Chapter 15: Using Dynamic Meshes

This tutorial is divided into the following sections:

15.1. 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. In this tutorial, such a case will be used to demonstrate the layering feature of the dynamic mesh capability in ANSYS Fluent.

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, the check valve is connected to a spring that acts to push the valve to the valve seat and to shut the flow. But when the pressure force on the valve is greater than the spring force, the valve 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 valve is typically neglected, thus allowing for a rigid body Fluid Structure Interaction (FSI) calculation, for which a user-defined function (UDF) is provided.

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.
- Compile a User-Defined Function (UDF) to specify flow-driven rigid-body motion.
- Calculate a solution using the pressure-based solver.

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)

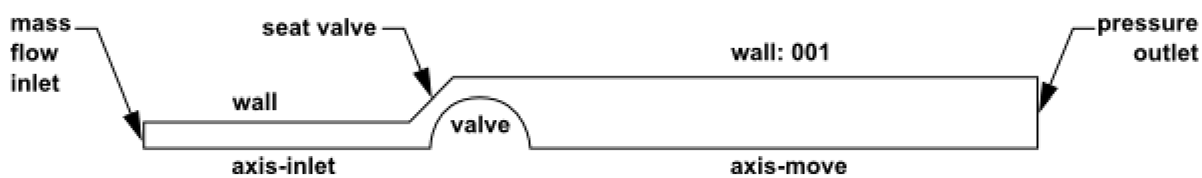
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 123)

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

15.3. Problem Description

The check valve problem to be considered is shown schematically in Figure 15.1: Problem Specification (p. 632). A 2D axisymmetric valve geometry is used, consisting of a mass flow inlet on the left, and a pressure outlet on the right, driving the motion of a valve. In this case, the transient motion of the valve due to spring force, gravity, and hydrodynamic force is studied. Note, however, that the valve in this case is not completely closed. Since dynamic mesh problems require that at least one layer remains in order to maintain the topology, a small gap will be created between the valve and the valve seat.

Figure 15.1: Problem Specification



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 **15.0** under **Version**.

5. Select this tutorial from the list.

6. Click **Files** to download the input and solution files.

7. Unzip `dynamic_mesh_R150.zip` to your working folder.

   The mesh and source files `valve.msh` and `valve.c` can be found in the `dynamic_mesh` directory created after unzipping the file.

   A user-defined function will be used to define the rigid-body motion of the valve geometry. This function has already been written (`valve.c`). You will only need to compile it within ANSYS Fluent.

8. Use the Fluent Launcher to start the **2D** version of ANSYS Fluent.

   Fluent Launcher displays your **Display Options** preferences from the previous session.

   Note that this tutorial has been generated using single precision, so you should ensure that **Double Precision** is disabled if you want to match the tutorial setup exactly.

   For more information about Fluent Launcher, see Starting ANSYS Fluent Using Fluent Launcher in the User's Guide.

9. Ensure that the **Display Mesh After Reading**, **Embed Graphics Windows**, and **Workbench Color Scheme** options are enabled.

10. Run in **Serial** under **Processing Options**.

15.4.2. Mesh

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

   File → **Read** → **Mesh...**

15.4.3. General Settings

1. Check the mesh.

   General → **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 mm.

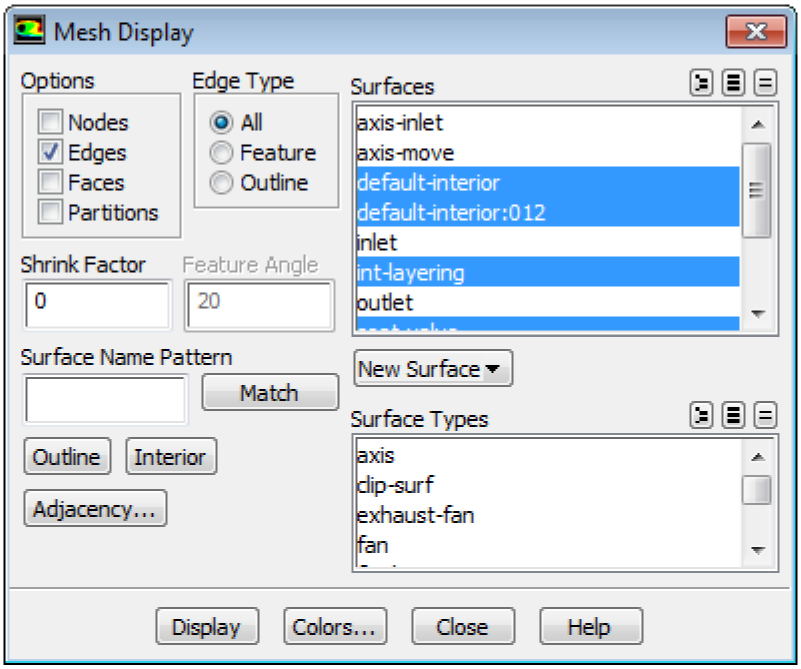
   General → **Units...**

   a. In the **Set Units** dialog box select **length** under **Quantities** and **mm** under **Units**.

   b. Close the **Set Units** dialog box.

3. Display the mesh (Figure 15.2: Initial Mesh for the Valve (p. 635)).

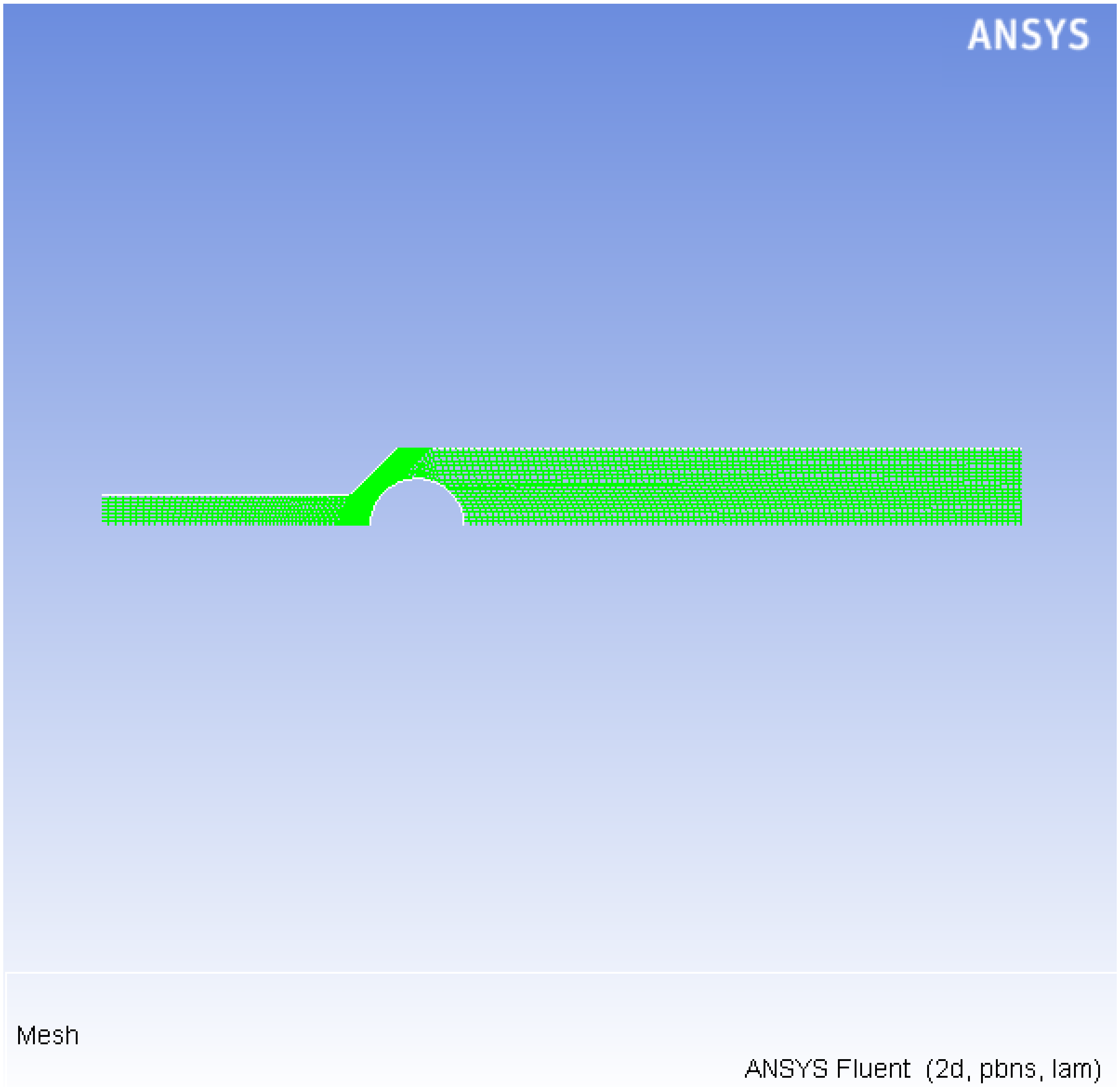
   General → **Display...**



   a. Deselect **axis-inlet**, **axis-move**, **inlet**, and **outlet** from the **Surfaces** selection list.

   b. Click **Display**.

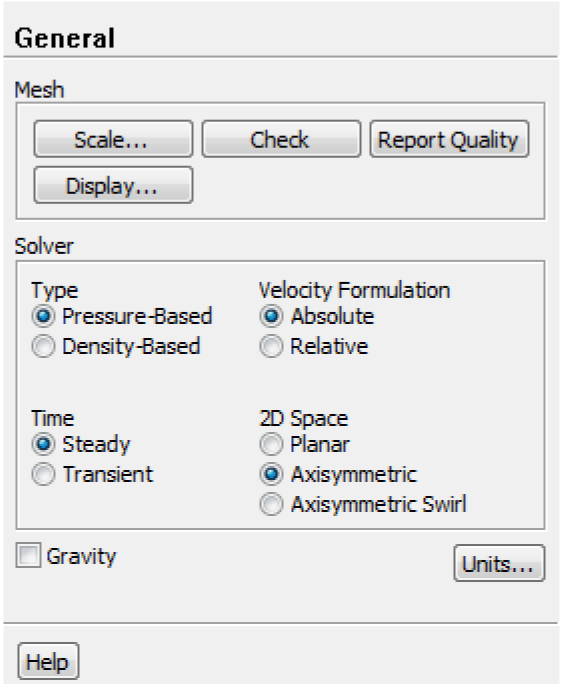
Figure 15.2: Initial Mesh for the Valve



c. Close the **Mesh Display** dialog box.

4. Enable an axisymmetric steady-state calculation.

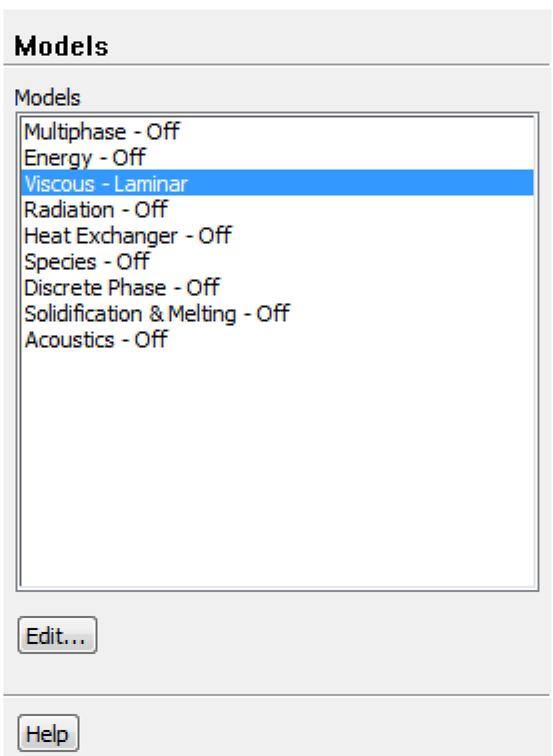
General



a. Select **Axisymmetric** from the **2D Space** list.

