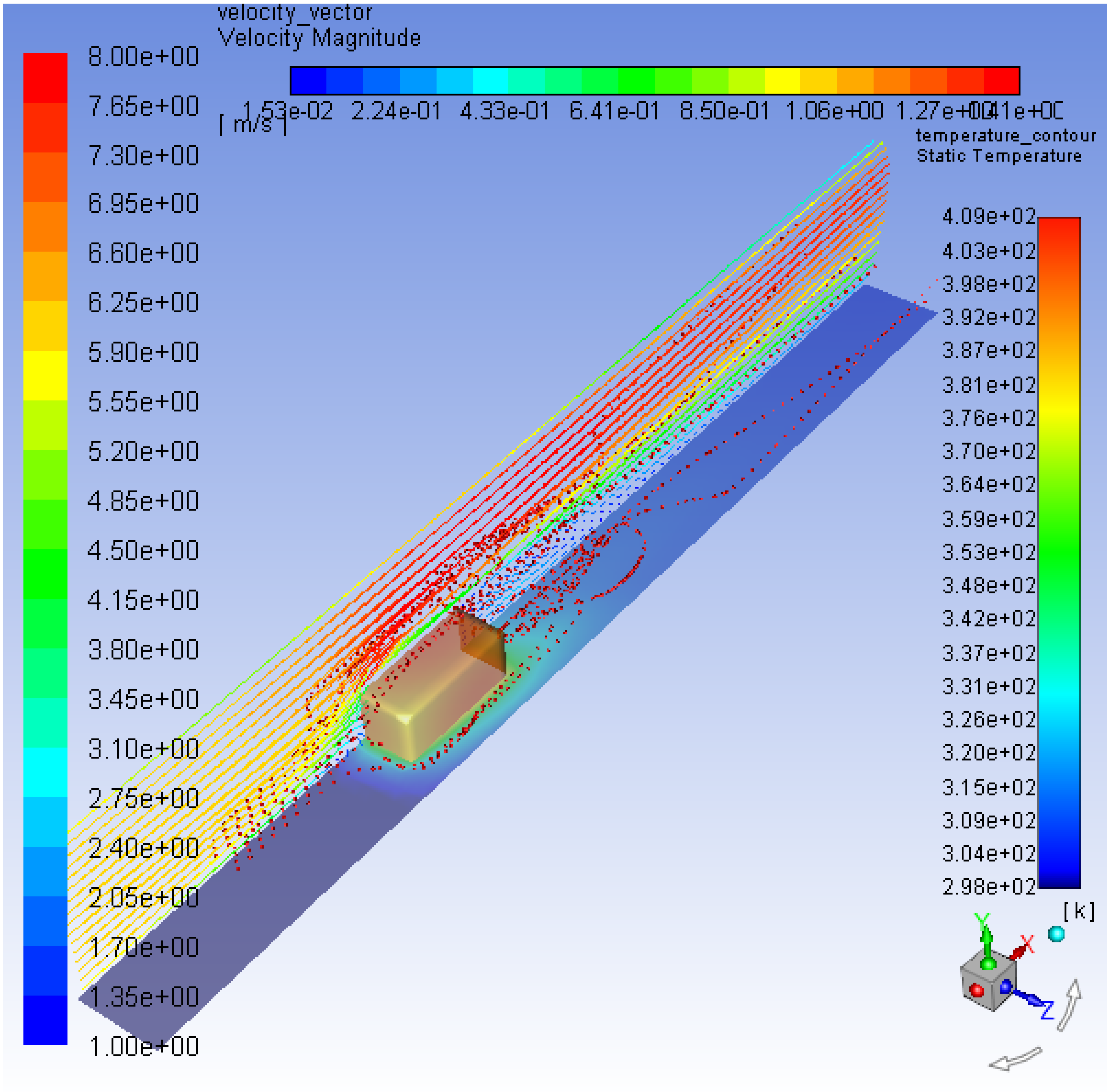


a. Disable **Draw Mesh** under **Options**.

b. Click **Display** and close the **Pathlines** dialog box.

c. Use the mouse to obtain the view that is shown in Figure 22.18: Overlay of Pathlines on Velocity Vectors and Temperature Contours Scene (p. 917).

Figure 22.18: Overlay of Pathlines on Velocity Vectors and Temperature Contours Scene



22.4.12. Creating Exploded Views

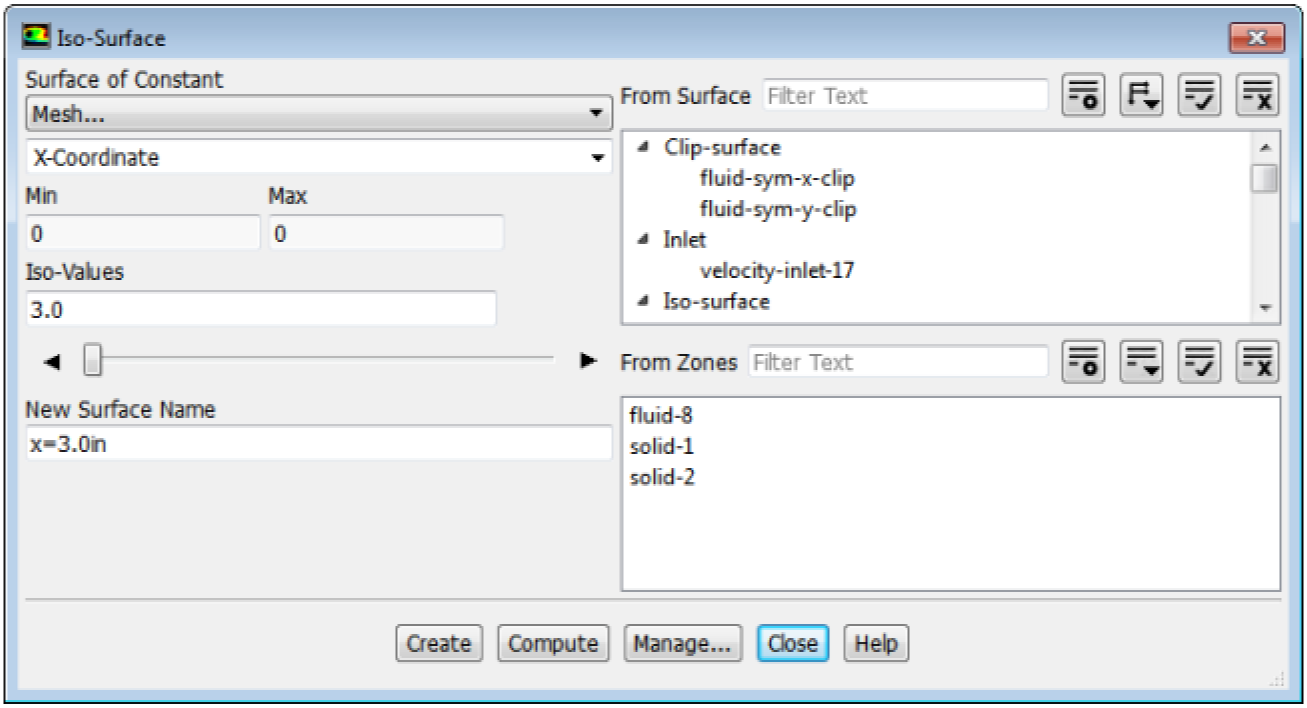
*The **Scene Description** dialog box stores each display that you request and allows you to manipulate the displayed items individually. This capability can be used to generate "exploded" views, in which results are translated or rotated out of the physical domain for enhanced display. As shown in the **Scene Description** dialog box, you can experiment with this capability by displaying "side-by-side" velocity vectors and temperature contours on a streamwise plane in the module wake.*

1. Clear the temperature contours, velocity vectors, and pathlines from the current display.

Click the **Close Tab** button () to clear the graphics window (located at on the right-hand side of the graphics window tab).

2. Create a plotting surface at X = 3 inches (named **x=3.0in**), just downstream of the trailing edge of the module.

Postprocessing → **Surface** → **Create** → **Iso-Surface...**

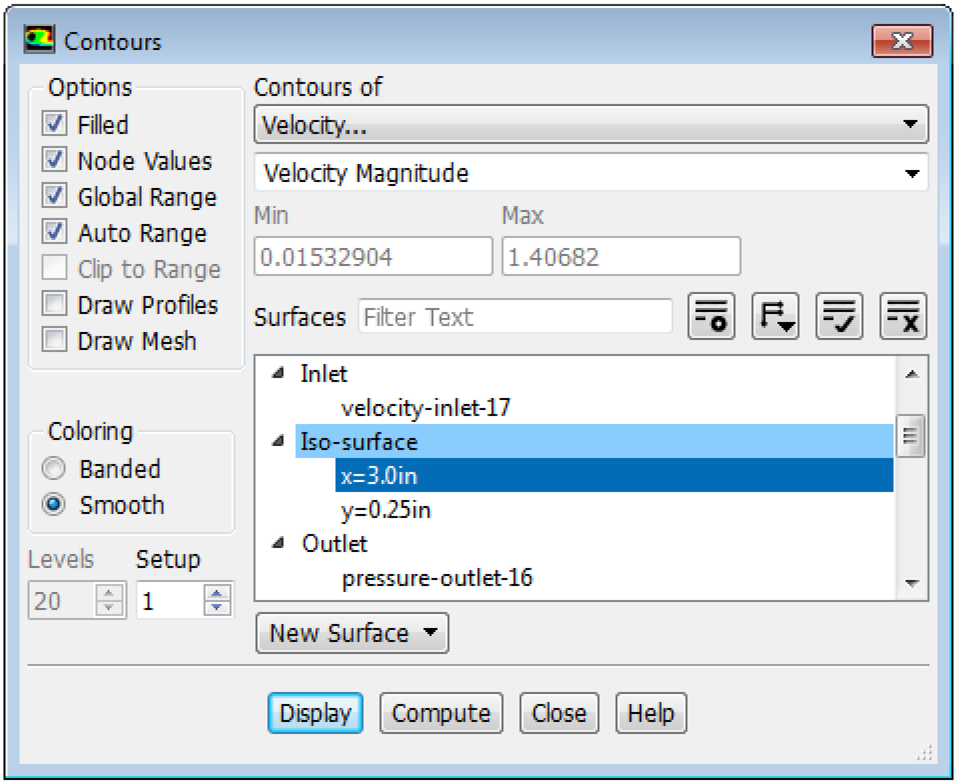


Tip

For details on creating an isosurface, see Creating Isosurfaces (p. 889).

3. Add the display of filled temperature contours on the **x=3.0in** surface.

Postprocessing → **Graphics** → **Contours** → **Edit...**



a. Disable **Draw Mesh** under **Options**.

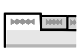
b. Deselect all surfaces in the **Surfaces** list.

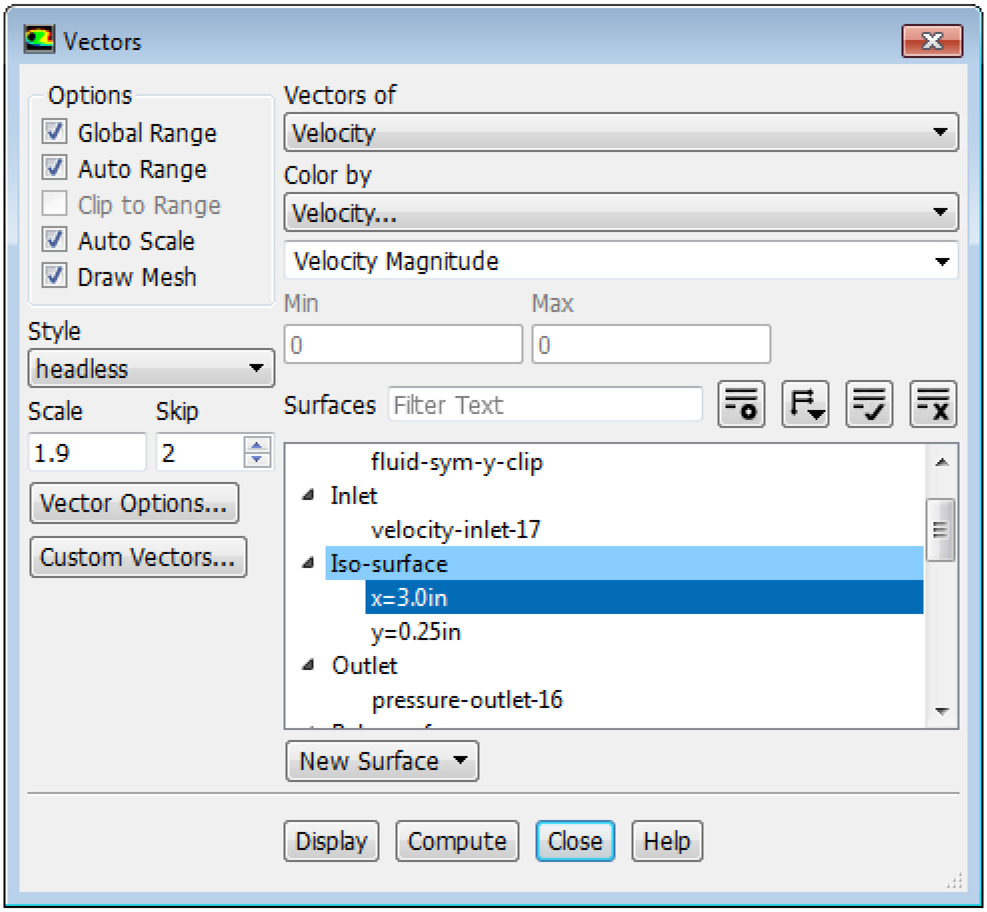
c. Select **x=3.0in** from the **Surfaces** selection list.

d. Click **Display** and close the **Contours** dialog box.

The filled temperature contours will be displayed on the ***x=3.0in*** *surface.*

4. Add the velocity vectors on the **x=3.0in** plotting surface.

 **Postprocessing** → **Graphics** → **Vectors** → **Edit...**



a. Enable the **Draw Mesh** option in the **Options** group box to open the **Mesh Display** dialog box.

   i. Retain the default settings.

   ii. Close the **Mesh Display** dialog box.

b. Enter `1.9` for **Scale**.

c. Set **Skip** to 2.

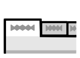
d. Deselect all surfaces.

e. Select **x=3.0in** from the **Surfaces** selection list.

f. Click **Display** and close the **Vectors** dialog box.

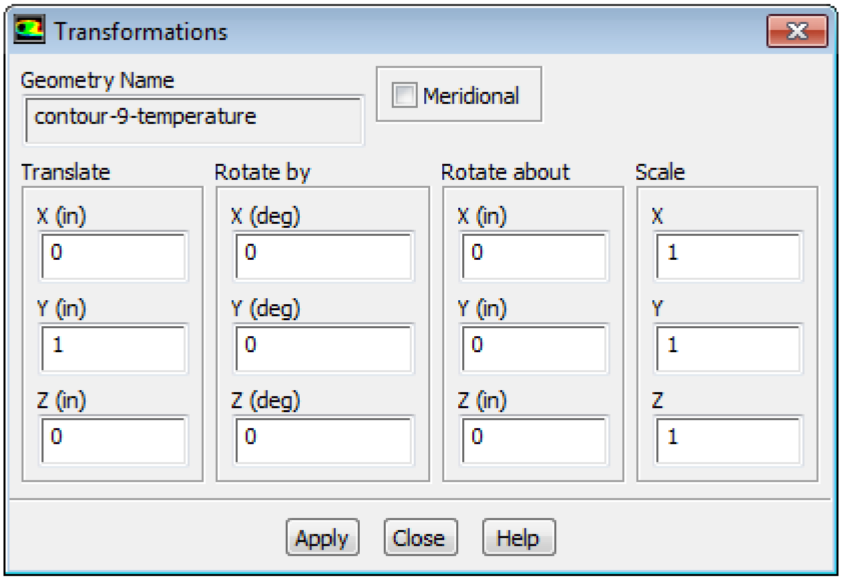
   The display will show the vectors superimposed on the contours of temperature at x=3.0 in.

5. Create the exploded view by translating the contour display, placing it above the vectors (Figure 22.19: Exploded Scene Display of Temperature and Velocity (p. 922)).

    **Viewing → Graphics → Compose...**

   a. Select **temperature_contour-9** from the **Names** selection list.

   b. Click the **Transform...** button to open the **Transformations** dialog box.



i. Enter 1 inch for **Y** in the **Translate** group box.

ii. Click **Apply** and close the **Transformations** dialog box.

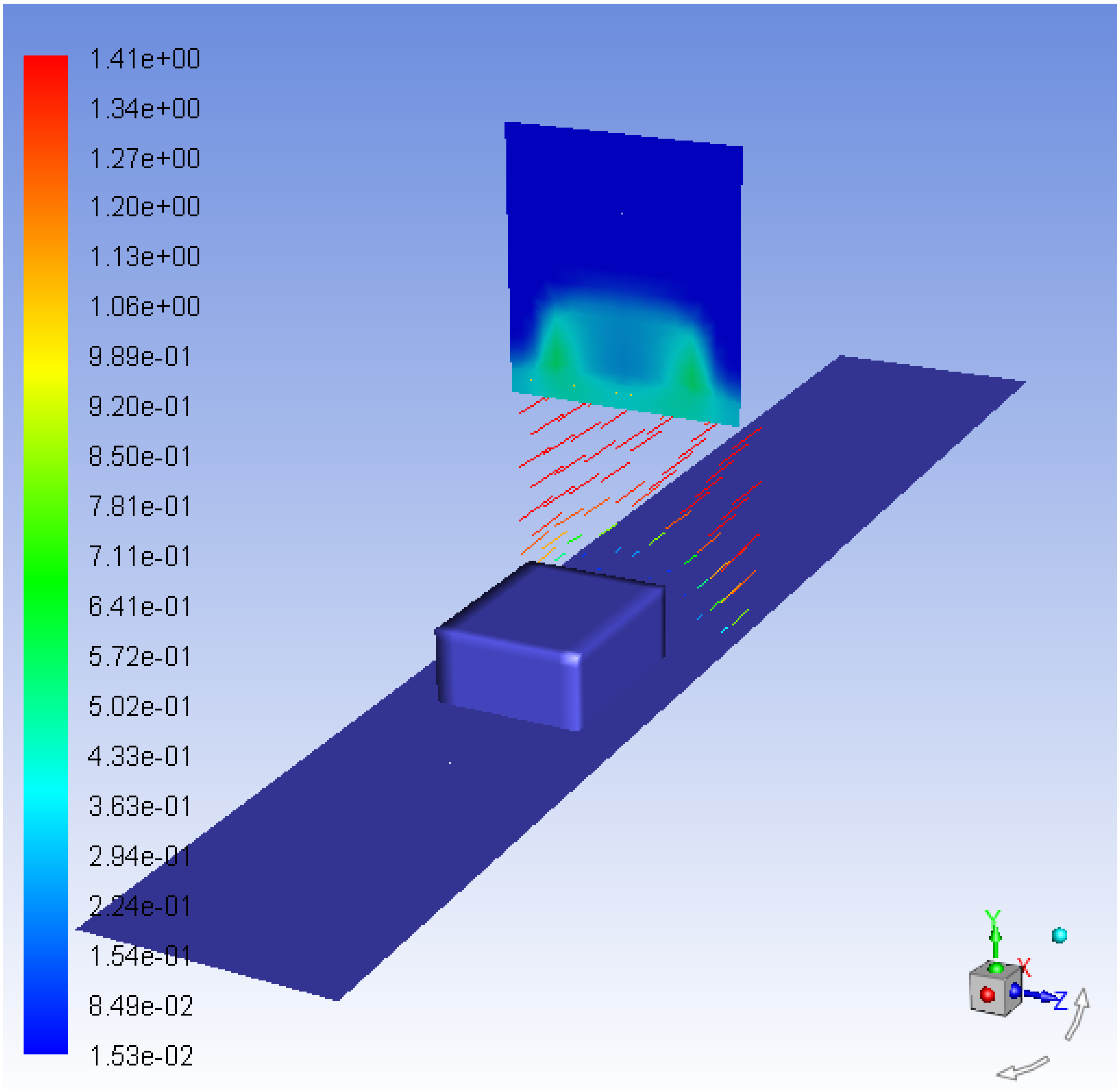
The exploded view allows you to see the contours and vectors as distinct displays in the final scene (Figure 22.19: Exploded Scene Display of Temperature and Velocity (p. 922)).

c. Deselect **Overlays**.

d. Click **Apply** and close the **Scene Description** dialog box.

e. Magnify the view, as shown in Figure 22.19: Exploded Scene Display of Temperature and Velocity (p. 922).

Figure 22.19: Exploded Scene Display of Temperature and Velocity



22.4.13. Animating the Display of Results in Successive Streamwise Planes

You may want to march through the flow domain, displaying a particular variable on successive slices of the domain. While this task could be accomplished manually, plotting each plane in turn, or using the ***Scene Description*** *and* ***Animate*** *dialog boxes, here you will use the* ***Sweep Surface*** *dialog box to facilitate the process. To illustrate the display of results on successive slices of the domain, you will plot contours of velocity magnitude on planes along the X axis.*

1. Delete the vectors and temperature contours from the display.

   Click the **Close Tab** button ( ) to clear the graphics window (located at on the right-hand side of the graphics window tab).

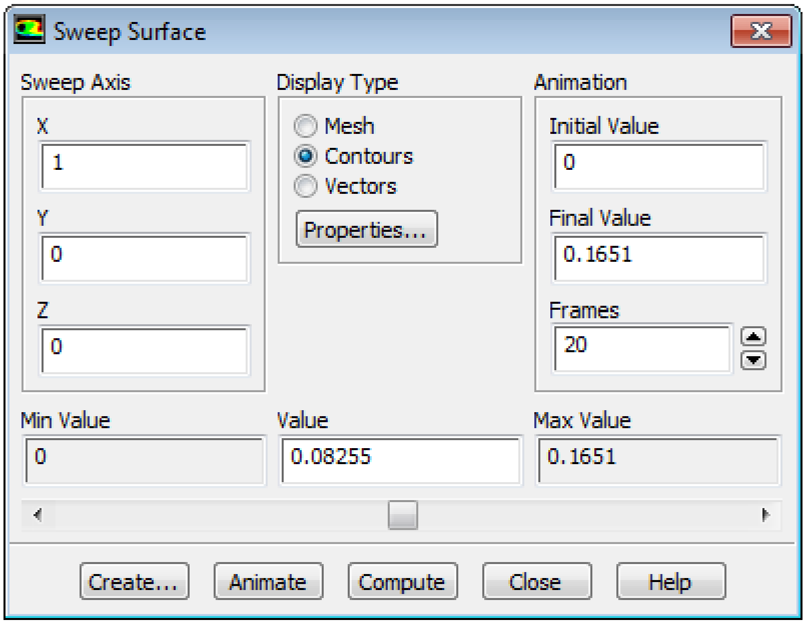
2. Display the mesh on surfaces **board-top** and **chip**.

   Postprocessing → **Graphics** → **Mesh** → **Edit...**

3. Use the mouse to reduce the magnification of the view in the graphics window so that the entire board surface is visible.

4. Generate contours of velocity magnitude and sweep them through the domain along the X axis.

   Postprocessing → **Animation** → **Sweep Surface...**



   a. Retain the default settings in the **Sweep Axis** group box.

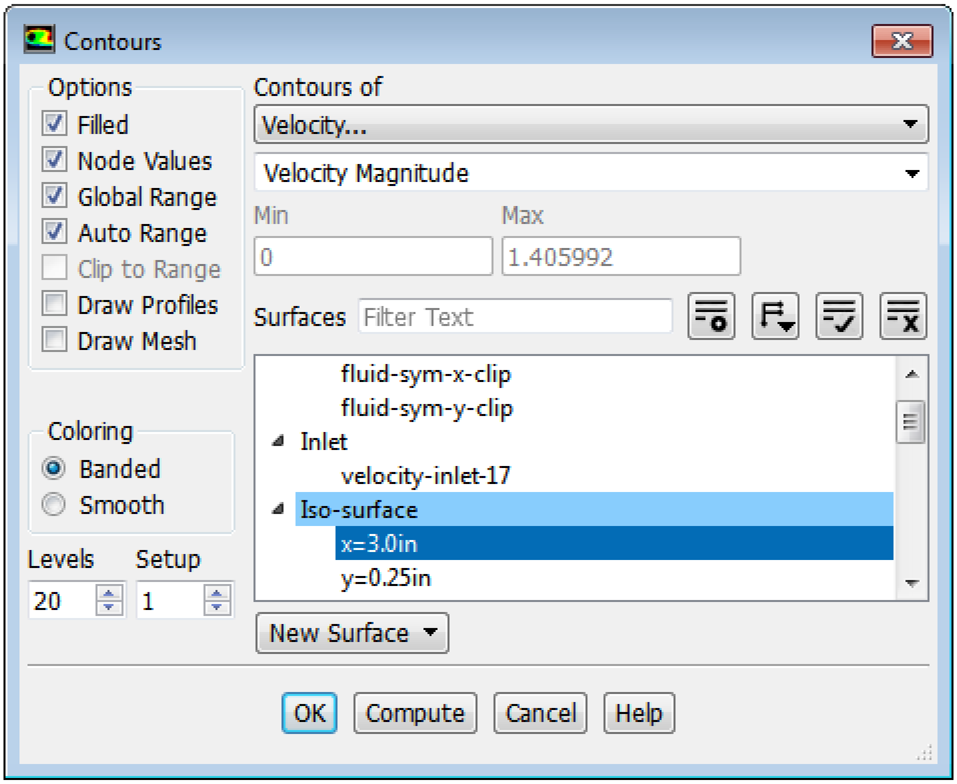
   b. Retain the default value of `0` m for **Initial Value** in the **Animation** group box.

   c. Retain `0.1651` m for **Final Value**.

   > **Warning**
   >
   > The units for the initial and final values are in meters, regardless of the length units being used in the model. Here, the initial and final values are set to the **Min Value** and **Max Value**, to generate an animation through the entire domain.

   d. Enter `20` for **Frames**.

   e. Select **Contours** from the **Display Type** list to open the **Contours** dialog box.



i. Disable **Draw Mesh** under **Options**.

ii. Select **Velocity...** and **Velocity Magnitude** from the **Contours of** drop-down lists.

iii. Click **OK** to close the **Contours** dialog box.

f. Click **Animate** and close the **Sweep Surface** dialog box.

You will see the velocity contour plot displayed at 20 successive streamwise planes. Fluent will automatically interpolate the contoured data on the streamwise planes between the specified end points. Especially on high-end graphics workstations, this can be an effective way to study how a flow variable changes throughout the domain.

Note

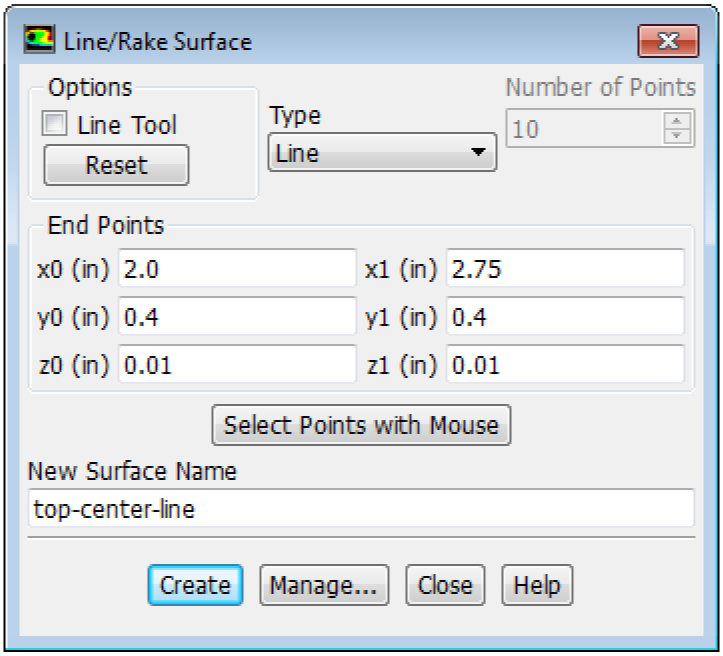
You can also make use of animation tools of Fluent for transient cases as demonstrated in Modeling Transient Compressible Flow (p. 267).

22.4.14. Generating XY Plots

XY plotting can be used to display quantitative results of your CFD simulations. Here, you will complete the review of the module cooling simulation by plotting the temperature distribution along the top centerline of the module.

1. Define the line along which to plot results.

Postprocessing → **Surface** → **Create** → **Line/Rake...**



a. Select **Line** from the **Type** drop-down list.

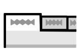
b. Enter the coordinates of the line using a starting coordinate of (`2.0`,`0.4`,`0.01`) and an ending coordinate of (`2.75`,`0.4`,`0.01`) in the **End Points** group box.

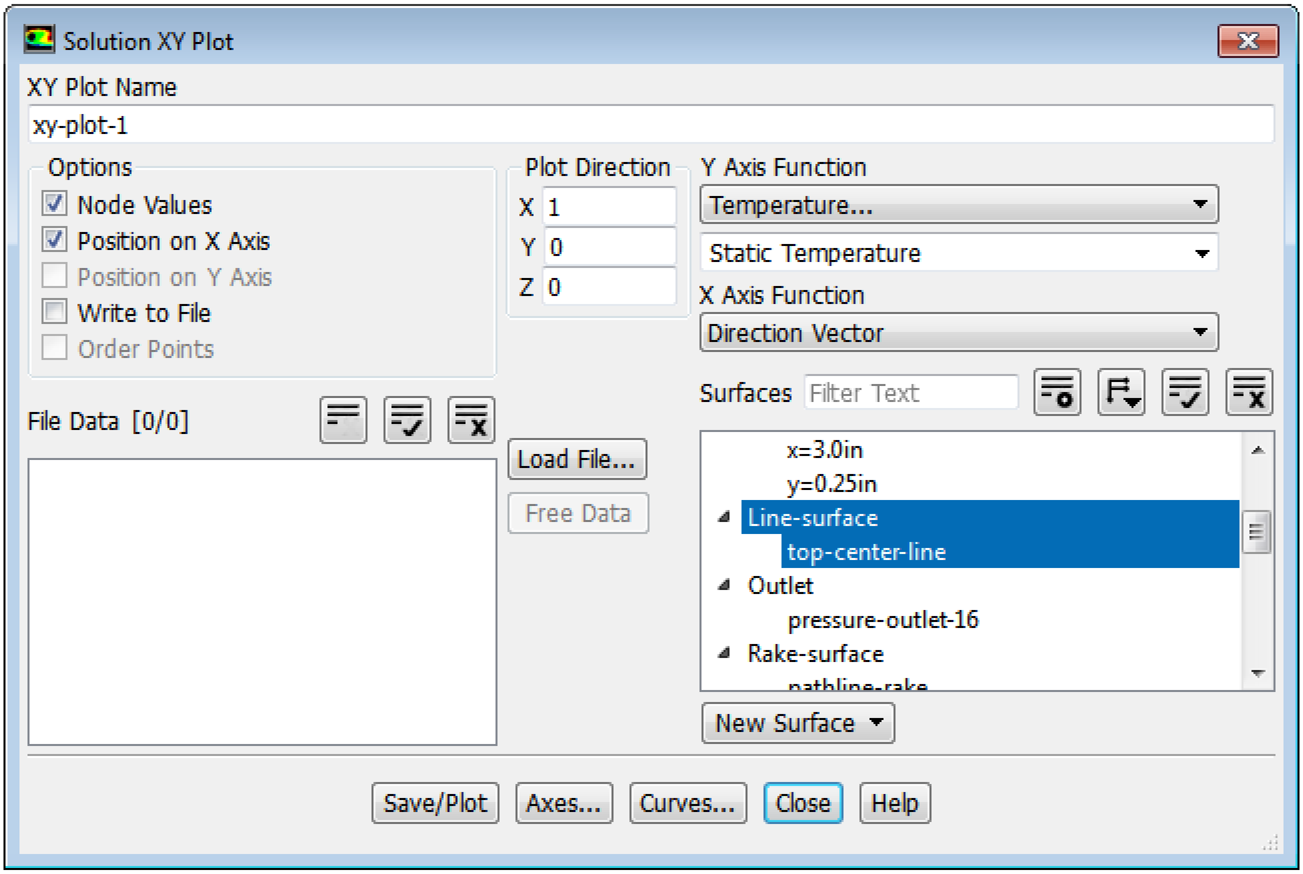
   These coordinates define the top centerline of the module.

c. Enter `top-center-line` for **New Surface Name**.

d. Click **Create** and close the **Line/Rake Surface** dialog box.

2. Plot the temperature distribution along the top centerline of the module (Figure 22.20: Temperature Along the Top Centerline of the Module (p. 928)).

 **Postprocessing** → **Plots** → **XY Plot** → **New...**



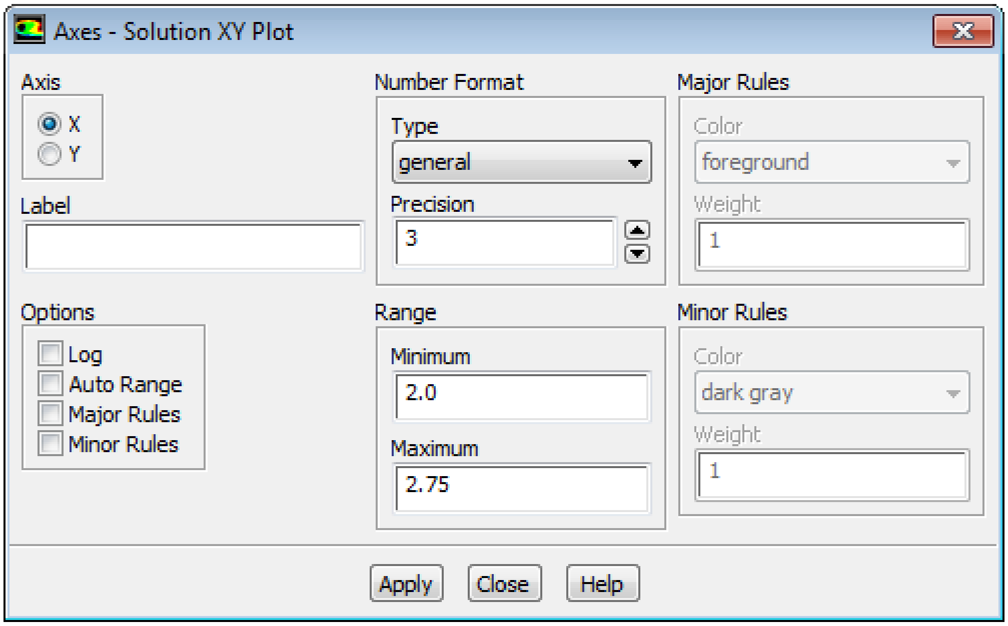
a. Retain the default **Plot Direction** of (1,0,0).

b. Select **Temperature...** and **Static Temperature** from the **Y Axis Function** drop-down lists.

c. Select **top-center-line** from the **Surfaces** selection list.

   *This will plot temperature vs. the X coordinate along the selected line (**top-center-line**).*

d. Click the **Axes...** button to open the **Axes - Solution XY Plot** dialog box.



i. Retain the selection of **X** in the **Axis** list.

ii. Disable **Auto Range** in the **Options** group box.

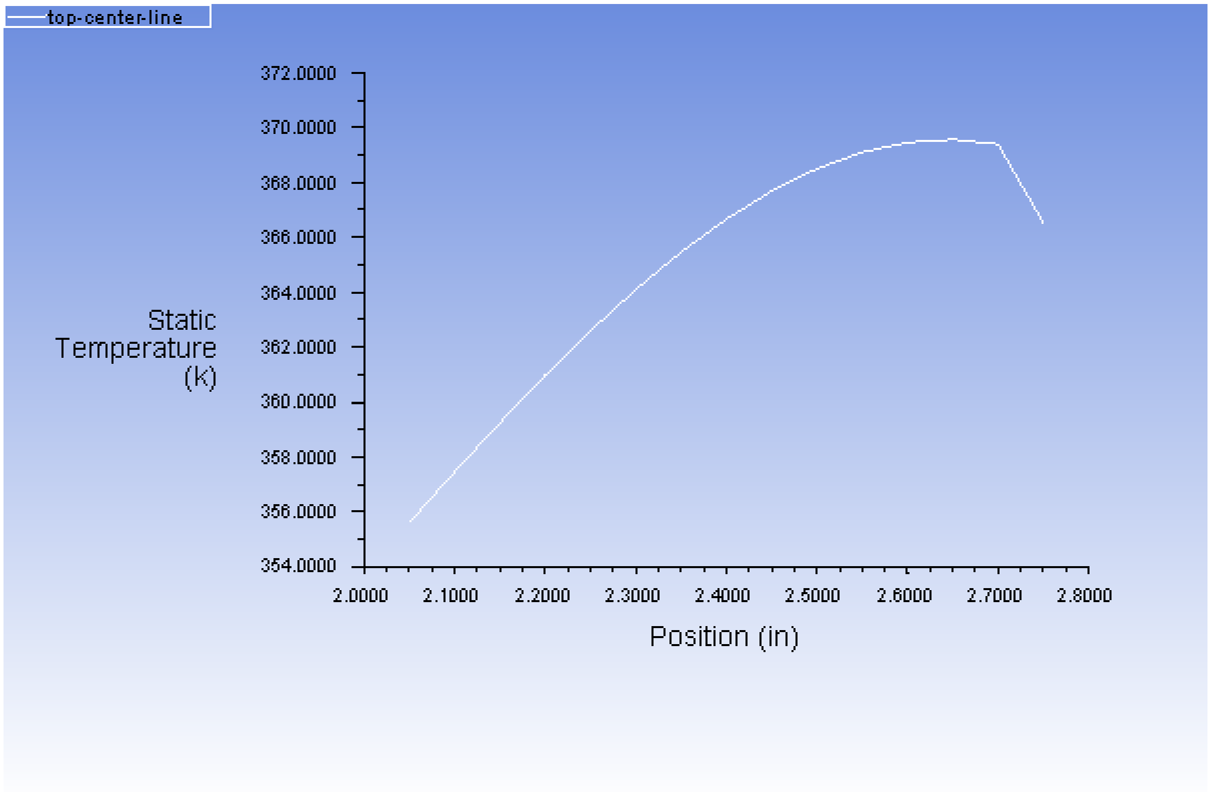
iii. Enter `2.0` for **Minimum** and `2.75` for **Maximum** in the **Range** group box.

iv. Click **Apply** and close the **Axes - Solution XY Plot** dialog box.

e. Click **Save/Plot** and close the **Solution XY Plot** dialog box.

The temperature distribution (Figure 22.20: Temperature Along the Top Centerline of the Module (p. 928)) shows the temperature increase across the module surface as the thermal boundary layer develops in the cooling air flow.

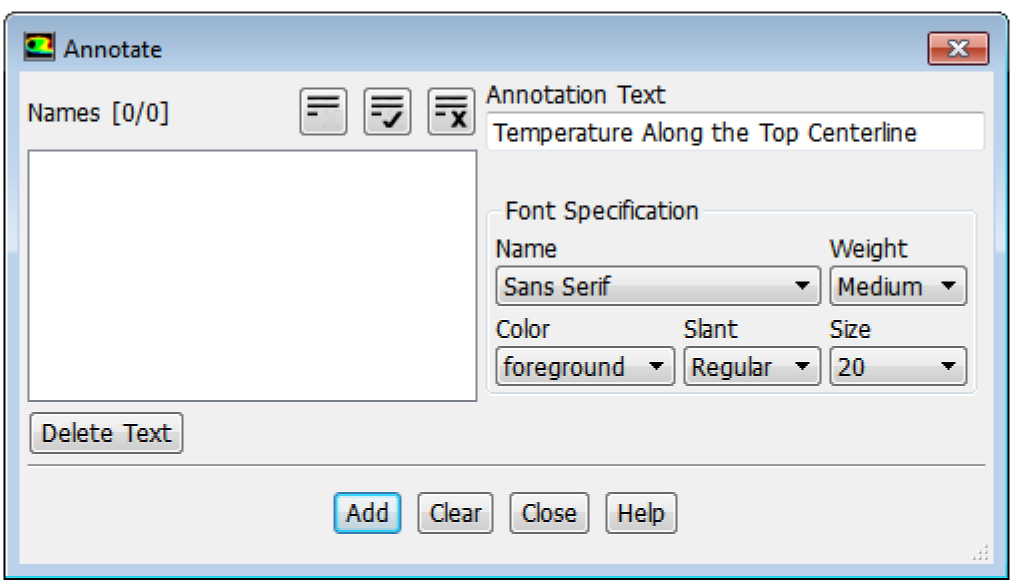
Figure 22.20: Temperature Along the Top Centerline of the Module



22.4.15. Creating Annotation

You can annotate the display with the text of your choice.

Viewing → **Graphics** → **Annotate...**



1. Enter the text describing the plot (for example, `Temperature Along the Top Centerline`) in the **Annotation Text** field.

2. Select **20** from the **Size** drop-down list in the **Font Specification** group box.

3. Click **Add**.

   *A **Working** dialog box will appear telling you to select the desired location of the text using the mouse-probe button.*

4. Click the right mouse button in the graphics window where you want the text to appear, and you will see the text displayed at the selected location (Figure 22.21: A Display with Annotation (p. 929)).