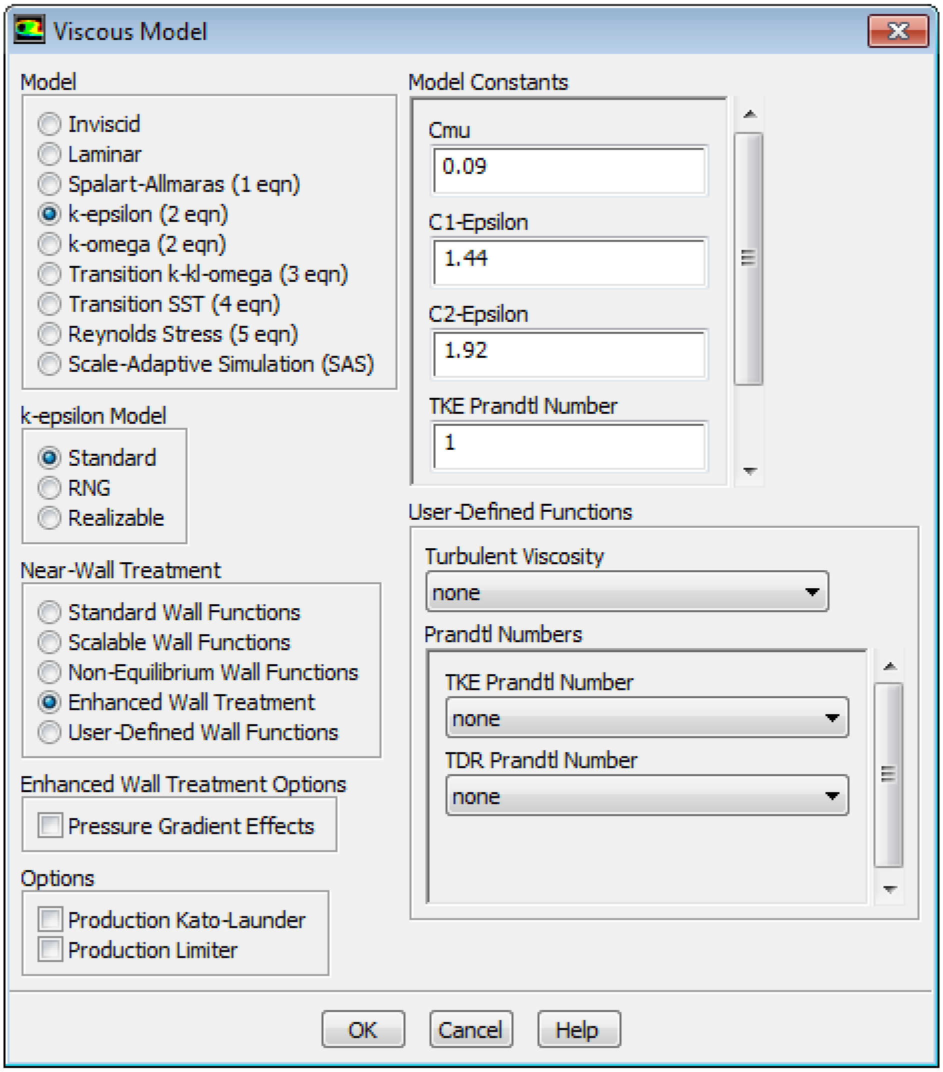
15.4.4. Models

Models



1. Enable the standard k- ε turbulence model.

Models → **Viscous** → **Edit...**



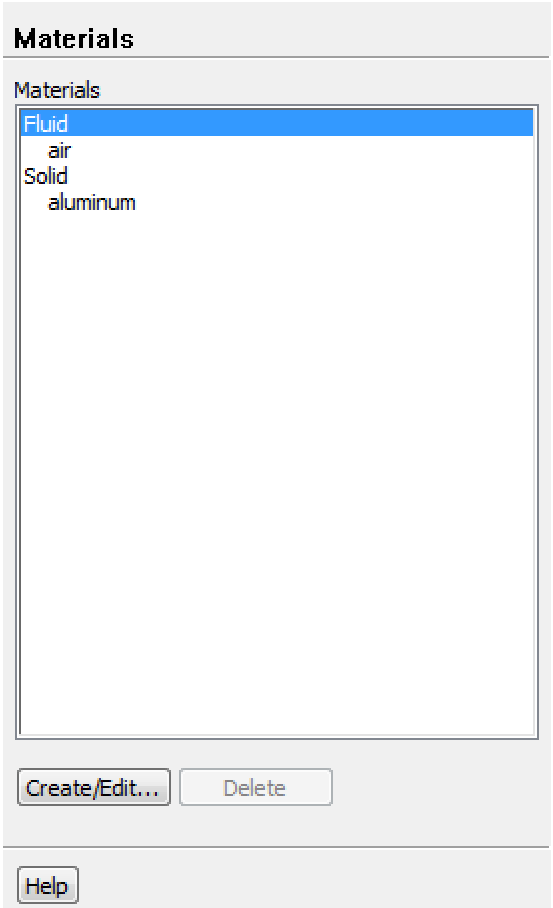
a. Select **k-epsilon (2 eqn)** from the **Model** list and retain the default selection of **Standard** in the **k-epsilon Model** group box.

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

c. Click **OK** to close the **Viscous Model** dialog box.

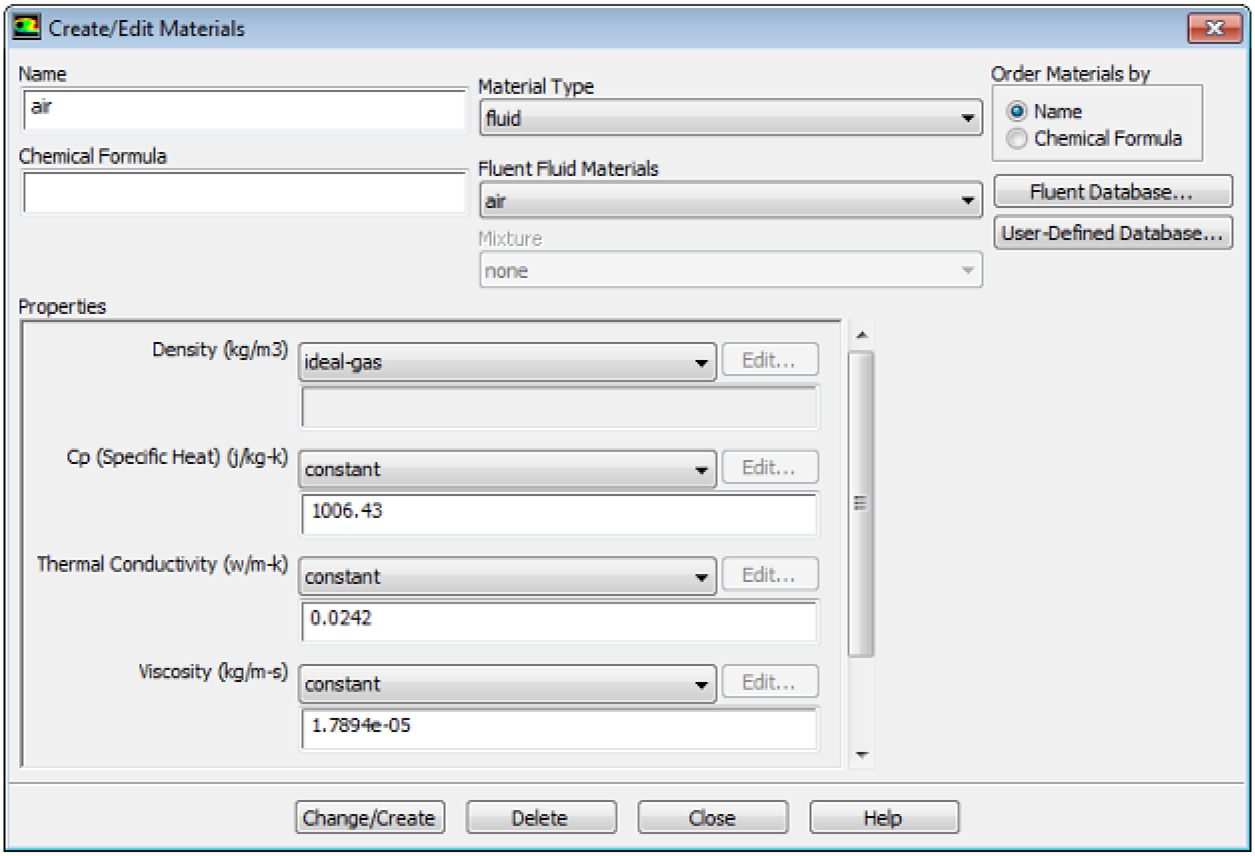
15.4.5. Materials

Materials



1. Apply the ideal gas law for the incoming air stream.

   Materials → **Fluid** → **Create/Edit...**



a. Select **ideal-gas** from the **Density** drop-down list.

b. Click **Change/Create**.

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

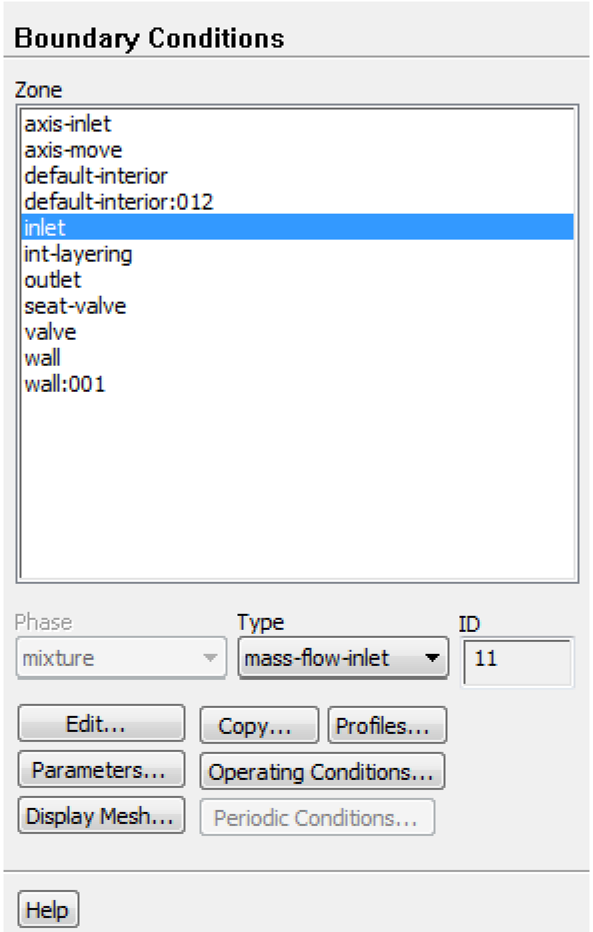
15.4.6. Boundary Conditions

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**).

   Boundary Conditions → **inlet**

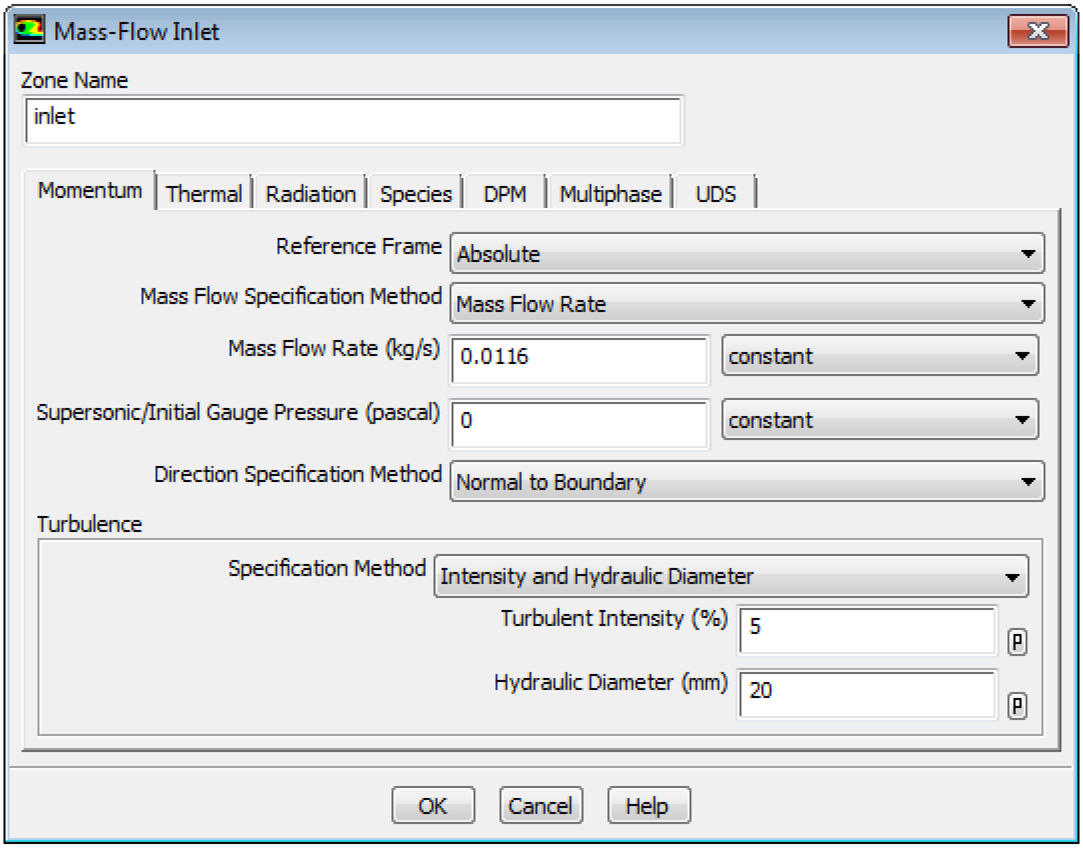
   Since the inlet boundary is assigned to a wall boundary type in the original mesh, you will need to explicitly assign the inlet boundary to a mass flow inlet boundary type in ANSYS Fluent.



a. Select **mass-flow-inlet** from the **Type** drop-down list in the **Boundary Conditions** task page.

b. Click **Yes** when ANSYS Fluent asks you if you want to change the zone type.