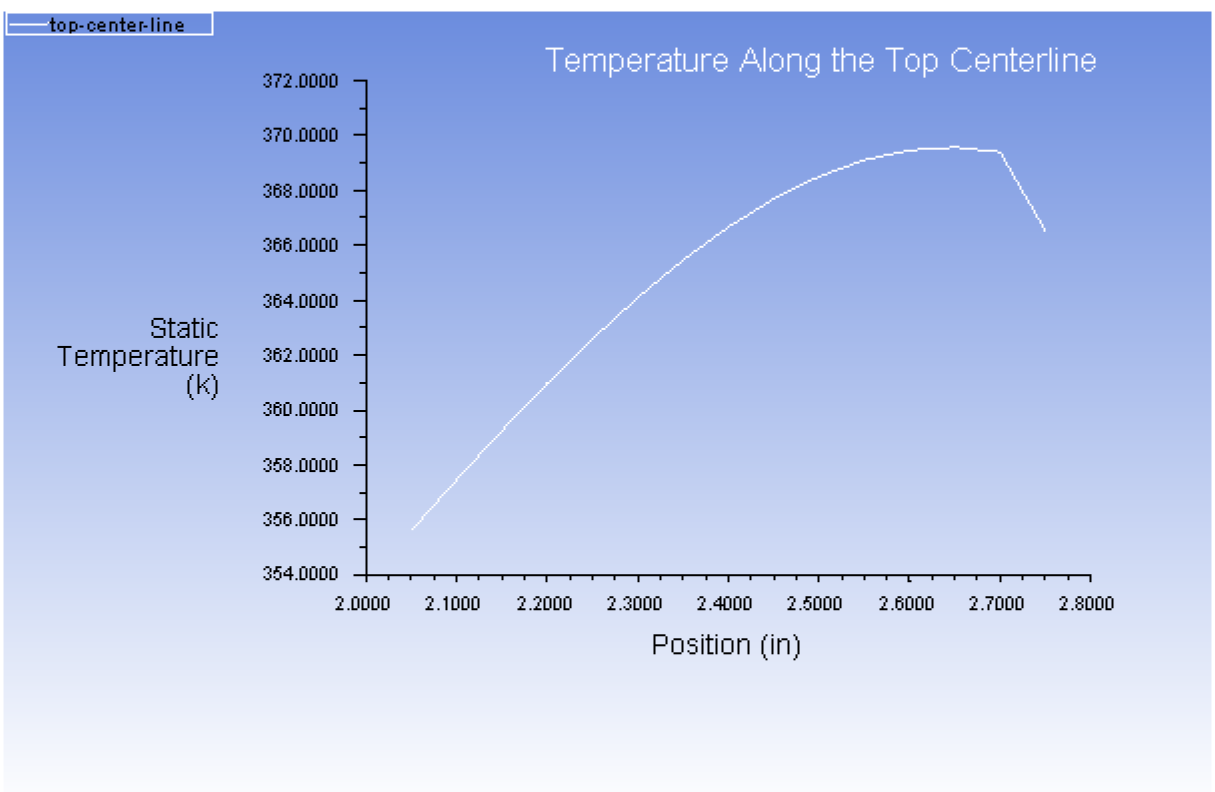
Extra

If you want to move the text to a new location on the screen, select the text in the **Names** selection list, click **Delete Text**, and click **Add** once again, defining a new position with the mouse.

Note

Depending on the size of the graphics window and the picture file format you choose, the font size of the annotation text you see on the screen may be different from the font size in a picture file of that graphics window. The annotation text font size is absolute, while the rest of the items in the graphics window are scaled to the proportions of the picture file.

Figure 22.21: A Display with Annotation

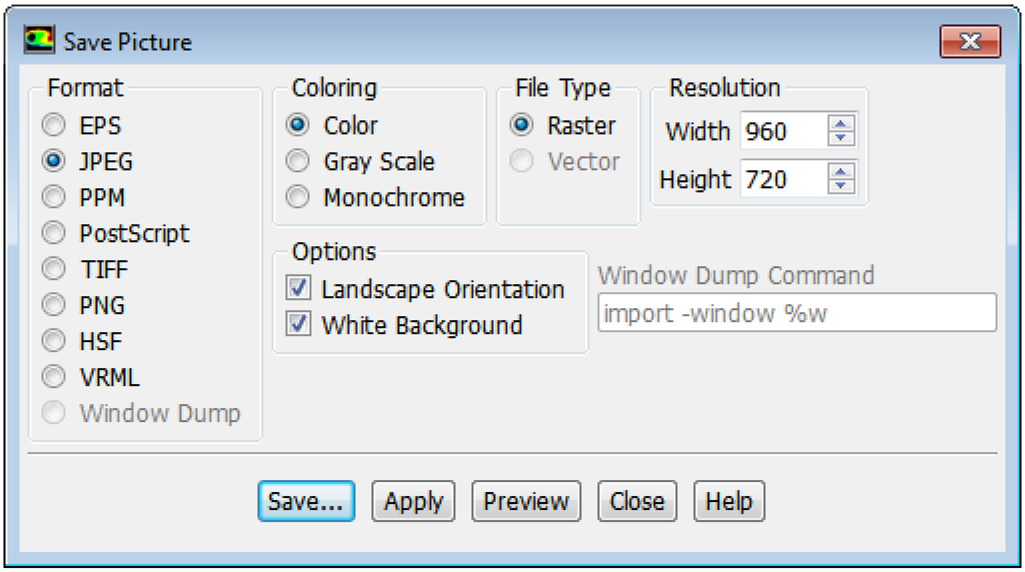


5. Close the **Annotate** dialog box.

22.4.16. Saving Picture Files

You can save picture files of the graphics window in many different formats, including PostScript, encapsulated PostScript, TIFF, PNG, PPM, JPEG, VRML and window dumps. Here, the procedure for saving a color PostScript file is shown.

Click the **Save Picture** icon- in the toolbar to open the **Save Picture** dialog box.



1. Select **JPEG** from the **Format** list.

2. Retain the default selection of **Color** from the **Coloring** list.

3. Click the **Save...** button to open the **Select File** dialog box.

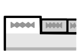
    a. Enter a name for **Hardcopy File**.

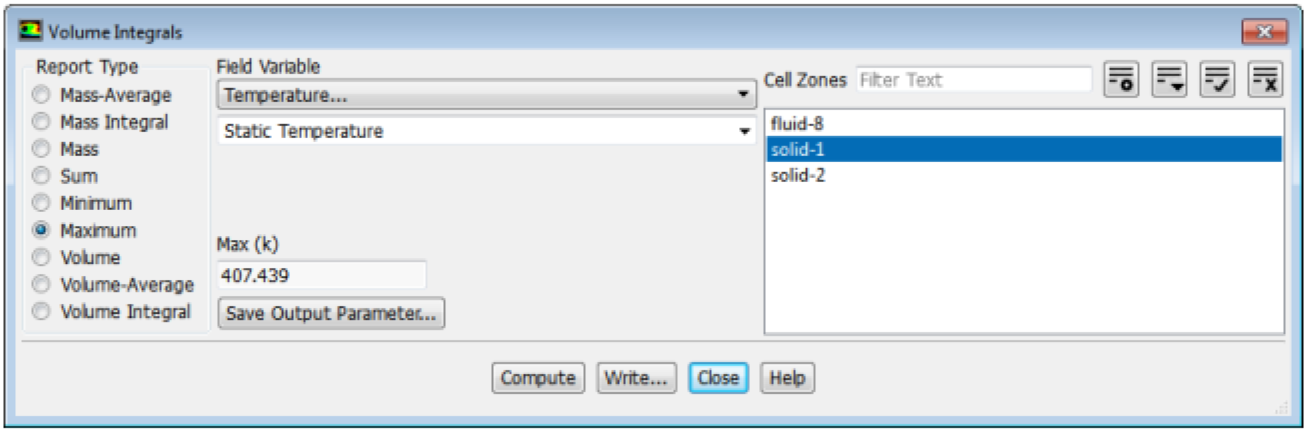
    b. Click **OK** to close the **Select File** dialog box.

4. Close the **Save Picture** dialog box.

22.4.17. Generating Volume Integral Reports

Reports of volume integrals can be used to determine the volume of a particular fluid region (that is, a fluid zone), the sum of quantities, or the maximum and minimum values of particular variables. Here we will use the volume integral reports to determine the maximum and minimum temperature in the chip, board, and the airflow.

 **Postprocessing** → **Reports** → **Volume Integrals...**



1. Select **Maximum** from the **Report Type** list.

2. Select **Temperature...** and **Static Temperature** from the **Field Variable** drop-down lists.

3. Select **solid-1** from the **Cell Zones** selection list.

4. Click **Compute** to calculate the maximum temperature.

   The maximum temperature in the solid-1 cell zone (the chip) is displayed.

5. Select **Minimum** in the **Report Type** group box and click **Compute**.

   The minimum temperature in the solid-1 cell zone (the chip) is displayed.

6. Repeat the operations to determine the maximum and minimum temperatures in the **solid-2** and **fluid-8** cell zones, corresponding to the board and fluid volume, respectively.

22.5. Summary

This tutorial demonstrated the use of many of the extensive postprocessing features available in Fluent.

For more information on these and related features, see reporting alphanumeric data and displaying graphics in the Fluent User's Guide.

Chapter 23: Using the Adjoint Solver – 2D Laminar Flow Past a Cylinder

This tutorial is divided into the following sections:

23.1. Introduction

ANSYS Fluent's adjoint solver is used to compute the sensitivity of quantities of interest in a fluid system with respect to the user-specified inputs, for an existing flow solution. Importantly, this also includes the sensitivity of the computed results with respect to the geometric shape of the system. The adjoint design change tool is a powerful component that can use the sensitivity information from one or more adjoint solutions to guide systematic changes that result in predictable improvements in the system performance, which can be made subject to various types of design constraints if desired.

This tutorial provides an example of how to generate sensitivity data for flow past a circular cylinder, how to postprocess the results, and how to use the data to perform a multi-objective design change that reduces drag and increases lift by morphing the mesh. The tutorial makes use of a previously computed flow solution, and demonstrates how to do the following:

- Load the adjoint solver add-on.
- Select the observable of interest.
- Access the solver controls for advancing the adjoint solution.
- Set convergence criteria and plot and print residuals.
- Advance the adjoint solver.
- Postprocess the results to extract sensitivity data.
- Use the design change tool to modify the cylinder shape to simultaneously reduce the drag and increase the lift.

23.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)

- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

and that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

23.3. Problem Description

The configuration is a circular cylinder, bounded above and below by symmetry planes. The flow is laminar and incompressible with a Reynolds number of 40, based on the cylinder diameter. At this Reynolds number, the flow is steady.

23.4. Setup and Solution

The following sections describe the setup steps for this tutorial:

23.4.1. Preparation

1. Set up a working folder on the computer you will be using.

2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

   Note

   If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

3. Enter the name of this tutorial into the search bar.

4. Narrow the results by using the filter on the left side of the page.

   a. Click **ANSYS Fluent** under **Product**.

   b. Click **18.0** under **Version**.

5. Select this tutorial from the list.

6. Click the **adjoint_cylinder_R180.zip** link to download the input files.

7. Unzip `adjoint_cylinder_R180.zip`.

   The files `cylinder_tutorial.cas` and `cylinder_tutorial.dat` can be found in the `adjoint_cylinder` folder created after unzipping the file.

8. Use Fluent Launcher to start the **2D** version of ANSYS Fluent, with the **Double Precision** and **Display Mesh After Reading** options enabled.

For more information about starting ANSYS Fluent using the Fluent Launcher, see the Fluent Getting Started Guide, available in the help viewer or on the ANSYS Customer Portal (http://support.ansys.com/documentation).

9. Load the converged case and data file for the cylinder geometry.

   File → **Read** → **Case & Data...**

   When prompted, browse to the location of the case and data files and select `cylinder_tutorial.cas` to load. The corresponding data file will automatically be loaded as well.

   Note

   After you read in the mesh, it will be displayed in the embedded graphics windows, since you enabled the appropriate display option in Fluent Launcher.

   The data file contains a previously computed flow solution that will serve as the starting point for the adjoint calculation. Part of the mesh and the velocity field are shown below:

Figure 23.1: Mesh Close to the Cylinder Surface

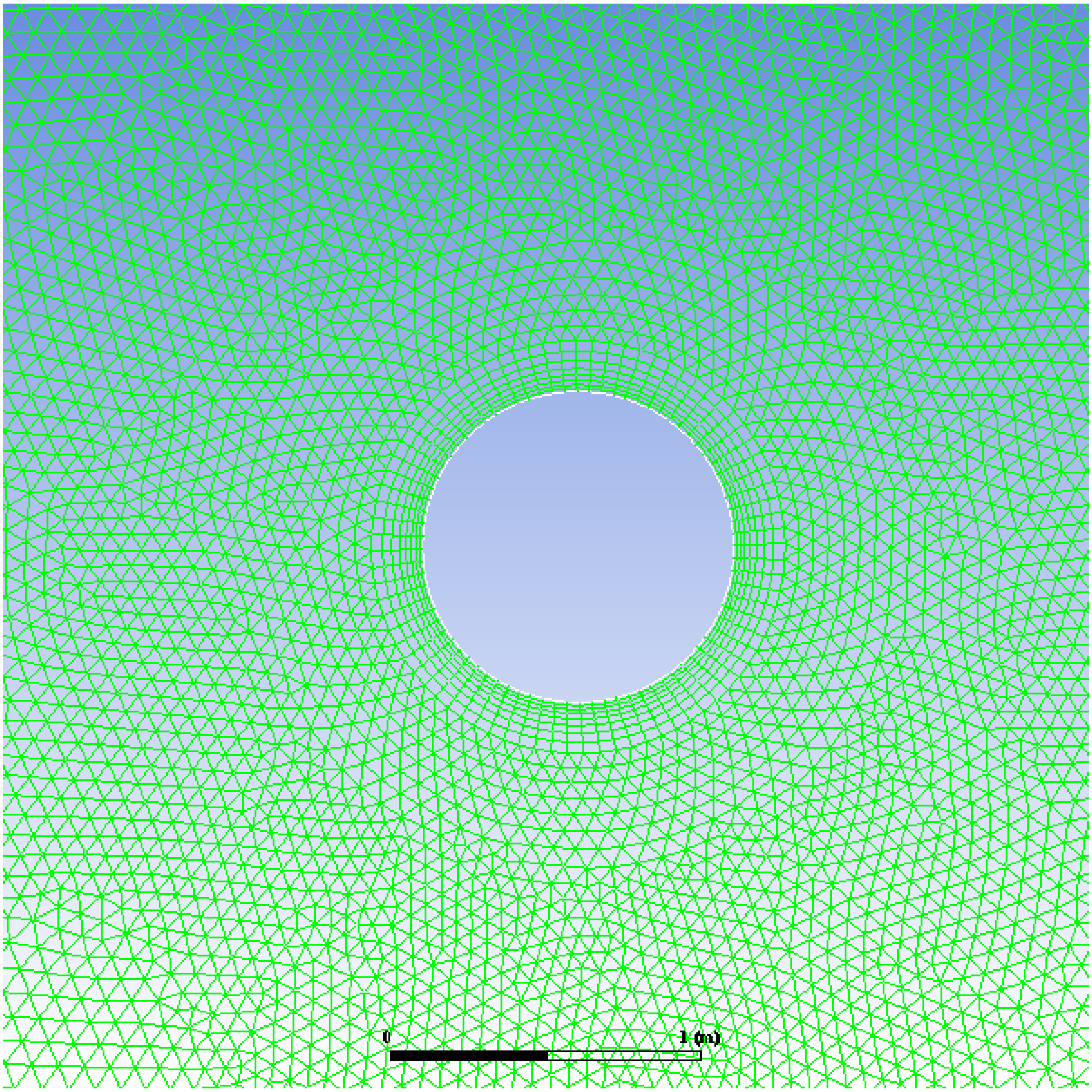
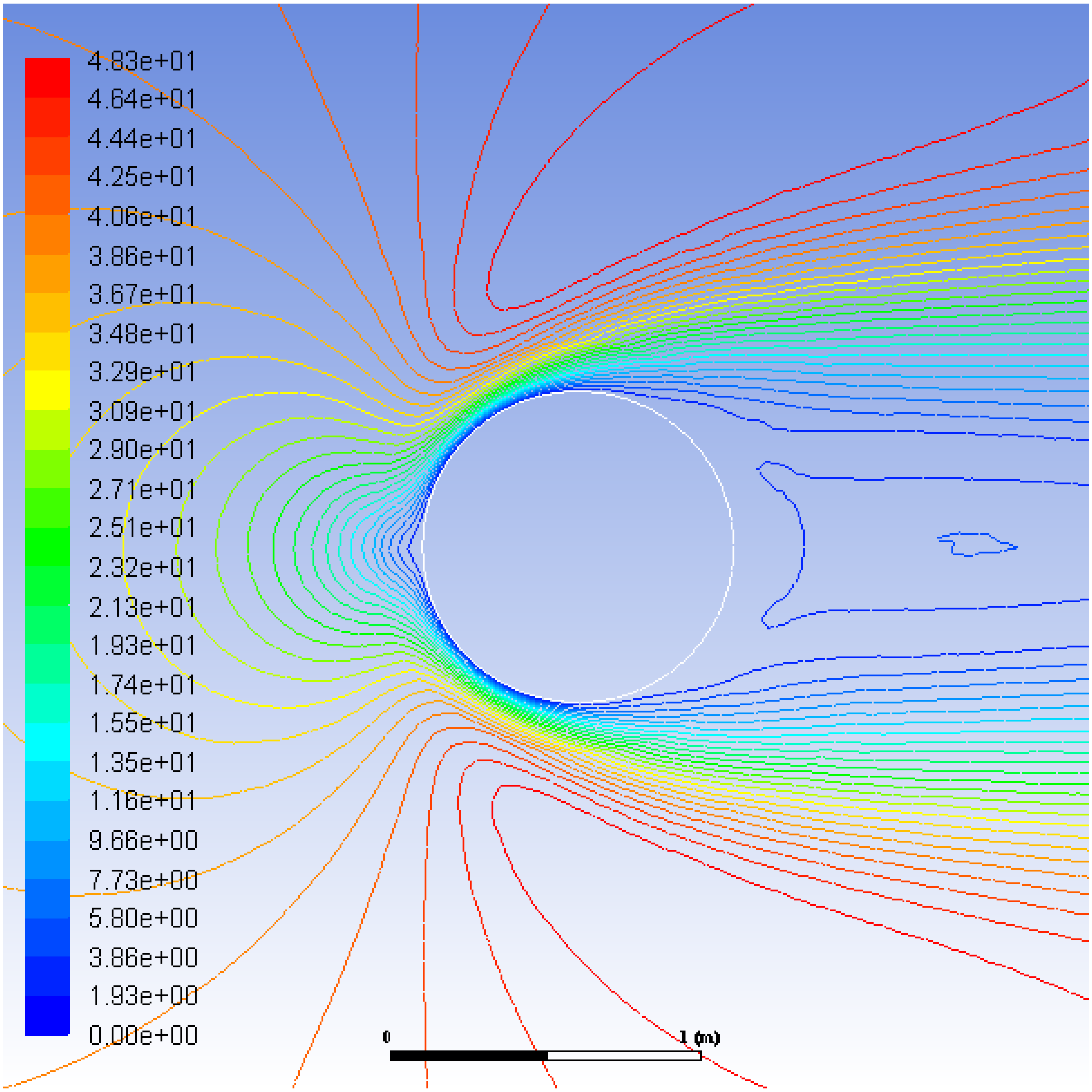


Figure 23.2: Contours of Velocity Magnitude



23.4.2. Step 1: Load the Adjoint Solver Add-on

The adjoint solver add-on is loaded into ANSYS Fluent by typing the following in the console:

```
>define/models/addon-module
```

to display the following:

```
Fluent Addon Modules:
      0. None
      1. MHD Model
      2. Fiber Model
      3. Fuel Cell and Electrolysis Model
      4. SOFC Model with Unresolved Electrolyte
      5. Population Balance Model
      6. Adjoint Solver
      7. Single-Potential Battery Model
      8. Dual-Potential MSMD Battery Model
      9. PEM Fuel Cell Model
```

```
       10. Macroscopic Particle Model
 Enter Module Number: [0] 6
```

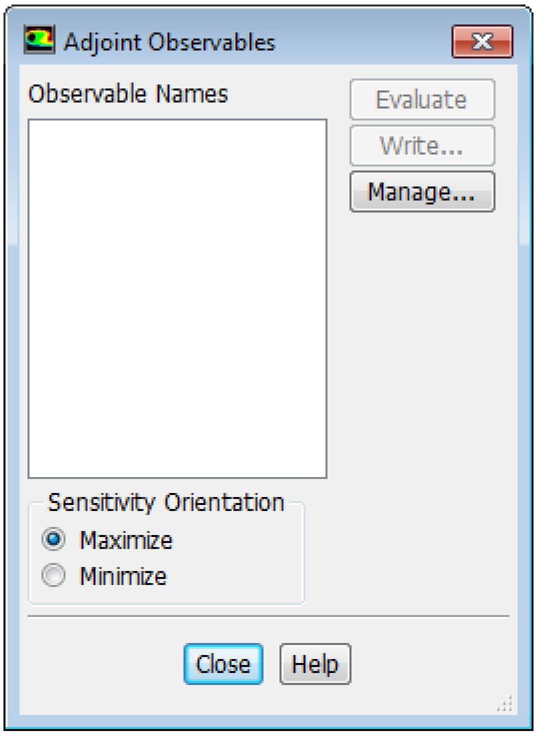
Enter 6 to load the adjoint module. After the adjoint module is loaded, buttons become available in the **Design** tab of the ribbon within ANSYS Fluent.

23.4.3. Step 2: Define Observables

Begin setting up the adjoint solver by opening the **Adjoint Observables** dialog box. Here you will create lift and drag observables.

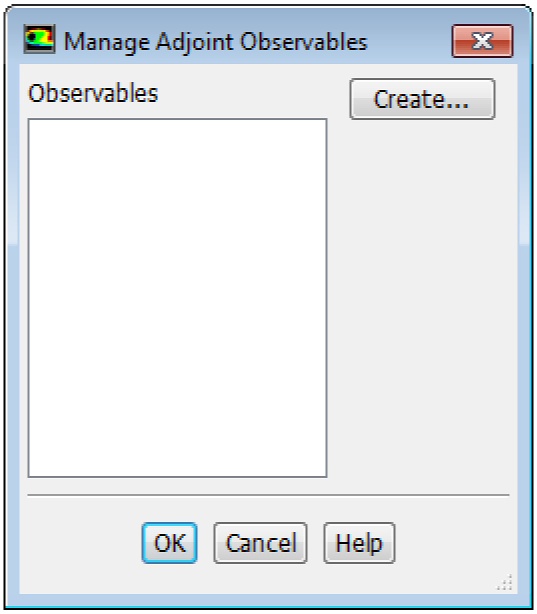
Design → **Adjoint-Based** → **Observable...**

Figure 23.3: Adjoint Observables Dialog Box



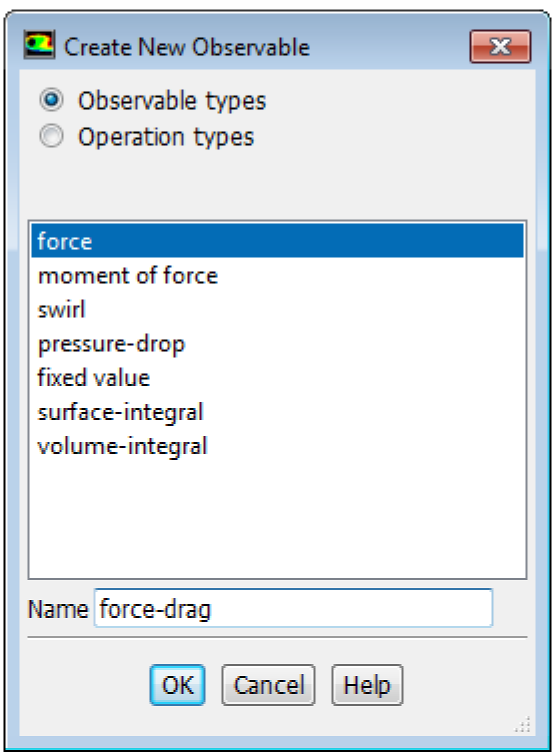
1. Click the **Manage...** button to open the **Manage Adjoint Observables** dialog box.

Figure 23.4: Manage Adjoint Observables Dialog Box



2. Click the **Create...** button in the **Manage Adjoint Observables** dialog box to open the **Create New Observable** dialog box.

Figure 23.5: Create New Observable Dialog Box



3. In the **Create New Observable** dialog box:

   a. Ensure that **Observable types** is selected.

b. Select **force** from the selection list.

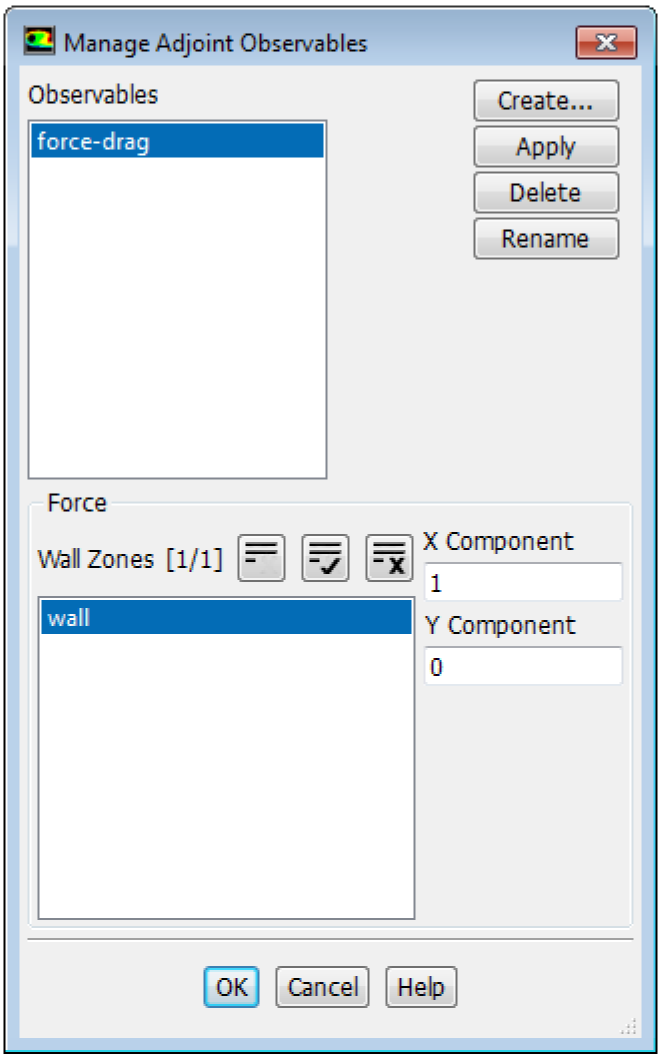
c. Enter `force-drag` for **Name**.

d. Click **OK** to close the dialog box.

4. In the **Manage Adjoint Observables** dialog box, the newly created **force-drag** observable appears and must now be configured. (Figure 23.6: Manage Observables Dialog Box (p. 940)):

a. Select **force-drag** in the **Observables** list.

Figure 23.6: Manage Observables Dialog Box



b. Select **wall** under **Wall Zones**. This is the cylinder wall on which you want the force to be evaluated.

c. Ensure that the **X-Component** direction is set to `1` and the **Y-Component** direction is set to `0`.

d. Click **Apply** to commit the settings for **force-drag**.

5. Repeat the process in the **Manage Adjoint Observables** dialog box to create a lift observable with the following settings:

Name	`force-lift`
Wall Zones	`wall`
X-Component	`0`
Y-Component	`1`

Tip

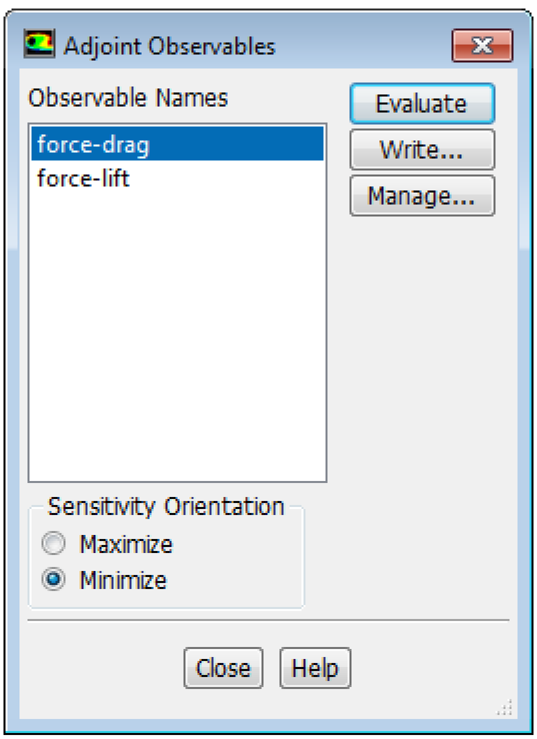
If the **Name** field is not available in the **Create New Observable** dialog box, select a different observable type and then select **force** again to make it available.

When you have configured the **force-lift** observable, click **OK** to commit the settings for **force-lift** and close the **Manage Adjoint Observables** dialog box.

23.4.4. Step 3: Compute the Drag Sensitivity

1. In the **Adjoint Observables** dialog box (Figure 23.7: Adjoint Observables Dialog Box (p. 941)) specify that you will solve for the drag sensitivity.

Figure 23.7: Adjoint Observables Dialog Box



   a. Select **force-drag** in the list of **Observable Names**.

      *The selection in the **Adjoint Obervables** dialog box determines the observable for which sensitivities will be computed. You will first compute the drag sensitivities.*

   b. Select **Minimize** from the **Sensitivity Orientation** list, because you are trying to reduce the drag force. This indicates that postprocessed results for the drag sensitivity will be displayed such that a reduction in drag is achieved by a design change in the positive sensitivity direction.

   c. Click **Evaluate** to print the value of the drag force on the wall in the console.

```
Observable name: force-drag
Observable Value (n) = 1271.7444
```

This value is in SI units, with n denoting Newtons.

2. Adjust the solution controls.

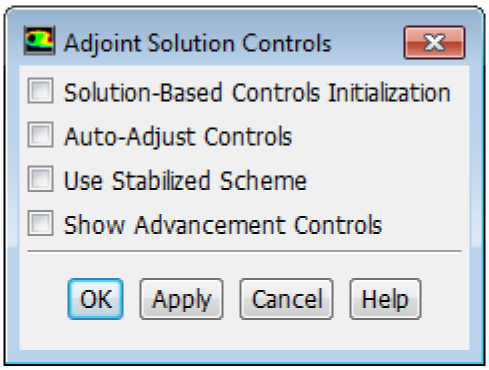
   The default solution control settings are chosen to provide robust solution advancement for a wide variety of problems, including those having complex geometry, high local flow rates, and turbulence. Given sufficient iterations, a converged result can often be obtained without modifying the controls.

   For this simple laminar flow case, more aggressive settings will yield faster convergence.

   Open the **Adjoint Solution Controls** dialog box (Figure 23.8: Adjoint Solution Controls Dialog Box (p. 942)).

   Design → **Adjoint-Based** → **Solver Controls...**

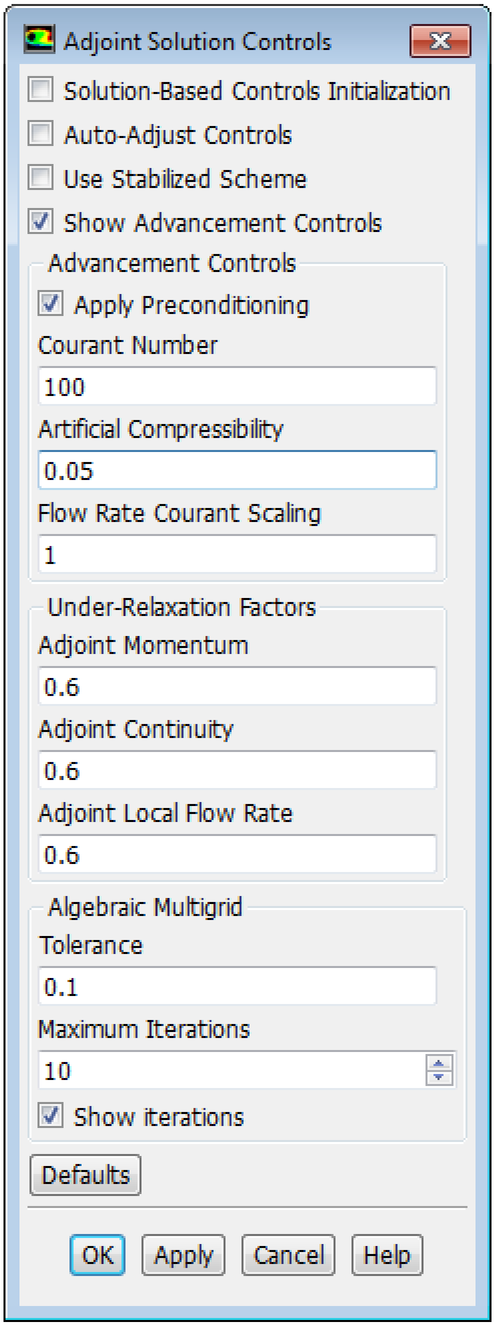
   Figure 23.8: Adjoint Solution Controls Dialog Box



   a. Disable the **Solution-Based Controls Initialization** and **Auto-Adjust Controls** options.

      This prevents Fluent from automatically choosing and adjusting the solution controls for you.

   b. Enable **Show Advancement Controls**.



c. Ensure that the **Apply Preconditioning** option is enabled.

   Preconditioning can help the calculation progress in a stable manner.

d. Enter `100` for **Courant Number**.

   Higher **Courant Number** values correspond to more aggressive settings / faster convergence, which is appropriate for a simple case such as this.

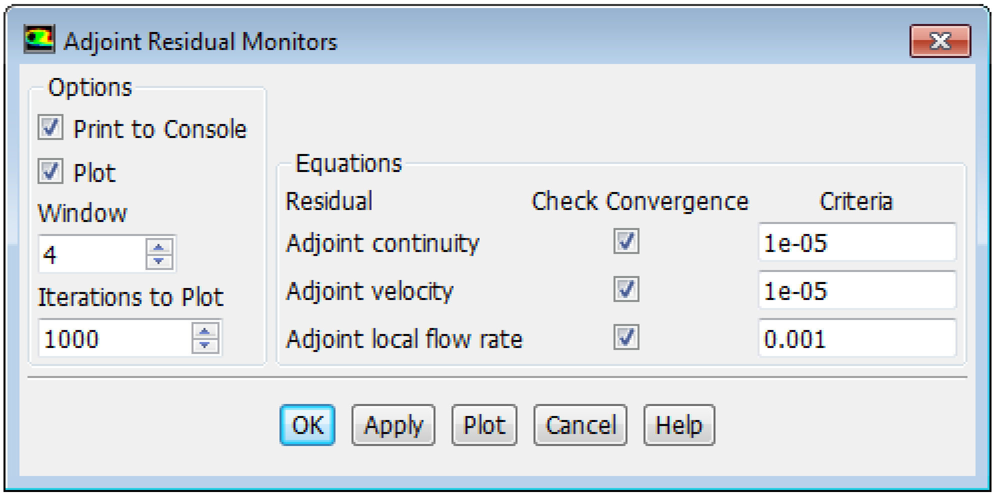
e. Enter `0.05` for **Artificial Compressibility**.

f. Click **OK** to close the dialog box.

3. Configure the adjoint solution monitors by opening the **Adjoint Residual Monitors** dialog box (Figure 23.9: Adjoint Residual Monitors Dialog Box (p. 944)).

   Design → Adjoint-Based → Monitors...

   Figure 23.9: Adjoint Residual Monitors Dialog Box



   In the **Adjoint Residual Monitors** dialog box, you set the adjoint equations that will be checked for convergence, as well as set the corresponding convergence criteria.

   a. Make sure that the **Print to Console** and **Plot** options are enabled.

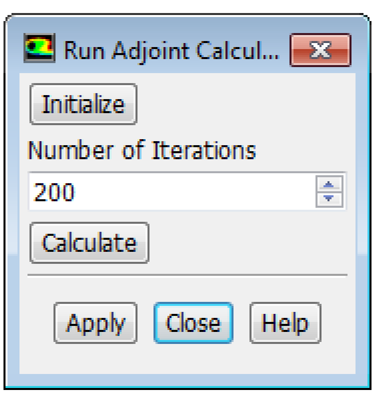
   b. Keep the default values of `1e-05` for **Adjoint continuity** and **Adjoint velocity**, and `0.001` for **Adjoint local flow rate**. These settings are adequate for most cases. Make sure that the **Check Convergence** options are enabled.

   c. Click **OK** to close the dialog box.

4. Run the adjoint solver using the **Run Adjoint Calculation** dialog box (Figure 23.10: Run Adjoint Calculation Dialog Box (p. 944)).

   Design → Adjoint-Based → Calculate...

   Figure 23.10: Run Adjoint Calculation Dialog Box

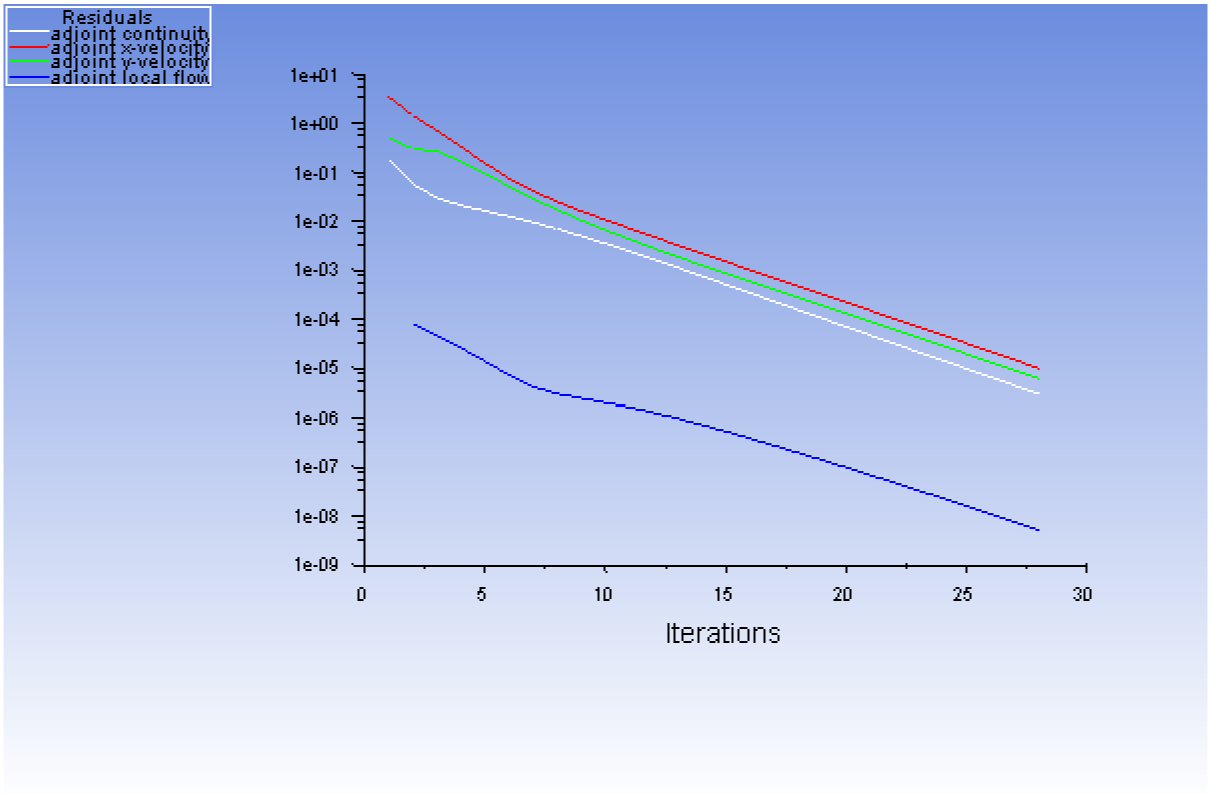


   a. Click the **Initialize** button. This initializes the adjoint solution everywhere in the problem domain to zero.

b. Set the **Number of Iterations** to `200`. The adjoint solver is fully configured to start running for this problem.

c. Click the **Calculate** button to advance the solver to convergence.

Figure 23.11: Residuals for the Converged Solution



d. When the calculation is complete, **Close** the **Run Adjoint Calculation** dialog box.

23.4.5. Step 4: Postprocess and Export Drag Sensitivity

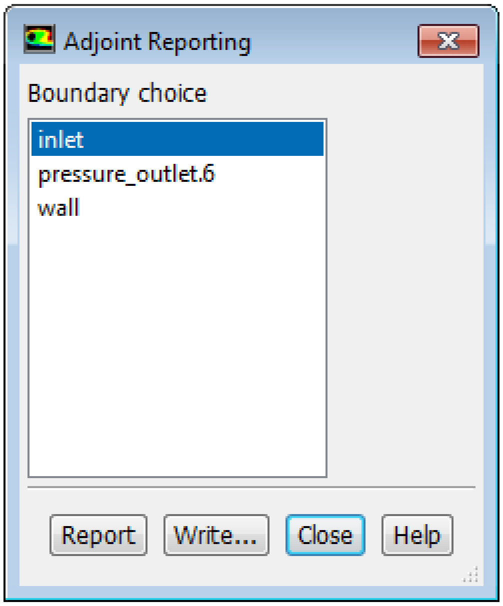
In this section, postprocessing options for the adjoint solution are presented.

23.4.5.1. Boundary Condition Sensitivity

1. Open the **Adjoint Reporting** dialog box (Figure 23.12: Adjoint Reporting Dialog Box (p. 946)).

   Design → **Adjoint-Based** → **Reporting...**

Figure 23.12: Adjoint Reporting Dialog Box



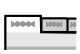
2. Select **inlet** under **Boundary choice** and click the **Report** button to display a report in the console of the available scalar sensitivity data on the inlet:

```
Updating shape sensitivity data.
Done.

Boundary condition sensitivity report: inlet
Observable: force-drag
Velocity Magnitude (m/s): 40  Sensitivity ((n)/(m/s)): 54.553894
Decrease Velocity Magnitude to decrease force-drag
```

3. Close the **Adjoint Reporting** dialog box.

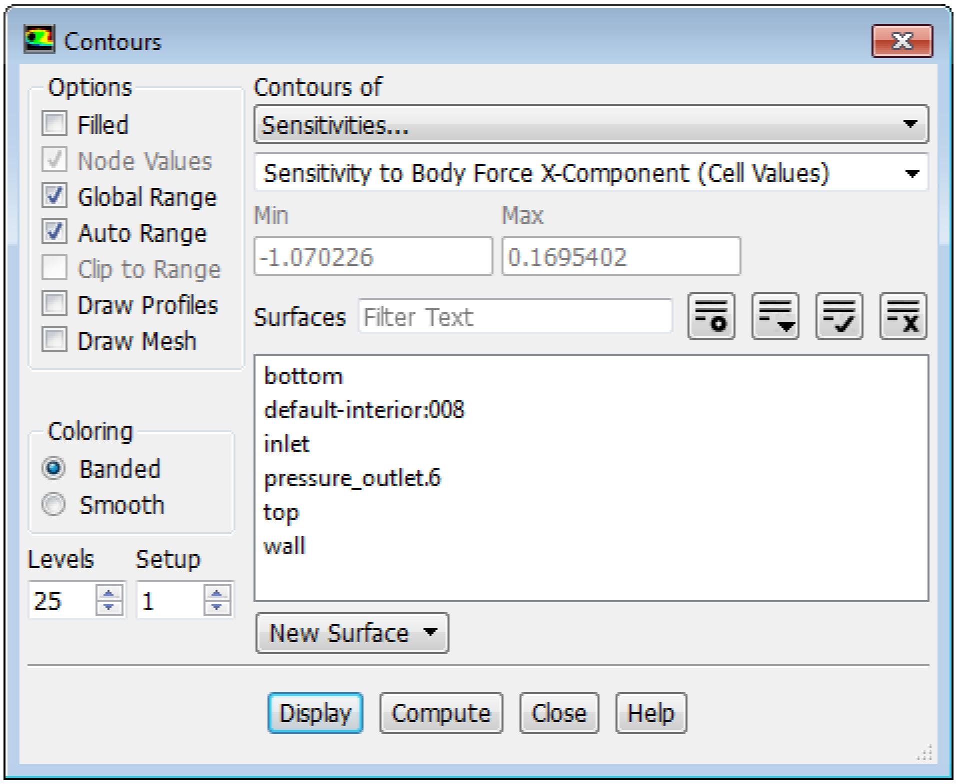
23.4.5.2. Momentum Source Sensitivity

1. Open the **Contours** dialog box.

 **Postprocessing** → **Graphics** → **Contours** → **Edit...**

2. Select **Sensitivities...** and **Sensitivity to Body Force X-Component (Cell Values)** from the **Contours of** drop-down lists.

Figure 23.13: Contours Dialog Box When Plotting Adjoint Fields



3. Enter 25 for **Levels**.

4. Click **Display** to view the contours (Figure 23.14: Adjoint Sensitivity to Body Force X-Component Contours (p. 948)) and then **Close** the **Contours** dialog box.

Figure 23.14: Adjoint Sensitivity to Body Force X-Component Contours

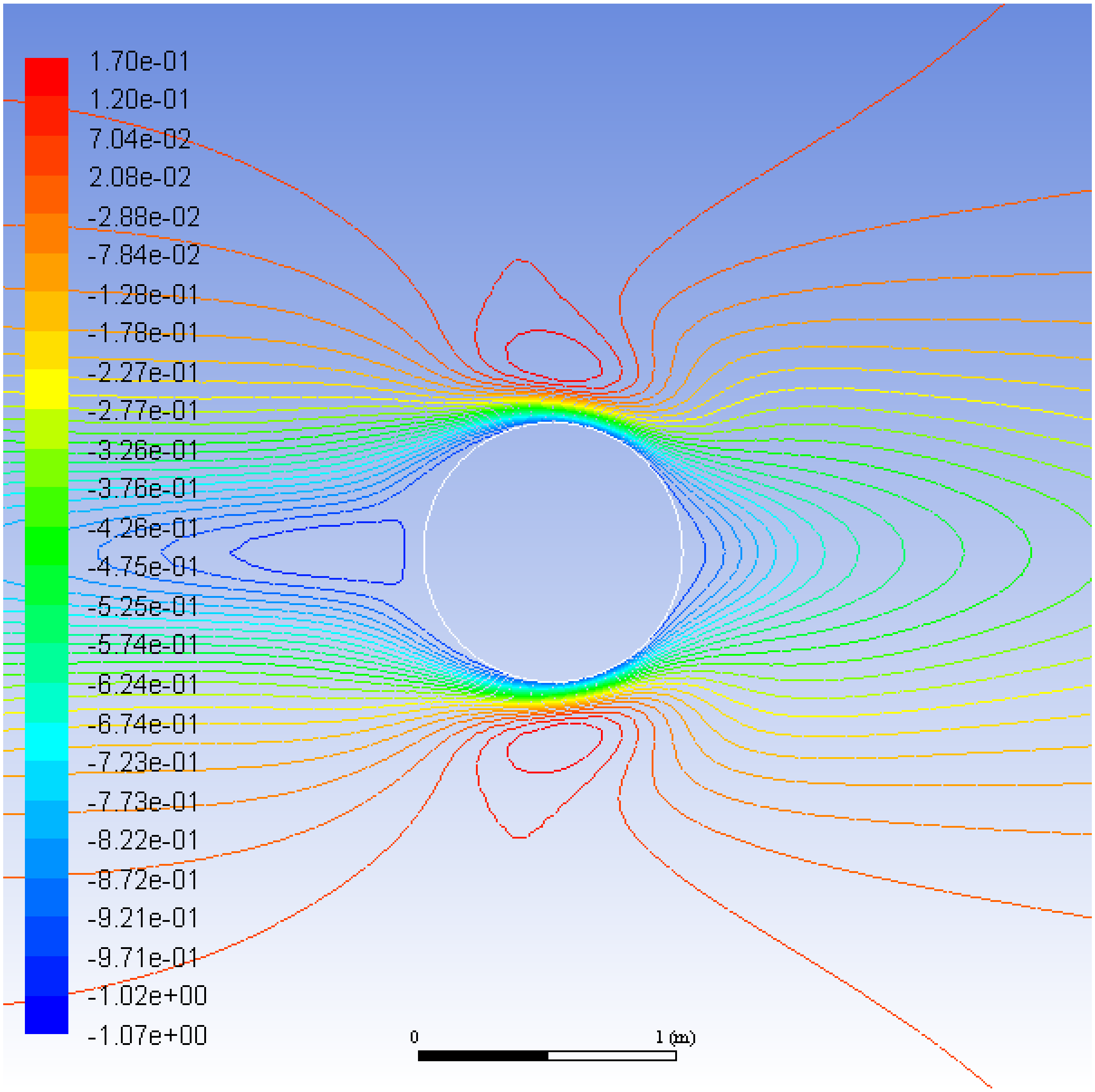


Figure 23.14: Adjoint Sensitivity to Body Force X-Component Contours (p. 948) shows how sensitive the drag on the cylinder is to the application of a body force in the X-direction in the flow. If a body force is applied directly upstream of the cylinder, for example, the disturbed flow is incident on the cylinder and modifies the force that it experiences.

The contours also show how the computed drag is affected by lack of convergence in the original X-momentum equation for the flow.

23.4.5.3. Shape Sensitivity

1. Open the **Vectors** dialog box (Figure 23.15: Vectors Dialog Box (p. 949))

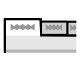
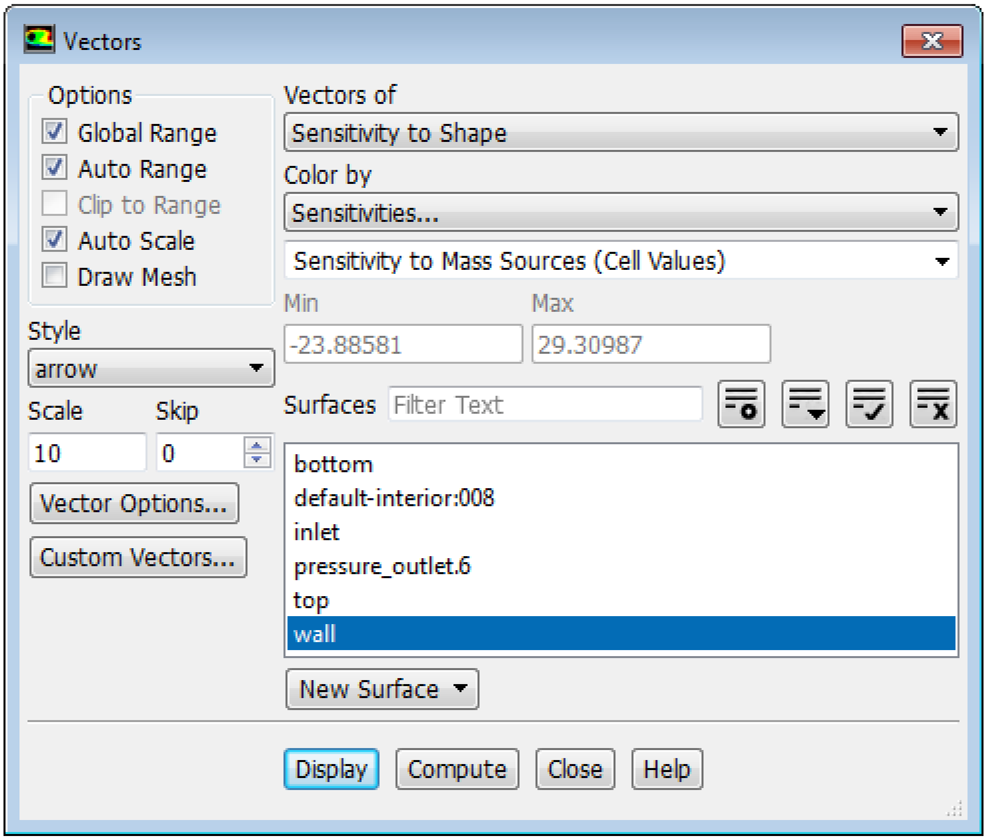
 **Postprocessing** → **Graphics** → **Vectors** → **Edit...**

Figure 23.15: Vectors Dialog Box



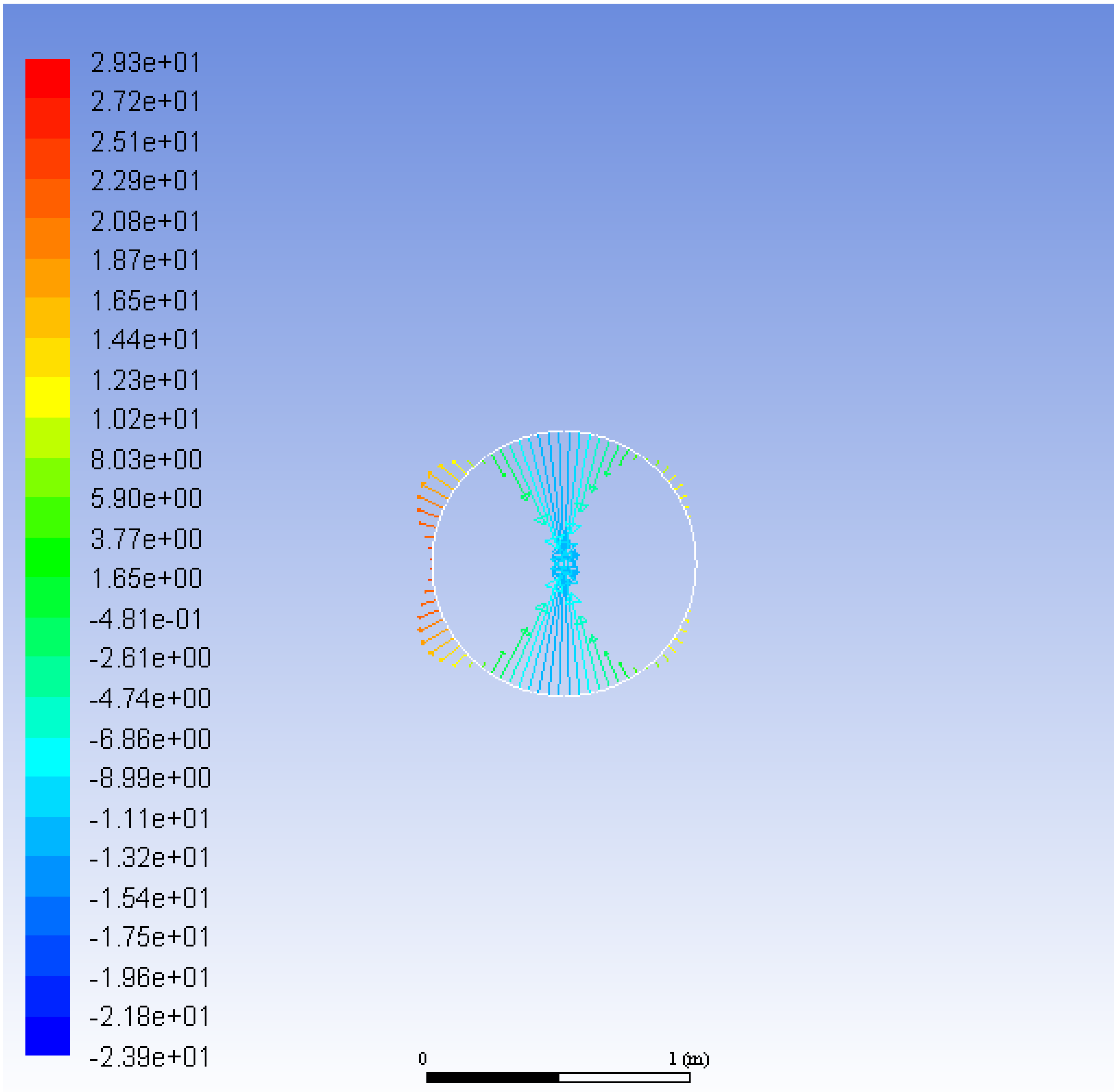
2. Select **Sensitivity to Shape** from the **Vectors of** drop-down list.

3. Select **Sensitivities...** and **Sensitivity to Mass Sources (Cell Values)** from the **Color by** drop-down lists.

4. Select **wall** from the **Surfaces** selection list.

5. Click the **Display** button to view the vectors (Figure 23.16: Shape Sensitivity Colored by Sensitivity to Mass Sources (Cell Values) (p. 950)) and then **Close** the **Vectors** dialog box.

Figure 23.16: Shape Sensitivity Colored by Sensitivity to Mass Sources (Cell Values)



This plot shows how sensitive the drag on the cylinder is to changes in the surface shape. The drag is affected more significantly if the cylinder is deformed on the upstream rather than the downstream side. Maximum effect is achieved by narrowing the cylinder in the cross-stream direction.

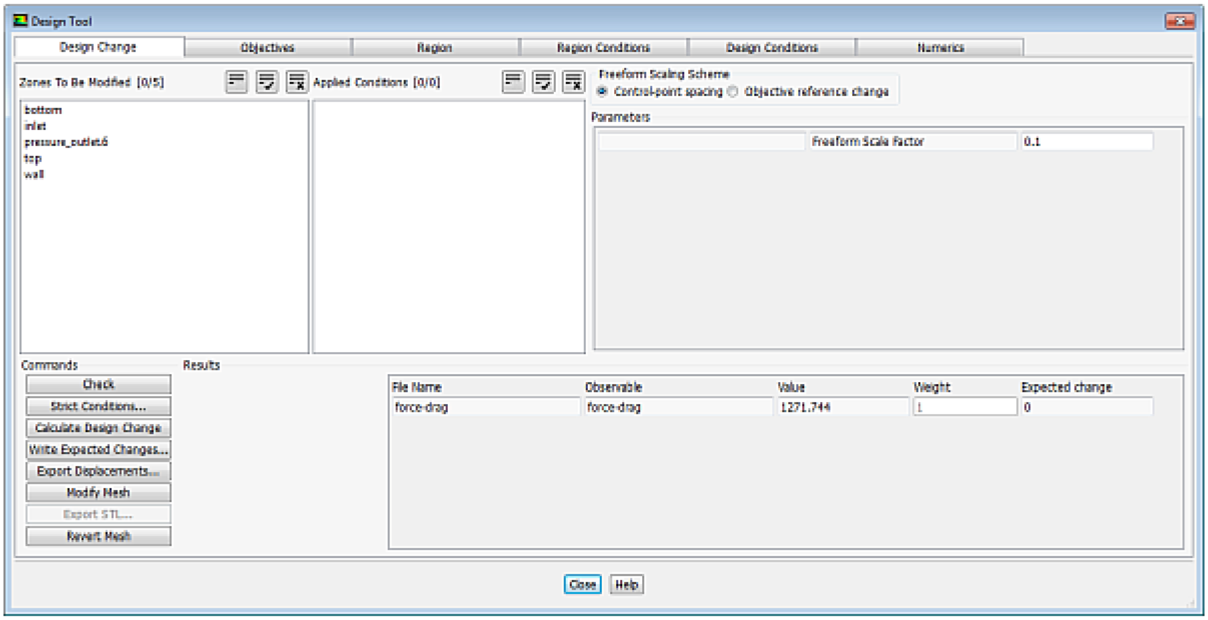
23.4.5.4. Exporting Drag Sensitivity Data

Before computing the sensitivity for the force-lift observable, you need to define the region that will be subject to geometry morphing, and export the drag sensitivity data so it can be used later in the multi-objective optimization.

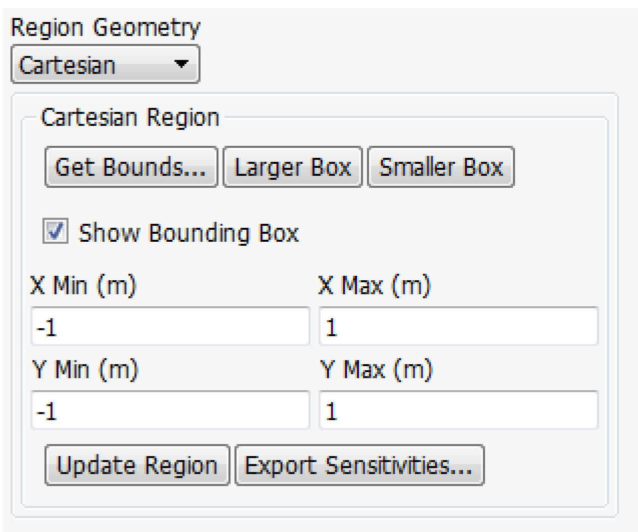
1. Open the **Design Tool** dialog box.

   Design → **Adjoint-Based** → **Design Tool...**

Figure 23.17: The Design Tool Dialog Box



2. In the **Region** tab, define the region that will be modified for the design change.



a. Ensure that **Cartesian** is selected from the **Region Geometry** drop-down list.

b. Click **Get Bounds...**.

c. Select **wall** in the **Bounding Box Definition** dialog box and click **OK**.

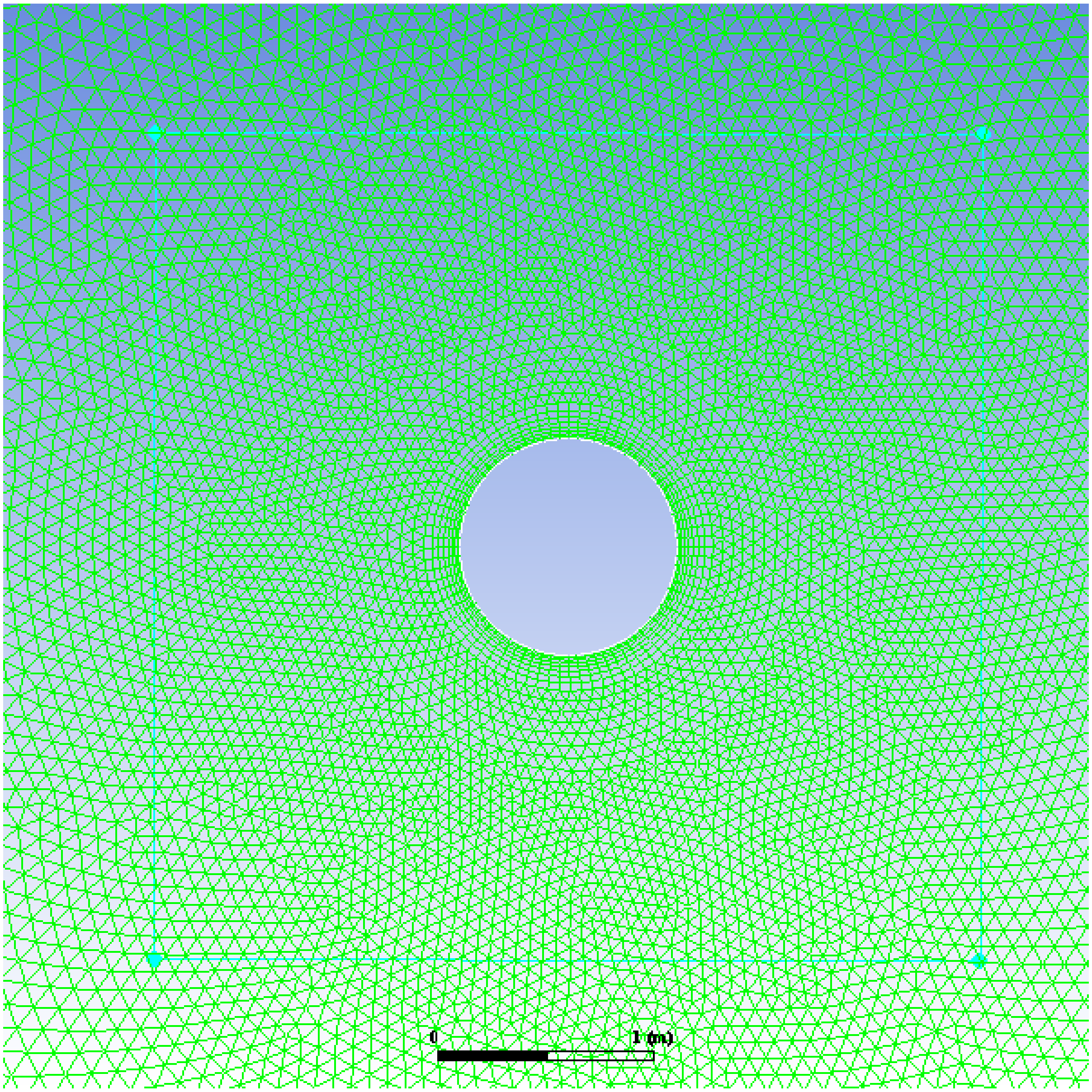
*This will initialize the morphing region to the bounding box around the cylinder **wall**.*

d. Click **Update Region** to update the view of the bounding box illustration in the graphics window.

*You can use the **Mesh Display** dialog box to also display the mesh, in order to review it prior to morphing.*

e. Click **Larger Box** several times until the X and Y Limits are ±1.907349 m (Figure 23.18: Morphing Region Around Cylinder (p. 952)).

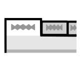
Figure 23.18: Morphing Region Around Cylinder

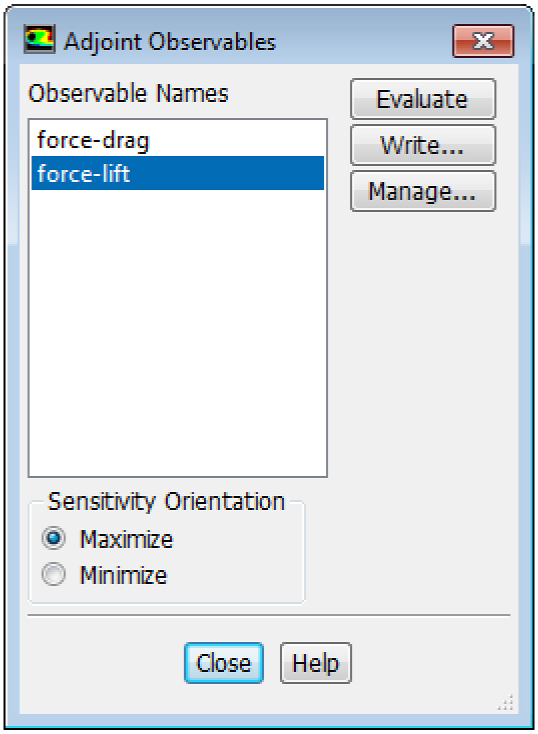


f. Click **Export Sensitivities...** and save the sensitivity data as `force-drag.s`.

23.4.6. Step 5: Compute Lift Sensitivity

1. Select **force-lift** from the **Observable Names** selection list and select **Maximize** from the **Sensitivity Orientation** list in the **Adjoint Observables** dialog box.

 **Design** → **Adjoint-Based** → **Observable...**



2. **Initialize** and **Calculate** the adjoint solution using the **Run Adjoint Calculation** dialog box to obtain the sensitivities for the **force-lift** observable.

   Design → **Adjoint-Based** → **Calculate...**

You can export the sensitivity data for the lift observable as you did for the drag, but it is not strictly necessary if you plan to perform the multi-objective optimization in the current Fluent session.

23.4.7. Step 6: Modify the Shape

In this section, you will load the previously saved **force-drag** sensitivity data and perform the multi-objective design change.

1. Open the **Design Tool** dialog box if it is not already open.

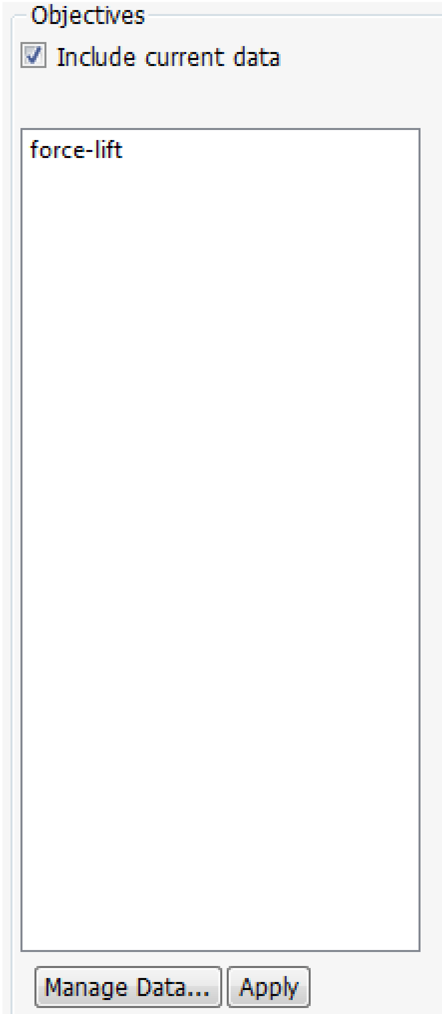
   Design → **Adjoint-Based** → **Design Tool...**

   force-lift is now displayed in the **Design Change** tab because it is the currently selected observable. The **Design Change** tab functions as a dashboard for the design modification, where you can select which boundaries are subject to modification, enable or disable conditions that you have defined, specify relative weighting if you have multiple freeform objectives, and view predicted results. You will return to it to perform the design change after you have configured the objectives and the morphing region.

2. Load the previously saved **force-drag** sensitivity data.

   a. Open the **Objectives** tab.

      The **force-lift** observable is already listed because **Include current data** is enabled.



b. Click **Manage Data...** to open the **Manage Sensitivity Data** dialog box.

c. Click **Import Sensitivities...** and select the **force-drag.s** file you created earlier. Click **OK**.

d. Close the **Manage Sensitivity Data** dialog box.

3. Define the objective for each observable.

   For this example, you will seek a design change that increases the lift and results in a 10% reduction in drag.

   a. In the **Objectives** tab, select the **force-lift** observable. The current value of the lift is displayed along with options to specify the objective for the lift.

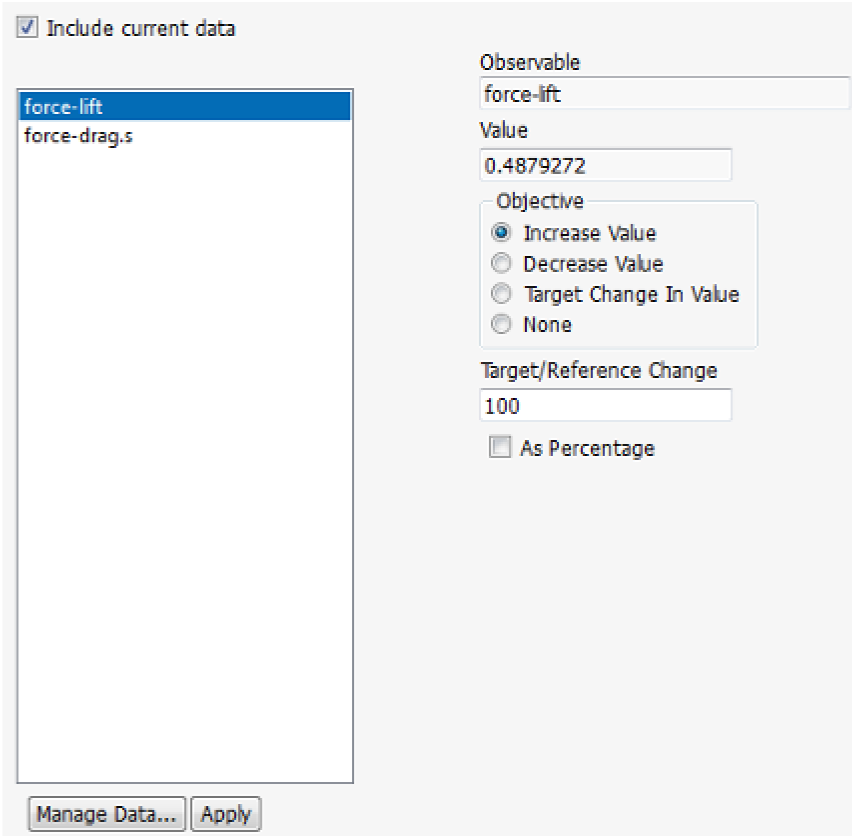
   b. Select **Increase Value** from the **Objective** list.

      This indicates that you want to increase the lift, but are not prescribing a specific target change.

   c. Enter `100` for **Target/Reference Change**.

      This setting is used to normalize the scale of the change in value of the observable, which can be important in cases where multiple observables are considered that may be of different scales.

d. Click **Apply**.



e. Select **force_drag.s** in the list of observables.

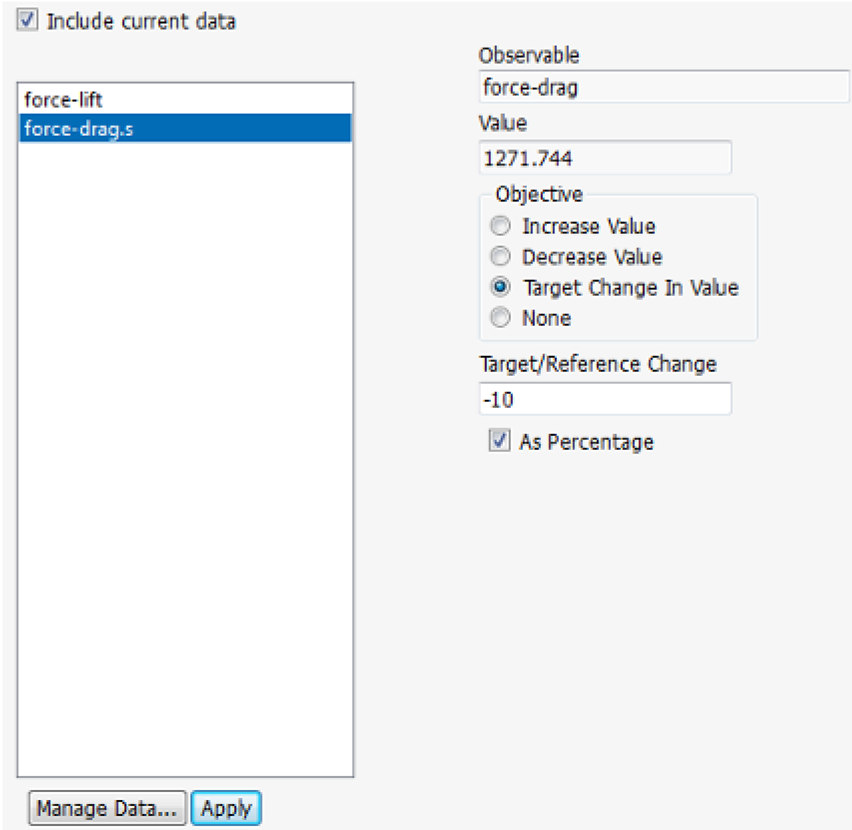
f. Select **Target Change In Value** from the **Objective** list.

This indicates that you are prescribing a specific change in the value of the observable, rather than a freeform increase or decrease.

g. Enter `-10` for **Target/Reference Change** and enable the **As Percentage** option.

10% is a generally a reasonable maximum target change for a design change. Using a target change that is too large may result in very large deformations and/or overshooting the local optimum.

h. Click **Apply**.



4. Configure the morphing region.

   You already specified the dimensions of the region earlier when exporting the **force-drag** sensitivity. Now you will also configure the control-point density.

   a. Click the **Region Conditions** tab in the **Design Tool** dialog box.

   b. Enter `30` for **Points** in the **X Motion** and **Y Motion** group boxes.

   c. Click **Apply**.

      *Many other settings are available in the **Region Conditions** tab, including constraints on control-point motion, symmetry conditions, and continuity conditions. For additional information, see the section on defining region conditions in the Fluent Advanced Add-On Modules manual, available in the help viewer or on the ANSYS Customer Portal (http://support.ansys.com/documentation).*

5. Compute the design change and modify the mesh.

   a. Return to the **Design Change** tab.

   b. Select **wall** in the **Zones To Be Modified** selection list.

      *Only zones that are selected in the **Zones To Be Modified** list (or that have prescribed motions applied) will be modified as part of the design change.*

c. If multiple freeform objectives were defined (that is, multiple objectives with **Increase Value** or **Decrease Value** selected in the **Objectives** tab), you would need to specify the **Weight** for each. In this case only one objective (**force-lift**) is freeform, so no input is required for **Weight**.

d. Retain the default settings of **Control-point spacing** for **Freeform Scaling Scheme**, and **0.1** for **Freeform Scale Factor**.

   *These settings allow you to adjust the magnitude of the attempted design change (**Freeform Scale Factor**) and the basis for the scaling (**Freeform Scaling Scheme**).*

e. Click **Calculate Design Change**.

   *The **Results** list is updated to reflect the **Expected change** for each observable.*

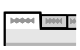
File Name	Observable	Value	Weight	Expected change
force-lift	force-lift	0.4879272	1	127.5843
force-drag.s	force-drag	1271.744	1	-127.1744

   Note that the drag is predicted to decrease by 10% as you requested, and the lift is predicted to increase.

f. Click the **Modify Mesh** button to apply the calculated mesh deformation that will reposition the boundary and interior nodes of the mesh. Information regarding the mesh modification is printed in the console:

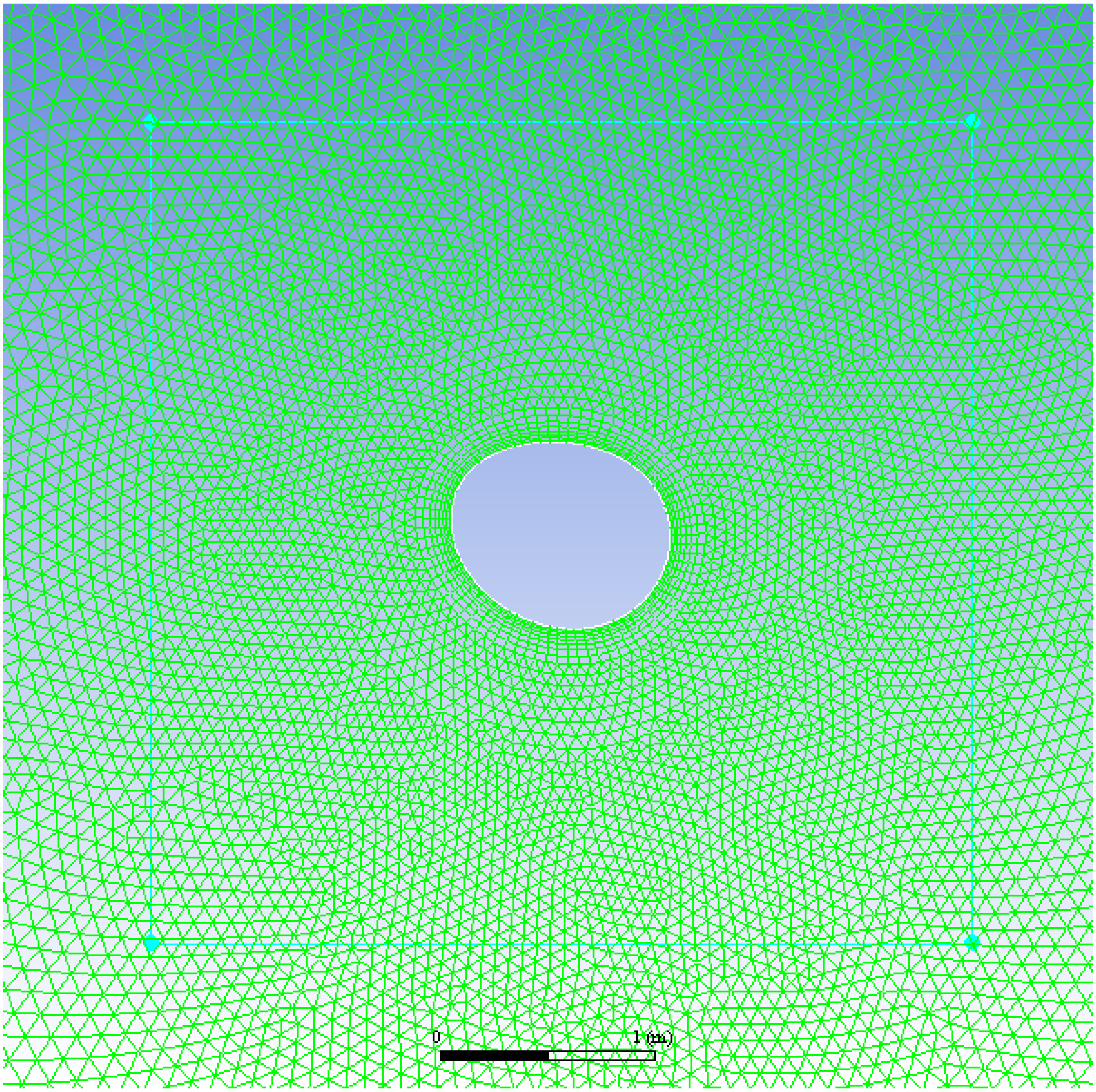
```
Updating mesh (steady, mesh iteration = 00001, pseudo time step 1.0000e+00)...
Dynamic Mesh Statistics:
Minimum Volume    = 3.46260e-04
Maximum Volume    = 6.36270e-01
Maximum Cell Skew = 3.69343e-01 (cell zone 11)
Minimum Orthogonal Quality = 6.30657e-01 (cell zone 11)
```

g. Display the new mesh geometry.

    **Postprocessing → Graphics → Mesh → Edit...**

   The effect on the mesh is shown in Figure 23.19: Mesh After Deformation (p. 958):

Figure 23.19: Mesh After Deformation



h. Re-converge the conventional flow calculation for this new geometry in the **Run Calculation** task page.

Solving → Run Calculation → Calculate

The currently loaded case file already has report definitions defined for lift and drag, or you can **Evaluate** the new values in the **Adjoint Observables** dialog box.

Design → Adjoint-Based → Observable...

The new values for drag and lift are reported to be:

```
Observable name: force-drag
Observable Value (n): 1151.3445
```

```
Observable name: force-lift
Observable Value (n): 122.92812
```

Note that the drag has changed by -120.4 N or -9.5% compared to the drag on the undeformed cylinder. This value compares very well with the change of -127.2 N (-10%) that was predicted from the adjoint solver. The lift has increased by 122.4 N, which again compares very well with the predicted change of 127.6 N.

23.5. Summary

This tutorial has demonstrated how to use the adjoint solver add-on to compute the sensitivity of the drag and lift on a circular cylinder to various inputs for a previously computed flow field. The process of setting up and running the adjoint solver was illustrated. The steps to perform various forms of postprocessing were also described. The design change tool was used to make a multi-objective change to the design that reduced the drag and increased the lift in a predictable manner.

This example considered multiple objectives at a single flow condition. Another powerful application of the design tool is to perform multi-objective design changes using sensitivities computed for multiple flow conditions. This allows you to identify design changes that improve performance across a range of anticipated operating conditions, potentially of differing importance. The design tool also offers a rich set of additional capabilities for including prescribed deformations, bounding planes / surfaces, and fixed-wall constraints in your multi-objective design change. For full details about how to use the design tool, refer to the Fluent Advanced Add-On Modules documentation, available in the help viewer or on the ANSYS Customer Portal (http://support.ansys.com/documentation).

Chapter 24: Simulating a Single Battery Cell Using the MSMD Battery Model

This tutorial is divided into the following sections:

24.1. Introduction

This tutorial is used to show how to set up a battery cell simulation in ANSYS Fluent.

This tutorial demonstrates how to do the following:

- Load the Battery module add-on
- Set up a battery cell simulation using the NTGK battery submodel
- Perform the calculations for different battery discharge rates and compare the results using the postprocessing capabilities of ANSYS Fluent
- Use the reduced order method (ROM) in a battery simulation
- Simulate a battery pulse discharge
- Introduce external and internal short-circuits in a battery simulation

24.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

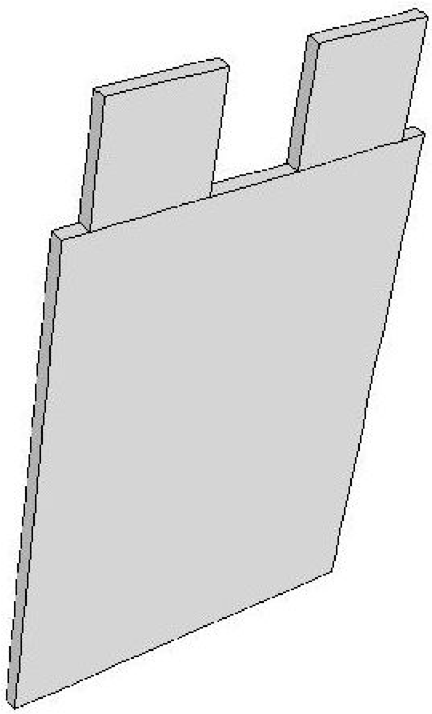
- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

and that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

24.3. Problem Description

The discharge behavior of a lithium-ion battery described in the Kim's paper [2] will be modeled in this tutorial. You will use the NTGK model. The battery is a 14.6 Ah $LiMn_2O_4$ cathode/graphite anode battery. The geometry of the battery cell is shown in Figure 24.1: Schematic of the Battery Cell Problem (p. 962). You will study the battery's behavior at different discharge rates.

Figure 24.1: Schematic of the Battery Cell Problem



For external and internal short-circuit treatment, you will consider an extreme case where external and internal short-circuits occur at the same time. You will simulate post-short-circuit battery processes. You can assume that the internal short is caused by a nail penetration occurring near the center of the battery.

24.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

24.4.1. Preparation

1. Set up a working folder on the computer you will be using.

2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

> **Note**
>
> If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

3. Enter the name of this tutorial into the search bar.

4. Narrow the results by using the filter on the left side of the page.

    a. Click **ANSYS Fluent** under **Product**.

    b. Click **18.0** under **Version**.

5. Select this tutorial from the list.

6. Click the **battery_cell_R180.zip** link to download the input files.

7. Unzip `battery_cell_R180.zip`.

    The input file `unit_battery.msh`, `unit_battery.cas`, and `unit_battery.dat` can be found in the `battery_cell` folder created after unzipping the file.

8. Use Fluent Launcher to start the **3D** version of ANSYS Fluent.

9. Enable **Double Precision**.

10. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.

11. Select either **Serial** or **Parallel** under **Processing Options**. In this tutorial, **Serial** is selected for demonstration only.

24.4.2. Reading and Scaling the Mesh

1. Read the mesh file `unit_battery.msh`.

    File → Read → Mesh...