*The **Mass-Flow Inlet** boundary condition dialog box will open.*



i. Enter `0.0116` kg/s for **Mass Flow Rate**.

ii. Select **Normal to Boundary** from the **Direction Specification Method** drop-down list.

iii. Select **Intensity and Hydraulic Diameter** from the **Specification Method** drop-down list in the **Turbulence** group box.

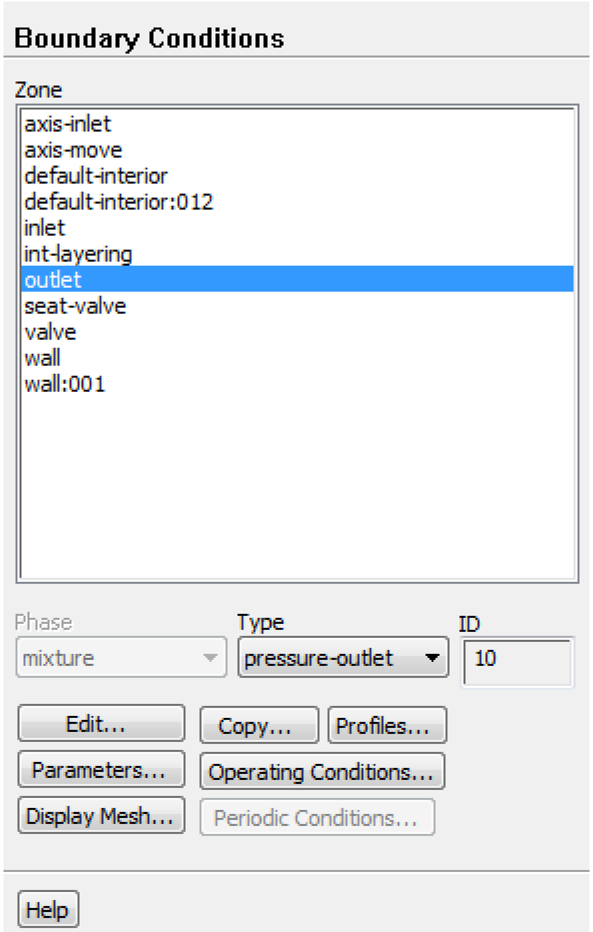
iv. Retain 5% for **Turbulent Intensity**.

v. Enter `20` mm for the **Hydraulic Diameter**.

vi. Click **OK** to close the **Mass-Flow Inlet** dialog box.

2. Set the conditions for the exit boundary (**outlet**).

Boundary Conditions → **outlet**

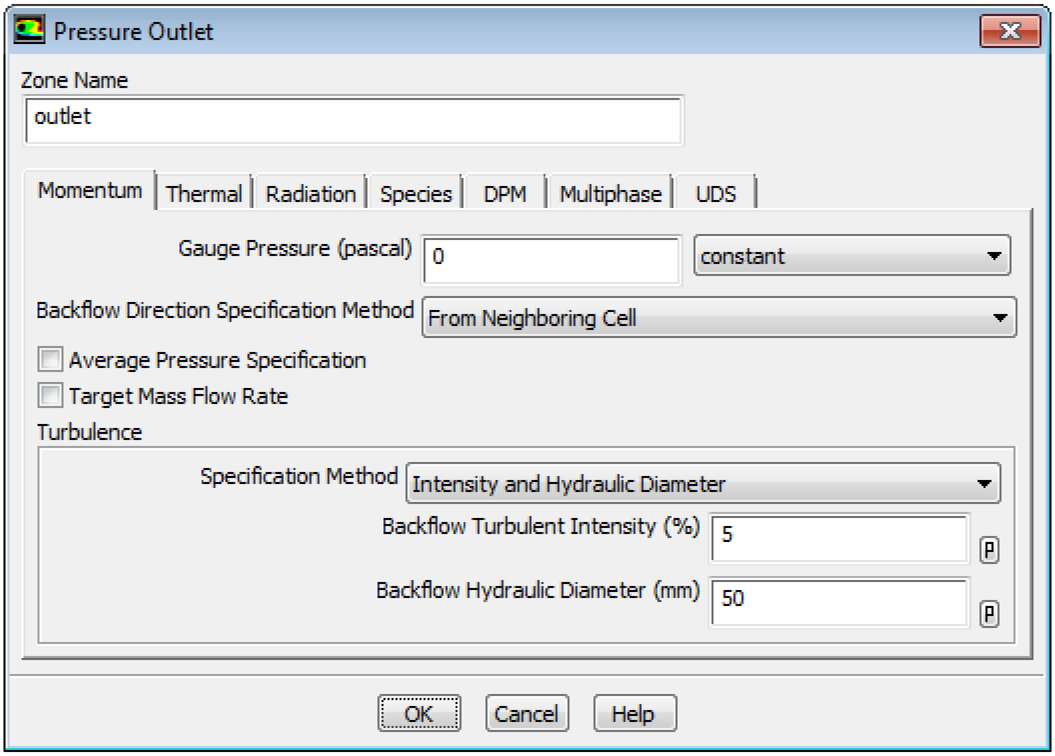


*Since the **outlet** boundary is assigned to a wall boundary type in the original mesh, you will need to explicitly assign the outlet boundary to a pressure outlet boundary type in ANSYS Fluent.*

a. Select **pressure-outlet** from the **Type** drop-down list in the **Boundary Conditions** task page.

b. Click **Yes** when ANSYS Fluent asks you if you want to change the zone type.

   *The **Pressure Outlet** boundary condition dialog box will open.*



i. Select **From Neighboring Cell** from the **Backflow Direction Specification Method** drop-down list.

ii. Select **Intensity and Hydraulic Diameter** from the **Specification Method** drop-down list in the **Turbulence** group box.

iii. Retain 5% for **Backflow Turbulent Intensity**.

iv. Enter 50 mm for **Backflow Hydraulic Diameter**.

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

3. Set the boundary type to **axis** for both the **axis-inlet** and the **axis-move** boundaries.

Boundary Conditions

*Since the **axis-inlet** and the **axis-move** boundaries are assigned to a wall boundary type in the original mesh, you will need to explicitly assign these boundaries to an axis boundary type in ANSYS Fluent.*

a. Select **axis-inlet** from the **Zone** list and select **axis** from the **Type** list.

b. Click **Yes** when ANSYS Fluent asks you if you want to change the zone type.

c. Retain the default **Zone Name** in the **Axis** dialog box and click **OK** to close the **Axis** dialog box.

d. Select **axis-move** from the **Zone** list and select **axis** from the **Type** list.

e. Click **Yes** when ANSYS Fluent asks you if you want to change the zone type.

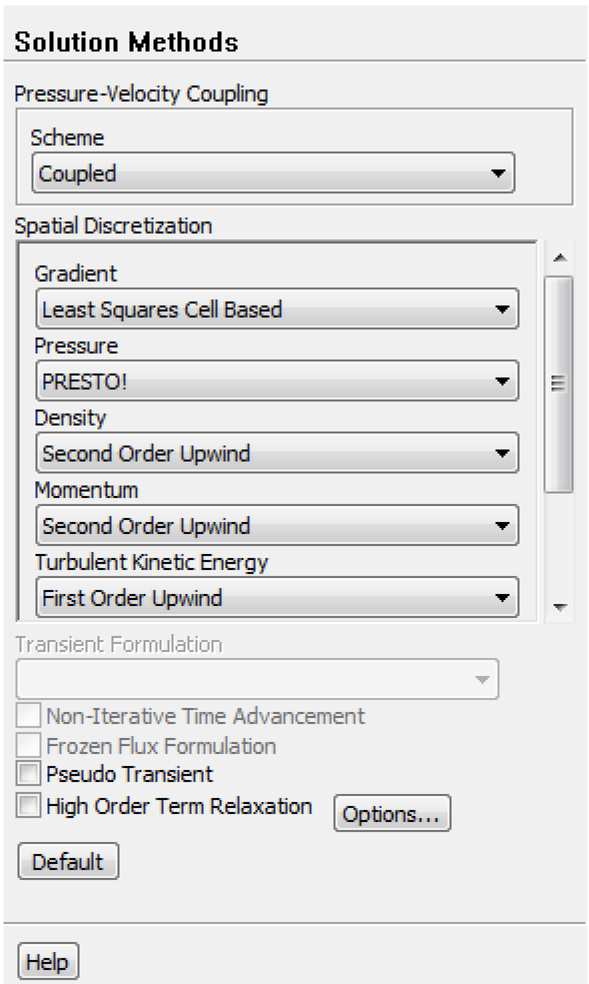
f. Retain the default **Zone Name** in the **Axis** dialog box and click **OK** to close the **Axis** dialog box.

15.4.7. 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.

   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.

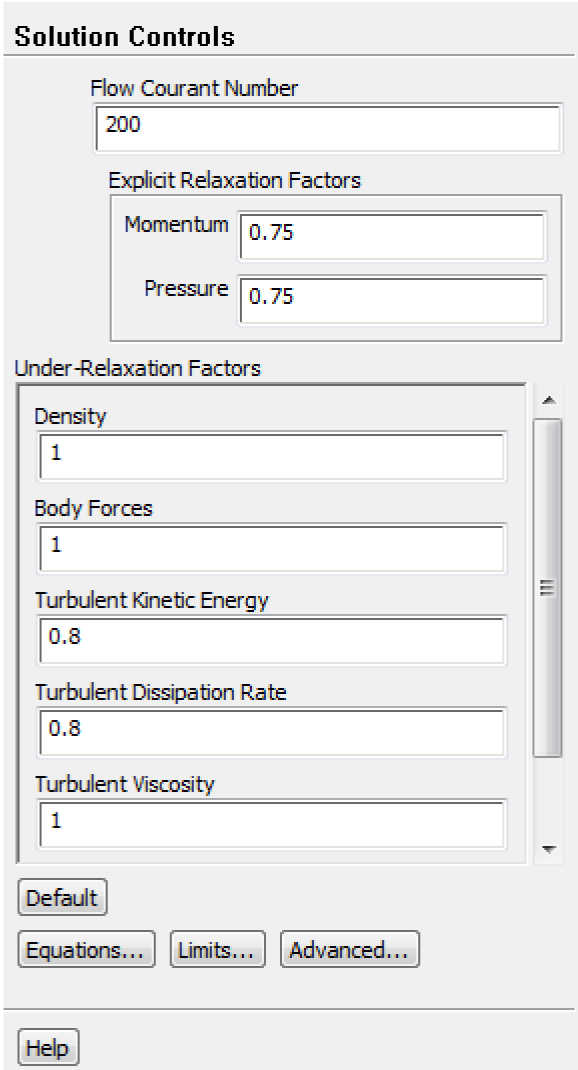
   d. Retain the default of **Second Order Upwind** in the **Momentum** drop-down list.

   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.

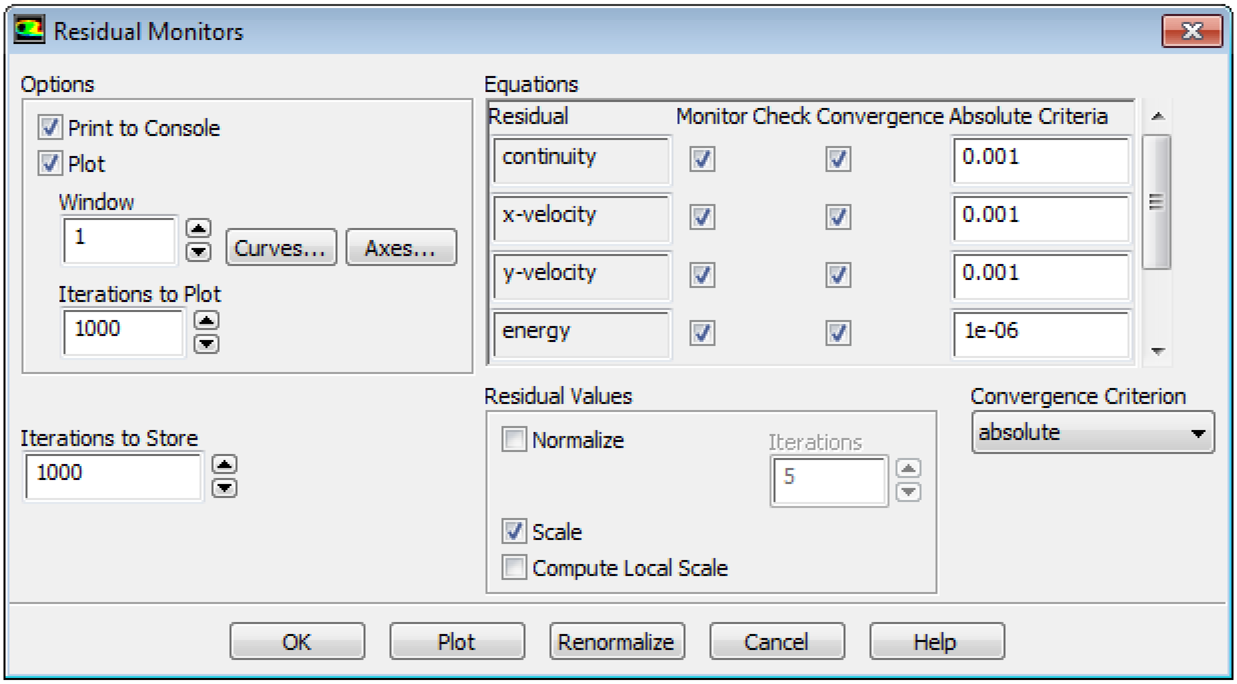
Solution Controls



a. Retain the default values for **Under-Relaxation Factors** in the **Solution Controls** task page.

3. Enable the plotting of residuals during the calculation.

Monitors → **Residuals** → **Edit...**

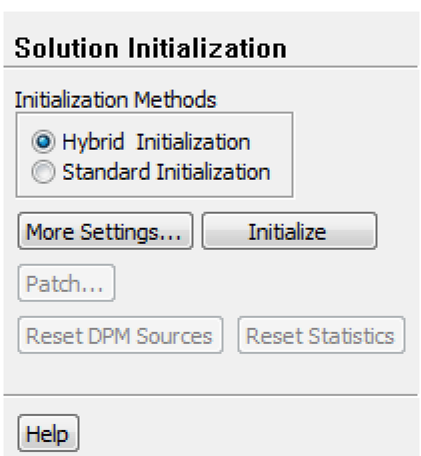


a. Ensure that **Plot** is enabled in the **Options** group box.

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

4. Initialize the solution.

Solution Initialization



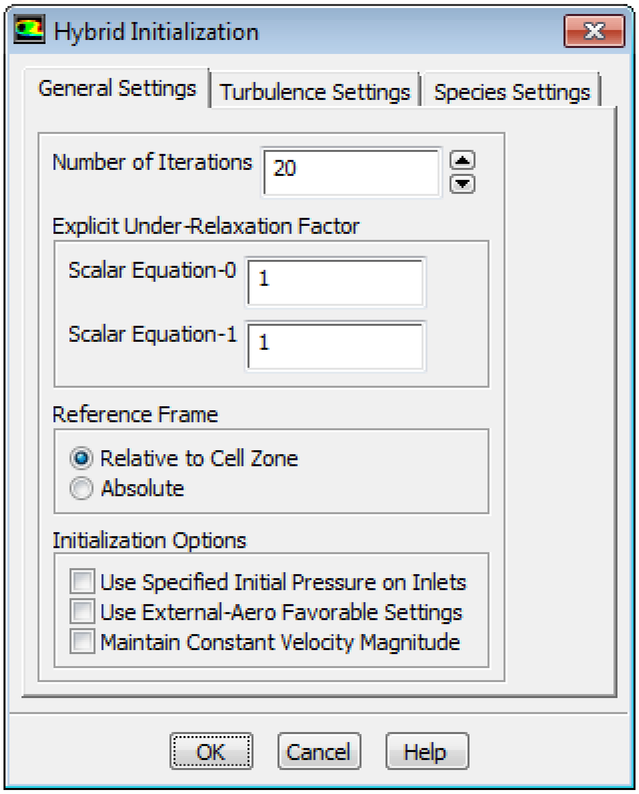
a. Retain the default **Hybrid Initialization** in the **Initialization Methods** group box.

b. Click **Initialize** in the **Solution Initialization** task page.

Note

A warning is displayed in the console stating that the convergence tolerance of 1.000000e-06 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 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** once more.

Note

Click **OK** in the **Question** dialog box, where it asks to discard the current data. The console displays that hybrid initialization is done.

Note

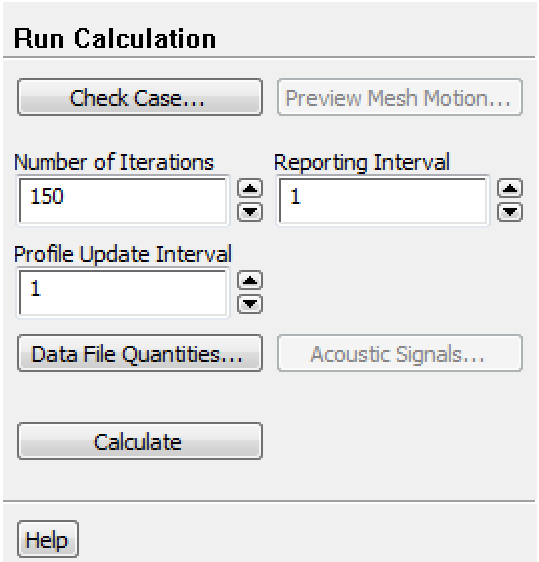
For flows in complex topologies, hybrid initialization will provide better initial velocity and pressure fields than standard initialization. This will help to improve the convergence behavior of the solver.

5. Save the case file (`valve_init.cas.gz`).

File → **Write** → **Case...**

6. Start the calculation by requesting 150 iterations.

   Run Calculation

   Click **Calculate**.

   The solution converges in approximately 115 iterations.

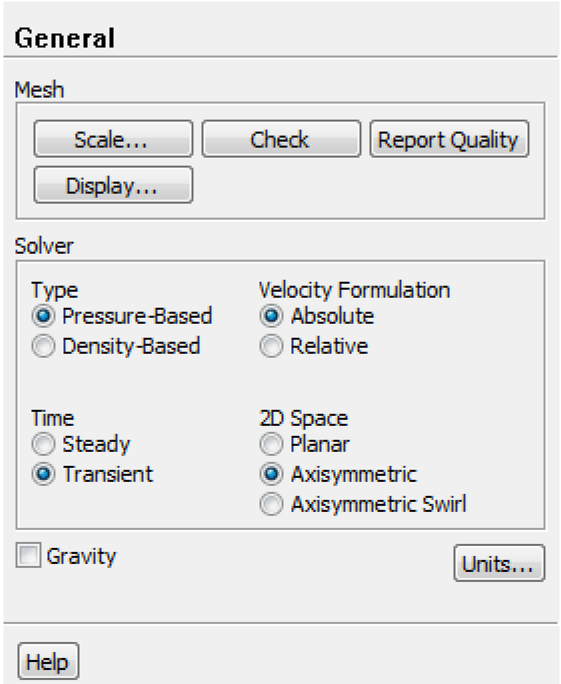
7. Save the case and data files (`valve_init.cas.gz` and `valve_init.dat.gz`).

   File → Write → Case & Data...

15.4.8. Time-Dependent Solution Setup

1. Enable a time-dependent calculation.

   General

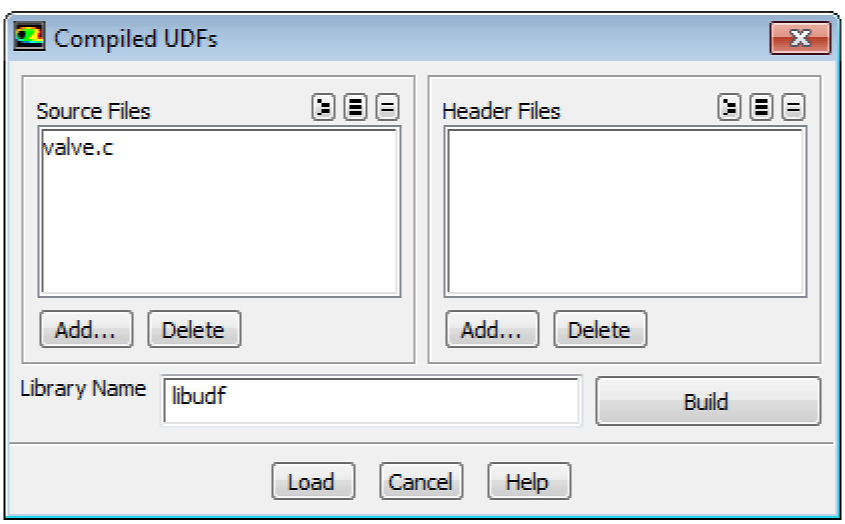


a. Select **Transient** from the **Time** list in the **General** task page.

15.4.9. Mesh Motion

1. Select and compile the user-defined function (UDF).

   Define → **User-Defined** → **Functions** → **Compiled...**



   a. Click **Add...** in the **Source Files** group box.

      *The **Select File** dialog box will open.*

      i. Select the source code **valve.c** in the **Select File** dialog box, and click **OK**.

   b. Click **Build** in the **Compiled UDFs** dialog box.

*The UDF is already defined, but it must be compiled within ANSYS Fluent before it can be used in the solver. Here you will create a library with the default name of `libudf` in your working folder. If you want to use a different name, you can enter it in the **Library Name** field. In this case you need to make sure that you will open the correct library in the next step.*

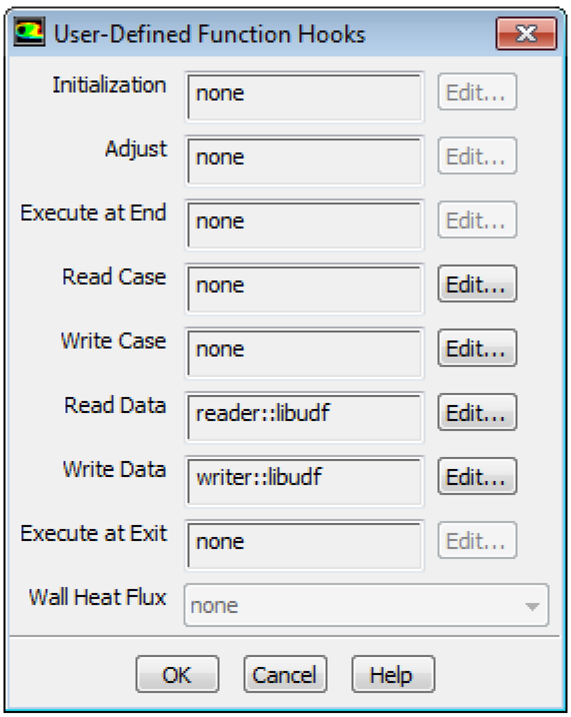
*A dialog box will appear warning you to make sure that the UDF source files are in the directory that contains your case and data files. Click **OK** in the warning dialog box.*

c. Click **Load** to load the UDF library you just compiled.

*When the UDF is built and loaded, it is available to hook to your model. Its name will appear as **valve::libudf** and can be selected from drop-down lists of various dialog boxes.*

2. Hook your model to the UDF library.

Define → **User-Defined** → **Function Hooks...**



a. Click the **Edit...** button next to **Read Data** to open the **Read Data Functions** dialog box.

   i. Select **reader::libudf** from the **Available Read Data Functions** selection list.

   ii. Click **Add** to add the selected function to the **Selected Read Data Functions** selection list.

   iii. Click **OK** to close the **Read Data Functions** dialog box.

b. Click the **Edit...** button next to **Write Data** to open the **Write Data Functions** dialog box.

   i. Select **writer::libudf** from the **Available Write Data Functions** selection list.

   ii. Click **Add** to add the selected function to the **Selected Write Data Functions** selection list.

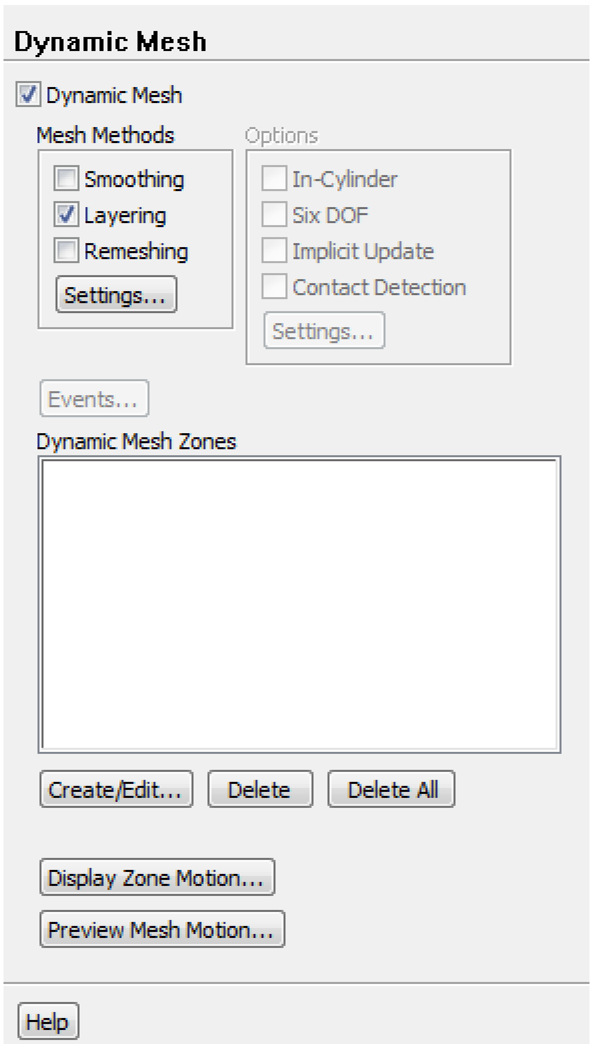
   iii. Click **OK** to close the **Write Data Functions** dialog box.