    When prompted, browse to the location of the `unit_battery.msh` and select the file.

    Once you read in the mesh, it is displayed in the embedded graphics windows.

    The geometry is already in the correct scale. You don't need to scale it.

2. Check the mesh.

    Setting Up Domain → Mesh → Check

24.4.3. Loading the MSMD battery Add-on

1. To load the MSMD battery add-on into ANSYS Fluent, type the following in the console window:

```
define/models/addon-module
```

A list of ANSYS Fluent add-on modules is displayed.

```
Fluent Addon Modules:
 0. none
 1. MHD Model
 2. Fiber Model
 3. Fuel Cell and Electrolysis Model
 4. SOFC Model with Unresolved Electrolyte
 5. Population Balance Model
 6. Adjoint Solver
 7. Single-Potential Battery Model
 8. Dual-Potential MSMD Battery Model
 9. PEM Fuel Cell Model
10. Macroscopic Particle Model
Enter Module Number: [0] 8
```

2. Enter 8 to load the MSMD battery itself.

   During the loading process, a scheme library containing the graphical and text user interface, and a library of user-defined functions (UDFs) containing a set of UDFs for the battery module are loaded into ANSYS Fluent. Fluent reports the progress in the console.

   Once the MSMD battery add-on is loaded, **MSMD Battery Model** appears in the **Models** task page and under the **Models** tree branch. The UDF library also becomes visible as a new entry in the UDF Library Manager dialog box.

24.4.4. NTGK Battery Model Setup

The following sections describe the setup steps for this tutorial:

24.4.4.1. Specifying Solver and Models

1. In the **Solver** group of the **Setting Up Physics** ribbon tab, enable a time-dependent calculation.

   Setting Up Physics → **Solver** → **Transient**

2. To solve for the temperature field, enable the **Energy** equation (in the **Models** group).

   Setting Up Physics → **Models** → **Energy**

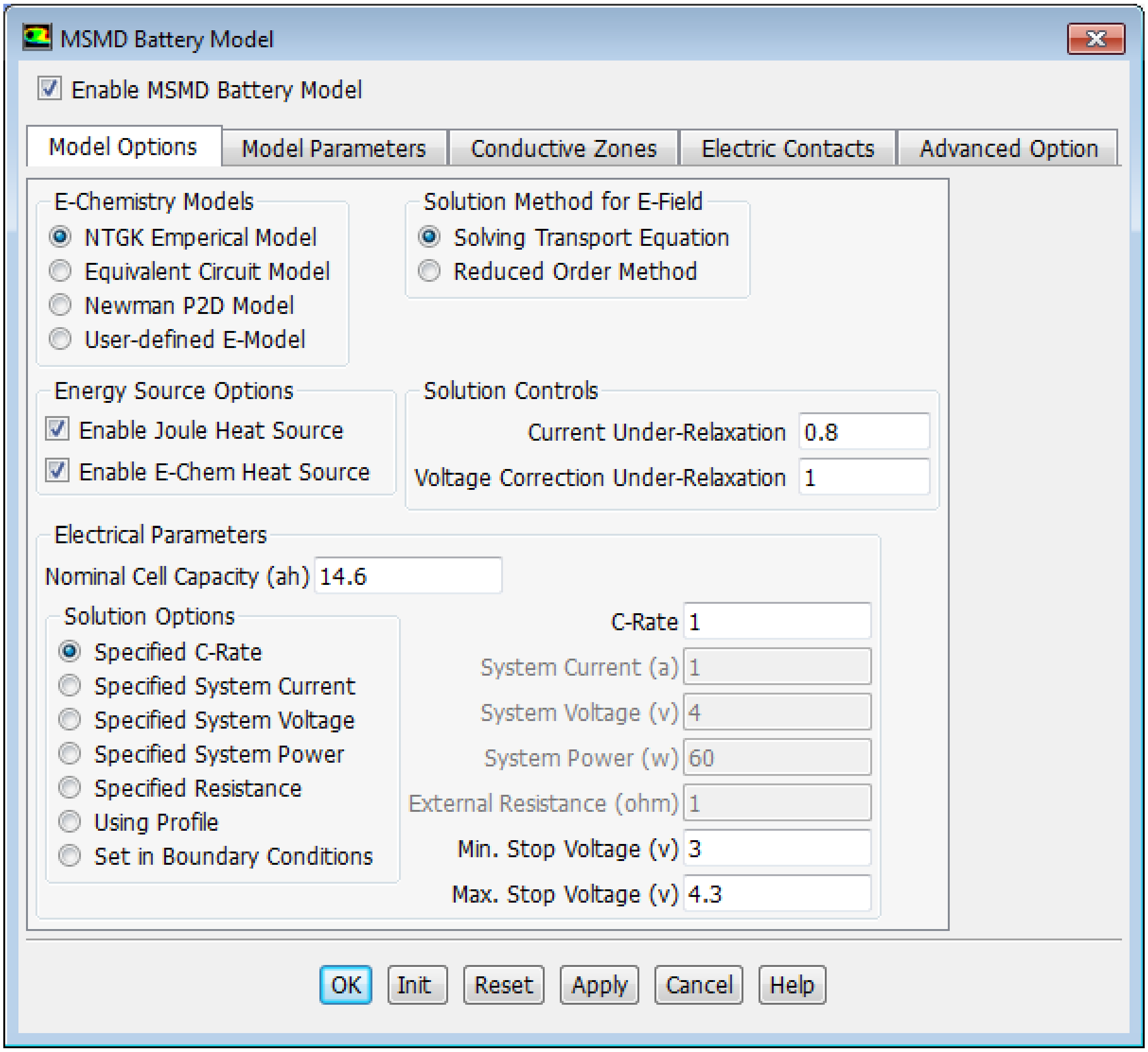
3. Enable the battery model.

   Setting Up Physics → **Models** → **More** → **Dual Potential MSMD Battery Model**

   a. In the **MSMD Battery Model** dialog box, select **Enable MSMD Battery Model**.

      The dialog box expands to display the battery model's settings.

Figure 24.2: Model Options



b. Under the **Model Options** tab (Figure 24.2: Model Options (p. 965)), configure the following battery operation conditions:

   i. Under **E-Chemistry Models**, retain the default selection of **NTGK Empirical Model**.

   ii. Ensure that **Solving Transport Equation** is selected for **Solution Method for E-field**.

   iii. Under **Electrical Parameters**, retain the default value of **14.6 Ah** for **Nominal Cell Capacity**.

   iv. Retain the default selection of **Specified C-Rate** and the value of **1** for **C-Rate**.

   v. Retain the default value of **3 V** for **Min. Stop Voltage**.

c. Under the **Model Parameters** tab, retain the default settings for Y and U coefficients. For details, see the Fluent Advanced Add-On Modules documentation on the ANSYS Customer Portal (http://support.ansys.com/documentation).

Note

- If in your case, Y and U functions are not in the same function form as in the Kim's paper, you need to modify the `cae_user.c` source code file. See the Fluent Advanced Add-On Modules documentation on the ANSYS Customer Portal (http://support.ansys.com/documentation) for details.

- For a given battery, you can perform a set of constant current discharging tests, and then use the battery's parameter estimation tool to obtain the Y and U functions. See the Fluent Advanced Add-On Modules documentation on the ANSYS Customer Portal (http://support.ansys.com/documentation) for details.

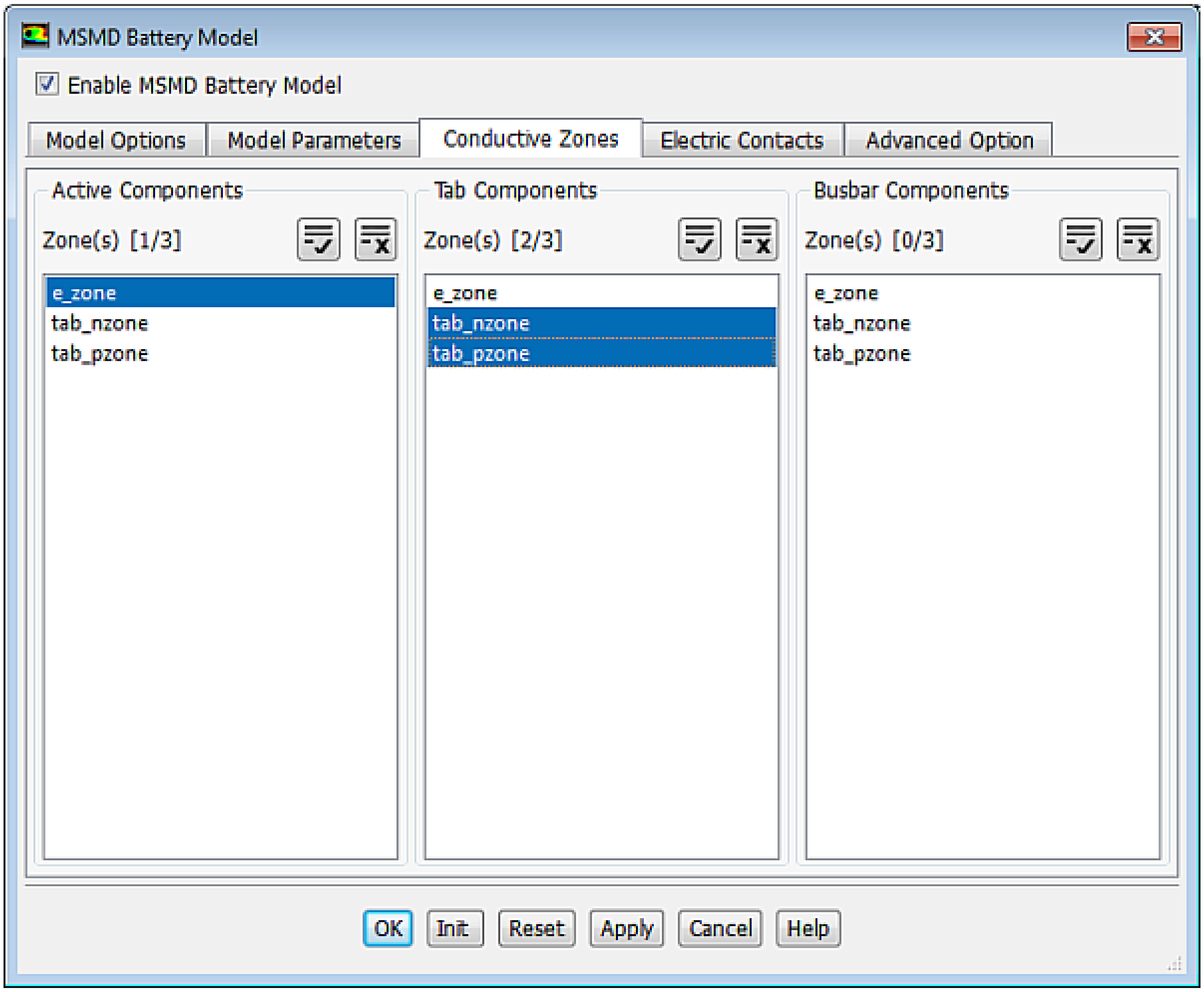
---

d. Under the **Conductive Zones** tab (Figure 24.3: Conductive Zones (p. 967)), configure the following settings:

Group	Control or List	Value or Selection
Active Components	Zone (s)	e-zone
Tab Components	Zone (s)	tab_nzone
		tab_pzone

For this single cell case, there are no busbar zones. Electro-chemical reactions occur only in the active zone. Battery tabs are usually modeled as passive zones, in which the potential field is also solved.

Figure 24.3: Conductive Zones



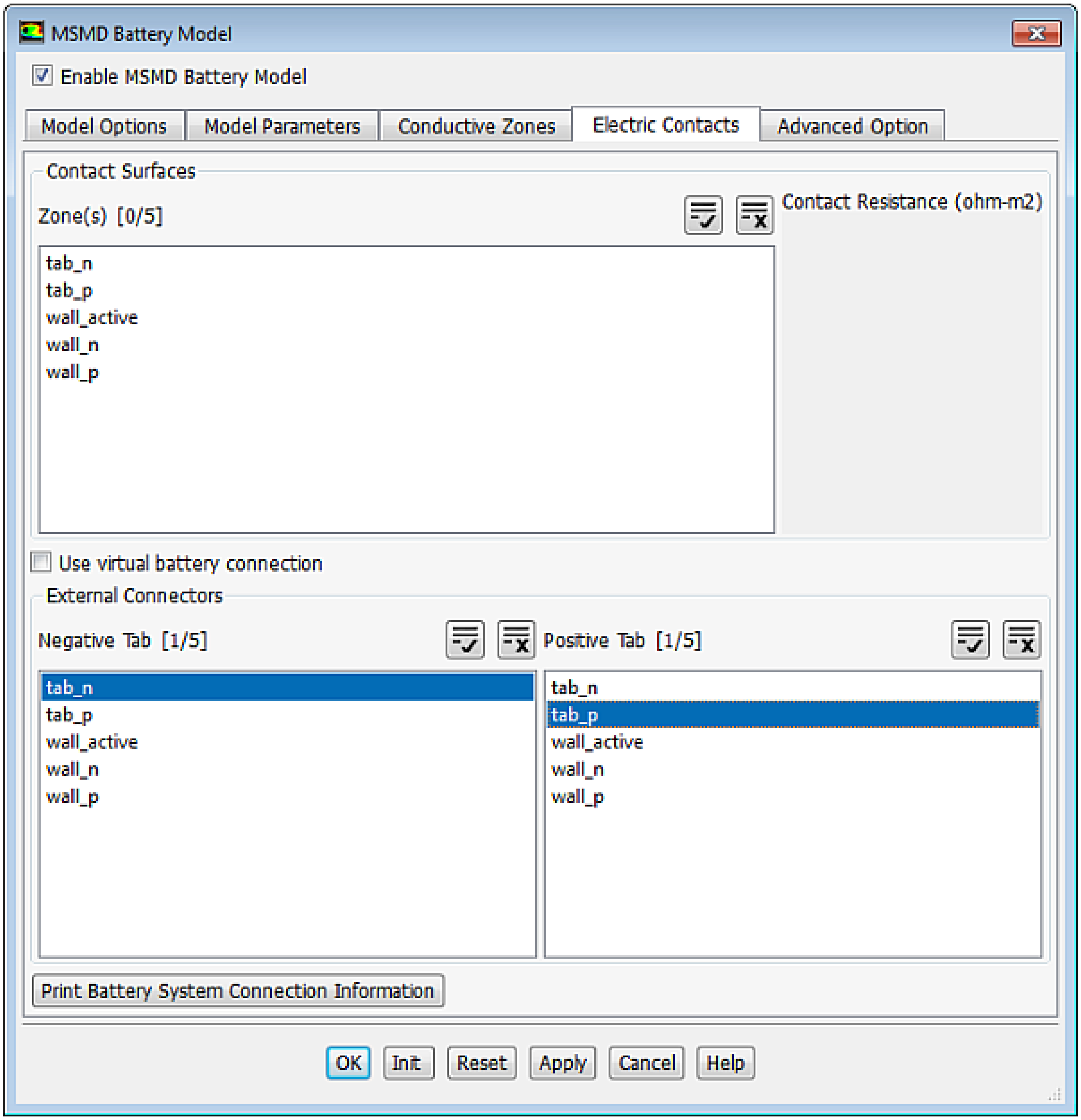
e. Under the **Electric Contacts** tab (Figure 24.4: Electric Contacts (p. 968)), configure the contact surface and external connector settings as follows:

Group	Control or List	Value or Selection
External Connectors	Negative Tap	tab_n
	Positive Tap	tab_p

The corresponding current or voltage boundary condition will be applied to those boundaries automatically.

Under the **Electric Contacts** tab, you can also define extra contact resistance for each zone.

Figure 24.4: Electric Contacts



f. Click the **Print Battery System Connection Information** button.

ANSYS Fluent prints the battery connection information in the console window:

```
Battery Network Zone Information:
-------------------------------------
 Battery Serial 1
   Parallel 1
        N-Tab  zone:  tab_nzone
        Active zone:  e_zone
        P-Tab  zone:  tab_pzone
-----------------------------------


 Active zone list at  odd level: e_zone
 Active zone list at even level:
```

```
Number of battery series stages =1; Number of batteries in parallel per series stage=1
****************END OF BATTERY CONNECTION INFO**************
```

g. Verify that the connection information is correct.

h. Click **OK** to close the **MSMD Battery Model** dialog box.

   In the background, Fluent automatically hooks all the necessary UDFs for the problem.

24.4.4.2. Defining New Materials for Cell and Tabs

Define the new e_material material for the battery's cell, p_material for the positive tab, and n_material for the negative tab.

In the battery model, two user defined scalars, uds0 and uds1, are solved for the positive and negative potentials, respectively. To specify the electric conductivity of the active material you need to define the UDS diffusivity.

Important

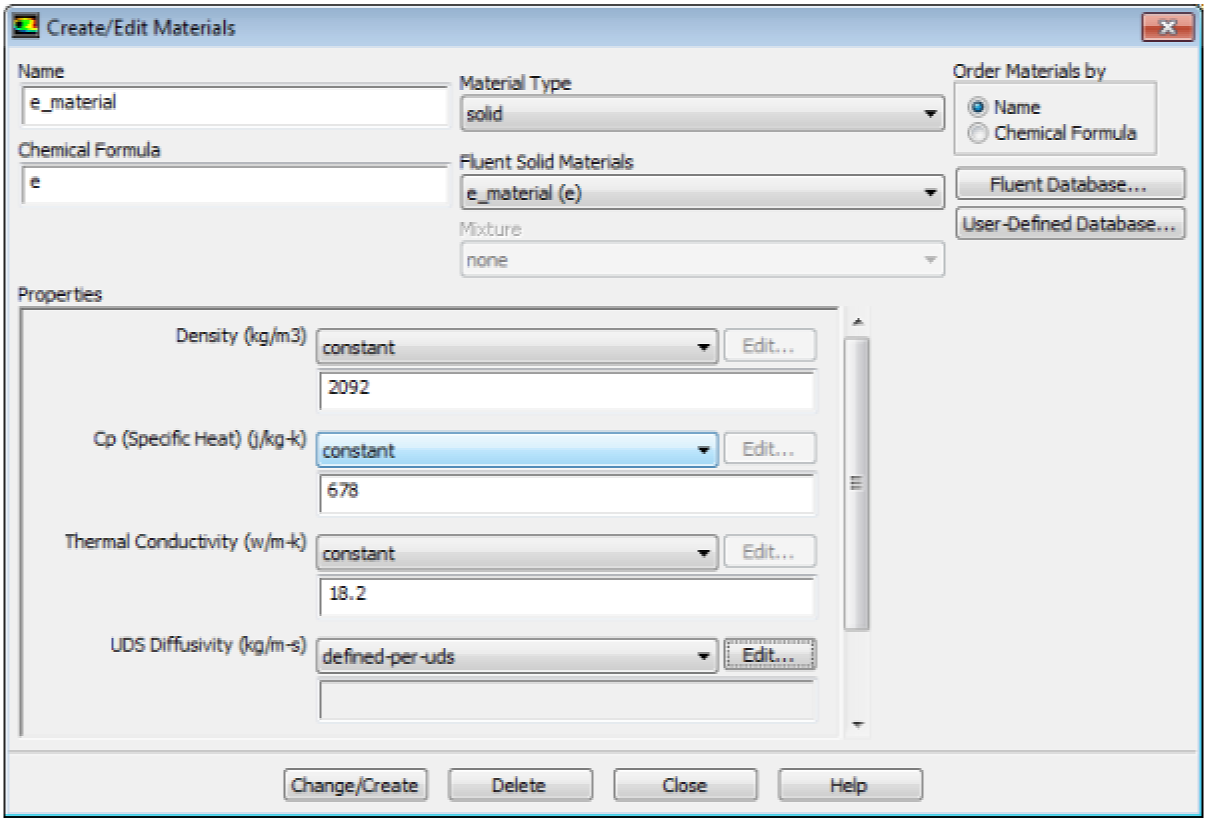
For the battery active material, you must define the electric conductivity via the UDS diffusivity on a per-scalar basis. ANSYS Fluent will use these two UDS scalars to solve the differential equations; for details, see the Fluent Advanced Add-On Modules documentation on the ANSYS Customer Portal (http://support.ansys.com/documentation).

For the battery passive conductive zones, such as tabs or busbars, you must define the UDS diffusivity using a user-defined function (UDF) and define the material's electric conductivity using the **Electrical Conductivity** entry field.

---

1. Create the electric material.

   Setting Up Physics → Materials → Create/Edit...



a. In the **Create/Edit Materials** dialog box, select **solid** from the **Material Type** drop-down list.

b. Enter **`e_material`** for **Name** and **e** for **Chemical Formula**.

c. Under **Properties**, set **Density** to **`2092`** [kg/m^3].

d. Set **CP** (**Specific Heat**) to **`678`** [J/kg-K].

e. Set **Thermal Conductivity** to **`18.2`** [W/m-K].

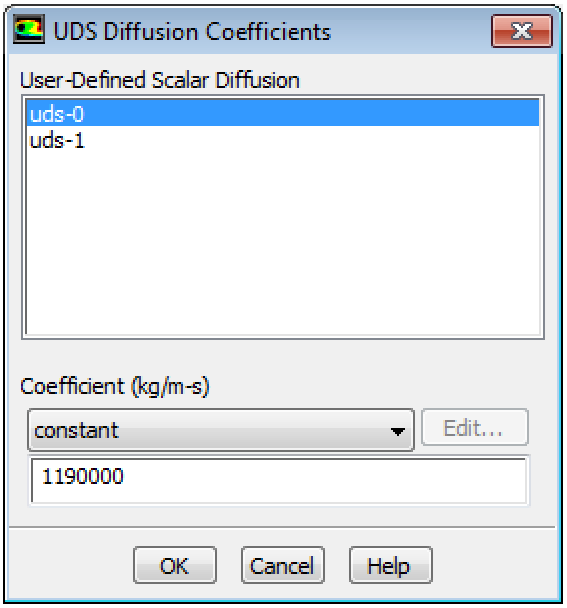
f. Ensure that **define-per-uds** is selected from the **UDS Diffusivity** drop-down list and click **Edit...** next to **UDS Diffusivity**.

Note

If the UDS Diffusion Coefficients are defined through the `defined-per-uds` option, the Fluent solver does not use the value for **Electrical Conductivity**.

---

g. In the **UDS Diffusion Coefficients** dialog box, specify the user-defined scalars.



i. Select **uds-0** in the **User-Defined Scalar Diffusion** list.

ii. Retain **constant** from the **Coefficient** drop-down list.

iii. Set **Coefficient** to `1.19e6` [1/ohm-m].

iv. In a similar way, set **uds-1** to `9.83e5` [1/ohm-m] and close the **UDS Diffusion Coefficients** dialog box.

Note

The units for UDS Diffusivity are `1/ohm-m`, and cannot be modified.

---

v. In the **Question** dialog box, click **No** to retain **aluminum** and add the new material (e_material) to the materials list.

Note

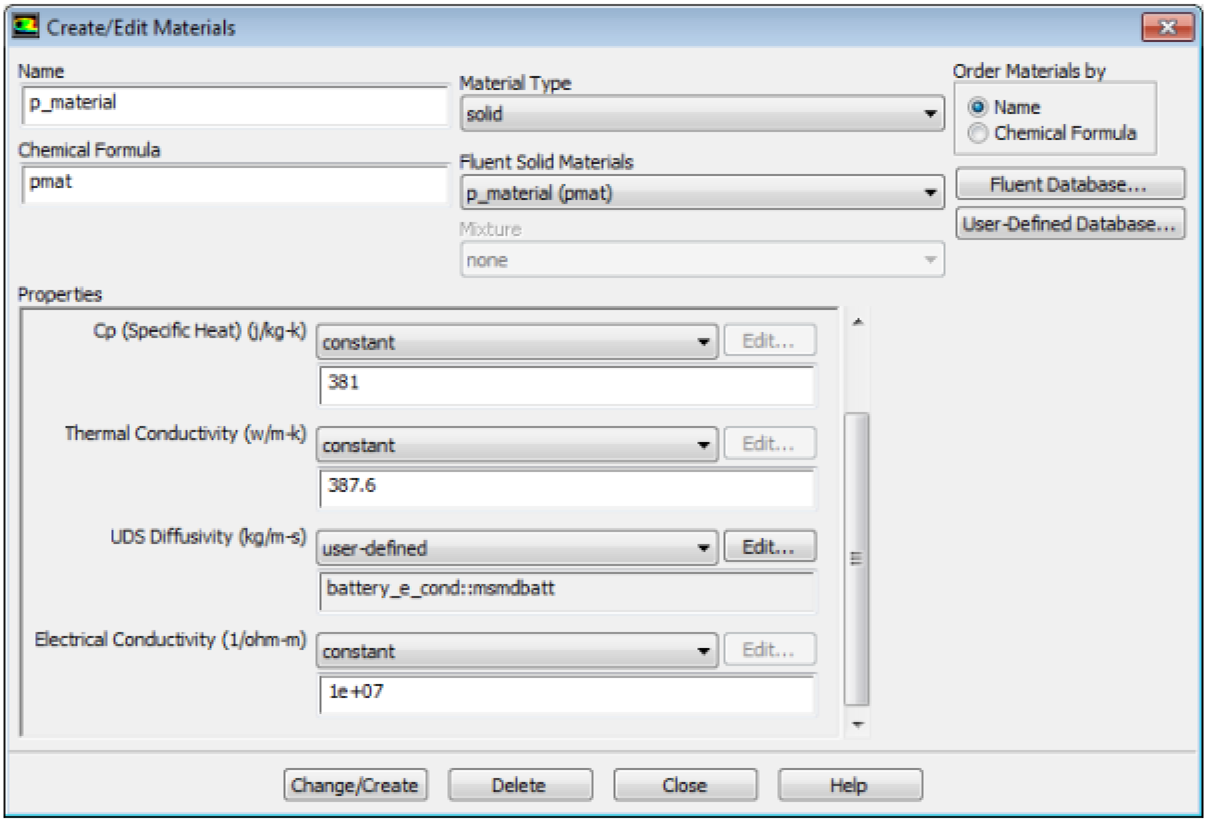
Refer to Appendix (p. 1001) for information on how to calculate the battery cell property values.

---

h. Ensure that `e_material (e)` is selected from the **Fluent Solid Materials** drop-down list.

i. Click **Change/Create**.

2. Create a new material for the positive tab by modifying copper from the solid material database.



The UDS Diffusivity for tab materials must be defined through the UDF as described below.

a. In the **Create/Edit Materials** dialog box, click **Fluent Database...**.

b. In the **Fluent Database Materials** dialog box, make sure that **solid** is selected for **Material Type**.

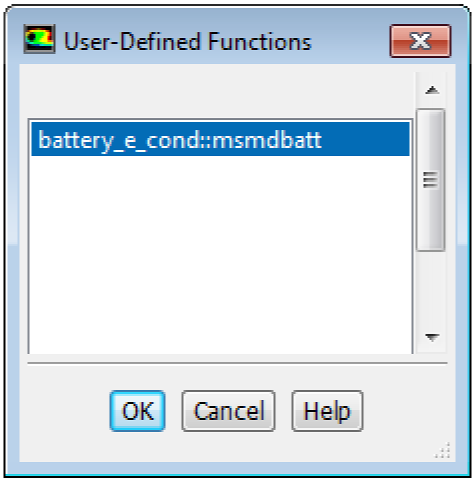
c. Select `copper` from **Fluent Solids Materials** and click **Copy** and then **Close**.

   The **Create/Edit Materials** dialog box now displays the copied properties for `copper`.

d. Enter **`p_material`** for **Name** and **`pmat`** for **Chemical Formula**.

e. Under **Properties**, select **user-defined** from the **UDS Diffusivity** drop-down list.

f. In the **User-Defined Functions** dialog box that opens, verify that **battery_e_cond::msmdbatt** is selected and click **OK**.



Fluent hooks the battery_e_cond::msmdbatt UDF to the busbar material.

g. In the **Question** dialog box, click **Yes** to overwrite **copper**.

   The new `material (p_material)` appears under **Materials**.

h. Set the **Electrical Conductivity** to **`1.0e7`** [1/ohm-m].

i. Click **Change/Create**.

3. Create a new material for the negative tab with the same properties as the material for the positive tab.

Note

You do not need to create two different materials for the positive and negative tabs if the positive and negative tabs are made of the same material. In this tutorial, the two different tab materials with the same physical properties have been created for demonstration purposes only.

---

a. From **Fluent Solid Materials** drop-down list, select `p_material`.

b. Enter **`n_material`** for **Name** and **`nmat`** for **Chemical Formula**.

c. Click **Change/Create**.

d. In the **Question** dialog box, click **No** to retain p_material and add the new material (n_material) to the materials list.

e. Close the **Create/Edit Materials** dialog box.

24.4.4.3. Defining Cell Zone Conditions

Assign e_material to the cell zone, p_material to the positive tab and n_material to the negative tab.

1. Assign e_material to the e_zone zone.

   Setup → **Cell Zone Conditions** → **e_zone** → **Edit...**

a. In the **Solid** dialog box, select `e_material` from the **Material Name** drop-down list.

b. Click **OK**.

2. In a similar manner, assign p_material to tab_pzone and n_material to tab_nzone.

24.4.4.4. Defining Boundary Conditions

Define the thermal boundary conditions for all walls for the cell, and positive and negative tabs.

1. Set the convection boundary condition for wall_active.

   Setup → **Boundary Conditions** → **wall_active** → **Edit...**

   a. In the **Wall** dialog box, under the **Thermal** tab, under **Thermal Conditions**, enable **Convection**.

   b. Set **Heat Transfer Coefficient** to **5** `[w/m²K]`.

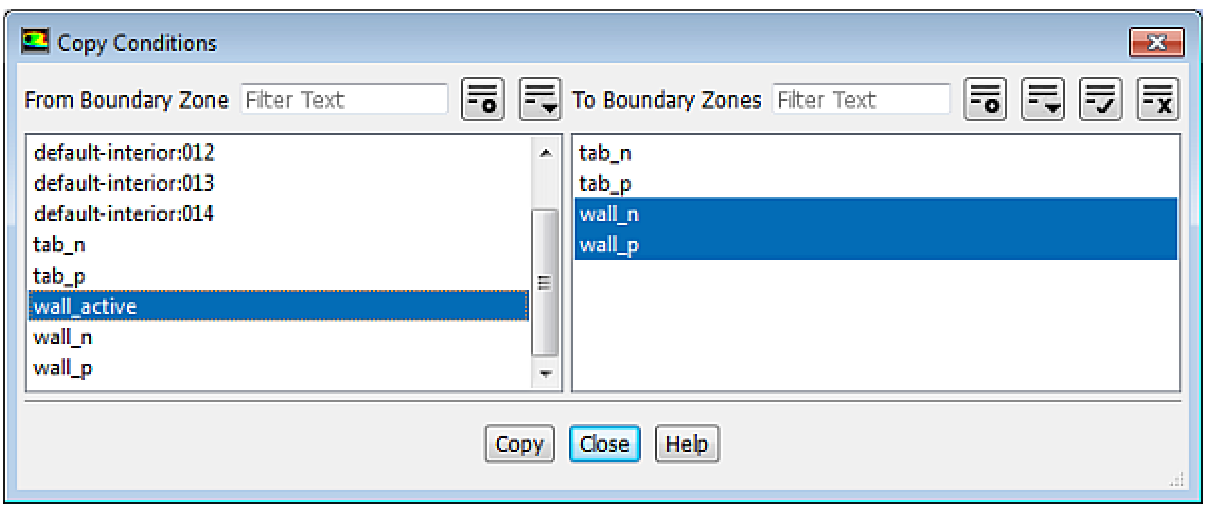
   c. Retain the default value of **300** `[K]` for **Free Stream Temperature**.

   d. Click **OK** to close the **Wall** dialog box.

   You do not need to change the settings under the UDS tab since the boundary conditions for the two UDS scalars have been set automatically when you defined the cell zone conditions.

2. Copy the boundary conditions for `wall_active` to `wall_p` and `wall_n`.

   Setup → **Boundary Conditions** → **wall_active** → **Copy...**



24.4.4.5. Specifying Solution Settings

1. Turn off the flow equation.

   Solving → **Controls** → **Equations...**

   a. In the **Equations** dialog box, deselect `Flow` from the **Equation** selection list.

   b. Click **OK**.

2. Remove the convergence criteria to ensure that automatic convergence checking does not occur.

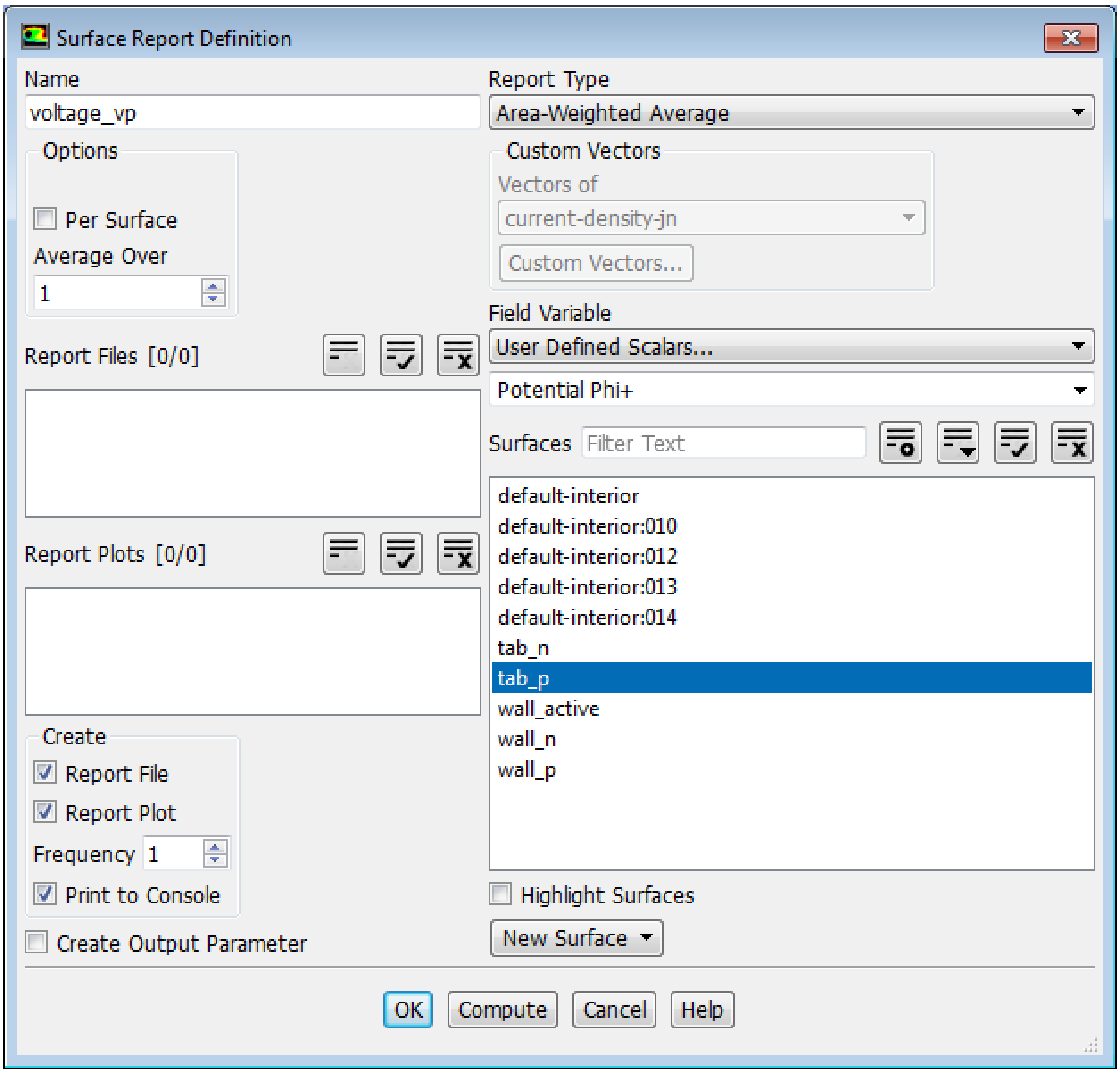
   Solving → **Reports** → **Residuals...**

   a. In the **Residual Monitors** dialog box, select **none** from the **Convergence Criterion** drop-down list.

   b. Click **OK**.

3. Create a surface report definition for the voltage at the positive tab.

   Solving → **Reports** → **Definitions** → **New** → **Surface Report** → **Area-Weighted Average**



   a. In the **Surface Report Definition** dialog box, enter `voltage_vp` for **Name**.

   b. Select **User Defined Scalars...** and **Potential Phi+** from the **Field Variable** drop-down lists.

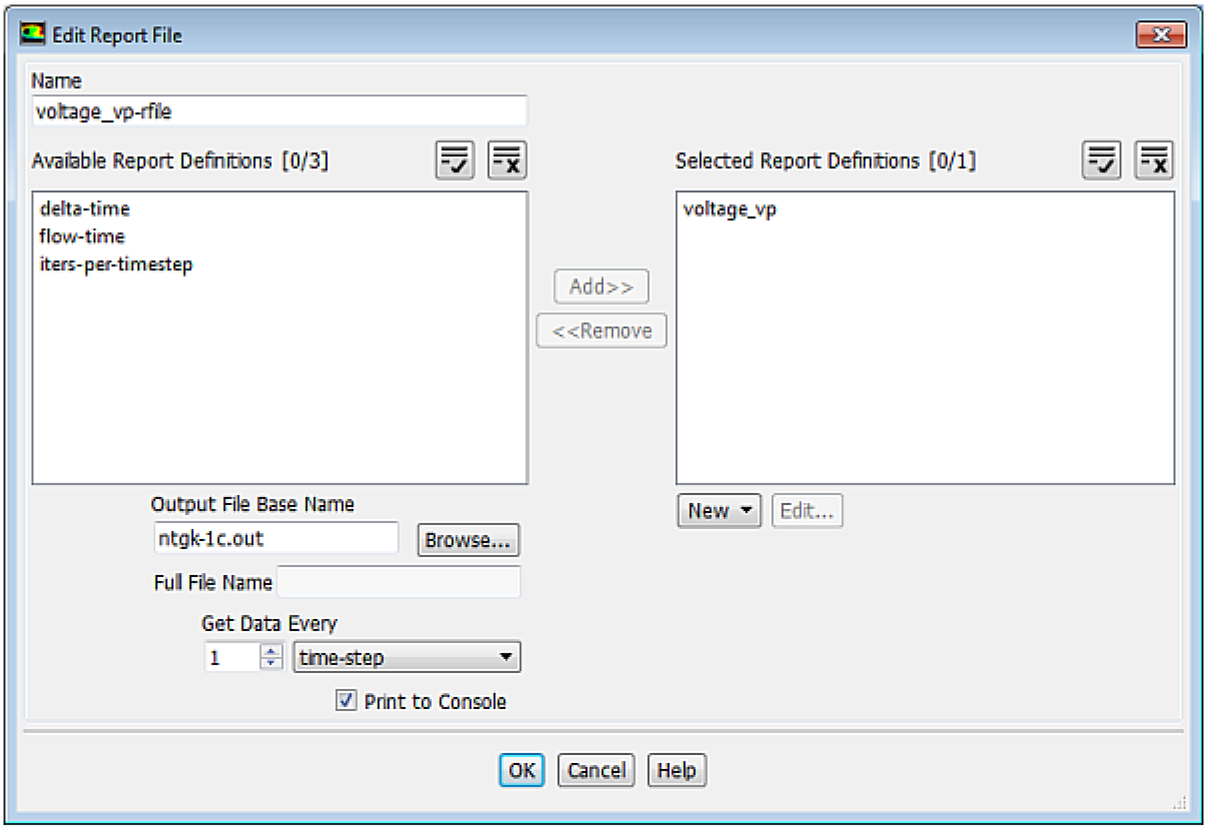
   c. From the **Surfaces** selection list, select **tab_p**.

d. In the **Create** group box, enable **Report File**, **Report Plot** and **Print to Console**.

e. Click **OK** to save the **voltage_vp** report definition and close the **Surface Report Definition** dialog box.

f. Rename the report output file.

Solution → **Monitors** → **Report Files** → **voltage_vp-rfile** **Edit...**



i. Enter `ntgk-1c.out` for **Output File Base Name**.

ii. Click **OK** to close the **Edit Report File** dialog box.

g. Modify the attributes of the plot axes.

Solution → **Monitors** → **Report Plots** → **voltage_vp-rplot** **Edit...**

i. In the **Edit Report Plot** dialog box, under the **Plot Window** group box, click the **Axes...** button to open the **Axes** dialog box.

ii. Select the **X** axis and set **Precision** to `0`.

iii. Click **Apply**.

iv. Select the **Y** axis and set **Precision** to **2**.

v. Set **Precision** to 2.

vi. Click **Apply** and close the **Axes** dialog box.

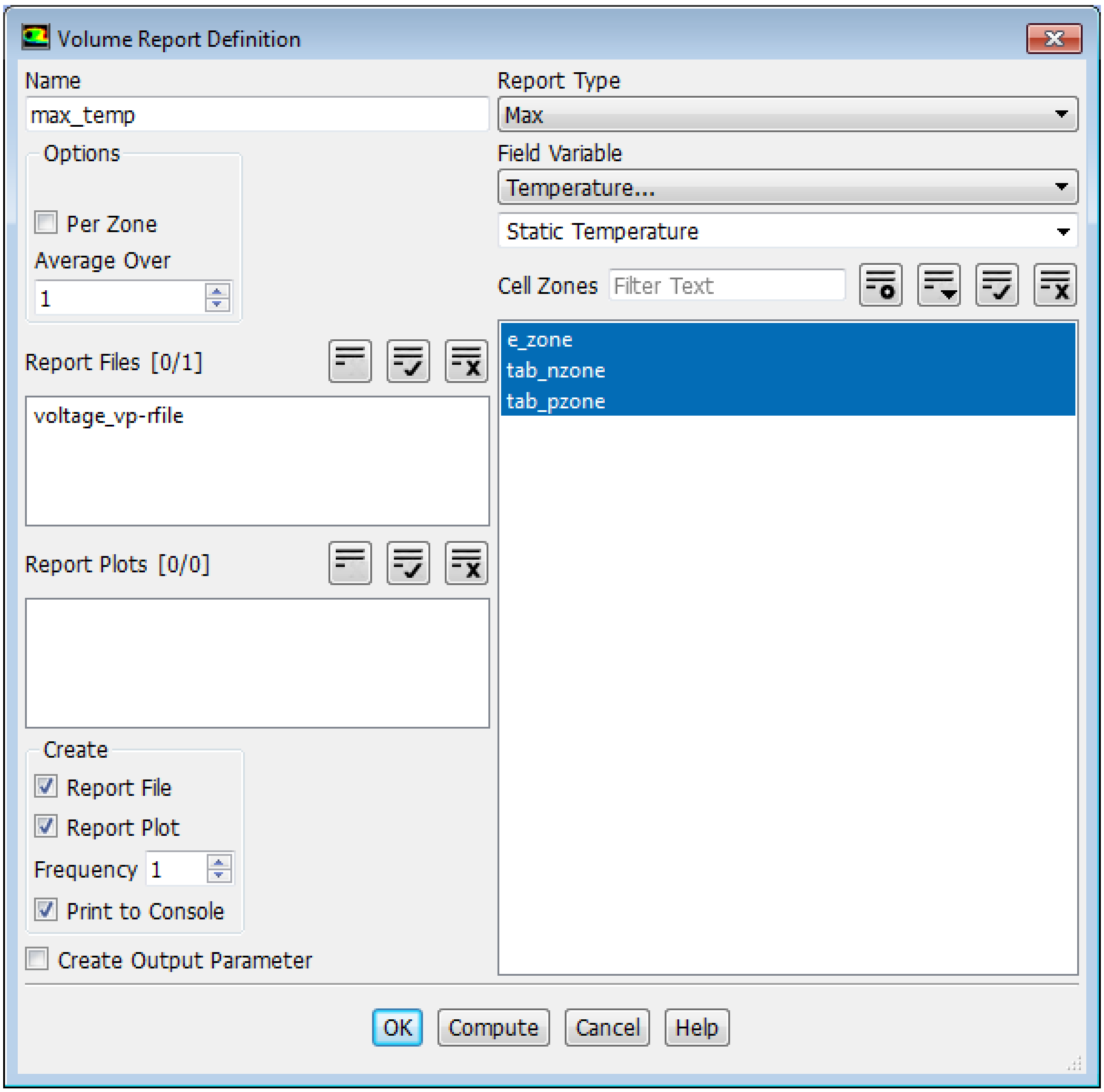
> **Note**
>
> You must click **Apply** to save the modified settings for each axis.

vii. Make sure that **time-step** is selected from the **Get Data Every** drop-down list.

viii. Click OK to close the **Edit Report Plot** dialog box.

4. Create a volume report definition for the maximum temperature in the domain.

Solving → **Reports** → **Definitions** → **New** → **Volume Report** → **Max...**



a. In the **Volume Report Definition** dialog box, enter `max_temp` for **Name**.

b. Select **Temperature...** and **Static Temperature** from the **Field Variable** drop-down lists.

c. From the **Cell Zones** selection list, select all zones.

d. In the **Create** group box, enable **Report File**, **Report Plot** and **Print to Console**.

e. Click **OK** to save the volume report definition settings and close the **Volume Report Definition** dialog box.

f. Rename the report output file.

Solution → **Monitors** → **Report Files** → **max_temp-rfile** **Edit...**

i. Enter `max-temp-1c.out` for **Output File Base Name**.

ii. Click **OK** to close the **Edit Report File** dialog box.

g. Modify the axis attributes by setting the **Precision** to `0` for the **X** axis and to **2** for the **Y** axes (in a manner similar to the surface plot definition).

h. Click **OK**.

5. Save the case file.

File → **Write** → **Case...**

24.4.4.6. Obtaining Solution

1. Initialize the field variables using the **Standard Initialization** method.

Solving → **Initialization**

a. Retain the selection of the **Standard** method (**Initialization** group).

b. Click **Initialize**.

You do not need to modify **Initial Values** in the **Solution Initialization** task page, because these values are not used for initialization. The ANSYS Fluent solver automatically computes the initial condition for UDS0 and UDS1.

2. Run the simulation.

Solving → **Run Calculation**

a. Set **Time Step Size** to `30` seconds and **No. of Time Steps** to `100`.

b. Click **Calculate**.

The residual plot, the report for voltage at the positive tap and the history of the maximum temperature in the domain are shown in Figure 24.5: Residual History of the Simulation (p. 979), Figure 24.6: Report Plot of Discharge Curve at 1 C (p. 979), and Figure 24.7: History of Maximum Temperature in the Domain (p. 980), respectively.

Figure 24.5: Residual History of the Simulation

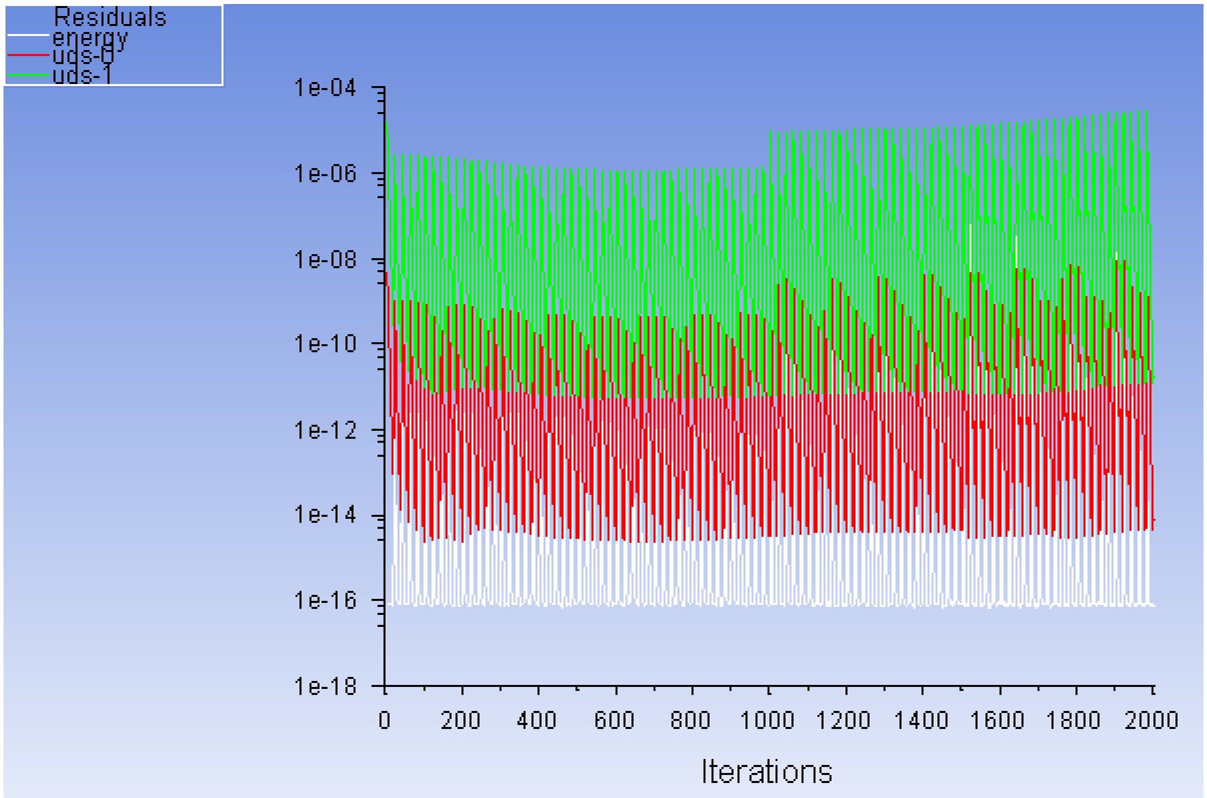


Figure 24.6: Report Plot of Discharge Curve at 1 C

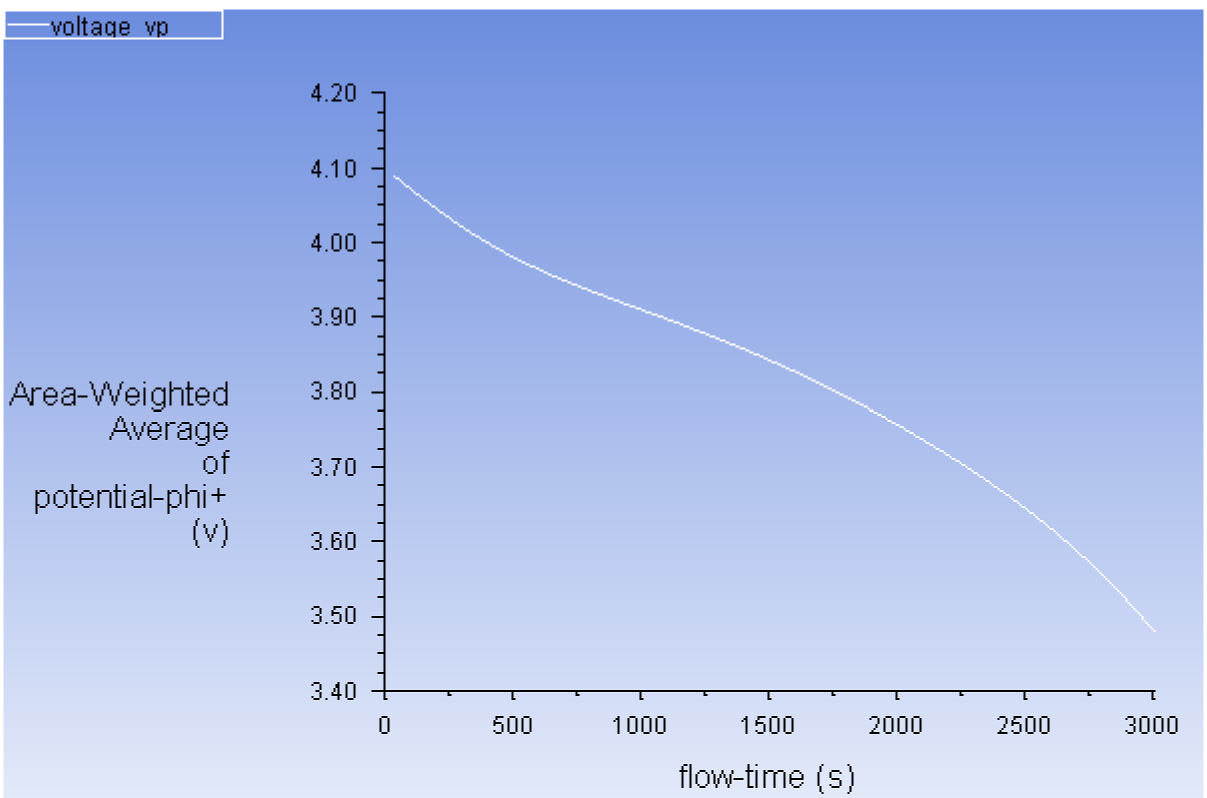
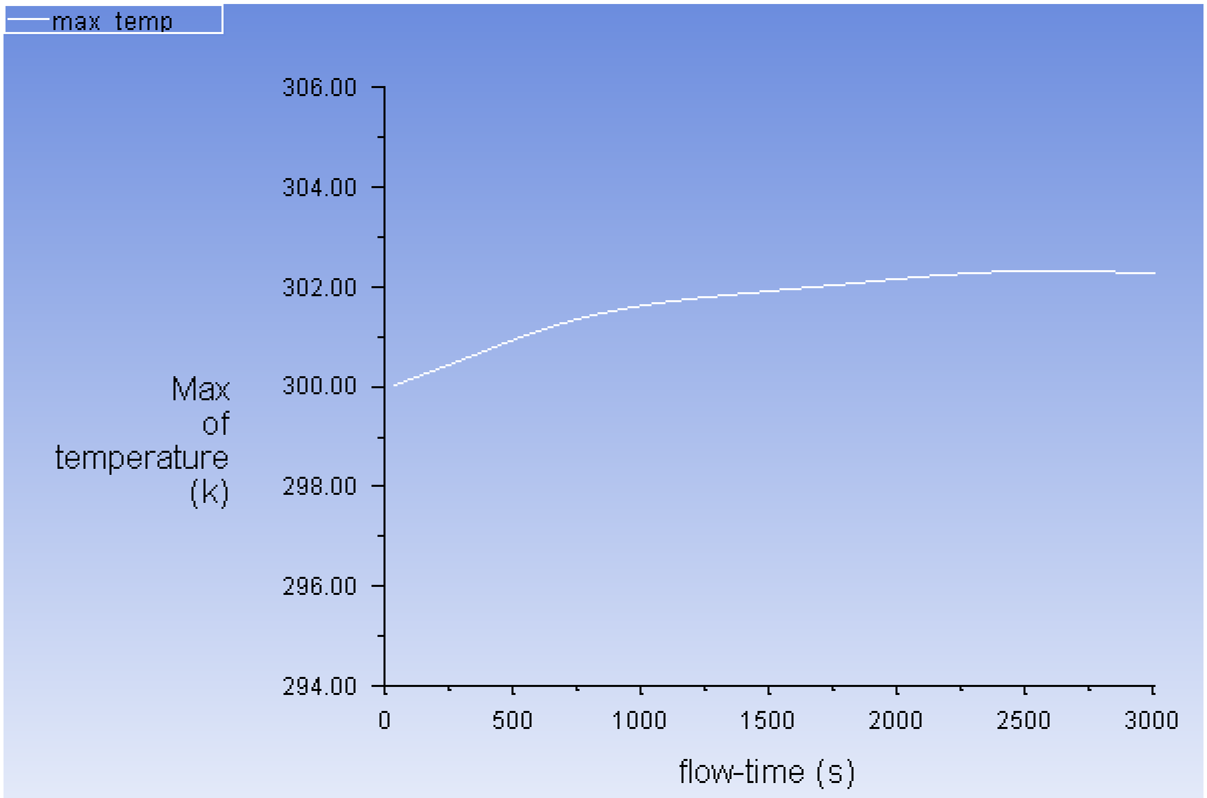


Figure 24.7: History of Maximum Temperature in the Domain



3. Save the case and data files (`unit_battery.cas.gz` and `unit_battery.cas.gz`).

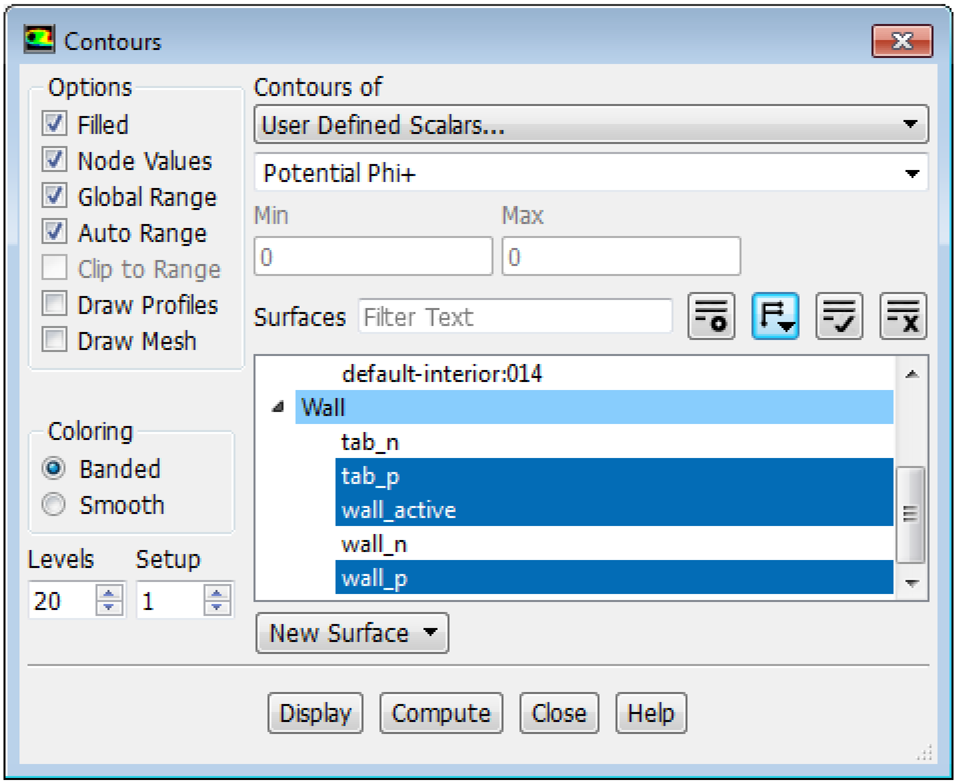
   File → **Write** → **Case & Data...**

24.4.5. Postprocessing

In this section, postprocessing capabilities for the MSMD battery model solution are demonstrated.

1. Display the contour plot of the phase potential for the positive electrode.

   Postprocessing → **Graphics** → **Contours** → **Edit...**



a. In the **Contours** dialog box, in the **Options** group box, enable **Filled**.

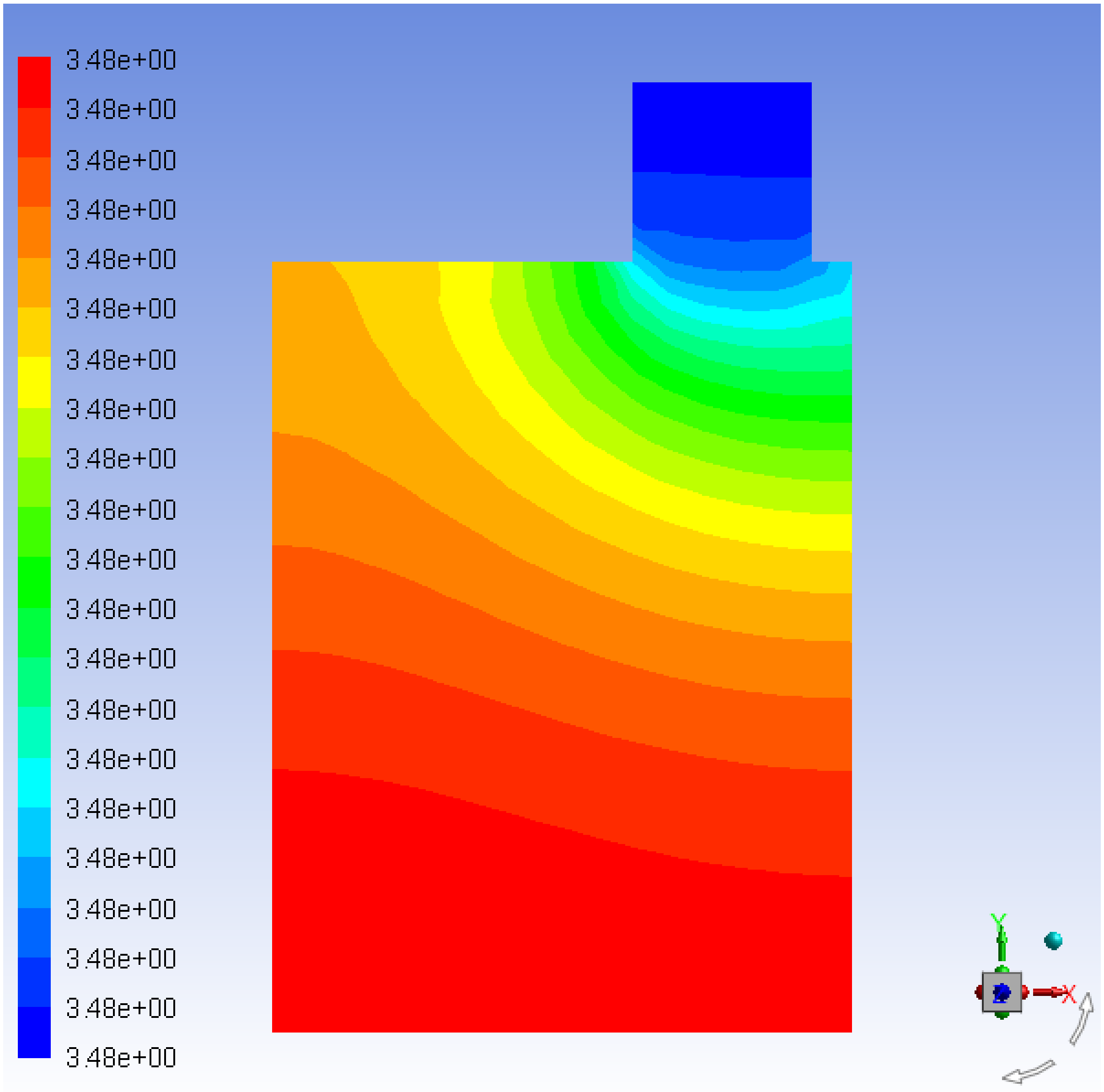
b. From the **Contours of** drop-down list, select **User Defined Scalars...** and **Potential Phi+**.

c. Click the **Toggle Tree View** button next to the **Surfaces** filter and from the drop-down list, select **Surface Type** (under **Group by**).

d. From the **Surfaces** selection list, under **Wall**, select **tab_p**, **wall_active**, and **wall_p**.

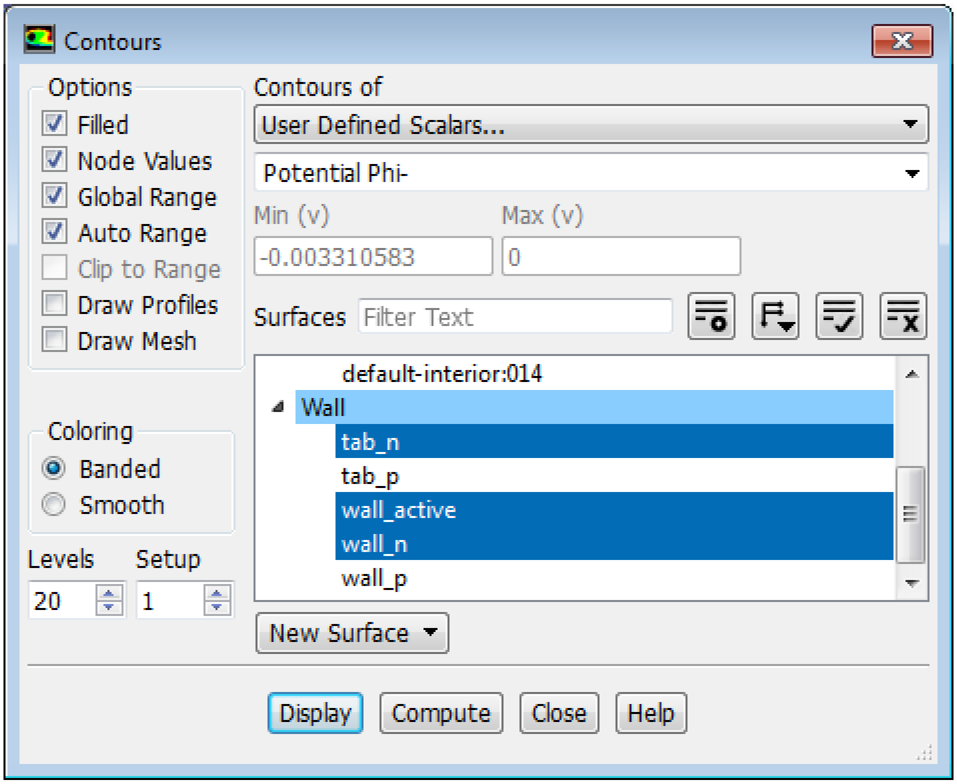
e. Click **Display** and close the **Contours** dialog box.

Figure 24.8: Contour Plot of Phase Potential for the Positive Electrode



2. In a similar manner, display the contour plot of the phase potential for the negative electrode.

   Postprocessing → **Graphics** → **Contours** → **Edit...**



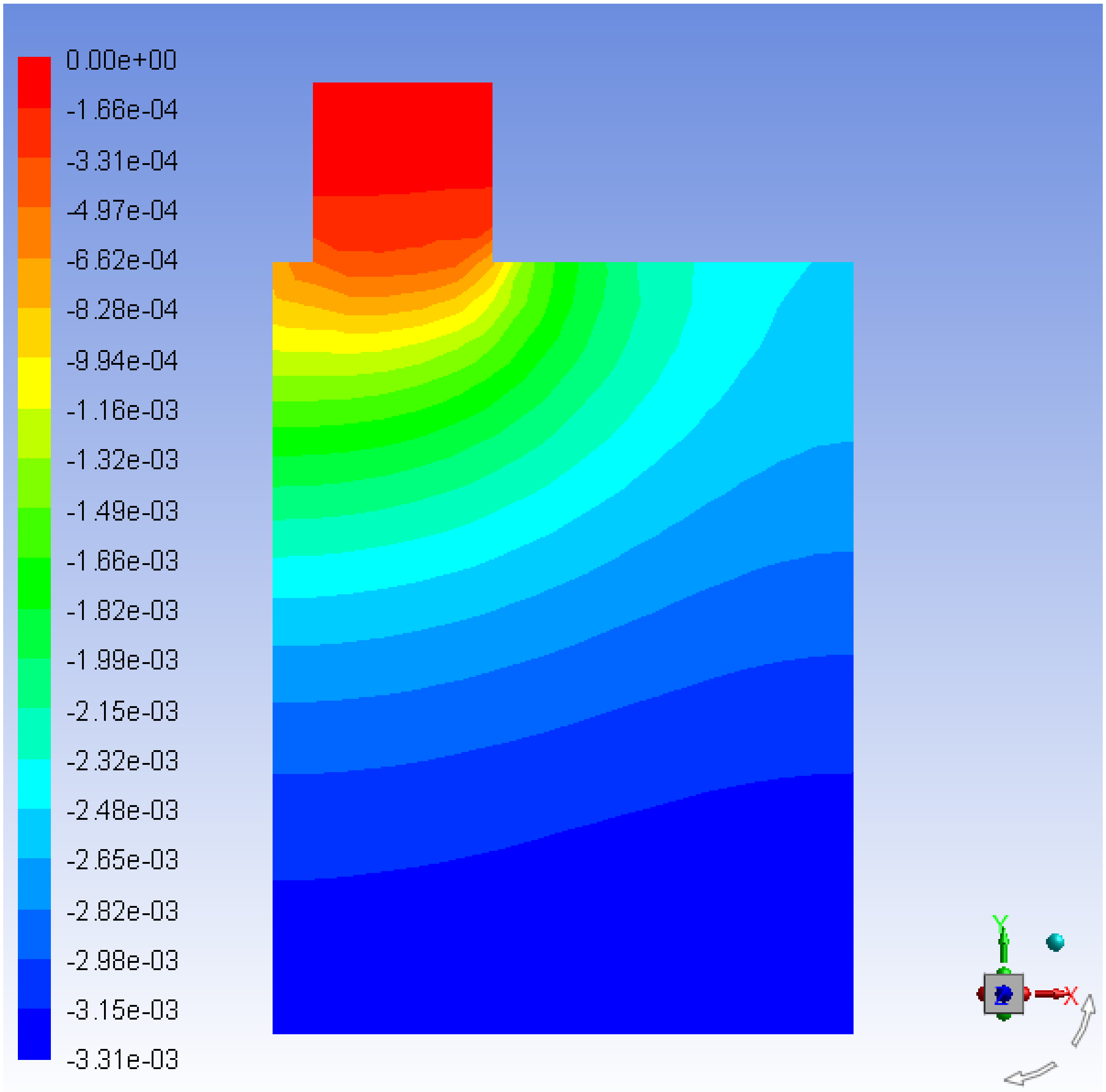
a. In the **Contours** dialog box, in the **Options** group box, enable **Filled**.

b. From the **Contours of** drop-down list, select **User Defined Scalars...** and **Potential Phi-**.

c. From the **Surfaces** selection list, select `tab_n`, `wall_active`, and `wall_n`.

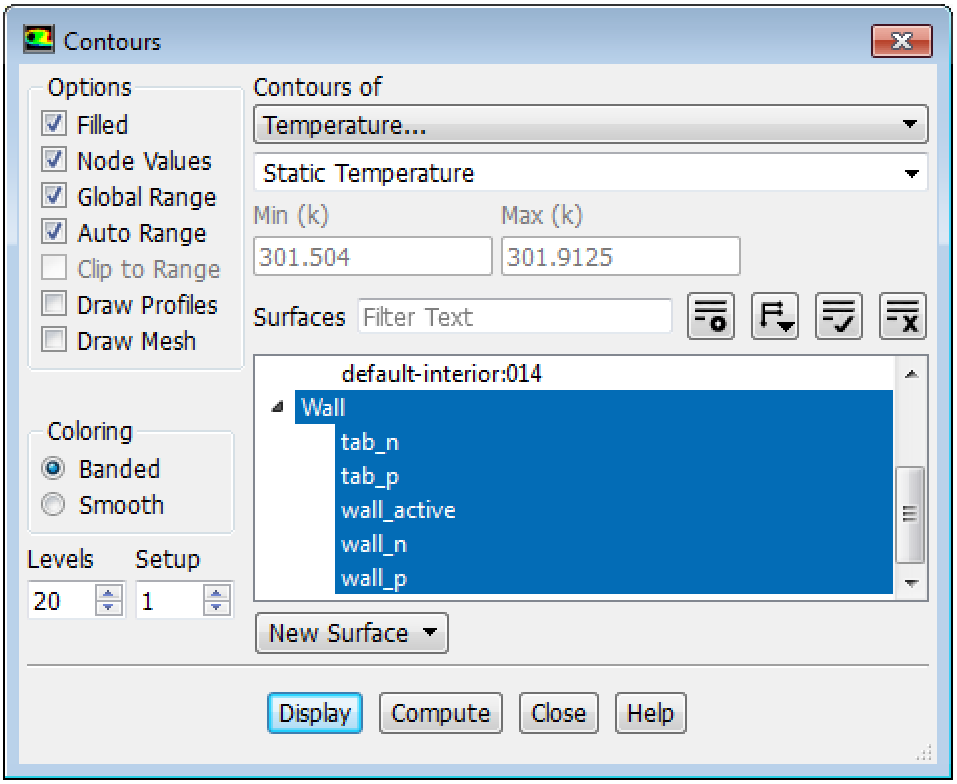
d. Click **Display** and close the **Contours** dialog box.

Figure 24.9: Contour Plot of Phase Potential for the Positive Electrode



3. Display the contour plot of the temperature.

   Postprocessing → **Graphics** → **Contours** → **Edit...**



a. In the **Contours** dialog box, in the **Options** group box, enable **Filled**.

b. From the **Contours of** drop-down list, select **Temperature...** and **Static Temperature**.

c. Select **Wall** in the **Surfaces** selection list.

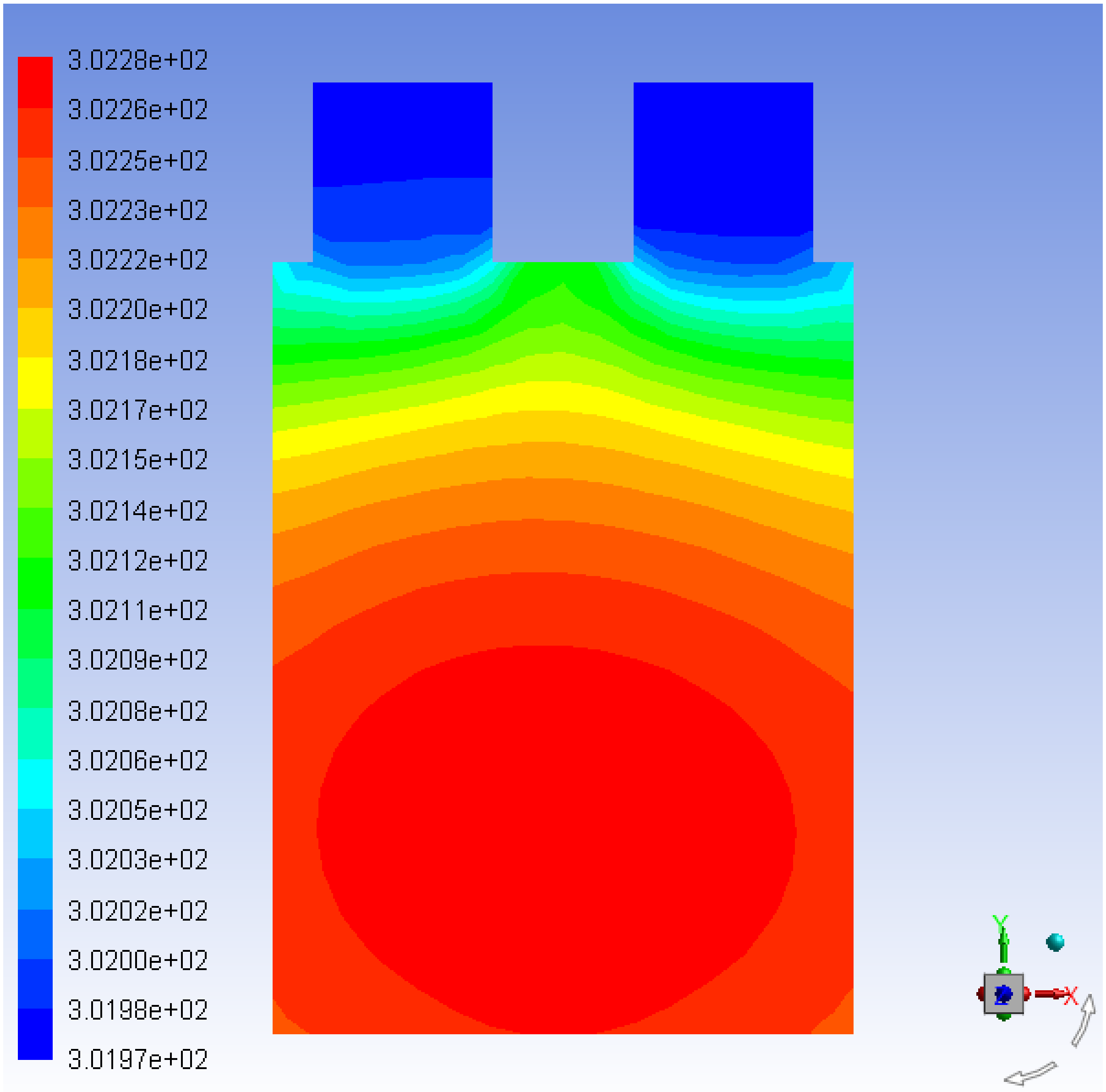
   The surfaces listed under **Wall** are automatically selected in the **Surfaces** list.

d. Click **Display** and close the **Contours** dialog box.

Note

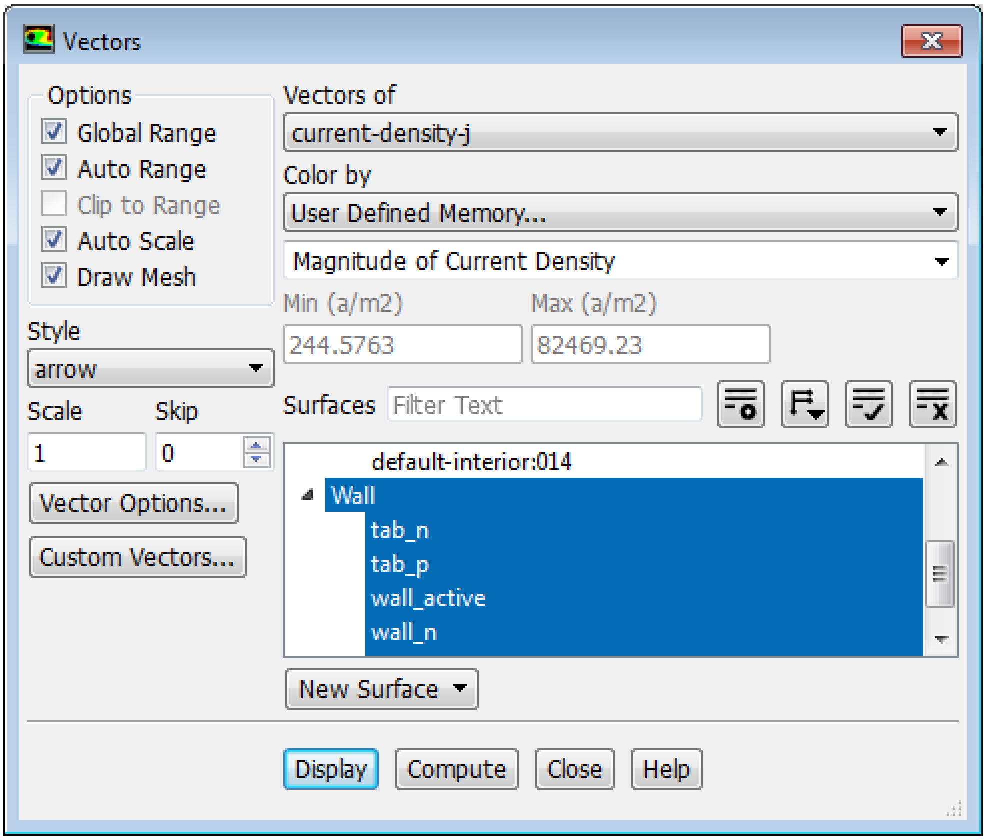
Use the **Axes** dialog box to set the precision for the colormap labels.

Figure 24.10: Contour Plot of Temperature



4. Display the vector plot of current density.

Postprocessing → Graphics → Vectors → Edit...



a. In the **Vectors** dialog box, select **current-density-j** from the **Vectors of** drop-down list.

b. Select **User Defined Memory...** and **Magnitude of Current Density** from the **Color by** drop-down list.

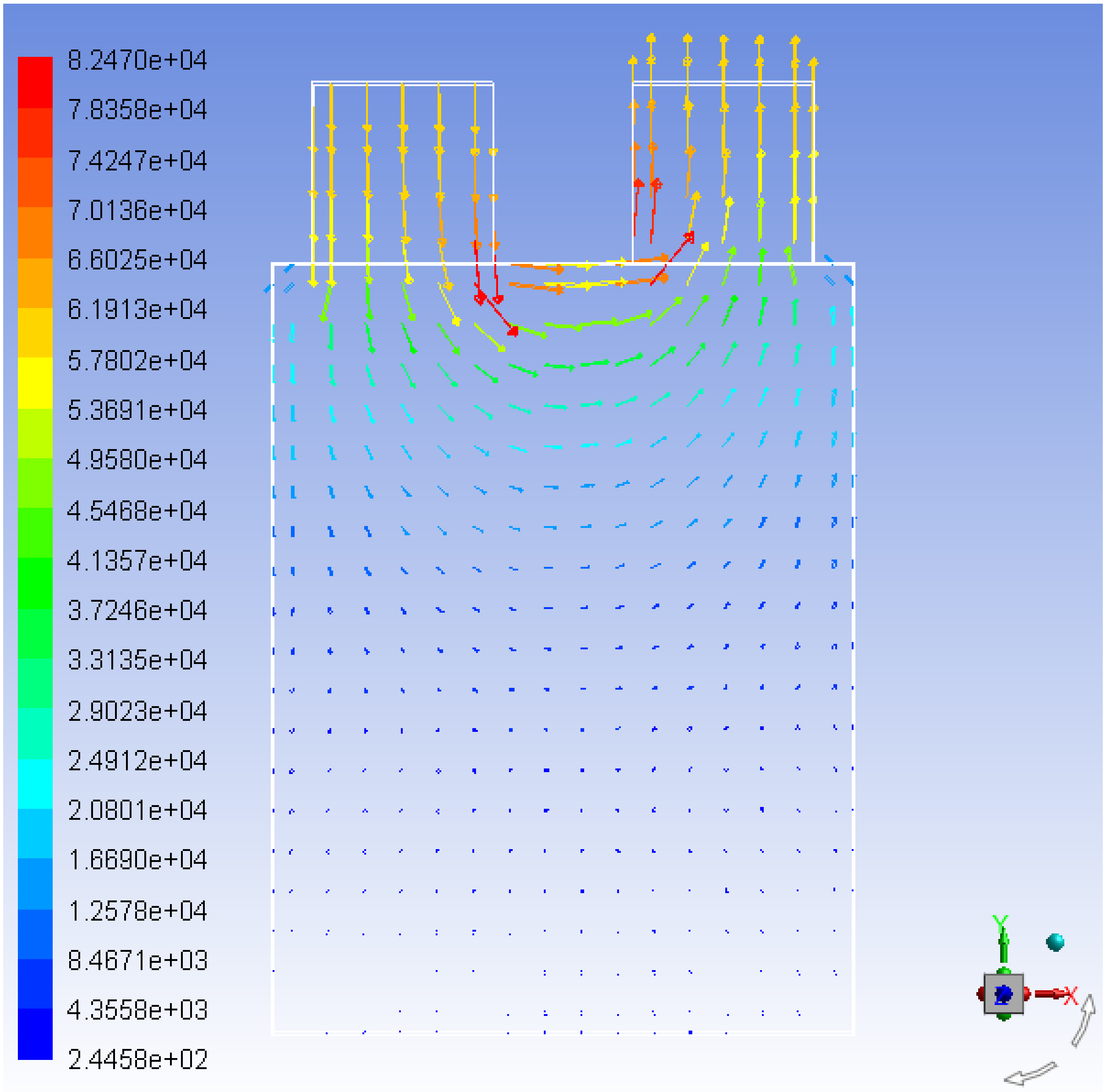
c. Click the **Toggle Tree View** button next to the **Surfaces** filter and from the drop-down list, select **Surface Type** (under **Group by**).

d. From the **Surfaces** selection list, select **Wall**.

e. In the **Options** group, enable **Draw Mesh** and in the **Mesh Display** dialog box, set the mesh display options as desired.

f. Click **Display** and close the **Vectors** dialog box.

Figure 24.11: Vector Plot of Current Density



5. Save the case file as `ntgk.cas.gz`. You will use this saved case later to treat electric short-circuits.

6. Repeat the simulation for the following charge rates and time steps:

C-Rate	Number of Time Steps
0.5 C	230
5 C	23

Make the following changes in the model's settings:

a. In the **MSMD Battery Model** dialog box, under the **Model Options** tab, specify the value listed in the above table for the **C-Rate**.

b. Modify the output filename for the **voltage_vp-rfile** report file by entering **`ntgk-`*`C-Rate`*`.out`** for **Output File Base Name** in the corresponding **Edit Report File** dialog box, where *`C-Rate`* is the value of the battery discharge rate. (For example, for C-Rate = 0.5 C, you will enter **`ntgk-0.5c.out`** for the filename).

c. Similarly, modify the output filename for **max_temp-rfile** by entering **`max-temp-`*`C-Rate`*`.out`** for **Output File Base Name** in the corresponding **Edit Report File** dialog box.

d. Initialize and run the solution for the number of the times steps specified in the above table.

Note

The Fluent solver will stop either after completing the specified number of time steps or when the **Min. Stop Voltage** condition is reached.

---

7. Display the discharge curves for the positive tab for the different discharge rates.

Postprocessing → **Plots** → **File...**

a. In the **File XY Plot** dialog box, click **Add...**.

b. In the **Select File** dialog box, change **File of Types** to `All Files (*)`, click `ntgk-0.5c.out` and click **OK**.

c. Under **Legend Entries**, in the lowest text-entry box, enter **`0.5c`** and click **Change Legend Entry**.

d. Do the same for `ntgk-1c.out` and `ntgk-5c.out` and change their legend entries accordingly.

e. In the **Legend Title** text-entry box, enter **`Discharge Rate`**.