These two functions will read/write the position of the center of gravity (CG) and velocity in the X direction to the data file. The location of the CG and the velocity are necessary for restarting a case. When starting from an intermediate case and data file, ANSYS Fluent needs to know the location of the CG and velocity, which are the initial conditions for the motion calculation. Those values are saved in the data file using the writer UDF and will be read in using the reader UDF when reading the data file.

c. Click **OK** to close the **User-Defined Function Hooks** dialog box.

3. Enable dynamic mesh motion and specify the associated parameters.

Dynamic Mesh



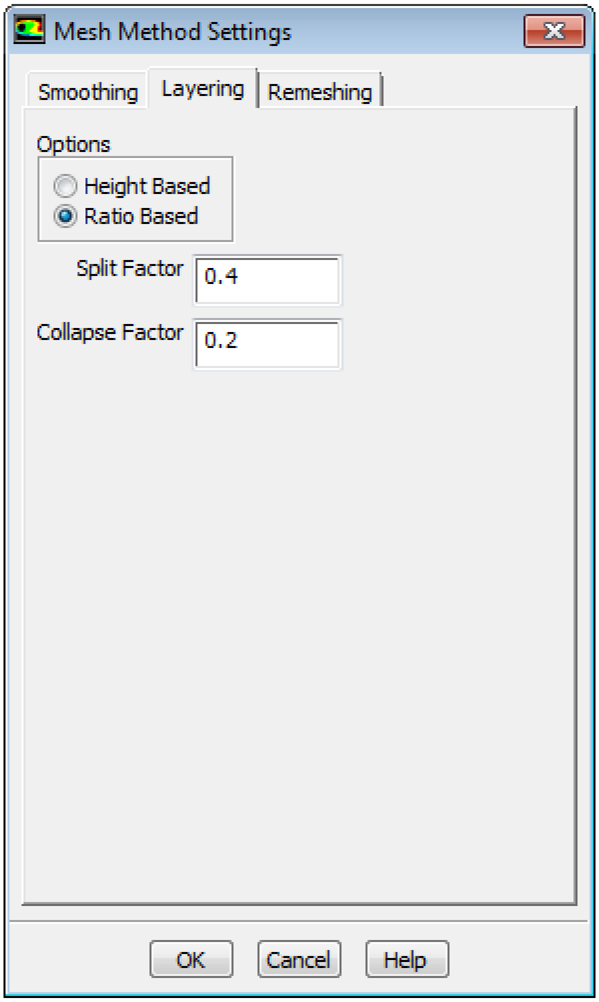
a. Enable **Dynamic Mesh** in the **Dynamic Mesh** task page.

For more information on the available models for moving and deforming zones, see Modeling Flows Using Sliding and Dynamic Meshes in the User's Guide.

b. Disable **Smoothing** and enable **Layering** in the **Mesh Methods** group box.

ANSYS Fluent will automatically flag the existing mesh zones for use of the different dynamic mesh methods where applicable.

c. Click the **Settings...** button to open the **Mesh Method Settings** dialog box.



i. Click the **Layering** tab.

ii. Select **Ratio Based** in the **Options** group box.

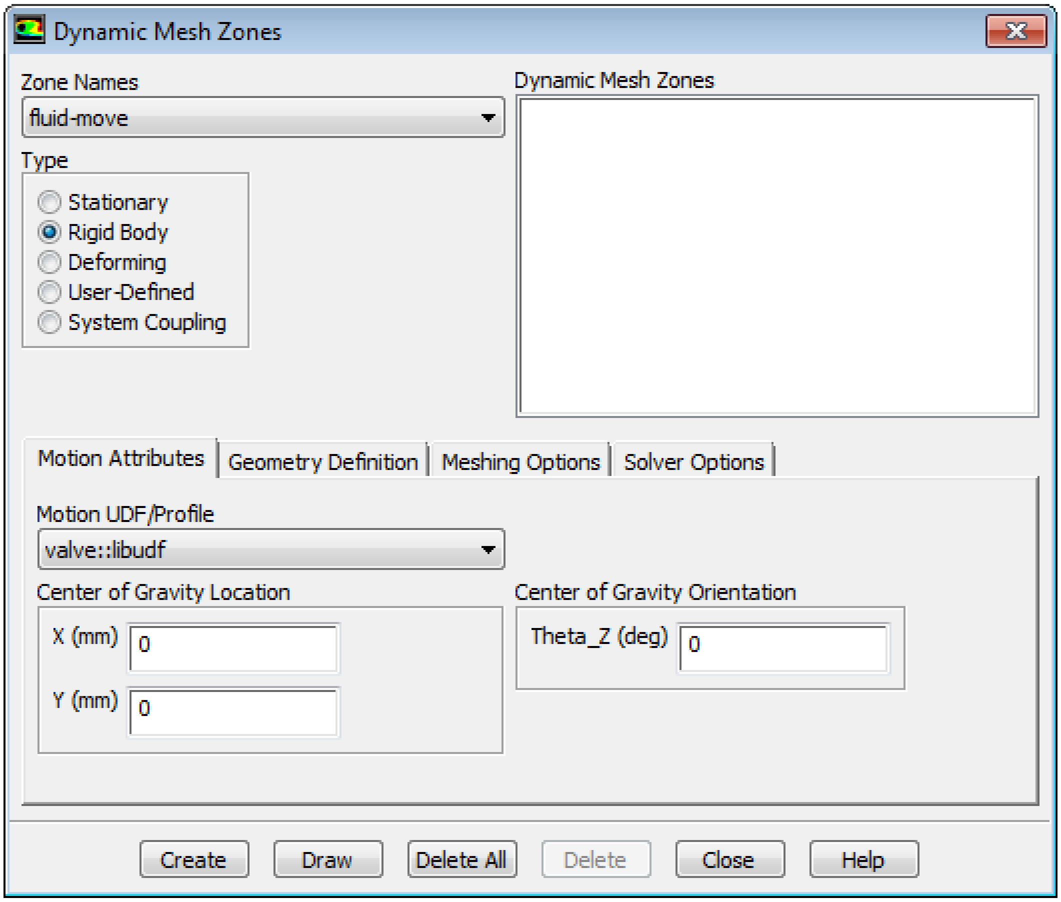
iii. Retain the default settings of `0.4` and `0.2` for **Split Factor** and **Collapse Factor**, respectively.

iv. Click **OK** to close the **Mesh Method Settings** dialog box.

4. Specify the motion of the fluid region (**fluid-move**).

Dynamic Mesh → **Create/Edit...**

The valve motion and the motion of the fluid region are specified by means of the UDF ***valve****.*



a. Select **fluid-move** from the **Zone Names** drop-down list.

b. Retain the default selection of **Rigid Body** in the **Type** group box.

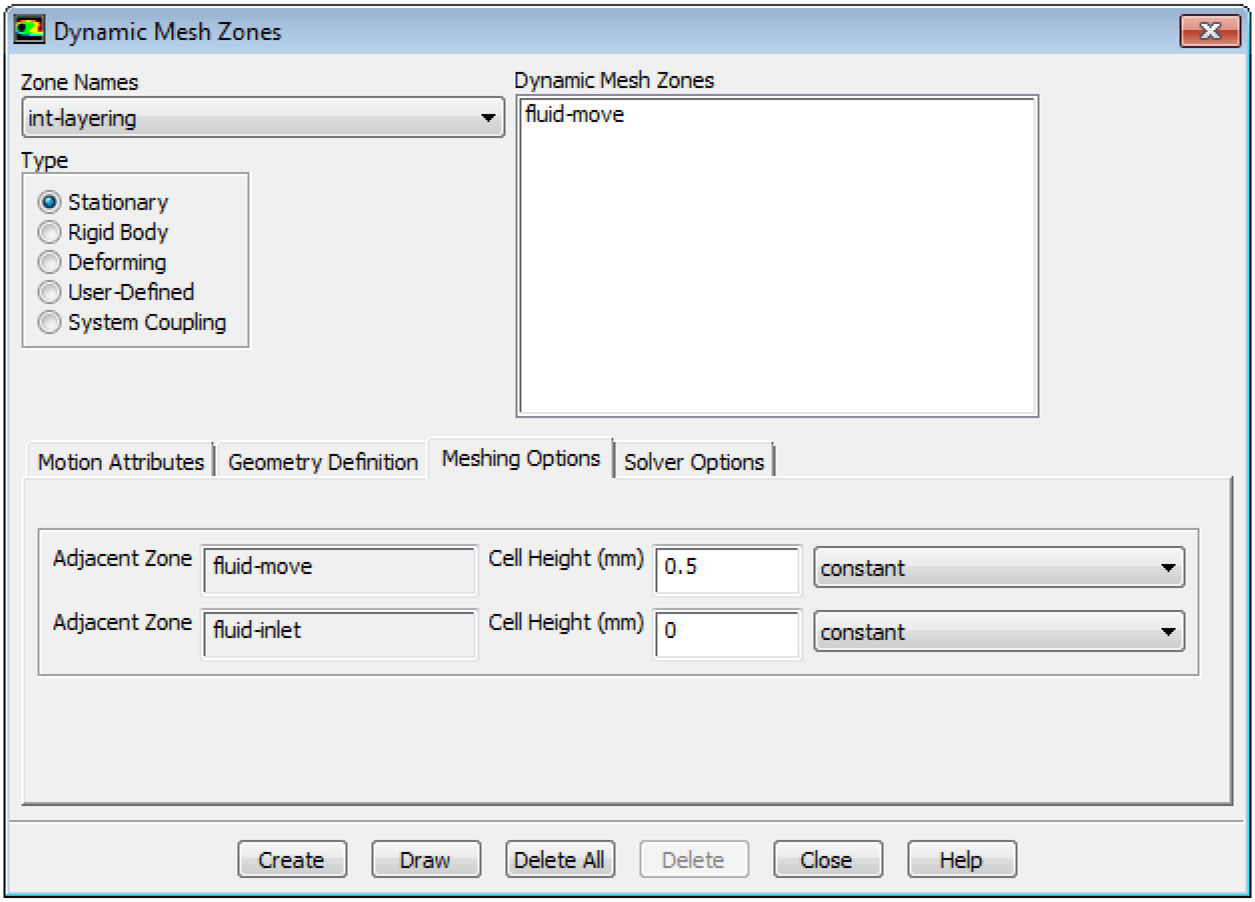
c. Ensure that **valve::libudf** is selected from the **Motion UDF/Profile** drop-down list in the **Motion Attributes** tab to hook the UDF to your model.

d. Retain the default settings of `(0, 0)` mm for **Center of Gravity Location**, and `0` for **Center of Gravity Orientation**.

   Specifying the CG location and orientation is not necessary in this case, because the valve motion and the initial CG position of the valve are already defined by the UDF.

e. Click **Create**.

5. Specify the meshing options for the stationary layering interface (**int-layering**) in the **Dynamic Mesh Zones** dialog box.



a. Select **int-layering** from the **Zone Names** drop-down list.

b. Select **Stationary** in the **Type** group box.

c. Click the **Meshing Options** tab.

   i. Enter `0.5` mm for **Cell Height** of the `fluid-move` **Adjacent Zone**.

   ii. Retain the default value of `0` mm for the **Cell Height** of the `fluid-inlet` **Adjacent zone**.

d. Click **Create**.

6. Specify the meshing options for the stationary outlet (**outlet**) in the **Dynamic Mesh Zones** dialog box.

   a. Select **outlet** from the **Zone Names** drop-down list.

   b. Retain the previous selection of **Stationary** in the **Type** group box.

   c. Click the **Meshing Options** tab and enter `1.9` mm for the **Cell Height** of the `fluid-move` **Adjacent Zone**.

   d. Click **Create**.

7. Specify the meshing options for the stationary seat valve (**seat-valve**) in the **Dynamic Mesh Zones** dialog box.

   a. Select **seat-valve** from the **Zone Names** drop-down list.

b. Retain the previous selection of **Stationary** in the **Type** group box.

c. Click the **Meshing Options** tab and enter `0.5` mm for **Cell Height** of the `fluid-move` **Adjacent Zone**.

d. Click **Create**.

8. Specify the motion of the valve (**valve**) in the **Dynamic Mesh Zones** dialog box.

a. Select **valve** from the **Zone Names** drop-down list.

b. Select **Rigid Body** in the **Type** group box.

c. Click the **Motion Attributes** tab.

i. Ensure that **valve::libudf** is selected from the **Motion UDF/Profile** drop-down list to hook the UDF to your model.

ii. Retain the default settings of `(0, 0)` mm for **Center of Gravity Location**, and `0` for **Center of Gravity Orientation**.

d. Click the **Meshing Options** tab and enter `0` mm for the **Cell Height** of the `fluid-move` **Adjacent zone**.