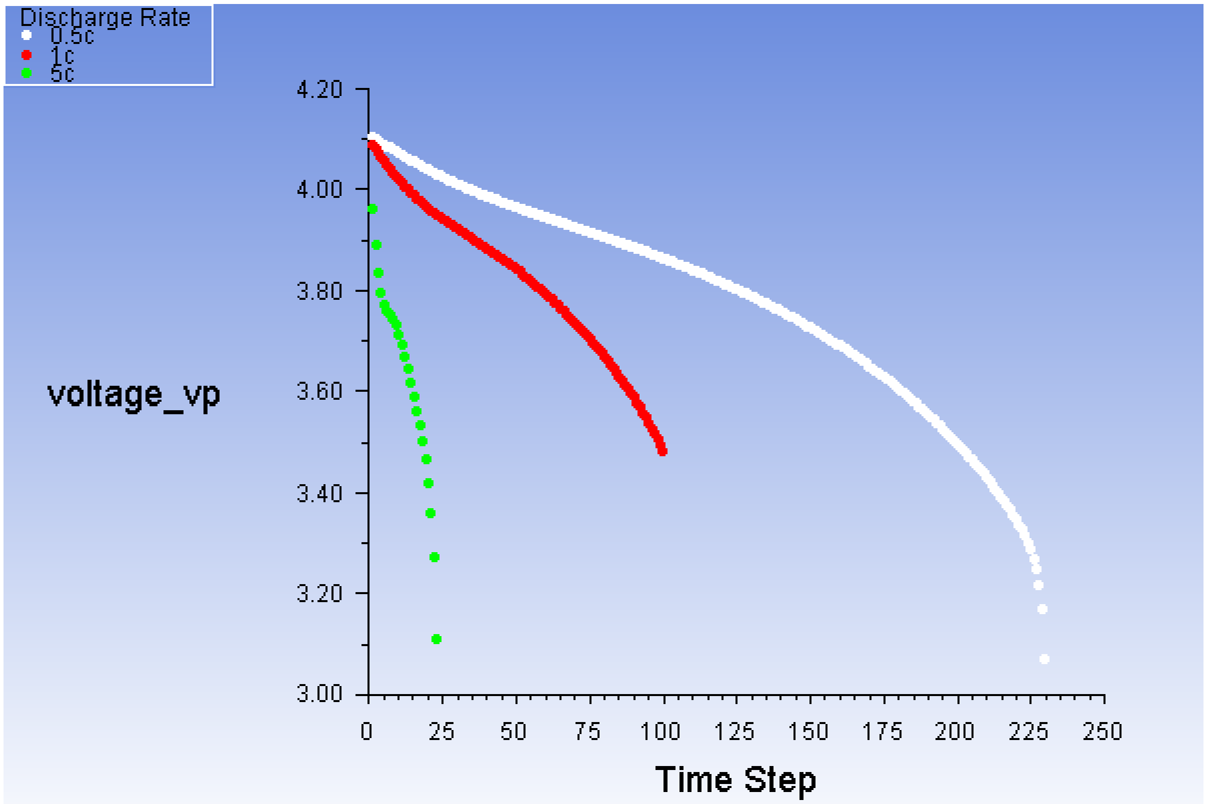
f. Click **Plot** and close the **File XY Plot** dialog box.

Note

Use the **Axes** dialog box to set the precision for the plot axes.

The Figure 24.12: NTGK Model: Discharge Curves (p. 990) shows the discharge curves for different discharge rates in the function of time.

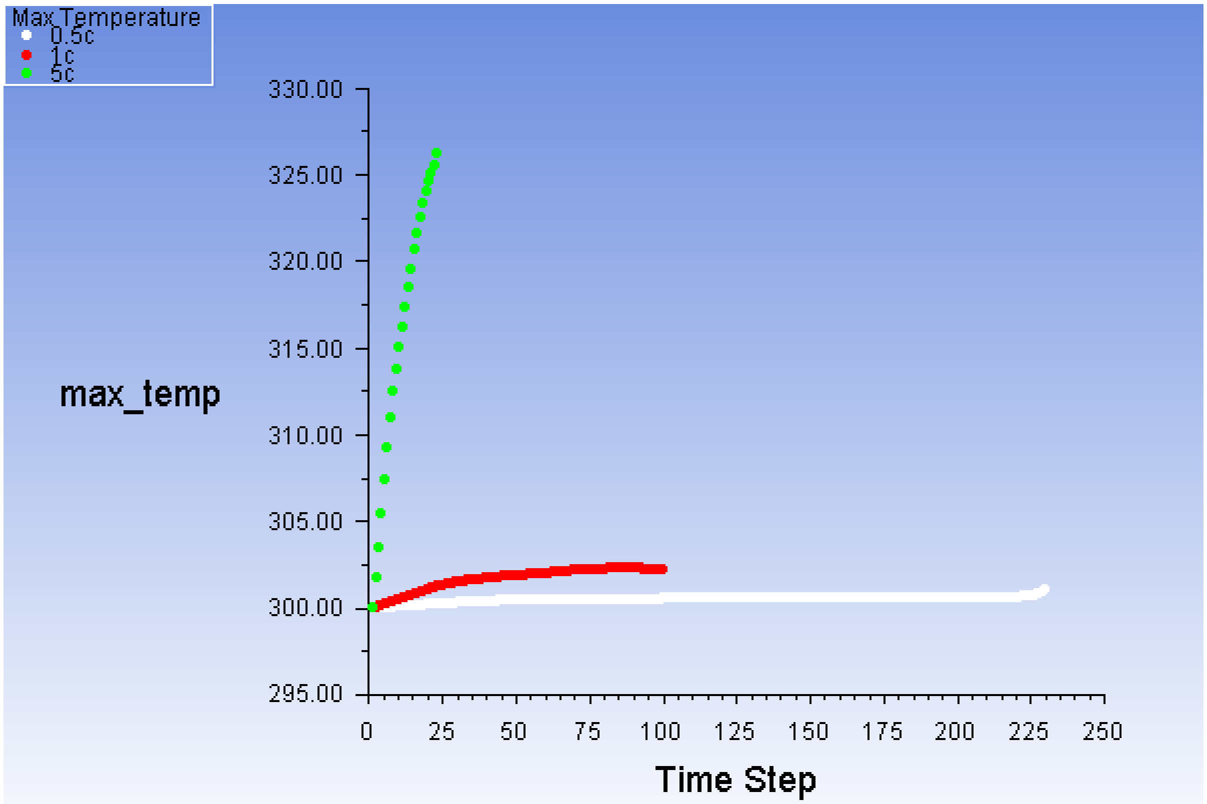
Figure 24.12: NTGK Model: Discharge Curves



8. In a manner similar to the previous step, load the files `max-temp-0.5c.out`, `max-temp-1c.out`, and `max-temp-5c.out` and display the maximum temperature curves in the domain.

   Figure 24.13: NTGK Model: Maximum Temperature in the Domain (p. 991) shows the maximum temperature curves in the simulation for different discharge rates.

Figure 24.13: NTGK Model: Maximum Temperature in the Domain



24.4.6. Simulating the Battery Pulse Discharge Using the ECM Model

1. In the **MSMD Battery Model** dialog box, under **E-Chemistry Models**, select **Equivalent Circuit Model**.

2. Under **Electrical Parameters**, retain the default value of **14.6 Ah** for **Nominal Cell Capacity**.

3. Retain the default selection of **Specified C-Rate** and enter **1** for **C-Rate**.

4. Under the **Model Parameters** tab, retain the battery specific parameters.

   For a given battery, these model parameters can be obtained using the battery's HPPC testing data. See Using the Dual-Potential MSMD Battery Model Text User Interface in the ANSYS Fluent Advanced Add-On Modules for details.

5. Click **OK** to apply the ECM battery model settings and close the **MSMD Battery Model** dialog box

6. Click **OK** in the **Warning** dialog box informing you that the re-initialization of the battery model is required.

7. Disable writing the maximum temperature in the domain over time to a file.

   Solutions → **Report Definitions** → **max_temp** **Edit...**

   a. In the **Volume Report Definition** dialog box, under **Report Files**, deselect **max_temp-rfile**.

   b. Click **OK** to close the **Volume Report Definition** dialog box.

8. In a similar manner, disable writing the time-dependent voltage at the positive tab to a file.

9. In the **Solution Initialization** task page, click **Initialize** to re-initialize the field variables.

10. Simulate the battery pulse discharge by changing the battery operating conditions each time after running the calculation for five minutes.

    a. In the **Run Calculation** task page, set **Number of Time Steps** to 10 and click **Calculate**.

    b. Once the calculation is complete, set **C-Rate** to `0` and run the calculation for 10 more time steps.

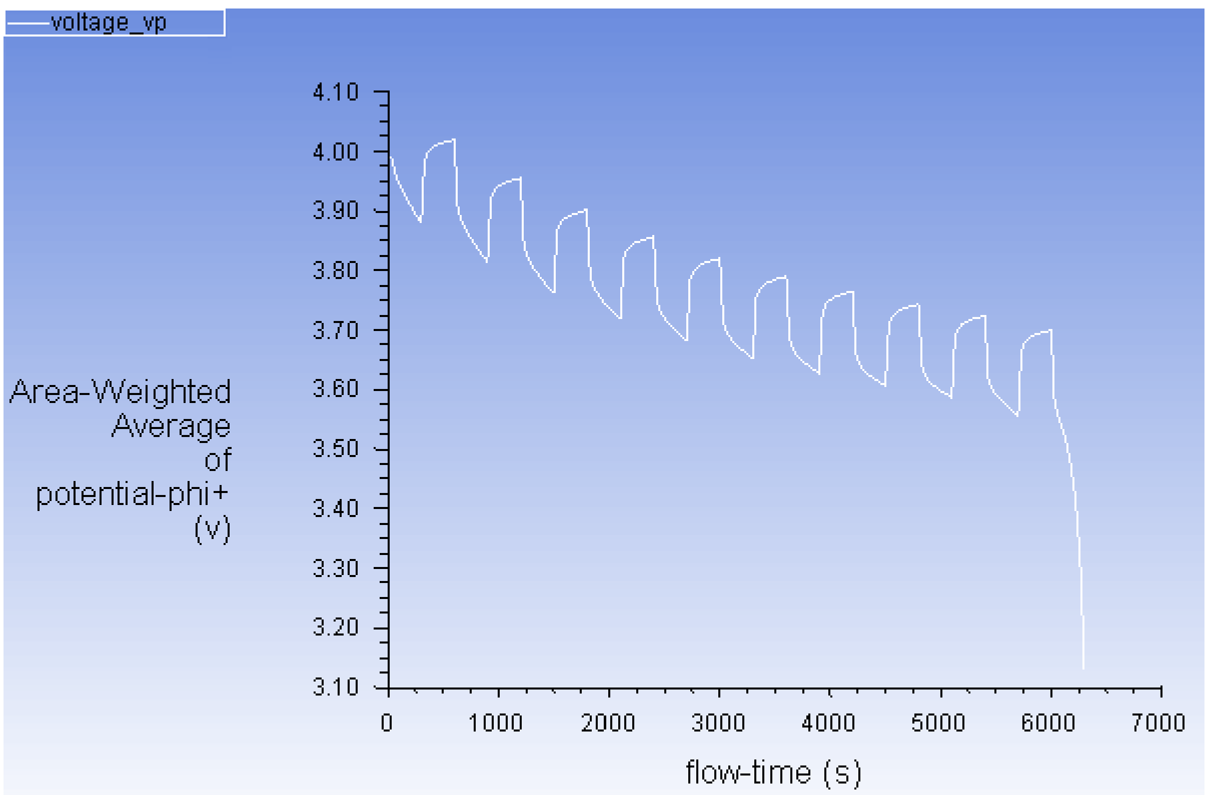
    c. Continue the simulation by alternating the value of **C-Rate** between 1 C and 0 C until, until the battery is fully discharged.

Note

Instead of doing this manually, you can use the **Using Profile** option in the **MSMD Battery Model** dialog box and load a profile file with specified C-rate fluctuations to drive the whole process. For more information about the usage of a profile file, refer to Specifying Battery Model Options in the ANSYS Fluent Advanced Add-On Modules..

The battery pulse discharge is summarized in Figure 24.14: Battery Pulse Discharge (p. 992).

Figure 24.14: Battery Pulse Discharge



24.4.7. Using the Reduced Order Method (ROM)

You will use the `ntgk.cas.gz` case file that you saved earlier to illustrate how to use the ROM for time-efficient calculations. This section assumes that you are already familiar with the ANSYS Fluent battery model; only the steps related specifically to using the ROM for problem solution are discussed here.

1. Read the NTGK model case file `ntgk.cas.gz`.

2. Initialize the problem.

3. In the **Run Calculation** task page, enter **3** for **Number of Time Steps** and click **Calculate**.

   Click **No** in the **Question** dialog box when asked if you would like to append the new data to the existing file, and then click **OK** in the **Warning** dialog box to overwrite the existing file.

4. Once the calculation is complete, enable the ROM.

   Setup → **Models** → **MSMD Battery Model** **Edit...**

   a. In the **MSMD Battery Model** dialog box, select **Reduced Order Method**.

   b. Set **Number of Sub-Steps/Time Step** to **`10`**.

5. Continue running your simulation from the direct method solution.

   The solution of the simulation using the ROM is significantly faster than when using the direct method without any changes in results.

24.4.8. External and Internal Short-Circuit Treatment

You will again use the `ntgk.cas.gz` case file that you saved earlier to illustrate how to treat external and internal short-circuits in a battery simulation. It is assumed that the battery is experiencing external and internal short-circuit simultaneously. This extreme case will be used to demonstrate the problem setup and postprocessing in a short simulation. This section assumes that you are already familiar with the ANSYS Fluent battery model, only the steps related to short simulation are emphasized here.

24.4.8.1. Setting up and Solving a Short-Circuit Problem

1. Read the NTGK model case file `ntgk.cas.gz`.

2. Set up the external electric short-circuit.

   a. In the **MSMD Battery Model** dialog box, under the **Model Options** tab, in the **Solution Options** group box, enable **Specified Resistance**.

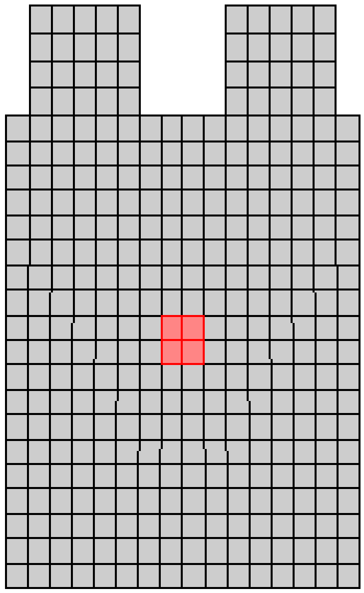
   b. For **External Resistance**, enter **`0.5 Ohm`** and click **OK**.

3. Set up the internal electric short-circuit in the center of the battery cell.

   a. Mark the short-circuit zone shown in Figure 24.15: Internal Short Circuit Region Marked for Patching (p. 994) using the region adaption feature.

      Setting Up Domain → **Adapt** → **Mark/Adapt Cells** → **Region...**

Figure 24.15: Internal Short Circuit Region Marked for Patching



i. In the **Region Adaption** dialog box, enter the following values for **Input Coordinates**.

X Min	X Max	Y Min	Y Max	Z Min	Z Max
-0.01	0.01	-0.01	0.02	-1	1

ii. Under **Options**, retain the default selection of **Inside**.

iii. Under **Shapes**, retain the default selection of **Hex**.

iv. Click **Mark**.

Fluent reports in the console that 12 cells were marked for refinement.

Extra

If you want to display the marked cells, perform the following steps:

A. In the **Region Adaption** dialog box, click **Manage...**.

B. In the **Manage Adaption Registers** dialog box, click **Options**.

C. In the **Adaption Display Option** dialog box, under **Options**, enable **Draw Mesh**.

D. In the **Mesh Display** dialog box, ensure that **Edges** and **Faces** are enabled under **Options**, and **All** is enabled under **Edge Type**, and click **Close**.

E. Ensure that **Wireframe** is enabled under **Refine** and click **OK**

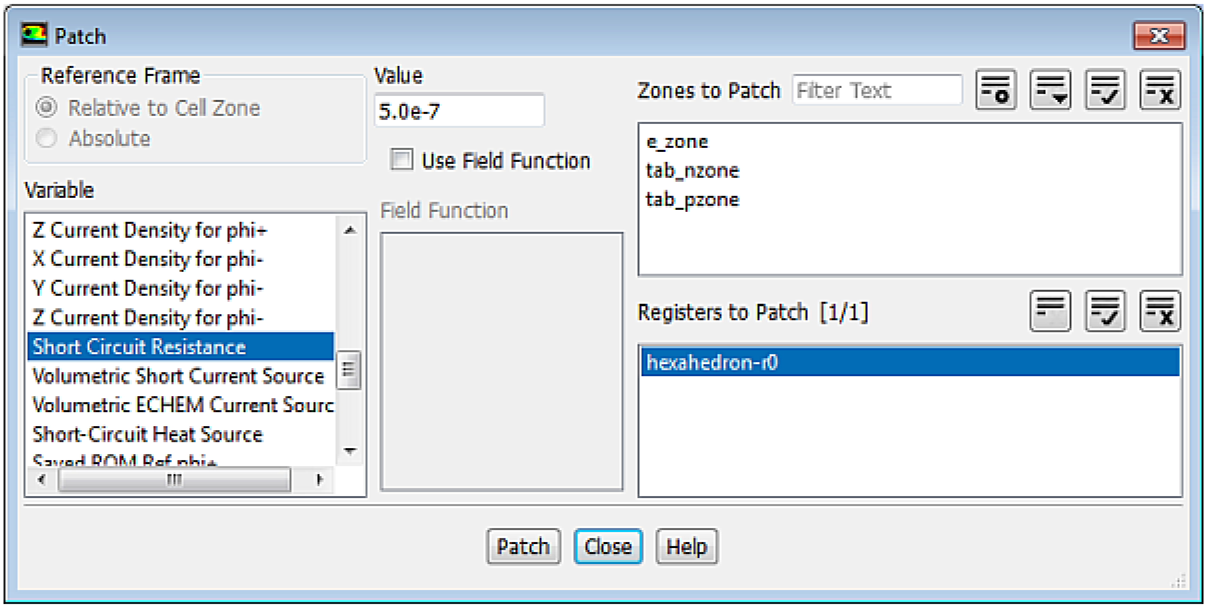
F. Click **Display**.

---

4. Initialize the field variables using the standard initialization method.

5. Patch the internal short circuit zone with the short resistance value.

   Solving → **Initialization** → **Patch...**



   a. In the **Solution Initialization** task page, click **Patch**.

   b. In the **Patch** dialog box, select **Short Circuit Resistance** under **Variable**.

   c. Select `hexehedron-r0` under **Registers to Patch**.

   d. For **Value**, enter **`5.0e-7`**.

   e. Click **Patch** and close the **Patch** dialog box.

6. Save the case file as `ntgk_short_circuit.cas.gz`.

7. Run the simulation for 5 seconds.

   Solving → **Run Calculation**

   a. Set **Time Step Size** to **1** second and **No. of Time Steps** to **5**.

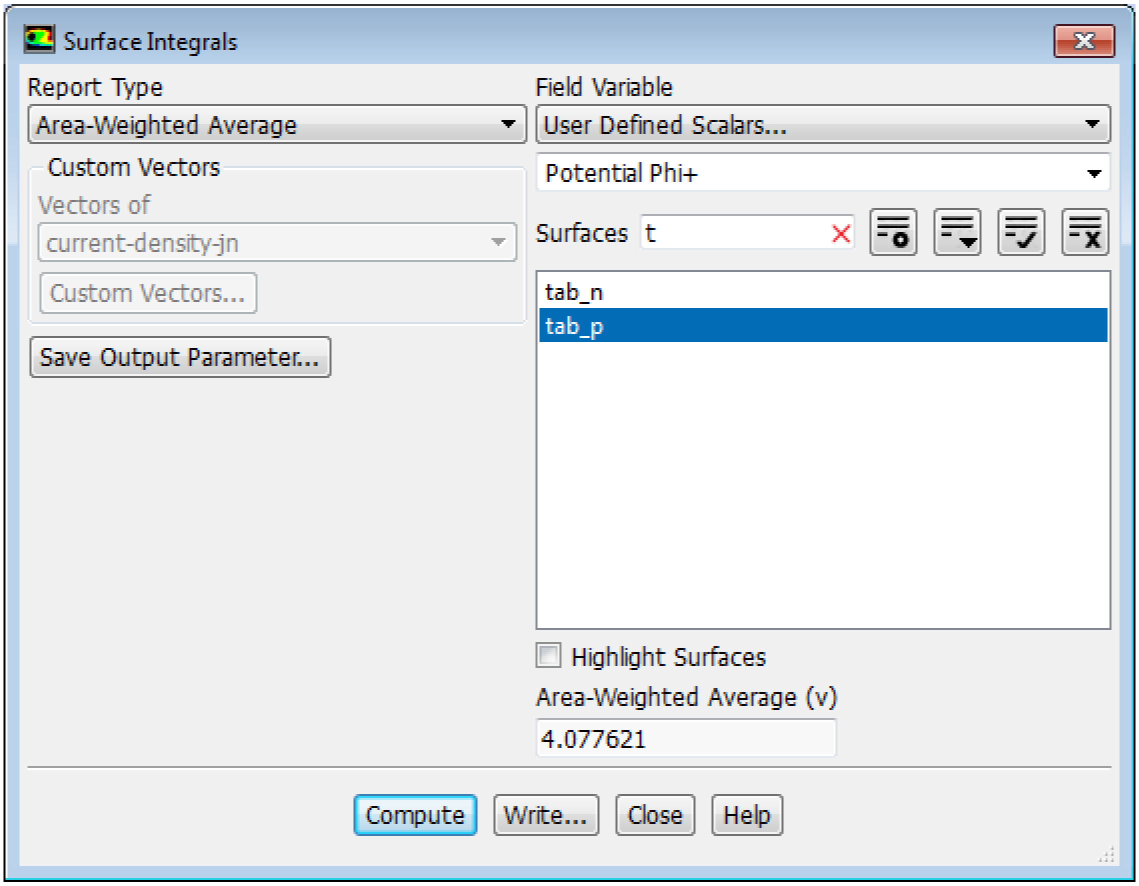
   b. Click **Calculate**.

8. Save the case and data files (`ntgk_short_circuit.cas.gz` and `ntgk_short_circuit.dat.gz`).

24.4.8.2. Postprocessing

1. Compute the battery tab voltage U_{tab}.

   Postprocessing → **Reports** → **Surface Integrals...**



a. In the **Surface Integrals** dialog box, from the **Report Type** drop-down list, select **Area-Weighted Average**.

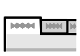
b. From the **Field Variable** drop-down lists, select **User Defined Scalar...** and **Potential Phi+**.

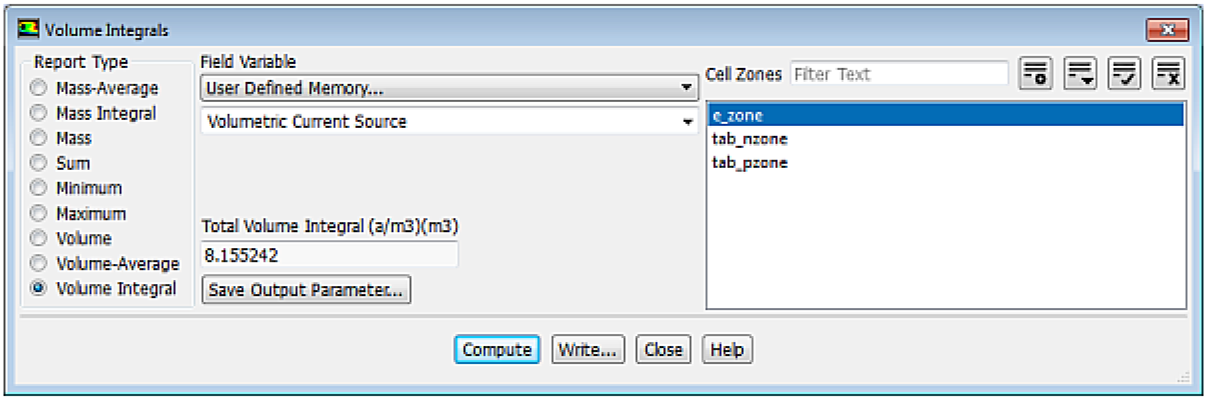
c. In the **Surfaces** filter, type **t** to display surface names that begin with "t" and select **tab_p** from the selection list.

d. Click **Compute**.

The battery tab voltage of approximately 4.078 V is printed in the **Area-Weighted Average** field and in the Fluent console.

2. Compute the battery tab current I_{tab}.

 **Postprocessing** → **Reports** → **Volume Integrals...**



a. In the **Report Type** group box, select **Volume Integrals**.

b. From the **Field Variable** drop-down lists, select **User Defined Memory...** and **Volumetric Current Source**.

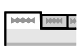
c. From the **Cell Zones** selection list, select e_zone.

d. Click **Compute**.

Fluent reports in the **Total Volume Integral** field and in the console that the total volume integral for the volumetric current source is approximately 8.155 A.

The computed values of the battery tab current and voltage satisfy the tab boundary condition $R_{load}=U_{tab}/I_{tab}$.

3. Display the vector plot of current at the positive and negative current collectors.

 **Postprocessing** → **Graphics** → **Vectors** → **Edit...**

a. In the **Vectors** dialog box, select **current-density-jp** from the **Vectors of** drop-down list.

b. Select **User Defined Memory...** and **Magnitude of Current Density** from the **Color by** drop-down lists.

c. From the **Surfaces** selection list, select **Wall**.

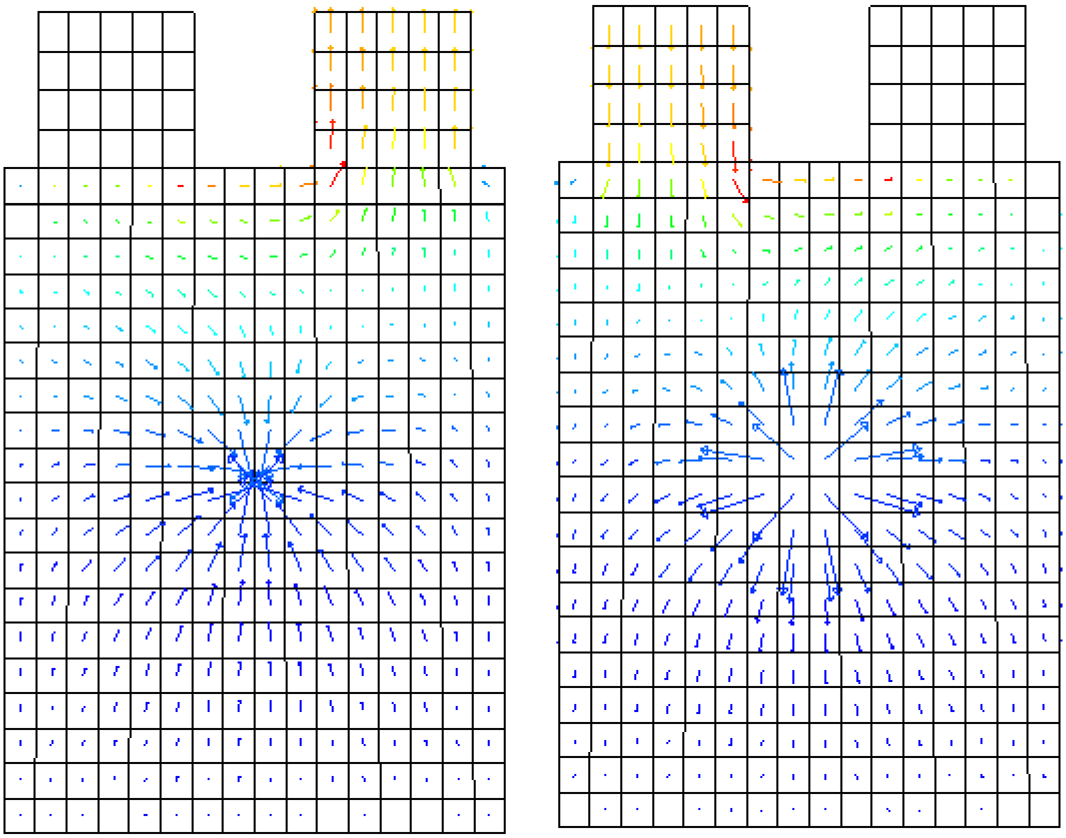
The surfaces of the "wall" type are automatically selected in the **Surfaces** list.

d. Click **Display**.

e. In a similar manner, display the current for the negative current collector by selecting **current-density-jn** from the **Vectors of** drop-down list.

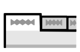
The Figure 24.16: The Vector Plots of Current at the Positive and Negative Current Collectors (p. 998) demonstrates the vector plots of electric current flow in the positive and negative current collectors in the battery cell side by side. The plots clearly show that besides providing tab current, short current flows form positive electrode to the negative electrode through the short area.

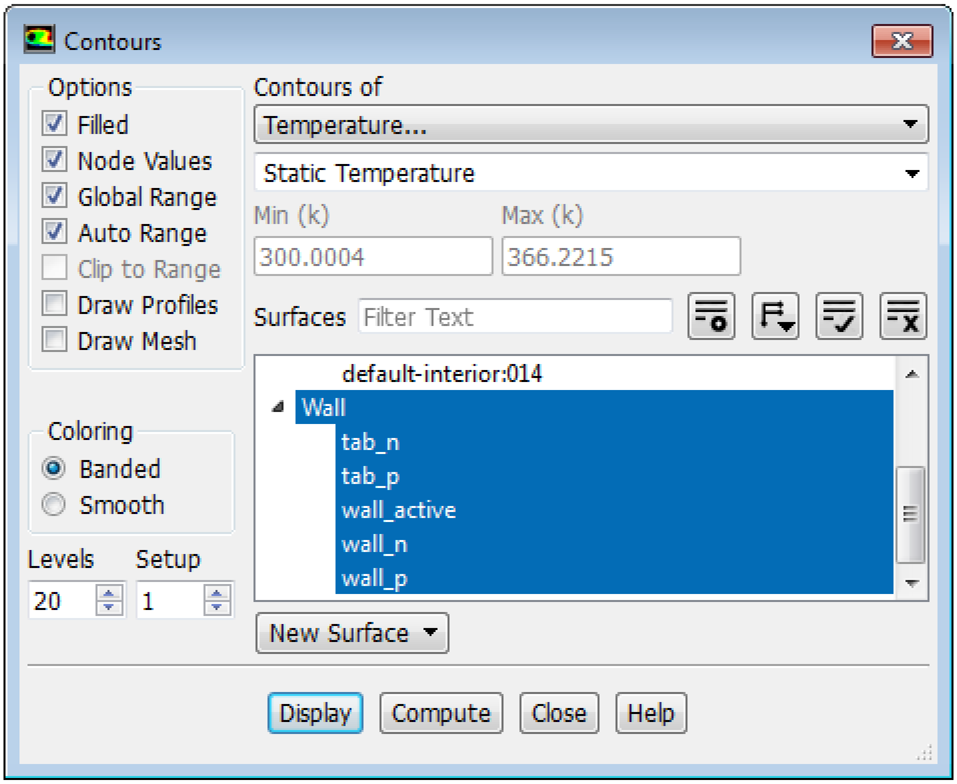
Figure 24.16: The Vector Plots of Current at the Positive and Negative Current Collectors



f. Close the **Vectors** dialog box.

4. Display the contour plot of the temperature as you did previously.

 **Postprocessing** → **Graphics** → **Contours** → **Edit...**



a. In the **Contours** dialog box, under **Options**, enable **Filled**.

b. From the **Contours of** drop-down list, select **Temperature...** and **Static Temperature**.

c. From the **Surface Types** selection list, select `wall`.

d. Click **Display** and close the **Contours** dialog box.

Figure 24.17: Contour Plot of Temperature

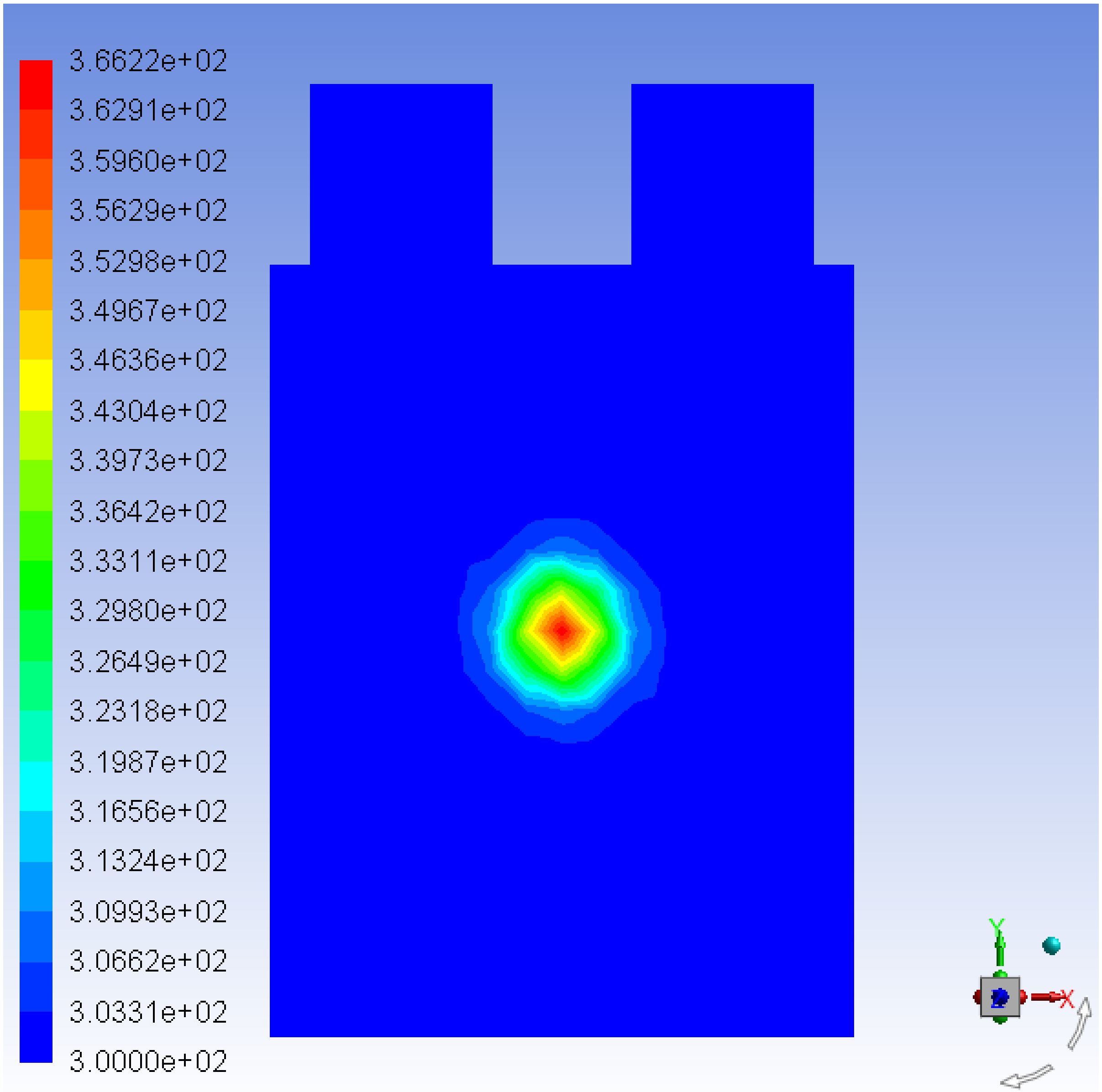


Figure 24.17: Contour Plot of Temperature (p. 1000) shows a temperature hotspot in the internal shorted area of the battery cell.

5. Check for different electric current flow rates in the manner described in step 2.

   a. Generate volume integral reports for the field variables listed in the table below.

Field Variable	Notation	Reported Value
Volumetric Short Current Source	I_{short}	15.844 A
Volumetric ECHEM Current Source	I_{echem}	23.999 A

   b. Verify that the total produced electric current equals to the sum of tab and short current, that is $I_{echem}=I_{tab}+I_{short}$.

6. Check for different types of heat generation rates.

    a. As you did for the current source reports, generate reports for the field variables listed in the table below.

Field Variable	Notation	Reported Value
Volumetric Ohmic Source	Q_{joule}	0.0339 W
Electrochemistry Source	Q_{echem}	1.092 W
Short-Circuit Heat Source	Q_{short}	64.607 W
Total Heat Generation Source	Q_{total}	65.733 W

    b. Verify that the total heat generation rate is the sum of different contributions, that is $Q_{total}=Q_{joule}+Q_{echem}+Q_{short}$.

Note that, as battery's temperature increases, thermal runaway may occur. If thermal runaway starts, some undesirable exothermic decomposition reactions will occur. For thermal runaway simulations, the default electrochemistry model cannot be used. Short treatment can only capture the thermal ramp-up process before the onset of thermal runaway.

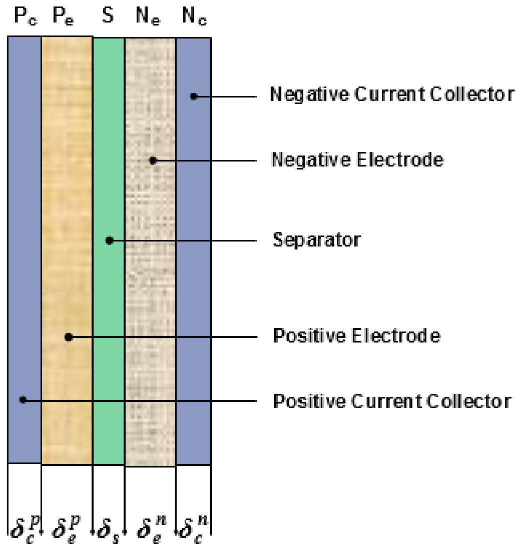
24.5. Summary

In this tutorial, you studied how to solve a battery cell problem using the NTGK submodel with the default settings. You then used the ROM to speed up the computation time of the battery model simulation. In addition, you learned how to use the MSMD model capability to treat external and internal short-circuits.

For more information about using the Dual-Potential MSMD Battery model, see the Fluent Advanced Add-On Modules documentation on the ANSYS Customer Portal (http://support.ansys.com/documentation).

24.6. Appendix

The battery cell cross-section is shown in the figure below.



You can estimate the material properties for your battery cell using the following correlations:

- For density ρ, heat capacity C_p, and thermal conductivity K:

$$x_{eff}=\frac{0.5x_c^p\delta_c^p+x_e^p\delta_e^p+x_s\delta_s+x_e^n\delta_e^n+0.5x_c^n\delta_c^n}{\delta_{total}}$$

$$\delta_{total}=0.5\delta_c^p+\delta_e^p+\delta_s+\delta_e^n+0.5\delta_c^n$$

  where x_{eff} is the effective property value of a material property (such as density, heat capacity, or thermal conductivity), is the thickness. The subscripts c, e, and s refer to current collector, electrode, and separator, respectively. The superscripts p and n refer to positive and negative, respectively.

- For electric conductivity σ:

$$\sigma_p=\frac{0.5\sigma_c^p\delta_c^p+\sigma_e^p\delta_e^p}{\delta_{total}}, \quad \sigma_n=\frac{0.5\sigma_c^n\delta_c^n+\sigma_e^n\delta_e^n}{\delta_{total}}$$

The material properties are taken from Kim's papers [2] and [1]. The computed material properties for the battery cell presented in the tutorial are shown in the table below.

Zone	P_c	P_e	S	N_e	N_c	**Total**
δ [um]	20	150	12	145	10	322
ρ [kg/m^3]	2700	1500	1200	2500	8960	2092
C_p [J/kg-K]	900	700	700	700	385	678
K [W/m-K]	238	5	1	5	398	18.2
σ [s/m]	3.83e7	13.9		100	6.33e7	σ_p = 1.19e6 σ_n = 9.83e5

24.7. References

1. U. S. Kim et al, "Effect of electrode configuration on the thermal behavior of a lithium-polymer battery", Journal of Power Sources, Volume 180 (2), pages 909-916, 2008.

2. U. S. Kim, et al., "Modeling the Dependence of the Discharge Behavior of a Lithium-Ion Battery on the Environmental Temperature", J. of Electrochemical Soc., Volume 158 (5), pages A611-A618, 2011.

Chapter 25: Simulating a 1P3S Battery Pack Using the MSMD Battery Model

This tutorial is divided into the following sections:

25.1. Introduction

This tutorial is used to show how to set up a battery pack (battery system connected in parallel/series pattern) simulation in ANSYS Fluent. All the three submodels are available for a pack simulation.

This tutorial illustrates how to do the following:

- Load the Battery module add-on
- Set up a battery pack simulation using the NTGK battery submodel in ANSYS Fluent
- Define active, tab, and busbar conductive zones
- Define electric contacts for the contact surface and external connectors
- Define electric conductivity for the active material using the user-defined scalars
- Define electric conductivity for the passive material using the user-defined function
- Obtain the battery pack simulation results and perform postprocessing activities

Most problem setup procedures are similar to the single cell simulation. The differences in the problem setup will be emphasized in this tutorial.

25.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

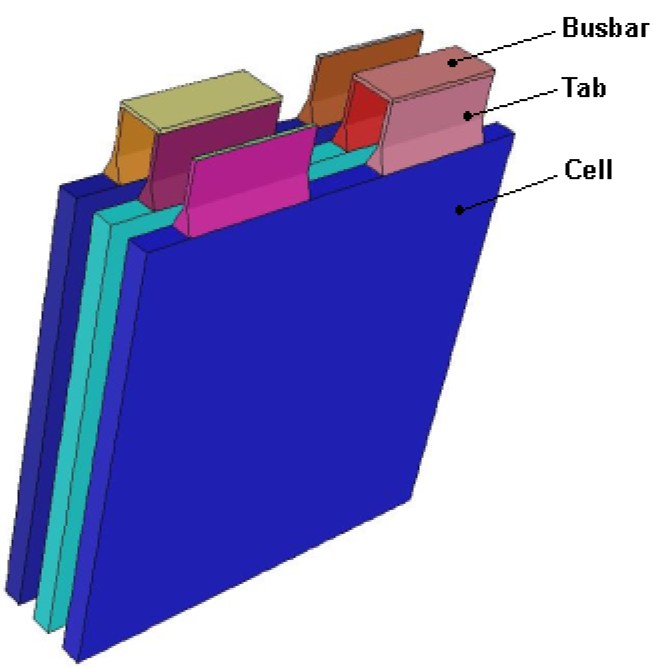
- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

and that you are familiar with the ANSYS Fluent tree and ribbon structure. Some steps in the setup and solution procedure will not be shown explicitly.

25.3. Problem Description

This problem considers a small 1P3S battery pack, that is, the three battery cells connected in series. A schematic of the problem is shown in Figure 25.1: Schematic of the Battery Pack Problem (p. 1006).

Figure 25.1: Schematic of the Battery Pack Problem



The discharging process of the battery pack is occurring under constant power of 200 W. The nominal cell capacity is 14.6 Ah.

You will create a material for the battery cells (an active material) and define the electric conductivity for the active material using the user-defined scalars (UDS). You will create a material for busbars and tabs (a passive material) and define the electric conductivity for the passive material using the provided user-defined function (UDF). You will use the same material for busbars and tabs.

In this tutorial, you will use the NTGK battery submodel to simulate the discharging process under constant power conditions.

25.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

25.4.1. Preparation

1. Set up a working folder on the computer you will be using.

2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

> **Note**
>
> If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

3. Enter the name of this tutorial into the search bar.

4. Narrow the results by using the filter on the left side of the page.

   a. Click **ANSYS Fluent** under **Product**.

   b. Click **18.0** under **Version**.

5. Select this tutorial from the list.

6. Click the **battery_pack_R180.zip** link to download the input files.

7. Unzip `battery_pack_R180.zip`.

   The input file `1P3S_battery_pack.msh`, `1P3S_battery_pack.cas`, and `1P3S_battery_pack.dat` can be found in the `battery_pack` folder created after unzipping the file.

8. Use Fluent Launcher to start the **3D** version of ANSYS Fluent.

9. Enable **Double Precision**.

10. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.

11. Select either **Serial** or **Parallel** under **Processing Options**. In this tutorial, **Serial** is selected for demonstration only.

25.4.2. Reading and Scaling the Mesh

1. Read the mesh file `1P3S_battery_pack.msh`.

   File → Read → Mesh...

   When prompted, browse to the location of the `1P3S_battery_pack.msh` and select the file.

   Once you read in the mesh, it is displayed in the embedded graphics windows.

2. Check the mesh.

   Setting Up Domain → Mesh → Check

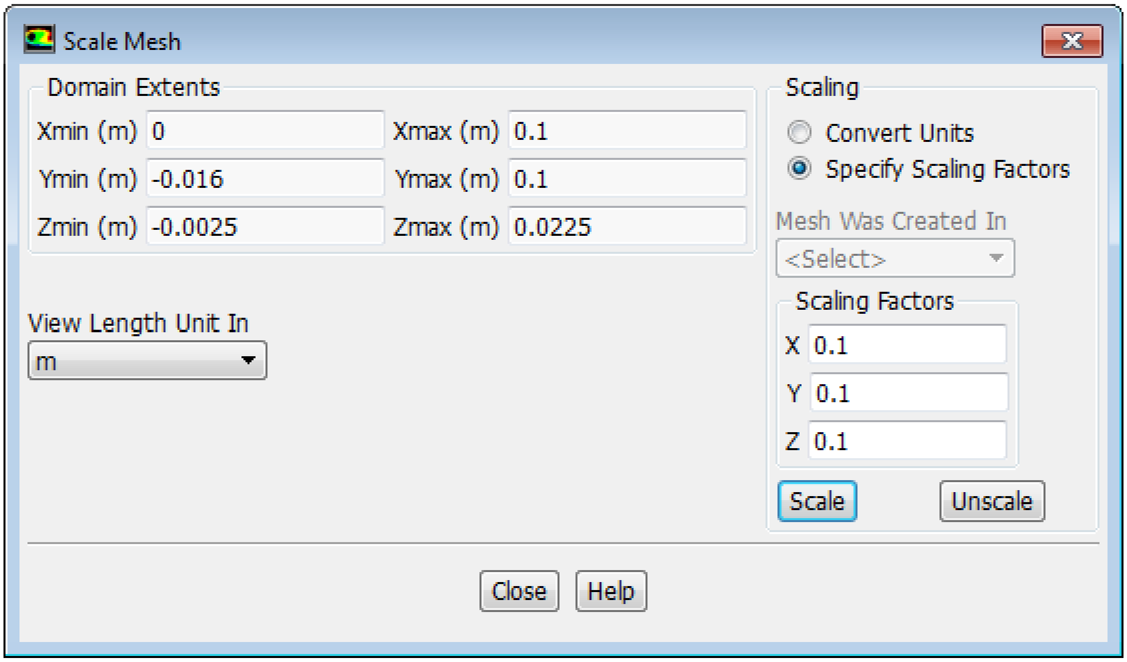
3. Scale the mesh.

   Setting Up Domain → Mesh → Scale

   a. In the **Scale Mesh** dialog box, select **Specify Scaling Factors** in the **Scaling** group.

b. Enter **`0.1`** for **X**, **Y** and **Z** in the **Scaling Factors** group.

c. Click **Scale** and close the **Scale Mesh** dialog box.



4. Check the mesh.

   Setting Up Domain → **Mesh** → **Check**

25.4.3. Loading the MSMD battery Add-on

1. To load the MSMD battery add-on into ANSYS Fluent, type the following in the console window:

```
define/models/addon-module
```

A list of ANSYS Fluent add-on modules is displayed.

```
Fluent Addon Modules:
 0. none
 1. MHD Model
 2. Fiber Model
 3. Fuel Cell and Electrolysis Model
 4. SOFC Model with Unresolved Electrolyte
 5. Population Balance Model
 6. Adjoint Solver
 7. Single-Potential Battery Model
 8. Dual-Potential MSMD Battery Model
 9. PEM Fuel Cell Model
10. Macroscopic Particle Model
Enter Module Number: [0] 8
```

2. Enter 8 to load the MSMD battery itself.

   During the loading process, a scheme library containing the graphical and text user interface, and a library of user-defined functions (UDFs) containing a set of UDFs for the battery module are loaded into ANSYS Fluent. Fluent reports the progress in the console.

Once the MSMD battery add-on is loaded, **MSMD Battery Model** appears in the **Model** task page and under the **Models** tree branch. The UDF library also becomes visible as a new entry in the UDF Library Manager dialog box.

25.4.4. Battery Model Setup

The following sections describe the setup steps for this tutorial:

25.4.4.1. Specifying Solver and Models

1. Enable a time-dependent calculation by selecting **Transient** in the **Solver** group of the **Setting Up Physics** tab.

   Setting Up Physics → Solver → Transient

2. To solve for the temperature field, enable the **Energy** equation (in the **Models** group).

   Setting Up Physics → Models → Energy

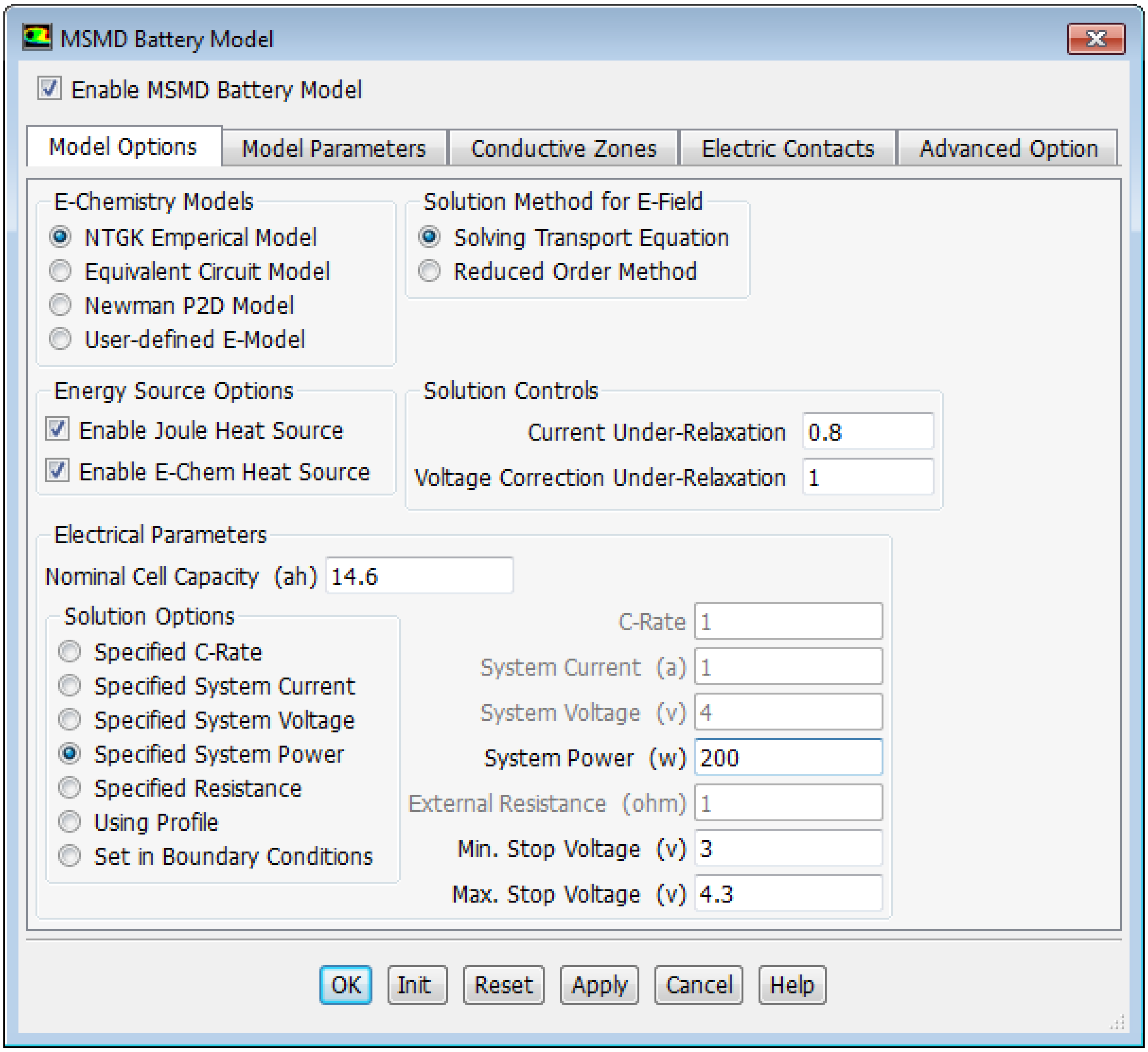
3. Enable the battery model.

   Setting Up Physics → Models → More → Dual Potential MSMD Battery Model

   a. In the **MSMD Battery Model** dialog box, select **Enable MSMD Battery Model**.

      The dialog box expands to display the battery model's settings.

Figure 25.2: Model Options



b. Under the **Model Options** tab (Figure 25.2: Model Options (p. 1010)), configure the following battery operation conditions:

   i. Under **E-Chemistry Models**, enable **NTGK Empirical Model**.

   ii. In the **Electrical Parameters** group, retain the default value of **`14.6` Ah** for **Nominal Cell Capacity**.

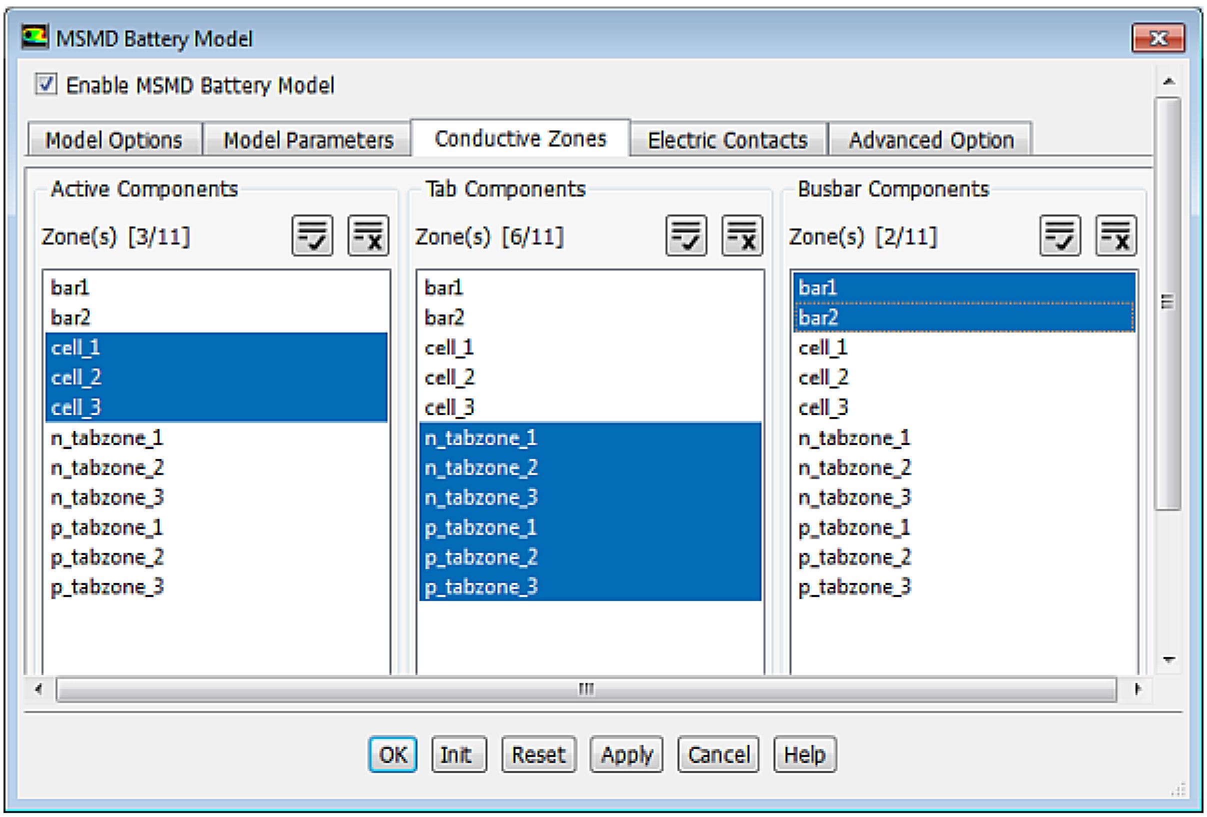
   iii. Enable **Specified System Power** in the **Solution Options** group and set **System Power** to **`200 W`**.

c. Under the **Model Parameters** tab, retain the default settings for Y and U coefficients. For details, see the Fluent Advanced Add-On Modules documentation on the ANSYS Customer Portal (http://support.ansys.com/documentation).

d. Under the **Conductive Zones** tab (Figure 25.3: Conductive Zones (p. 1011)), configure the following settings:

Group	Control or List	Value or Selection
Active Components	Zone (s)	cell_1
		cell_2
		cell_3
Tab Components	Zone (s)	n_tabzone_1
		n_tabzone_2
		n_tabzone_3
		p_tabzone_1
		p_tabzone_2
		p_tabzone_3
Busbar Components	Zone (s)	bar1
		bar2

Figure 25.3: Conductive Zones

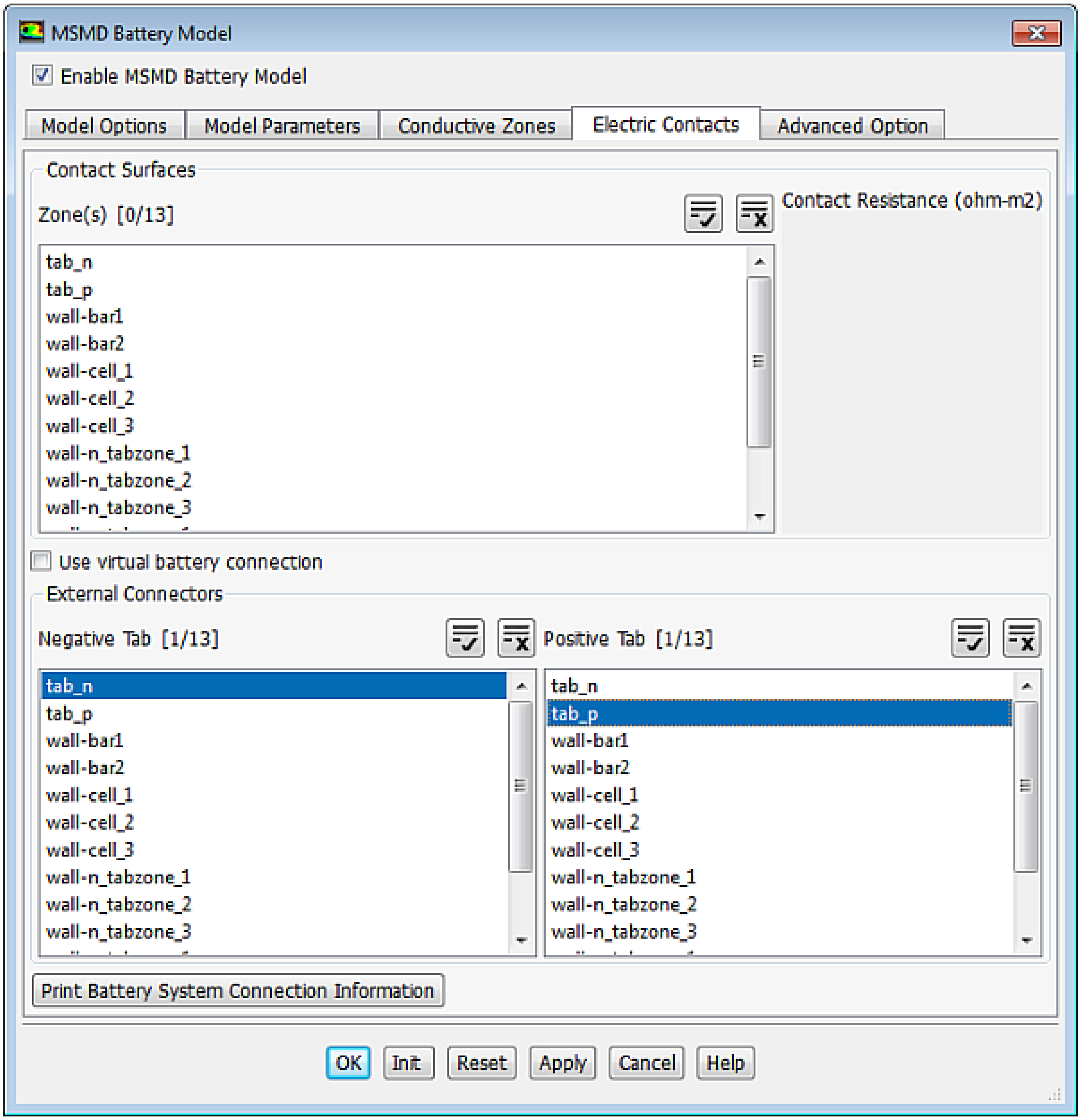


e. Under the **Electric Contacts** tab (Figure 25.4: Electric Contacts (p. 1012)), configure the contact surface and external connector settings as follows:

Group	Control or List	Value or Selection
External Connectors	Negative Tap	tab_n
	Positive Tap	tab_p

The corresponding current or voltage boundary condition will be applied to those boundaries automatically.

Figure 25.4: Electric Contacts



f. Click the **Print Battery System Connection Information** button.

ANSYS Fluent prints the battery connection information in the console window:

```
Battery Network Zone Information:
-------------------------------------
 Battery Serial 1
   Parallel 1
        N-Tab  zone:  n_tabzone_1
        Active zone:  cell_1
        P-Tab  zone:  p_tabzone_1
 Battery Serial 2
   Parallel 1
      N-Tab  zone:  n_tabzone_2
```

```
      Active zone:  cell_2
      P-Tab  zone:  p_tabzone_2
 Battery Serial 3
   Parallel 1
      N-Tab  zone:  n_tabzone_3
      Active zone:  cell_3
      P-Tab  zone:  p_tabzone_3
-----------------------------------
 Busbar zone 1: bar1
 Busbar zone 2: bar2

 Active zone list at  odd level: cell_1 cell_3
 Active zone list at even level: cell_2

 Number of battery seriel =3; Number of parallel per serial=1
****************END OF BATTERY CONNECTION INFO**************
```

g. Verify that the connection information is correct. If an error message appears or if the connections are not what you want, redefine the conductive zones in the **Conductive Zones** tab (Figure 25.3: Conductive Zones (p. 1011)). Repeat this process until you confirm that the battery connections are set correctly.

Important

To set a valid connection, you must connect the negative tab to the positive tab through conductive zones.

---

h. Click **OK** to close the **MSMD Battery Model** dialog box.

   In the background, Fluent automatically hooks all the necessary UDFs for the problem.

25.4.4.2. Defining New Materials

Define the new e_material material for all the battery's cells and busbar_material material for the battery pack's busbars and tabs.

Important

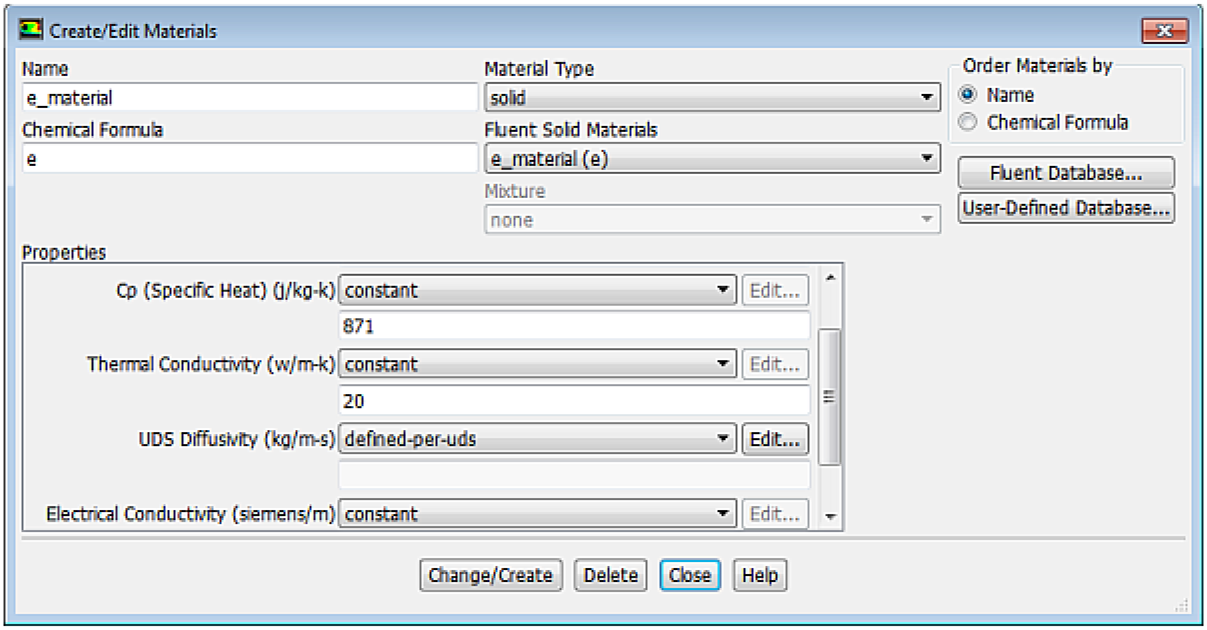
For the battery active material, you must define the electric conductivity via the UDS diffusivity on a per-scalar basis. ANSYS Fluent will use these two UDS scalars to solve the differential equations; for details, see the Fluent Advanced Add-On Modules documentation on the ANSYS Customer Portal (http://support.ansys.com/documentation).

For the battery passive material(s), you must define the UDS diffusivity using a user-defined function (UDF).

---

1. Create the electric material.

   Setup → **Materials** → **Solid** → **aluminum** **Edit...**

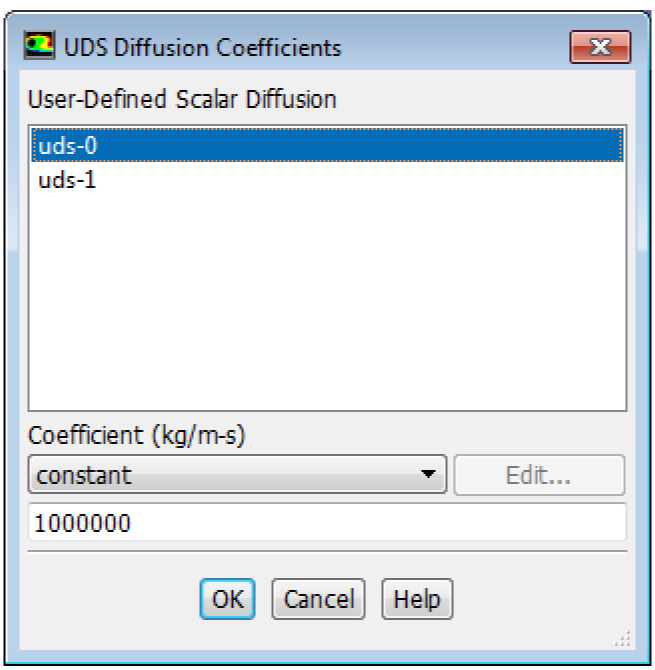


a. In the **Create/Edit Materials** dialog box, enter **`e_material`** for **Name** and **e** for **Chemical Formula**.

b. Set **Thermal Conductivity** to **`20`**.

c. Under **Properties**, ensure that **define-per-uds** is selected from the **UDS Diffusivity** drop-down list and click **Edit...** next to **UDS Diffusivity**.

d. In the **UDS Diffusion Coefficients** dialog box, set the constant value of 1.0 e6 for the both user-defined scalars.



i. Select **uds-0** in the **User-Defined Scalar Diffusion** list.

ii. Retain **constant** from the **Coefficient** drop-down list.

iii. Set **`1.0 e6`** [s/m] for **Coefficient**.

iv. In a similar way, set **uds-1** to `1.0 e6` and close the **UDS Diffusion Coefficients** dialog box.

Note

- The units for UDS Diffusivity are `s/m`, and cannot be modified.
- In this tutorial, the electric conductivities are the same for the two scalars UDS0 and UDS1, so you will need to define only one material for all cell zones. If the electric conductivities are different for positive and negative electrodes, you will need to define two different materials, as described in the Fluent Advanced Add-On Modules documentation on the ANSYS Customer Portal (http://support.ansys.com/documentation).

---

v. In the **Question** dialog box, click **No** to retain **aluminum** and add the new material (e_material) to the materials list.

e. Ensure that `e_material (e)` is selected from the **Fluent Solid Materials** drop-down list.

f. Click **Change/Create**.

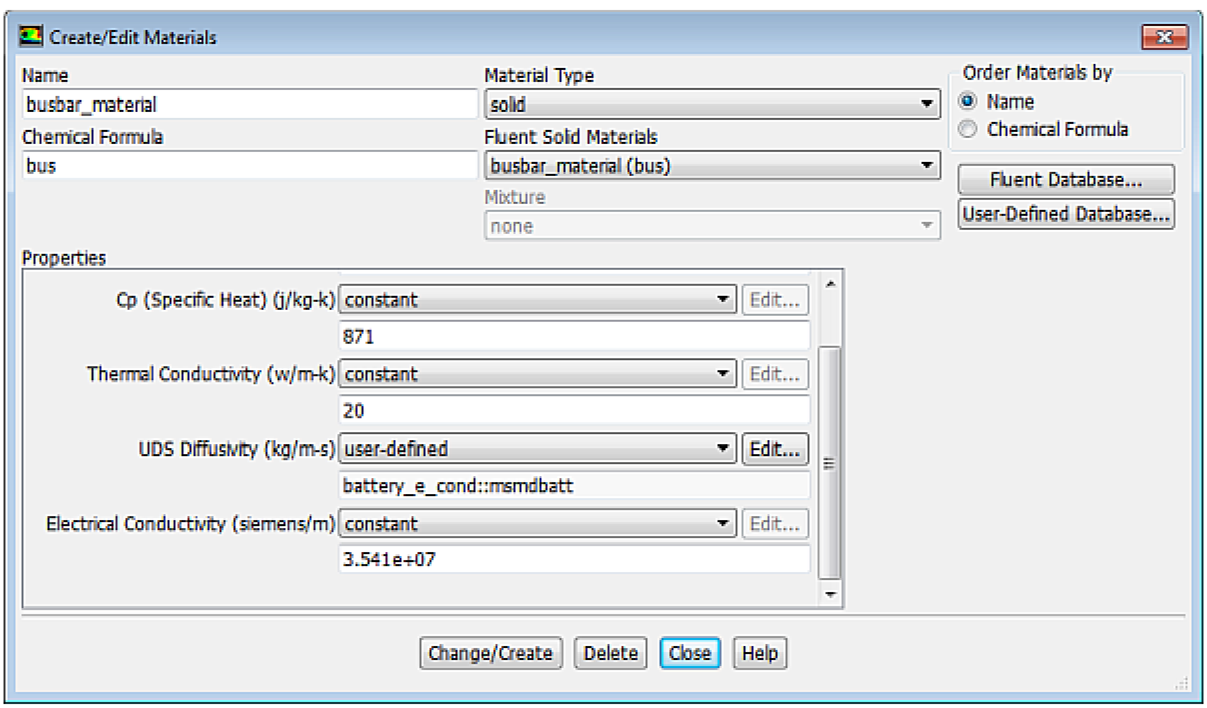
Note

If the UDS Diffusion Coefficients are defined through the `defined-per-uds` option, the Fluent solver does not use the value for **Electrical Conductivity**.

---

2. Create the busbar_material material for busbars and tabs by modifying e-material you have created in the previous step.

Setup → **Materials** → **Solid** → **e-material** **Edit...**



As stated in the problem description, you will use the same material for busbars and tabs.

Note

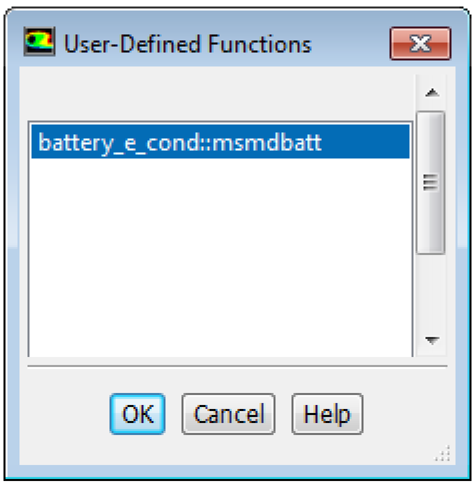
If the busbar and tab materials are different, you need to define the two different materials and assign them to the busbars and tabs, respectively.

The UDS Diffusivity for both busbar and tab materials must be defined through the UDF as described below.

a. In the **Create/Edit Materials** dialog box, enter **`busbar_material`** for **Name** and **`bus`** for **Chemical Formula**.

b. Under **Properties**, select **user-defined** from the **UDS Diffusivity** drop-down list.

c. In the **User-Defined Functions** dialog box that opens, verify that **battery_e_cond::msmdbatt** is selected and click **OK**.



d. In the **Question** dialog box, click **No** to retain **e-material** and add the new busbar_material material to the materials list.

   Fluent hooks the battery_e_cond::msmdbatt UDF to the busbar material.

e. Ensure that `busbar_material (bus)` is selected from the **Fluent Solid Materials** drop-down list.

f. Set **Thermal Conductivity** to **`20`**.

g. Retain the default value of **`3.541 e7`** [1/ohm-m] for **Electrical Conductivity**.

h. Click **Change/Create**.

i. Close the **Create/Edit Materials** dialog box.

25.4.4.3. Defining Cell Zone Conditions

Assign e_material to all the cell zones and busbar_material to all the tabs and busbars.

1. Assign e_material to the cell_1 cell zone.

Setup → **Cell Zone Conditions** → **cell_1** → **Edit...**

a. In the **Solid** dialog box, select `e_material` from the **Material Name** drop-down list.

b. Click **OK**.

2. Copy the cell zone condition for the cell_1 zone to the cell_2 and cell_3 cell zones.

Setup → **Cell Zone Conditions** → **cell_1** → **Copy...**

a. In the **Copy Conditions** dialog box, select `cell_1` in the **From Cell Zone** list.

b. In the **To Cell Zones** list, select `cell_2` and `cell_3`.

c. Click **Copy**.

d. Click **OK** in the **Question** dialog box to copy the cell zone conditions and close the **Copy Conditions** dialog box.

3. In a similar manner, assign busbar_material to all the tabs and busbars cell zones.

25.4.4.4. Defining Boundary Conditions

Define the thermal boundary conditions for all walls for the cells, busbars, and tabs. The boundary conditions for the two UDSs have been set automatically when you defined the cell zone conditions.

1. Set the convection boundary condition for wall-cell_1.

Setting Up Physics → **Zones** → **Boundaries**

a. In the **Boundary Conditions** task page, double-click **wall-cell_1**.

b. In the **Wall** dialog box, under the **Thermal** tab, configure the following settings:

i. Under **Thermal Conditions**, enable **Convention**.

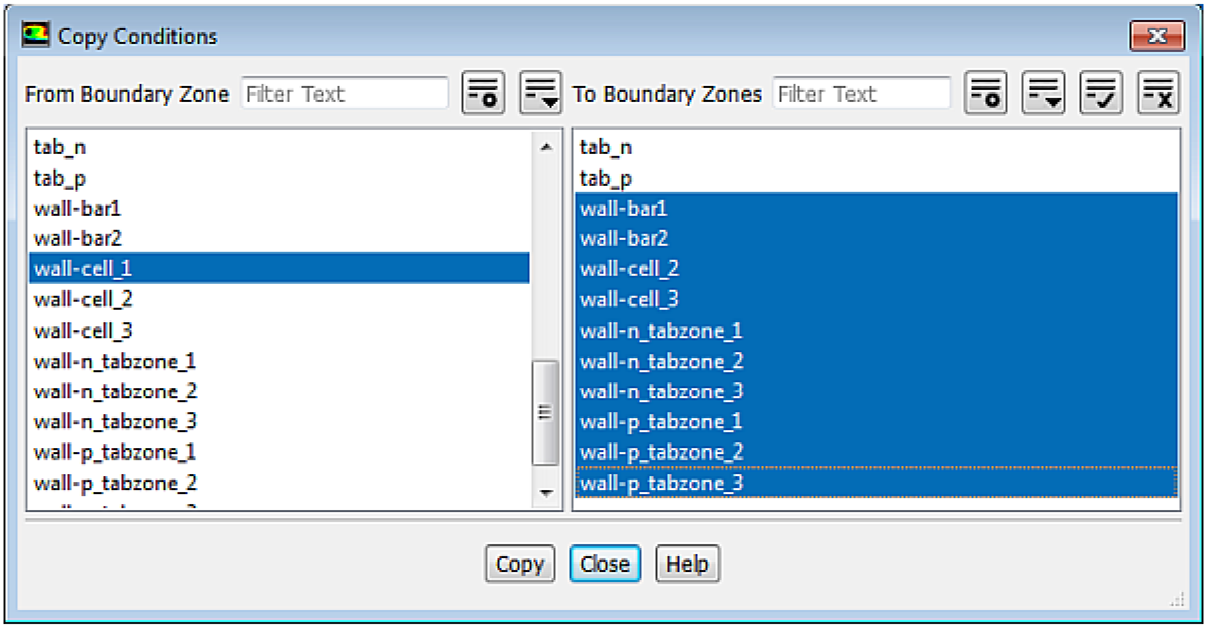
ii. Set **Heat Transfer Coefficient** to **5 [w/m^{2}K]**.

iii. Set **Free Stream Temperature** to **300 [K]**.

iv. Click **OK** to close the **Wall** dialog box.

2. Copy the boundary conditions for `wall-cell_1` to `wall-cell_2`, `wall-cell_3` and all the tab and busbar wall zones (a boundary zones that have names starting with the "wall" string and containing the "bar" or "tabzone" string).

Setup → **Boudnary Conditions** → **wall-cell_1** → **Copy...**



25.4.4.5. Specifying Solution Settings

1. Turn off the flow equation.

   Solving → **Controls** → **Equations...**

   a. In the **Equations** dialog box, deselect `Flow` from the **Equation** selection list.

   b. Click **OK**.

2. Remove the convergence criteria to ensure that automatic convergence checking does not occur.

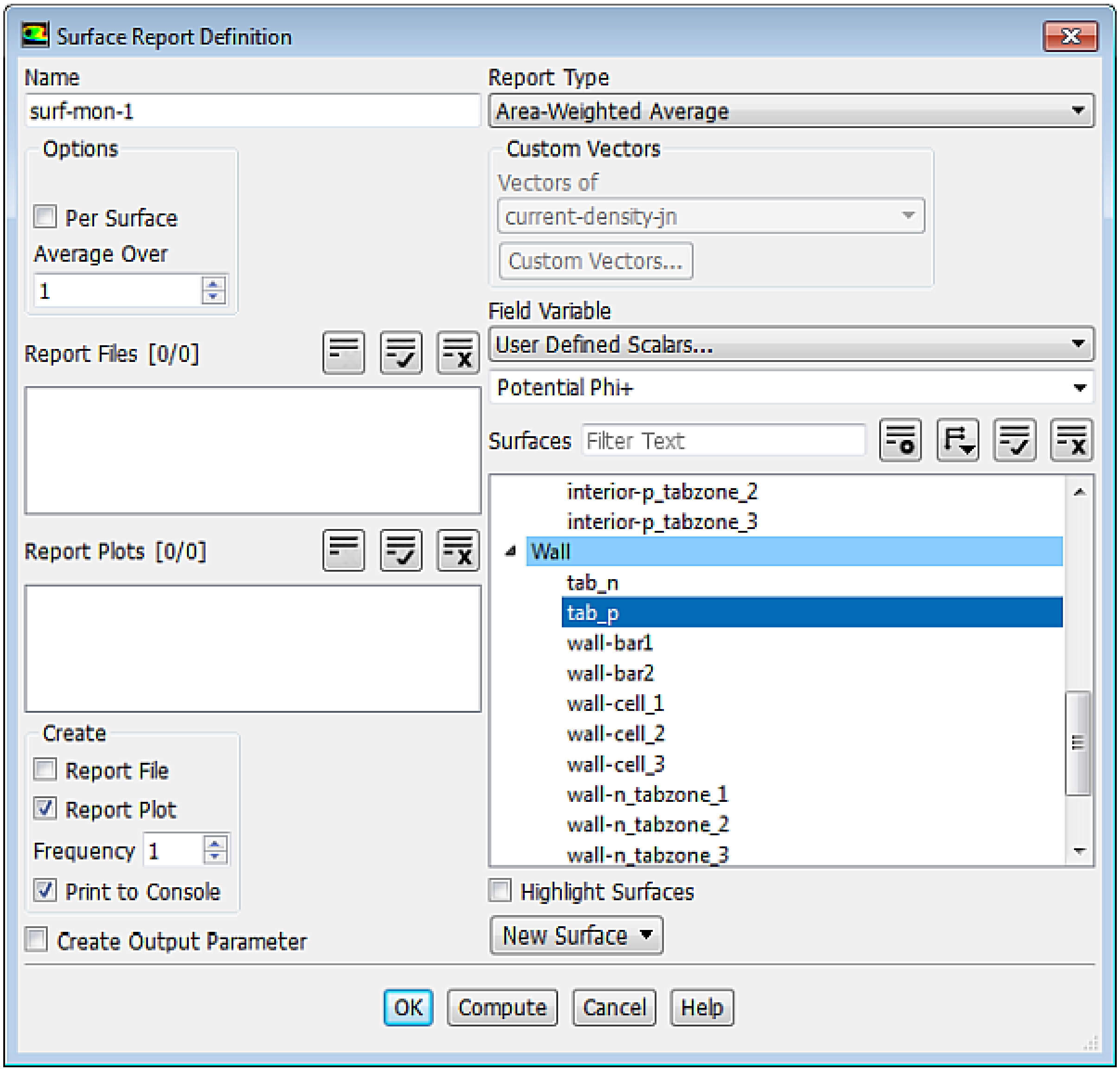
   Solving → **Reports** → **Residuals...**

   a. In the **Residual Monitors** dialog box, select **none** from the **Convergence Criterion** drop-down list.

   b. Click **OK**.

3. Create a surface report definition for the voltage at the positive tab.

   Solving → **Reports** → **Definitions** → **New** → **Surface Report** → **Area-Weighted Average**



a. In the **Surface Report Definition** dialog box, enter **`surf-mon-1`** for **Name**.

b. Select **User Defined Scalars...** and **Potential Phi+** from the **Field Variable** drop-down lists.

c. From the **Surfaces** selection list, select `tab_p`.

d. In the **Create** group box, enable **Report Plot** and **Print to Console**.

e. Click **OK** to save the surface report definition and close the **Surface Report Definition** dialog box.

f. Modify the attributes of the plot axes.

Solution → **Monitors** → **Report Plots** → **surf-mon-1-rplot** **Edit...**

i. In the **Edit Report Plot** dialog box, under the **Plot Window** group box, click the **Axes...** button to open the **Axes** dialog box.

ii. Select the **X** axis and set **Precision** to **`0`**.

iii. Click **Apply**.

iv. Select the **Y** axis and set **Precision** to **2**.

v. Set **Precision** to 2.

vi. Click **Apply** and close the **Axes** dialog box.

Note

You must click **Apply** to save the modified settings for each axis.