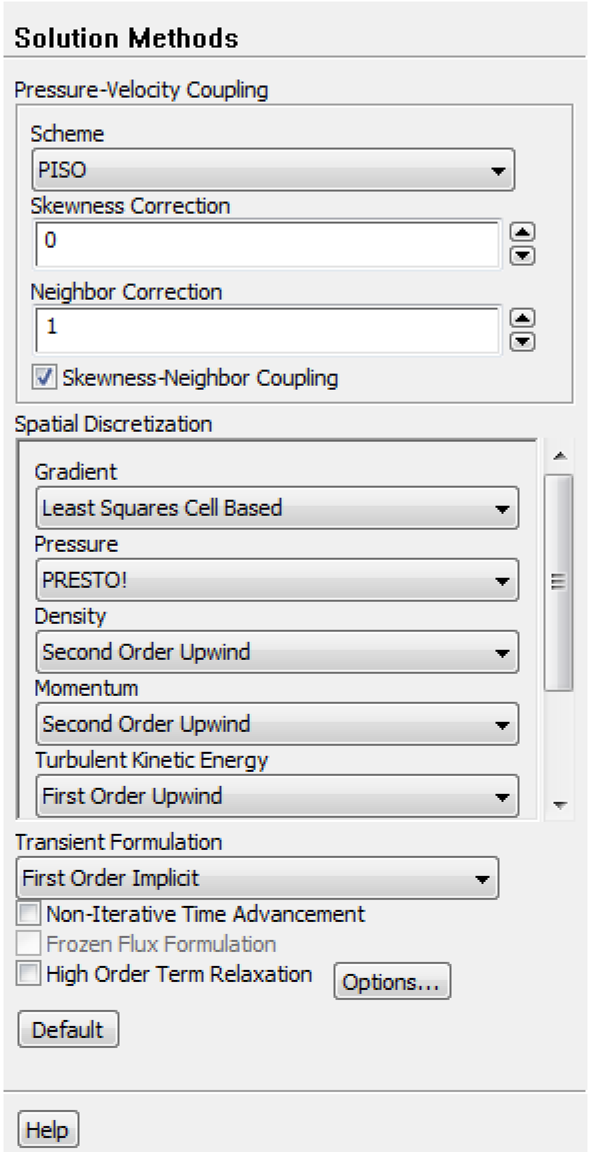
e. Click **Create** and close the **Dynamic Mesh Zones** dialog box.

In many MDM problems, you may want to preview the mesh motion before proceeding. In this problem, the mesh motion is driven by the pressure exerted by the fluid on the valve and acting against the inertia of the valve. Hence, for this problem, mesh motion in the absence of a flow field solution is meaningless, and you will not use this feature here.

15.4.10. Time-Dependent Solution

1. Set the solution parameters.

Solution Methods



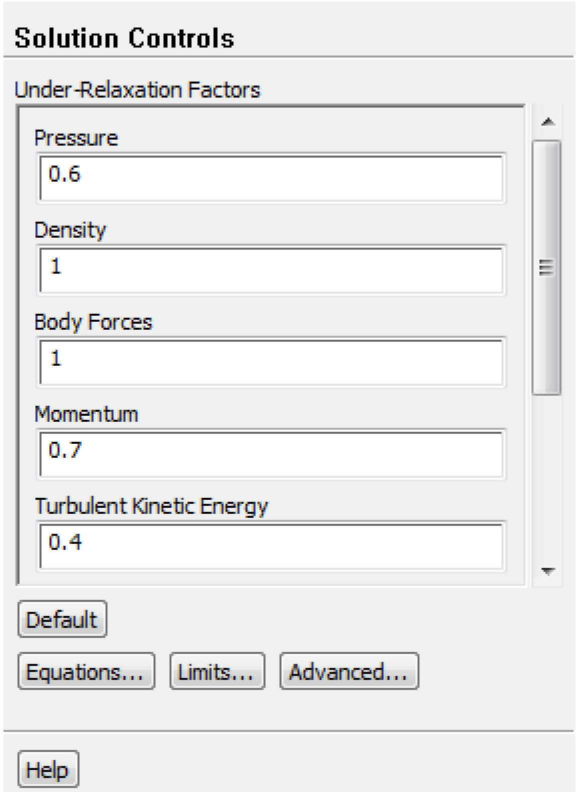
a. Select **PISO** from the **Scheme** drop-down list in **Pressure-Velocity Coupling** group box.

b. Enter `0` for **Skewness Correction**.

c. Retain all of the other previously set schemes and defaults.

2. Set the relaxation factors.

Solution Controls



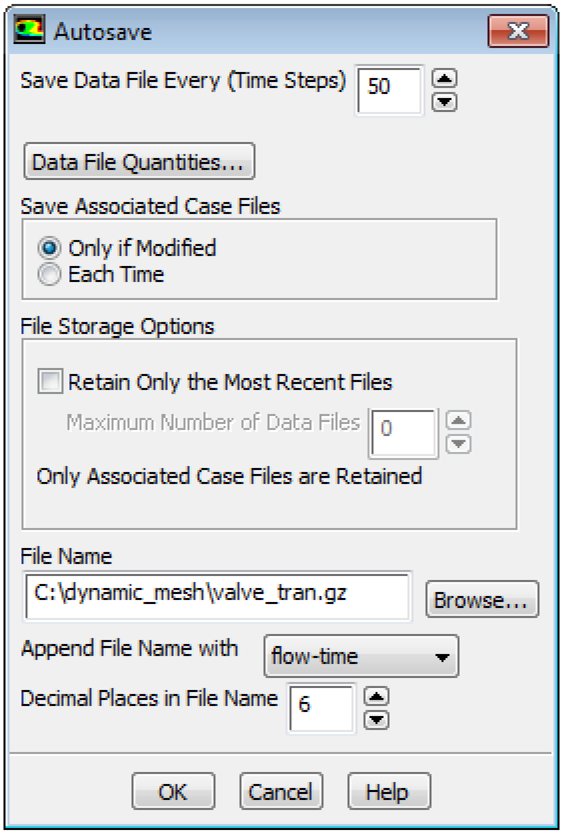
a. Enter `0.6` for **Pressure** in the **Under-Relaxation Factors** group box.

b. Enter `0.4` for **Turbulent Kinetic Energy**.

c. Enter `0.4` for **Turbulent Dissipation Rate**.

3. Request that case and data files are automatically saved every 50 time steps.

Calculation Activities (Autosave Every (Time Steps)) → **Edit...**



a. Enter `50` for **Save Data File Every (Time Steps)**.

b. Enter `valve_tran.gz` in the **File Name** text box.

c. Select **flow-time** from the **Append File Name with** drop-down list.

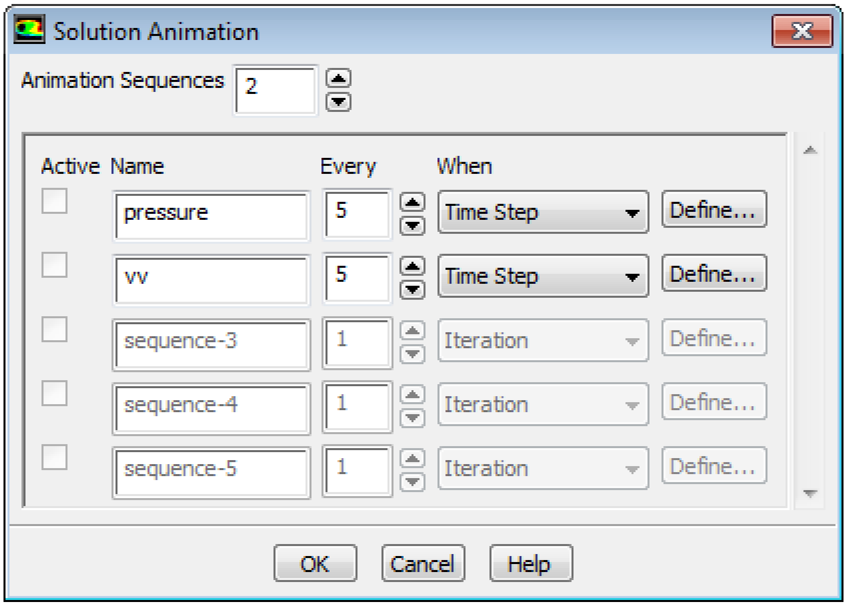
   When ANSYS Fluent saves a file, it will append the flow time value to the file name prefix (`valve_tran`). The gzipped standard extensions (`.cas.gz` and `.dat.gz`) will also be appended.

d. Click **OK** to close the **Autosave** dialog box.

4. Create animation sequences for the static pressure contour plots and velocity vectors plots for the valve.

   Calculation Activities (Solution Animations) → Create/Edit...

   Use the solution animation feature to save contour plots of temperature every five time steps. After the calculation is complete, you use the solution animation playback feature to view the animated temperature plots over time.



a. Set **Animation Sequences** to 2.

b. Enter `pressure` in the **Name** text box for the first animation.

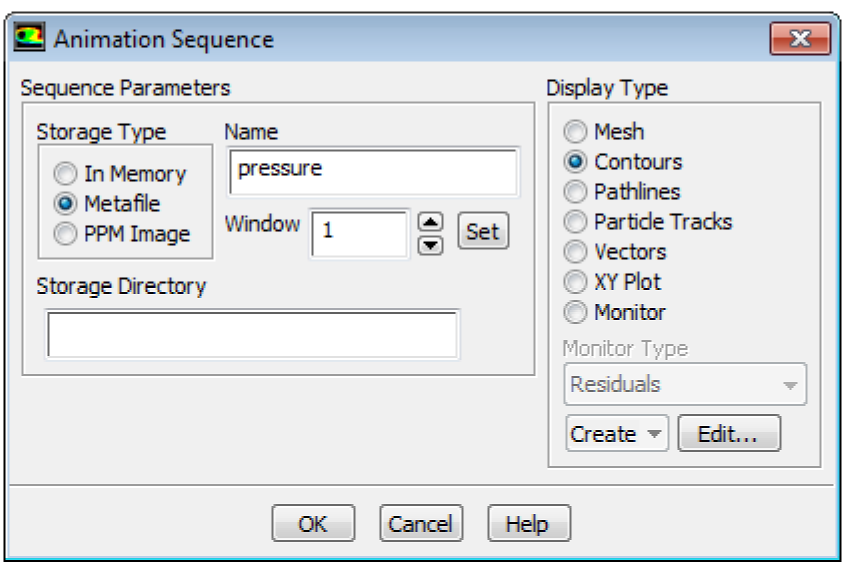
c. Enter `vv` in the **Name** text box for the second animation.

d. Set **Every** to 5 for both animation sequences.

   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.

e. Select **Time Step** from the **When** drop-down list for `pressure` and `vv`.

f. Click the **Define...** button next to `pressure` to open the **Animation Sequence** dialog box.



i. Retain the default selection of **Metafile** in the **Storage Type** group box.

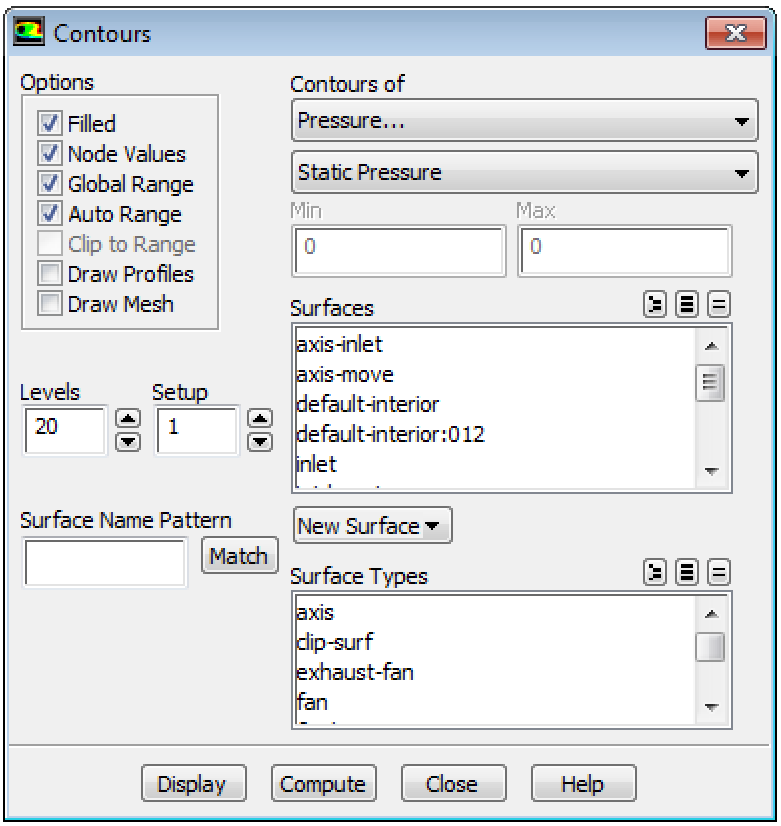
Note

If you want to store the plots in a folder other than your working folder, enter the folder path in the **Storage Directory** text box. If this field is left blank (the default), the files will be saved in your working folder (that is, the folder where you started ANSYS Fluent).