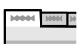
---

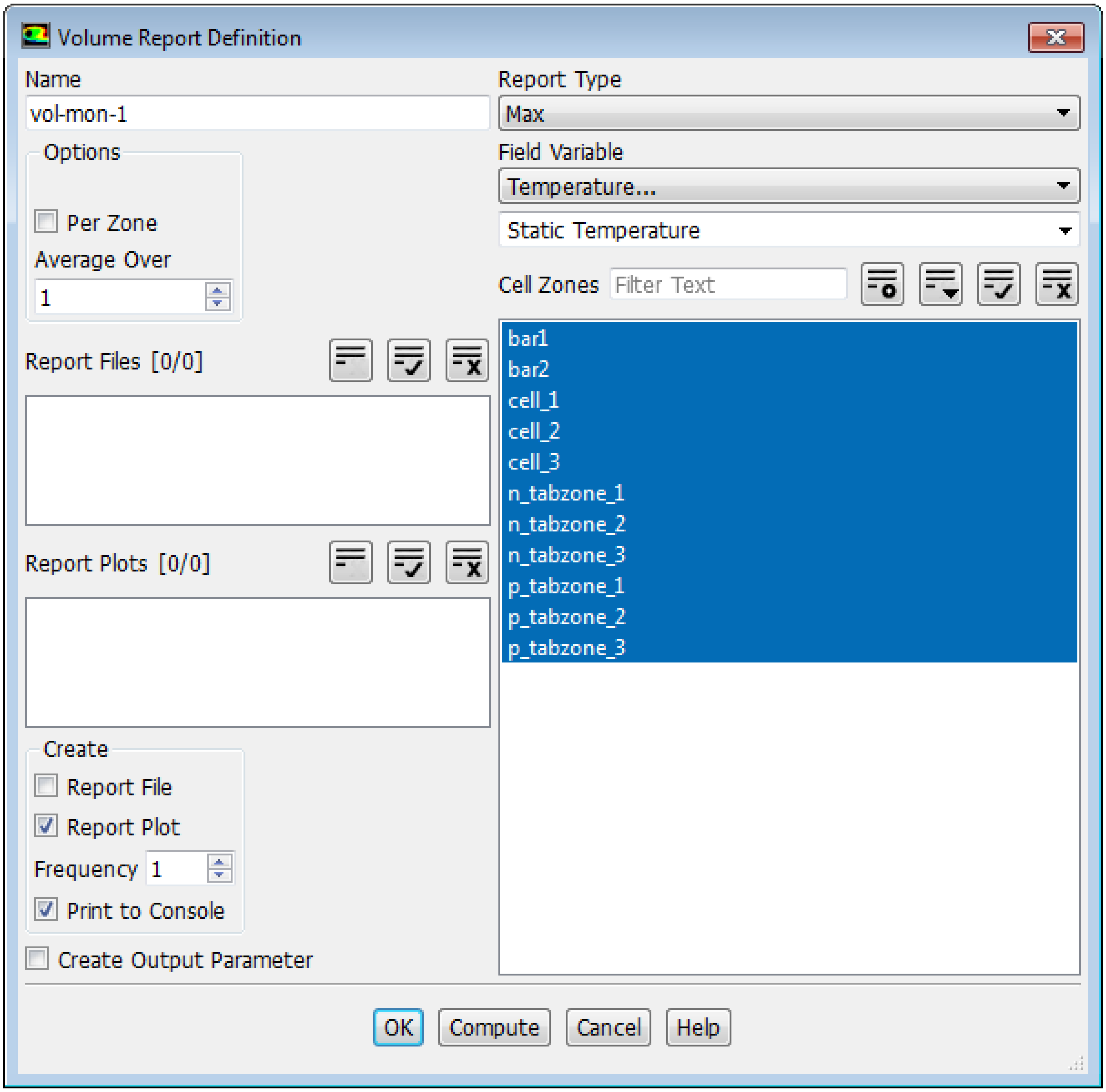
vii. Click OK to close the **Edit Report Plot** dialog box.

g. Ensure that **time-step** is selected from the **Get Data Every** drop-down list.

h. Click **OK**.

4. Create a volume report definition to monitor the maximum temperature in the domain.

 **Solving** → **Reports** → **Definitions** → **New** → **Volume Report** → **Max...**



a. In the **Volume Report Definition** dialog box, enter `vol-mon-1` for **Name**.

b. Select **Temperature...** and **Static Temperature** from the **Field Variable** drop-down lists.

c. From the **Cell Zones** selection list, select all zones.

d. In the **Create** group box, enable **Report Plot** and **Print to Console**.

e. Click **OK** to save the volume report definition settings and close the **Volume Report Definition** dialog box.

f. Modify the attributes of the plot axes.

Solution → **Monitors** → **Report Plots** → **vol-mon-1-rplot** **Edit...**

i. In the **Edit Report Plot** dialog box, under the **Plot Window** group box, click the **Axes...** button to open the **Axes** dialog box.

ii. Select the **X** axis and set **Precision** to **0**.

iii. Click **Apply**.

iv. Select the **Y** axis and set **Precision** to **2**.

v. Click **Apply** and close the **Axes** dialog box.

vi. Ensure that **time-step** is selected from the **Get Data Every** drop-down list.

vii. Click OK to close the **Edit Report Plot** dialog box.

5. Save the case file.

   File → **Write** → **Case...**

25.4.4.6. Obtaining Solution

1. Initialize the field variables using the **Standard Initialization** method.

   Solution → **Solution Initialization**

   a. Retain the selection of **Standard** from the **Initialization Methods** group box.

   b. Click **Initialize**.

      You do not need to modify the **Initial Values** in the **Solution Initialization** task page, because these values are not used for initialization. The ANSYS Fluent solver automatically computes the initial condition for UDS0 and UDS1.

2. Run the simulation.

   Solving → **Run Calculation**

   a. Set **Time Step Size** to **30** seconds and **No. of Time Steps** to **50**.

   b. Click **Calculate** and run the simulation up to 1500 seconds.

      The residual plot, the history of the voltage at the positive tap and the history of the maximum temperature in the domain are shown in Figure 25.5: Residual History of the Simulation (p. 1023), Figure 25.6: Surface Report Plot of Discharge Curve at 200W (p. 1023), and Figure 25.7: Volume Report Plot of Maximum Temperature in the Domain (p. 1024), respectively.

Figure 25.5: Residual History of the Simulation

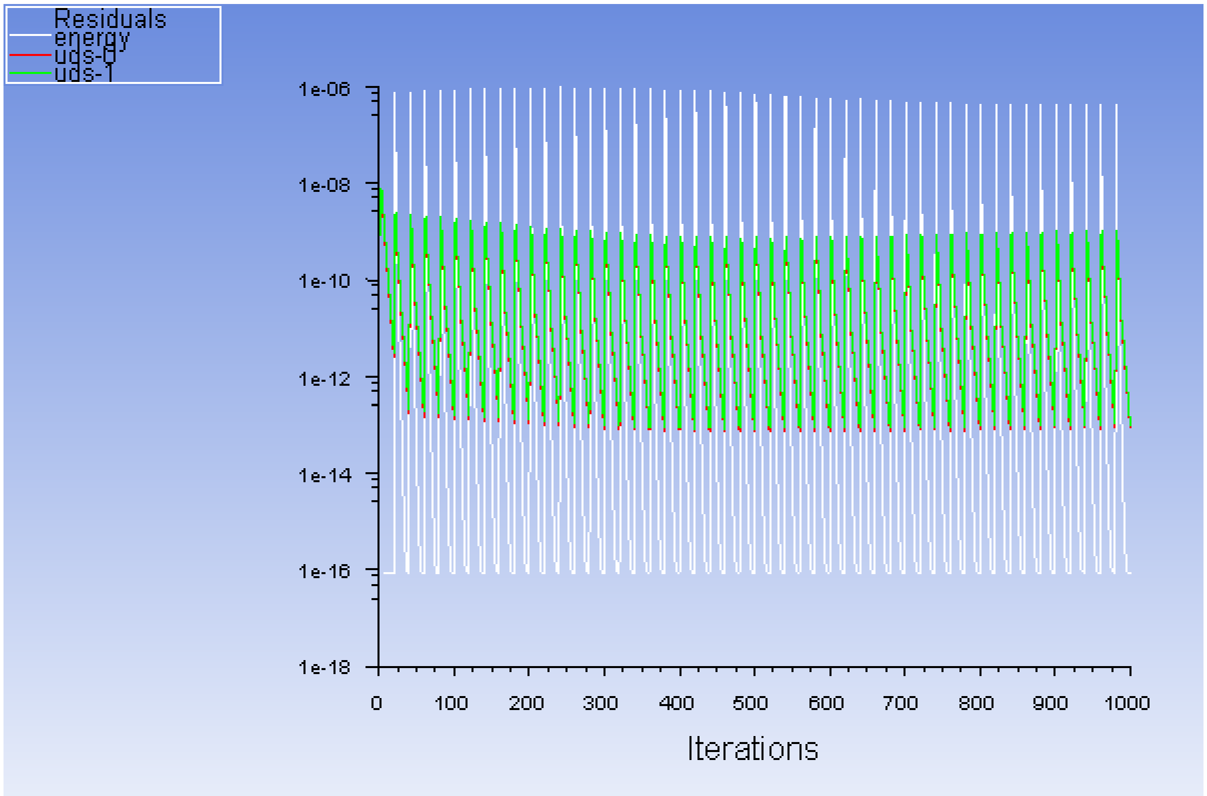


Figure 25.6: Surface Report Plot of Discharge Curve at 200W

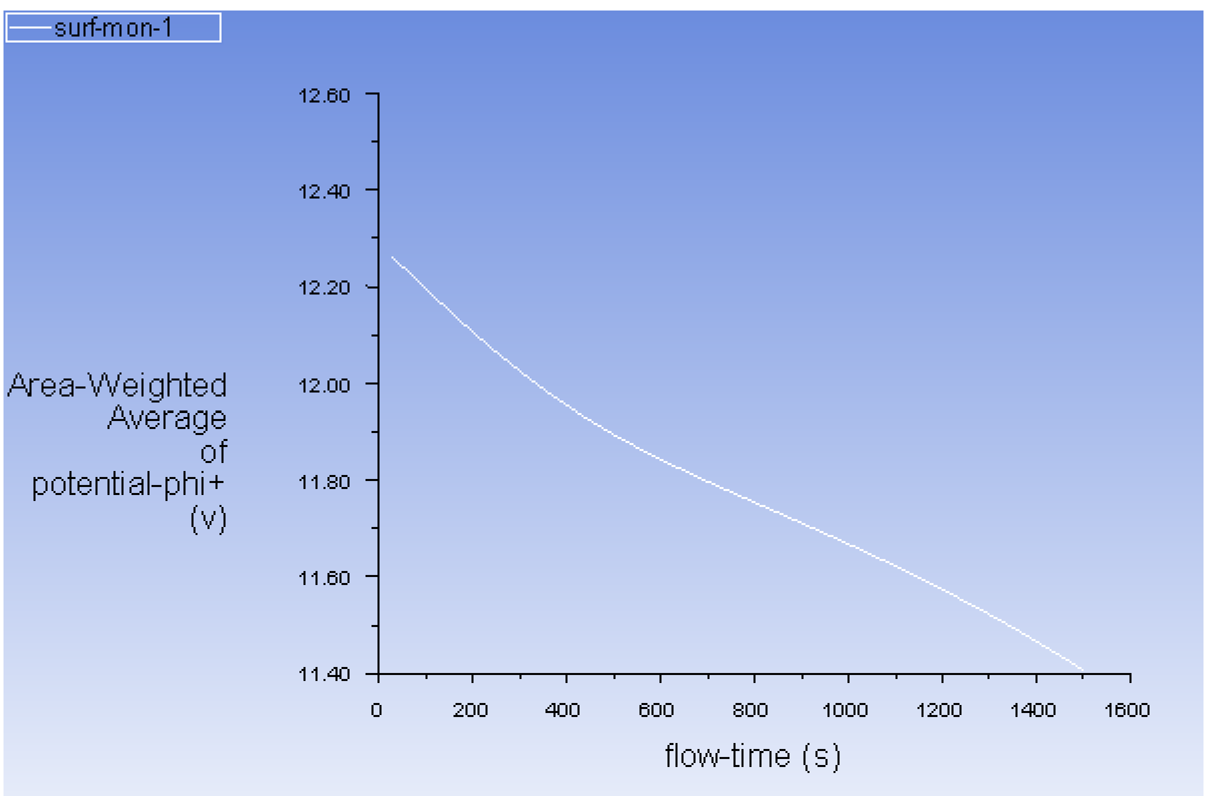
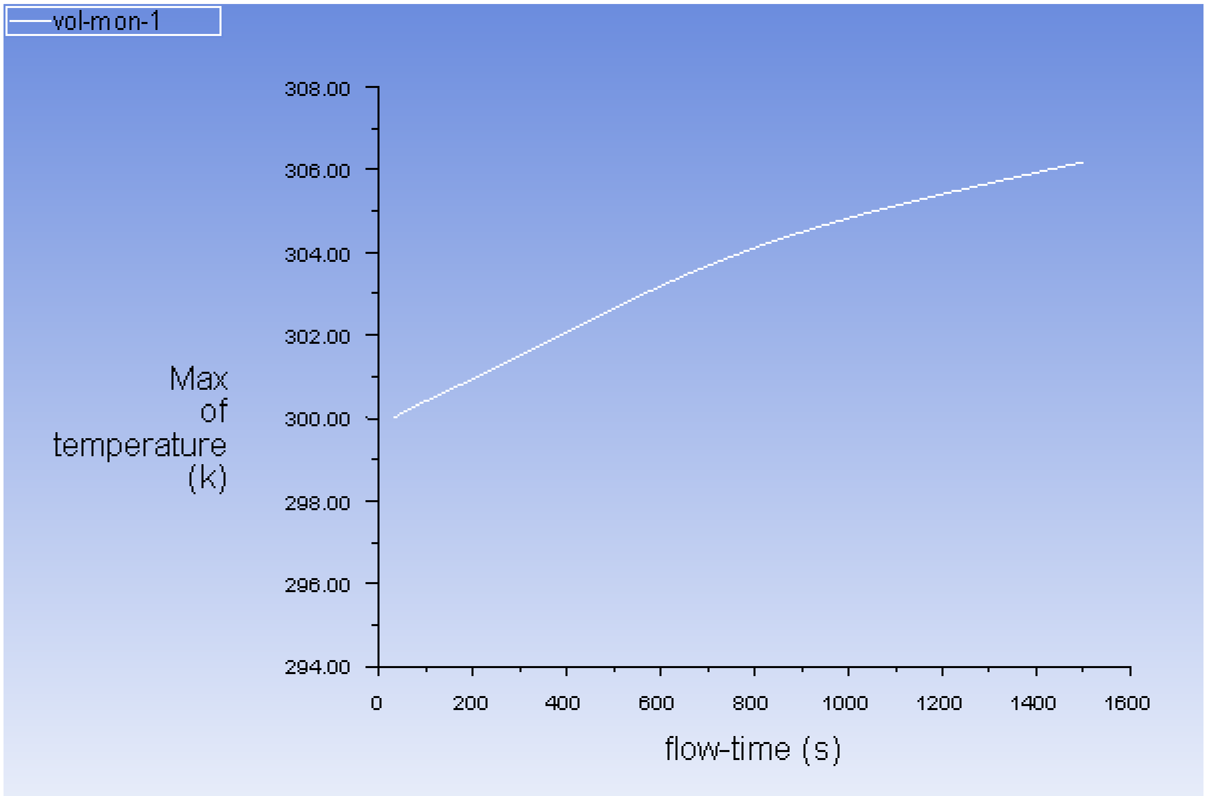


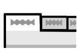
Figure 25.7: Volume Report Plot of Maximum Temperature in the Domain

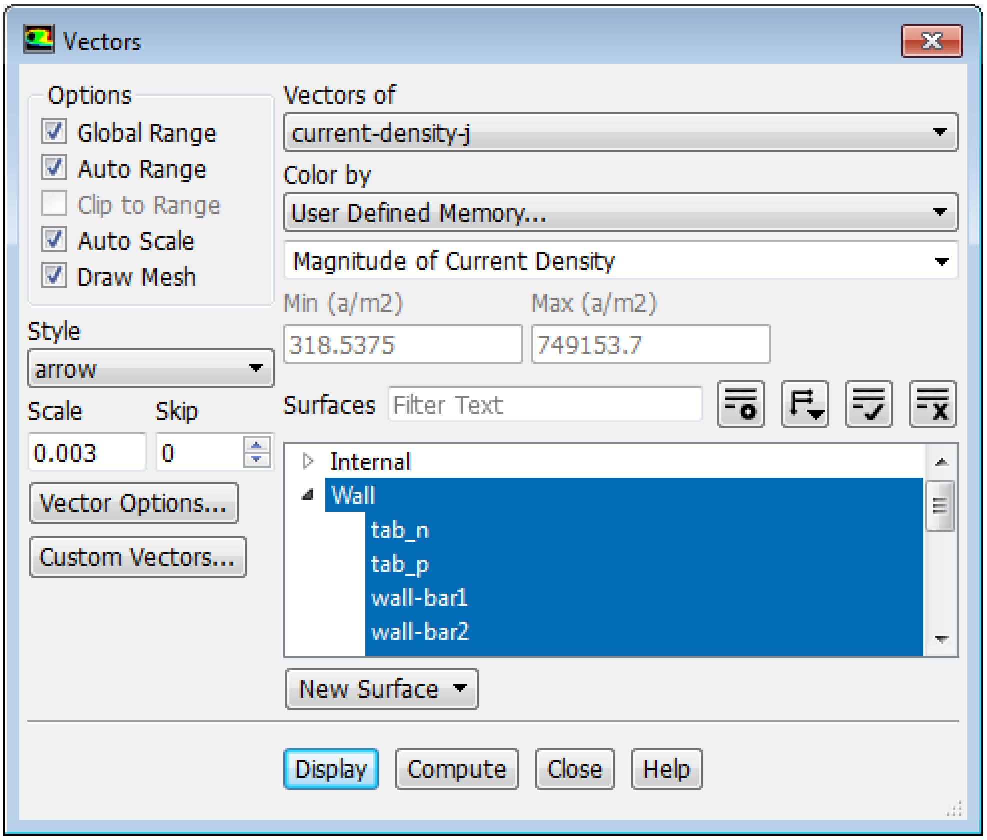


25.4.5. Postprocessing

In this section, postprocessing options for the MSMD battery model solution are presented.

1. Display the vector plot of current density.

    **Postprocessing** → **Graphics** → **Vectors** → **Edit...**



a. In the **Vectors** dialog box, select **current-density-j** from the **Vectors of** drop-down list.

b. Select **User Defined Memory...** and **Magnitude of Current Density** from the **Color by** drop-down list.

c. Click the **Toggle Tree View** button next to the **Surfaces** filter and from the drop-down list, select **Surface Type** (under **Group By**).

d. From the **Surfaces** selection list, select **Wall**.

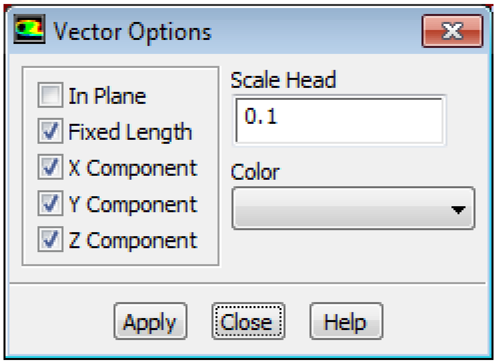
   The surfaces of the "wall" type are automatically selected in the **Surfaces** list.

e. In the **Options** group, enable **Draw Mesh** and set the mesh display options as desired.

f. Set Scale to `0.003`.

g. Click **Vector Options...**.

   i. In the **Vector Options** dialog box, enable **Fixed Length**.

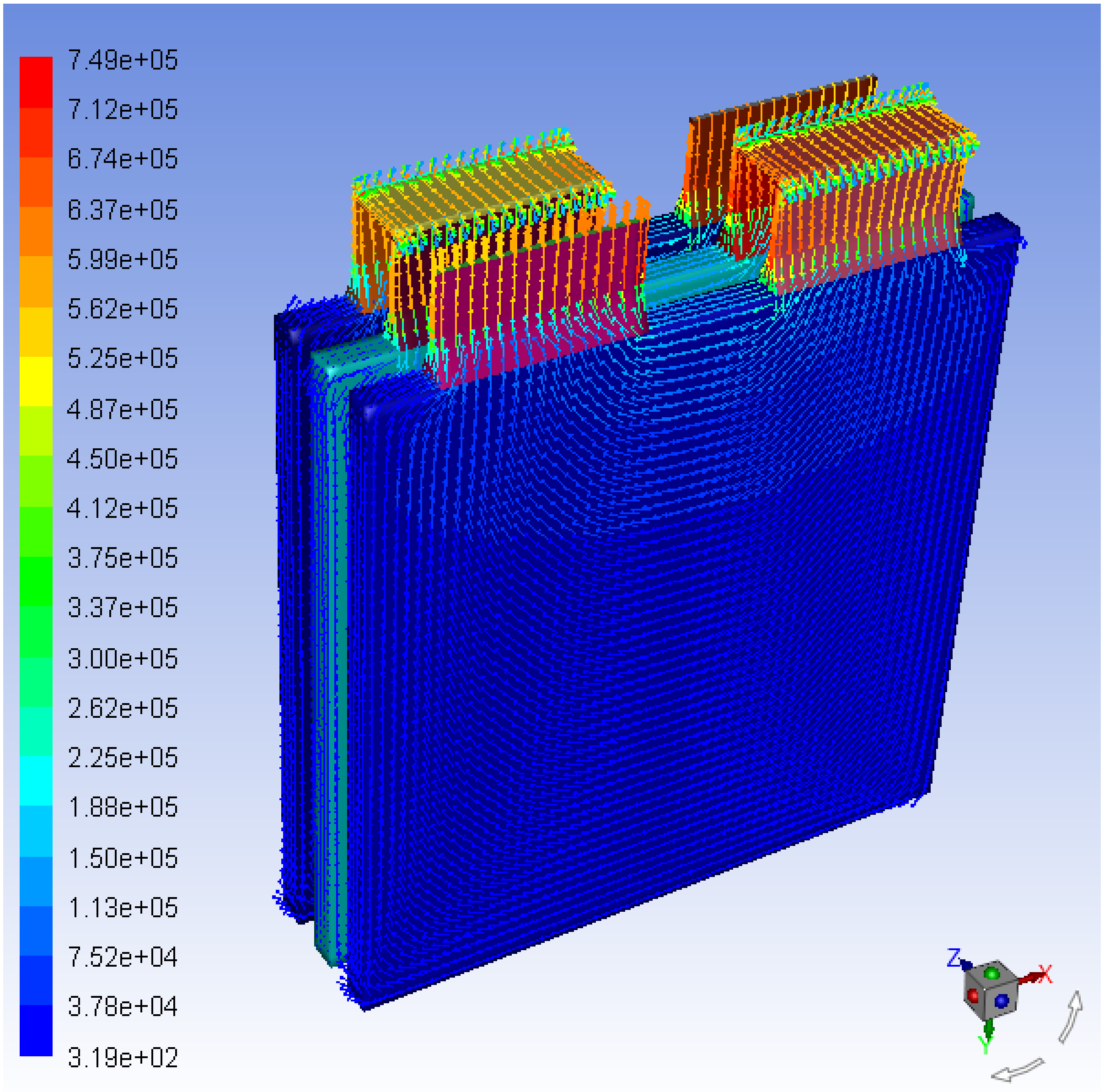


All vectors in your plot will be displayed with the same lengths.

ii. Click **Apply** and close the **Vector Options** dialog box.

h. Click **Display** and close the **Contours** dialog box.

Figure 25.8: Vector Plot of Current Density

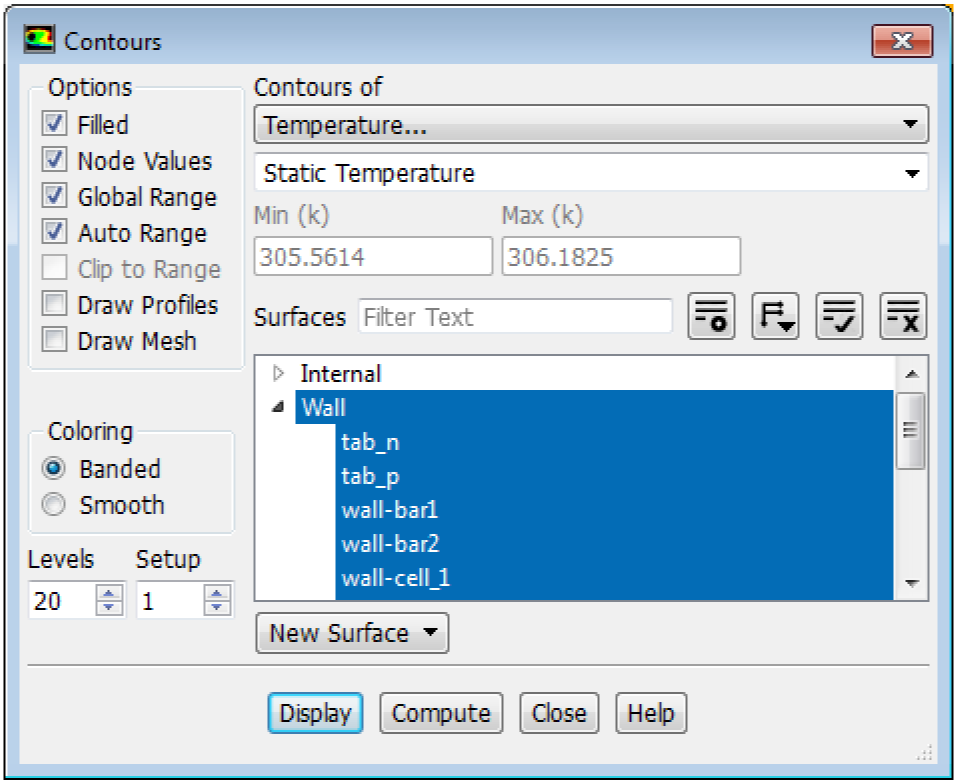


Note

Use the **Headlight** and **Lighting** display options under the **Viewing** ribbon tab to manipulate the graphics display.

2. Display the contour plot of the temperature.

Postprocessing → **Graphics** → **Contours** → **Edit...**



a. In the **Contours** dialog box, in the **Options** group box, enable **Filled**.

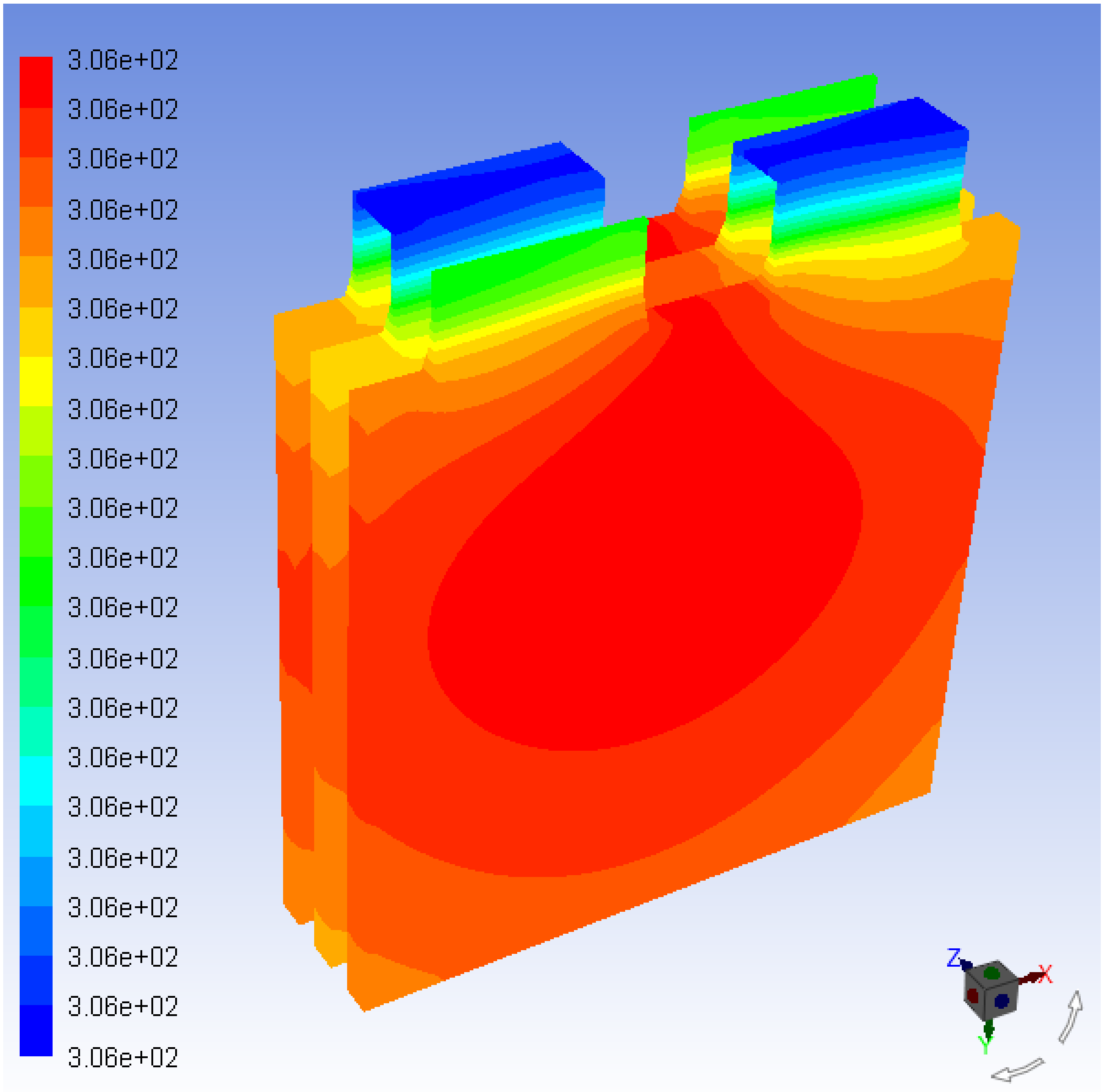
b. From the **Contours of** drop-down list, select **Temperature...** and **Static Temperature**.

c. Click the **Toggle Tree View** button next to the **Surfaces** filter and from the drop-down list, select **Surface Type** (under **Group By**).

d. From the **Surfaces** selection list, select **Wall**.

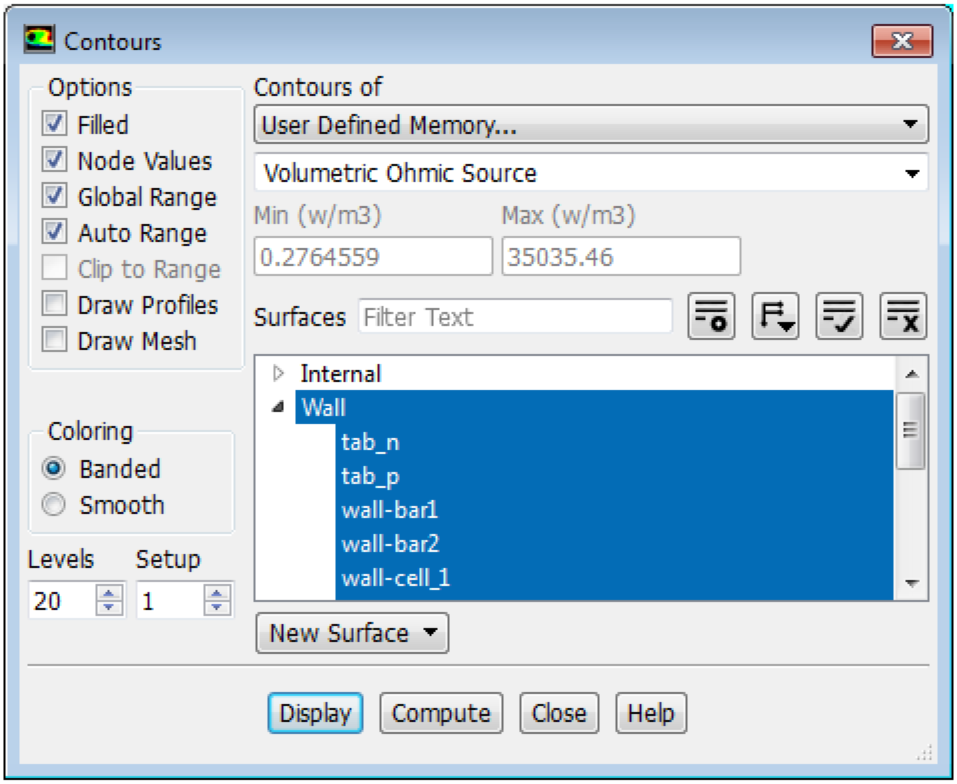
e. Click **Display** and close the **Contours** dialog box.

Figure 25.9: Contour Plot of Temperature



3. In a similar manner, display the contour of Ohmic heat source.

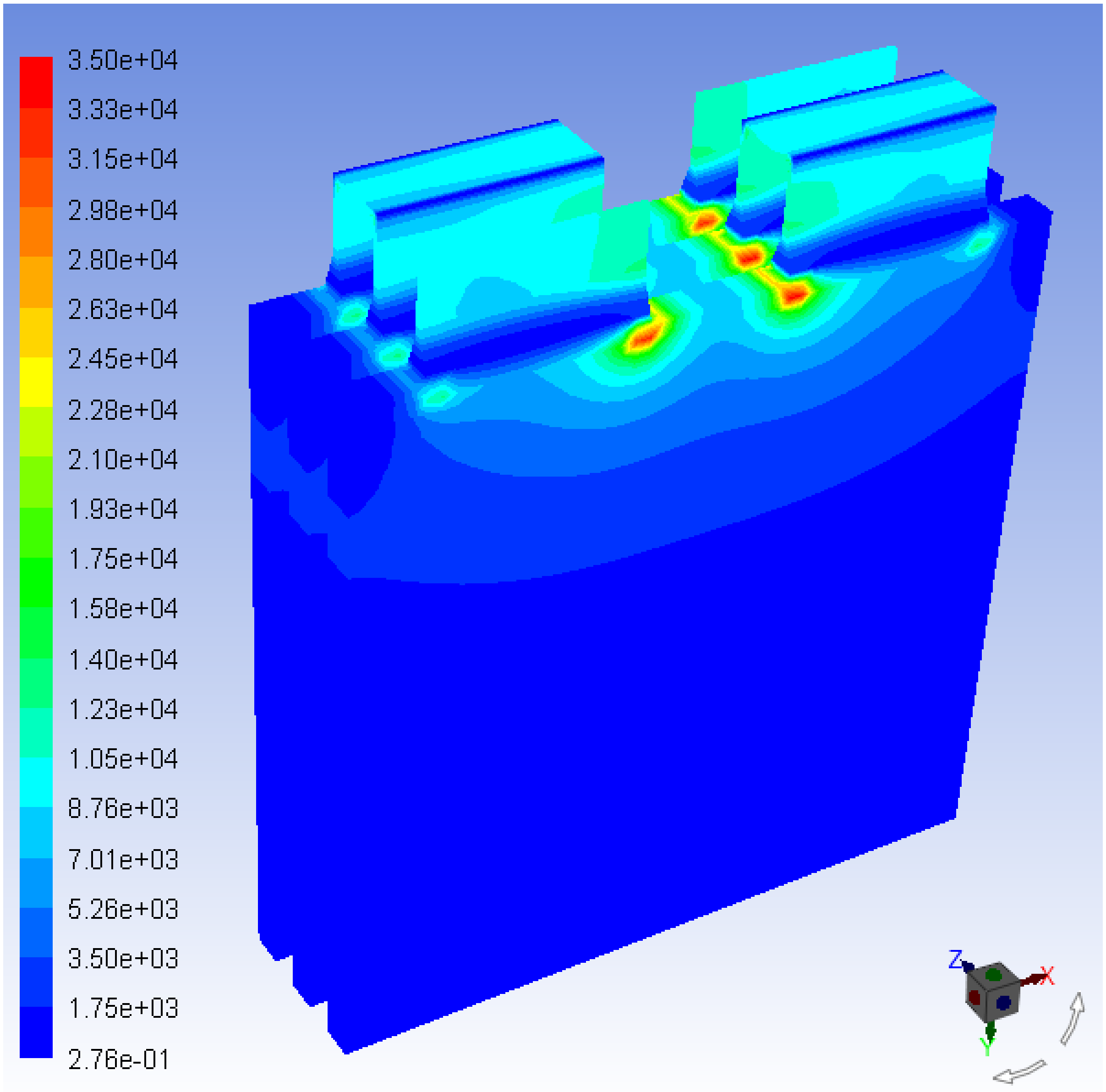
   Postprocessing → Graphics → Contours → Edit...



a. From the **Contours of** drop-down list, select **User Defined Memory...** and **Volumetric Ohmic Source**.

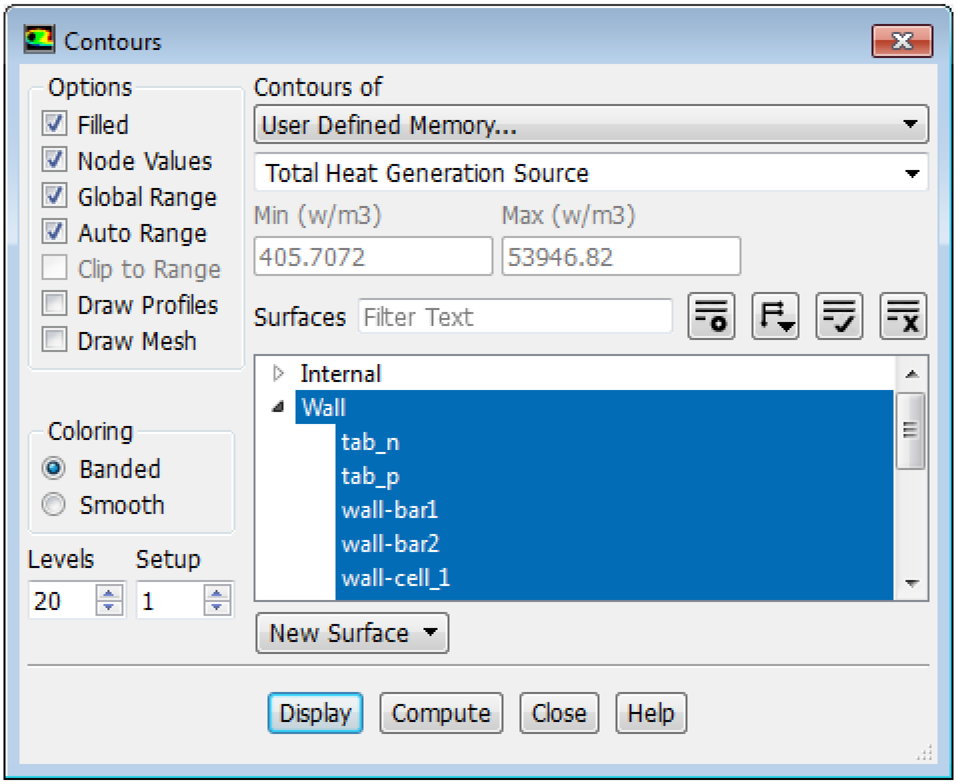
b. Click **Display**.

Figure 25.10: Ohmic Heat Generation Rate



4. In a similar manner, display the contour of the total heat source.

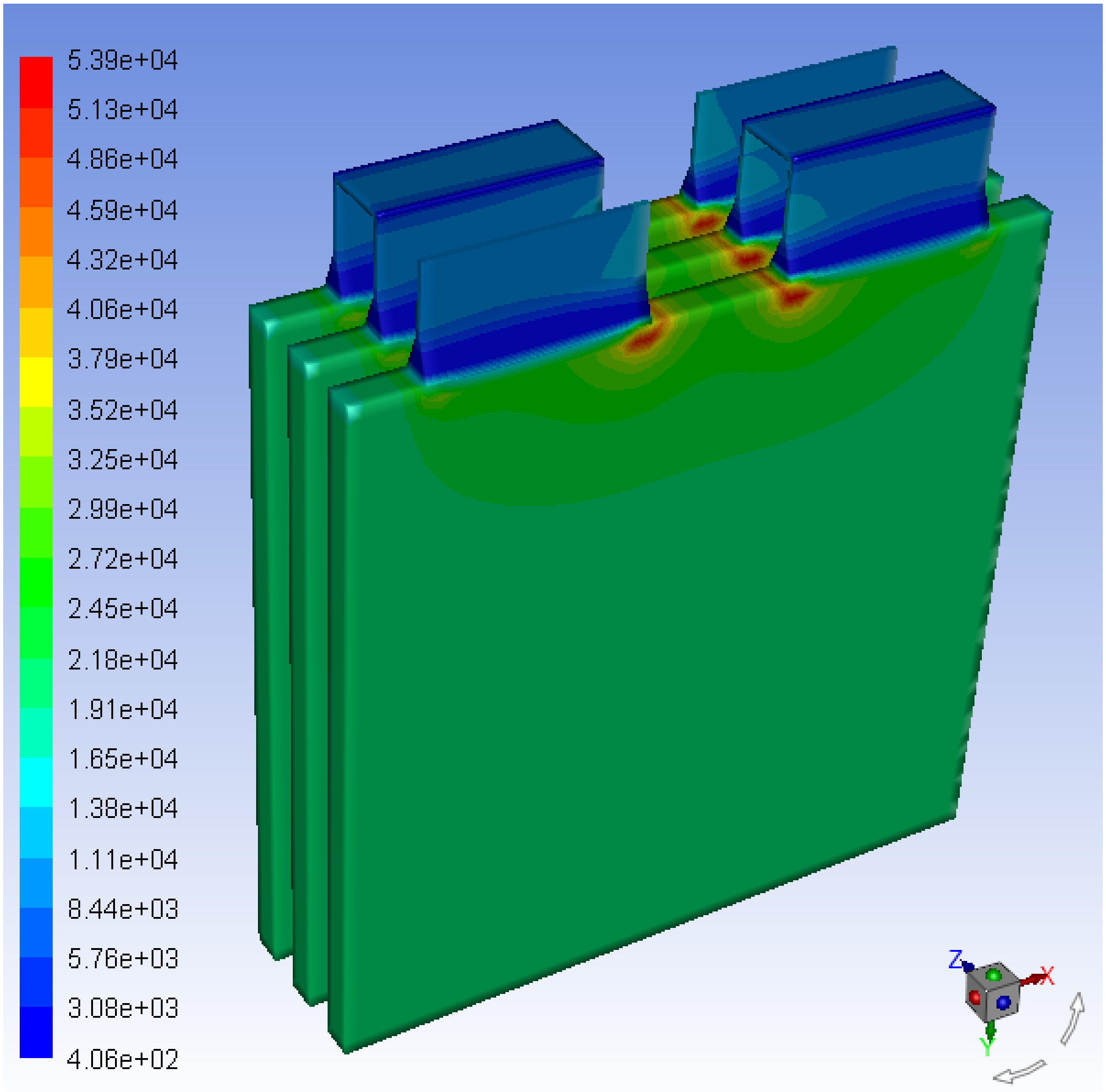
   Postprocessing → **Graphics** → **Contours** → **Edit...**



a. From the **Contours of** drop-down list, select **User Defined Memory...** and **Total Heat Generation Source**.

b. Click **Display**.

Figure 25.11: Total Heat Generation Rate



25.5. Summary

This tutorial has demonstrated the use of the MSMD battery add-on to perform electrochemical and heat transfer simulations for battery packs. You have learned how to set up and solve the problem for the battery pack of the 1P3S configuration using the NTGK Battery submodel. You have also learned some of the postprocessing capabilities available in the MSMD battery model.

As an exercise, you can obtain solutions using the Equivalent Circuit Model and Newman P2D Model. For details, see the Fluent Advanced Add-On Modules documentation on the ANSYS Customer Portal (http://support.ansys.com/documentation).