---

ii. Set **Window** number to 1 and click **Set**.

iii. Select **Contours** in the **Display Type** group box to open the **Contours** dialog box.



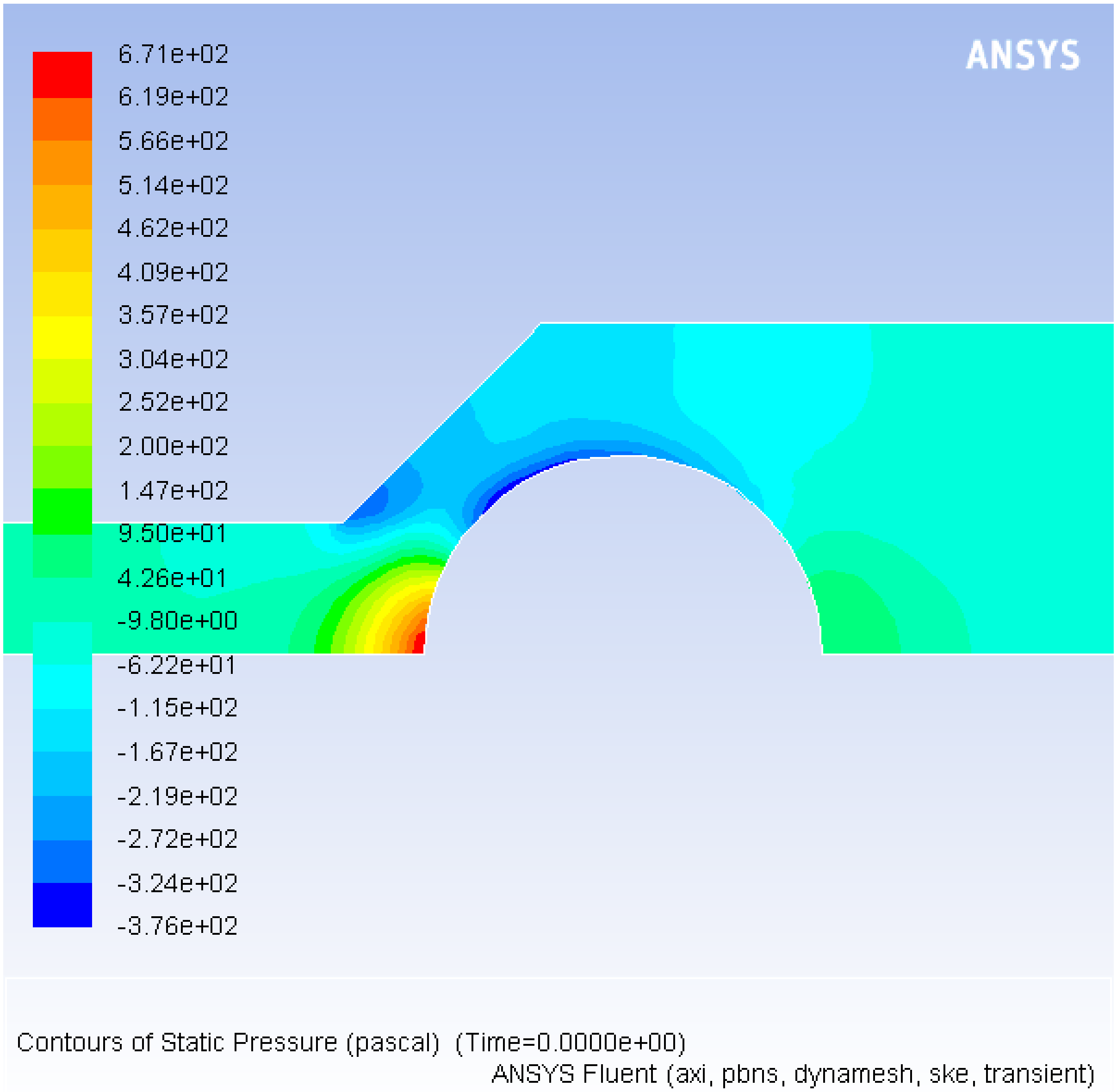
A. Enable **Filled** in the **Options** group box.

B. Retain the default selection of **Pressure...** and **Static Pressure** from the **Contours of** drop-down lists.

C. Click **Display** (Figure 15.3: Contours of Static Pressure at t=0 s (p. 661)).

D. Close the **Contours** dialog box.

Figure 15.3: Contours of Static Pressure at t=0 s



iv. Click **OK** in the **Animation Sequence** dialog box.

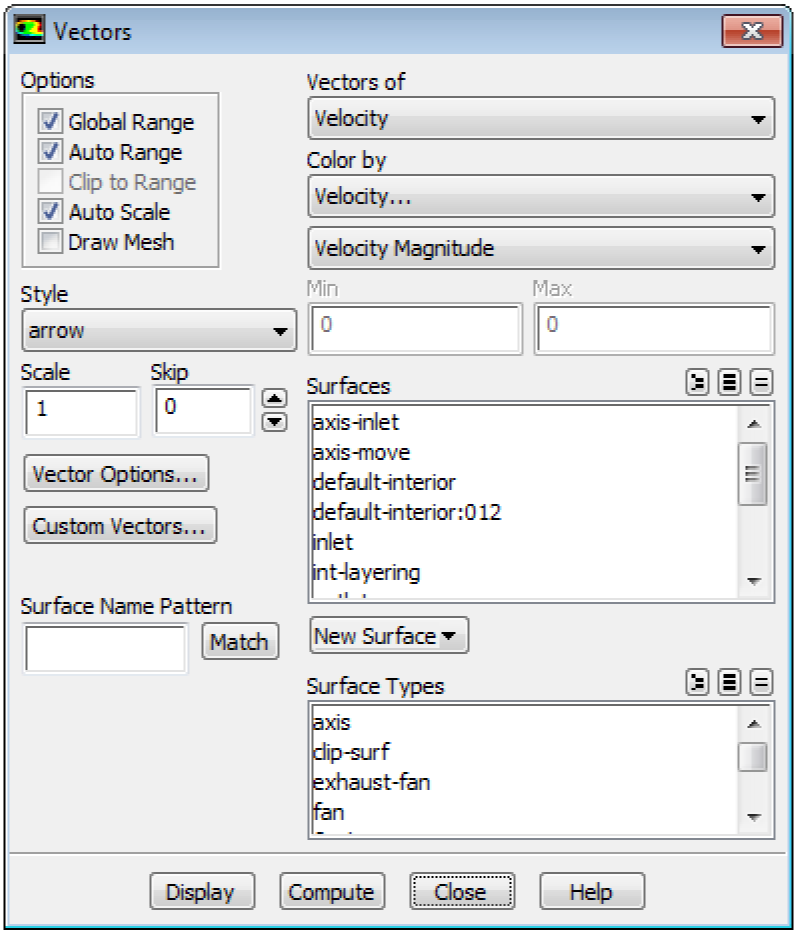
*The **Animation Sequence** dialog box will close, and the check box in the **Active** column next to **pressure** in the **Solution Animation** dialog box will be enabled.*

g. Click the **Define...** button next to vv to open the **Animation Sequence** dialog box.

i. Retain the default selection of **Metafile** in the **Storage Type** group box.

ii. Set **Window** to 2 and click **Set**.

iii. Select **Vectors** in the **Display Type** group box to open the **Vectors** dialog box.

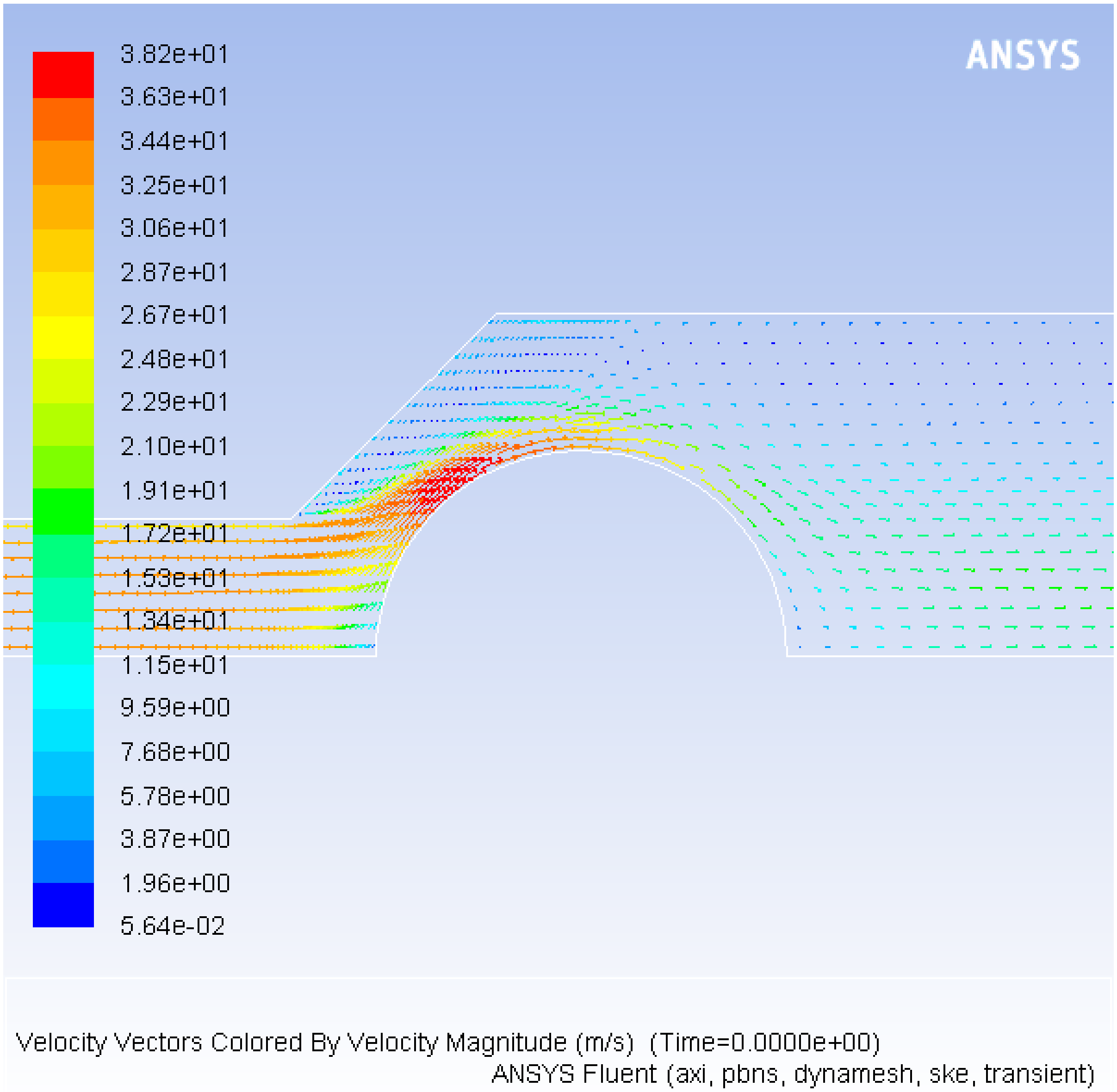


A. Retain all the other default settings.

B. Click **Display** (Figure 15.4: Vectors of Velocity at t=0 s (p. 663)).

C. Close the **Vectors** dialog box.

Figure 15.4: Vectors of Velocity at t=0 s



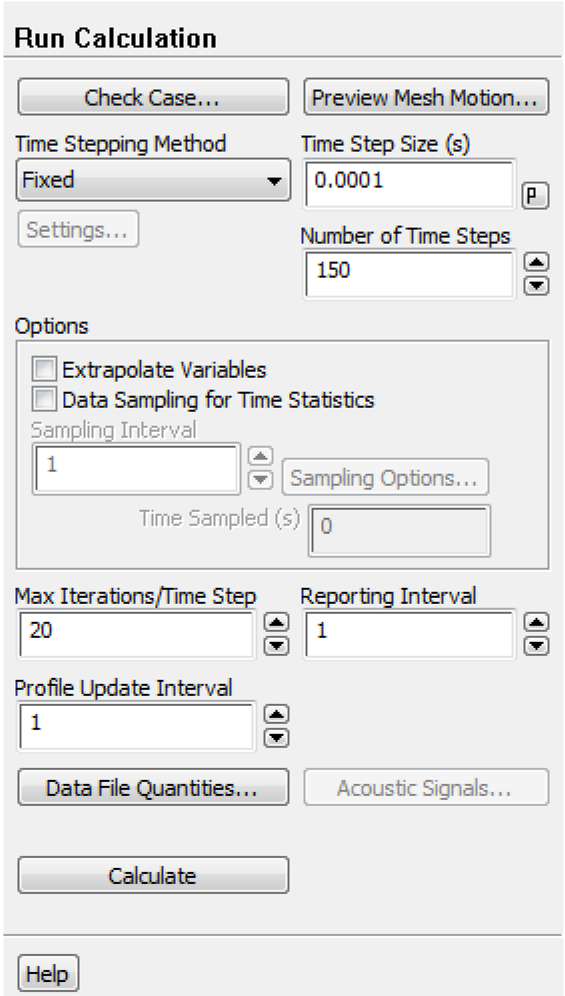
iv. Click **OK** in the **Animation Sequence** dialog box.

*The **Animation Sequence** dialog box will close, and the check box in the **Active** column next to **vv** in the **Solution Animation** dialog box will be enabled.*

h. Click **OK** to close the **Solution Animation** dialog box.

5. Set the time step parameters for the calculation.

Run Calculation



a. Enter `0.0001` s for **Time Step Size**.

b. 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.

6. 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...**

7. Request 150 time steps and calculate a solution.

   Run Calculation

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.

The UDF (`valve.c`) that is provided can be edited and customized by changing the parameters as required for your case. In this tutorial, the values necessary for this case were preset in the source code. These values may be modified to best suit your model.

15.4.11. Postprocessing

1. Inspect the solution at the final time step.

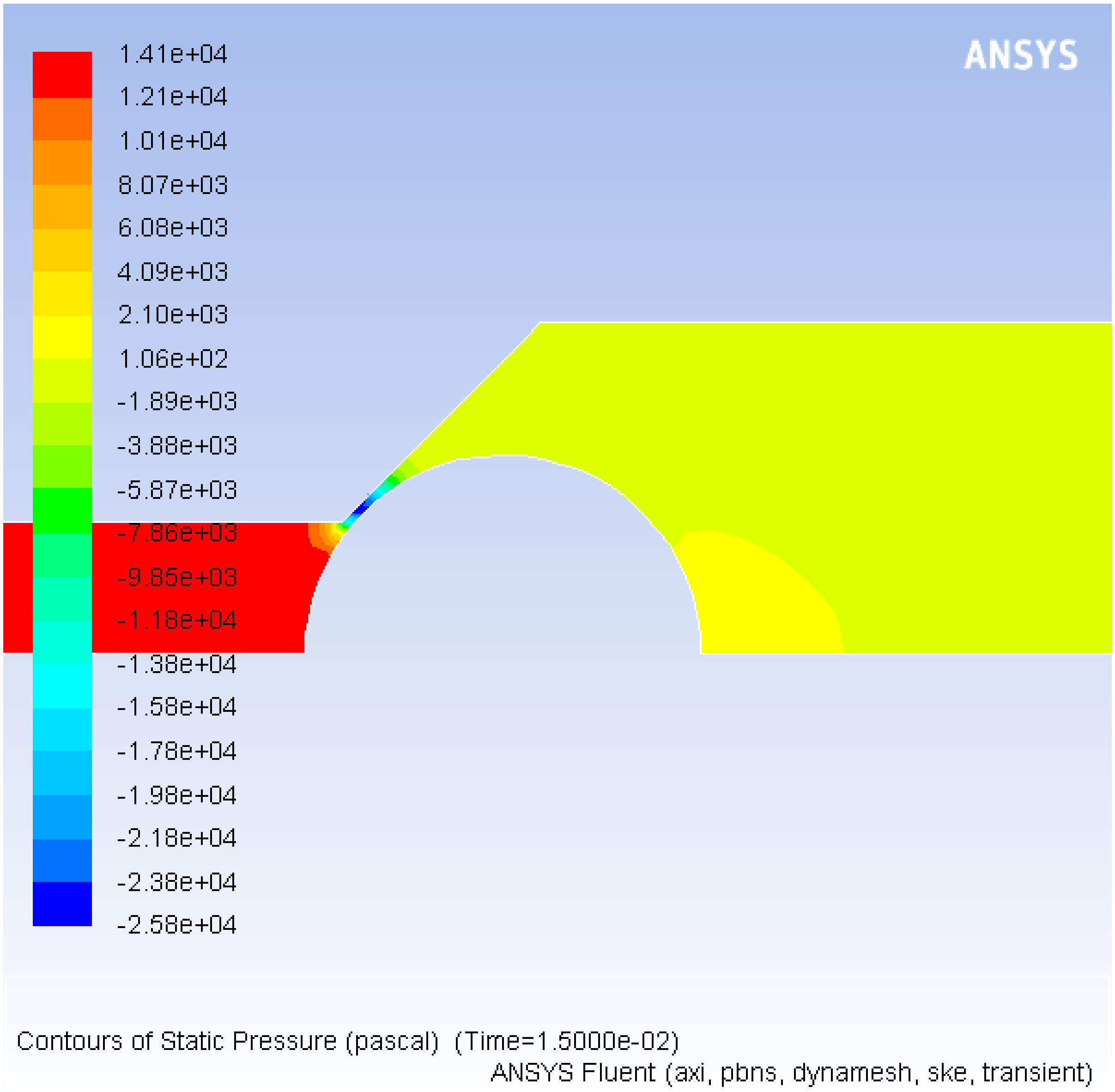
   a. Inspect the contours of static pressure in the valve (Figure 15.5: Contours of Static Pressure After 150 Time Steps (p. 666)).

      Graphics and Animations → **Contours** → **Set Up...**

      Note

      You may need to switch to Window 1 (using the drop-down list at the upper left corner of the graphics window) to view the contour plot.

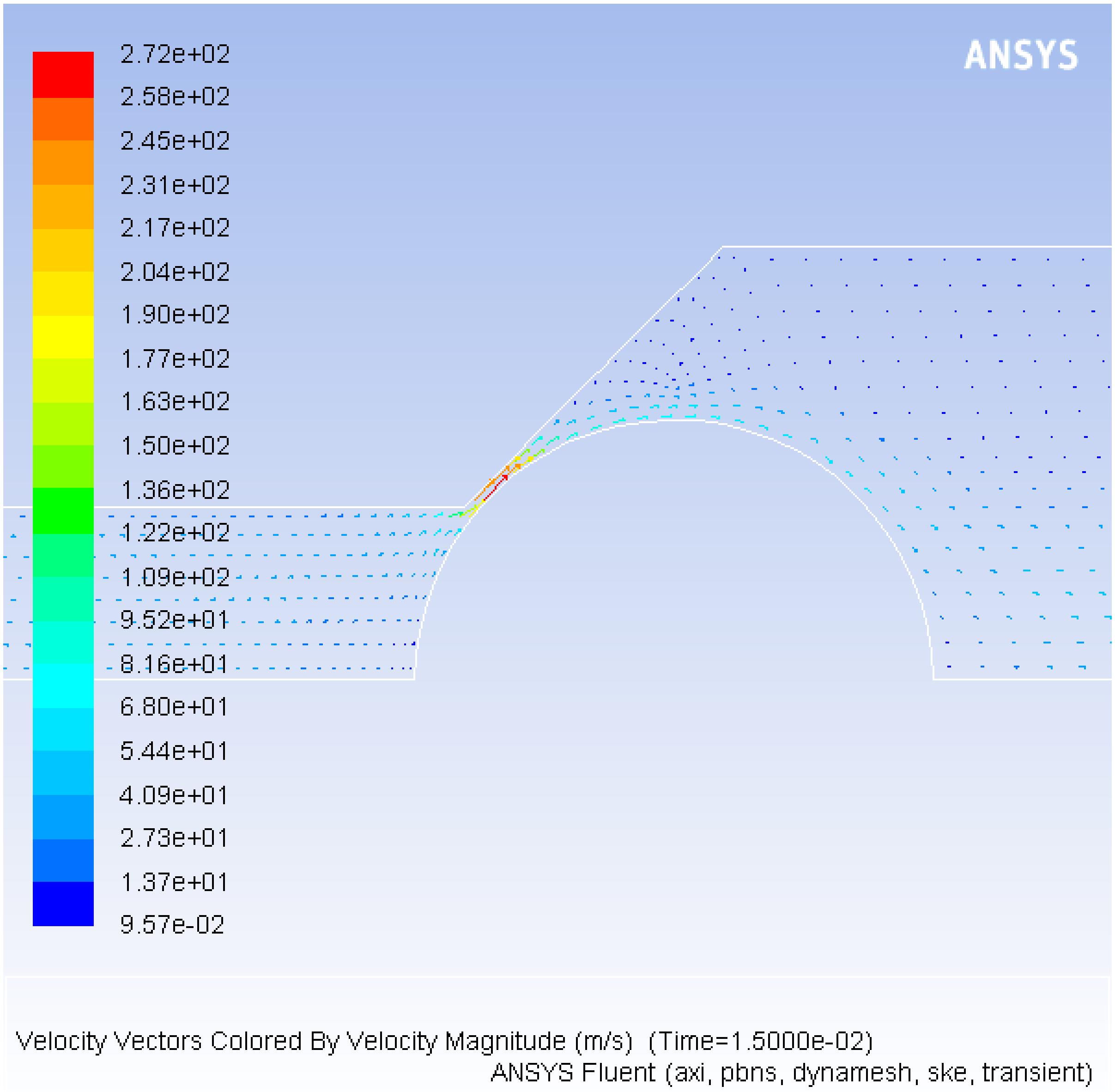
Figure 15.5: Contours of Static Pressure After 150 Time Steps



b. Inspect the velocity vectors near the point where the valve meets the seat valve (Figure 15.6: Velocity Vectors After 150 Time Steps (p. 667)).

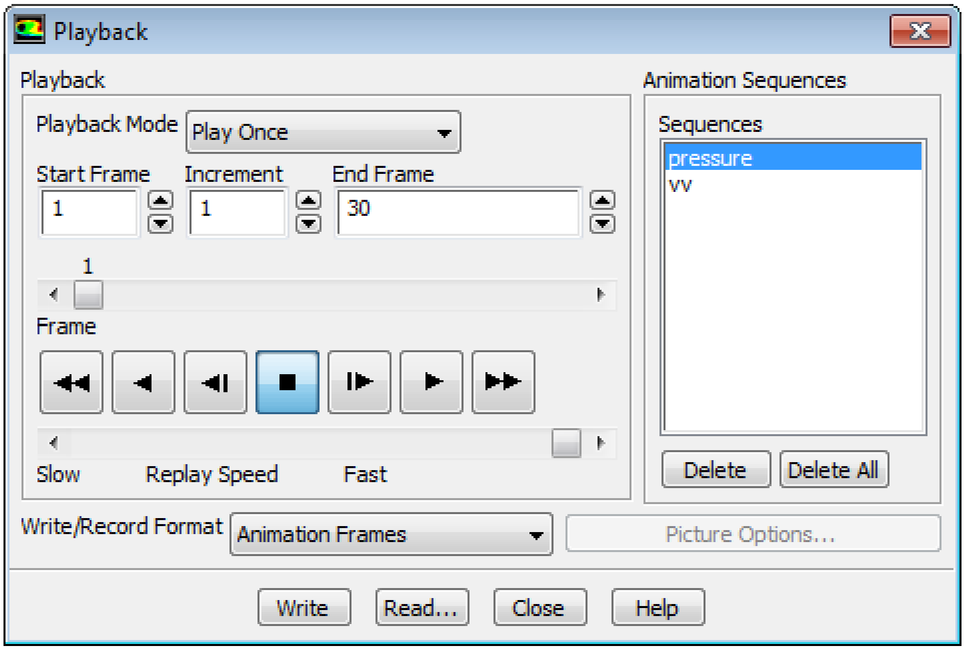
Graphics and Animations → **Vectors** → **Set Up...**

Figure 15.6: Velocity Vectors After 150 Time Steps



2. Play the animation of the pressure contours.

   a. **Graphics and Animations** → **Solution Animation Playback** → **Set Up...**



b. Select `pressure` from the **Sequences** list in the **Animation Sequences** box of the **Playback** dialog box.

*If the **Sequences** list is empty, click **Read...** to select the `pressure.cxa` sequence file from your working directory.*

The playback control buttons will become active.

c. Set the slider bar above **Replay Speed** about halfway in between **Slow** and **Fast**.

d. Retain the default settings in the rest of the dialog box and click the  button.

*You may have to change the Viewer window to see the animation. In the drop-down menu at the top of the Viewer, set the window number to 1, which corresponds to the **Window** number for `pressure` that you set in the **Animation Sequence** dialog box.*

3. Play the animation of the velocity vectors.

a. Select `vv` from the **Sequences** list in the **Animation Sequences** box of the **Playback** dialog box.

*If the **Sequences** list does not contain `vv`, click **Read...** to select the `vv.cxa` sequence file from your working directory.*

b. Retain the default settings in the rest of the dialog box and click the  button.

*You may have to change the Viewer window to see the animation. In the drop-down menu at the top of the Viewer, set the window number to 2, which corresponds to the **Window** number for `vv` that you set in the **Animation Sequence** dialog box.*

For additional information on animating the solution, see Modeling Transient Compressible Flow (p. 257) and see Animating the Solution of the 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.

15.5. 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 (1DoF) rigid body FSI by means of a user-defined function (UDF). ANSYS Fluent can also perform a more general six degrees of freedom (6DoF) rigid body FSI using a built-in 6DoF solver.

If you decide to run this tutorial in parallel, make sure you use **Principal Axes** as the partitioning method.

15.6. 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.