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:

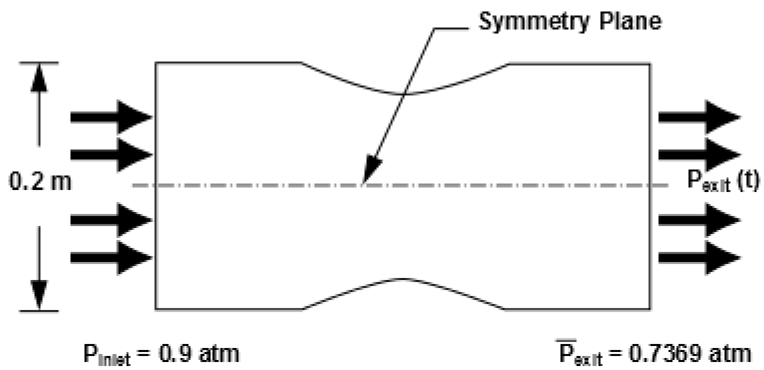
- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

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.

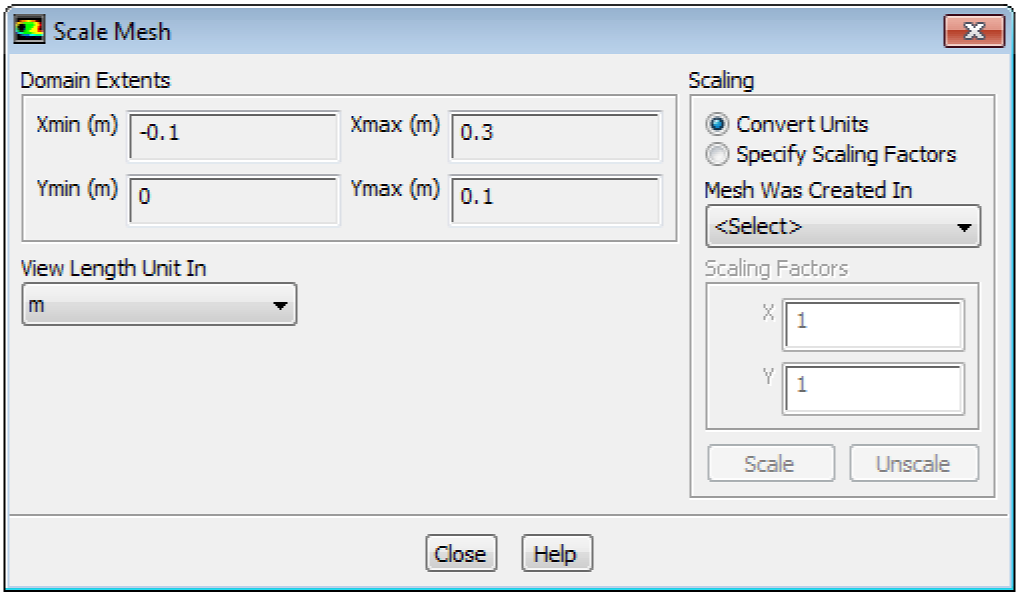
2. Check the mesh.

   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.

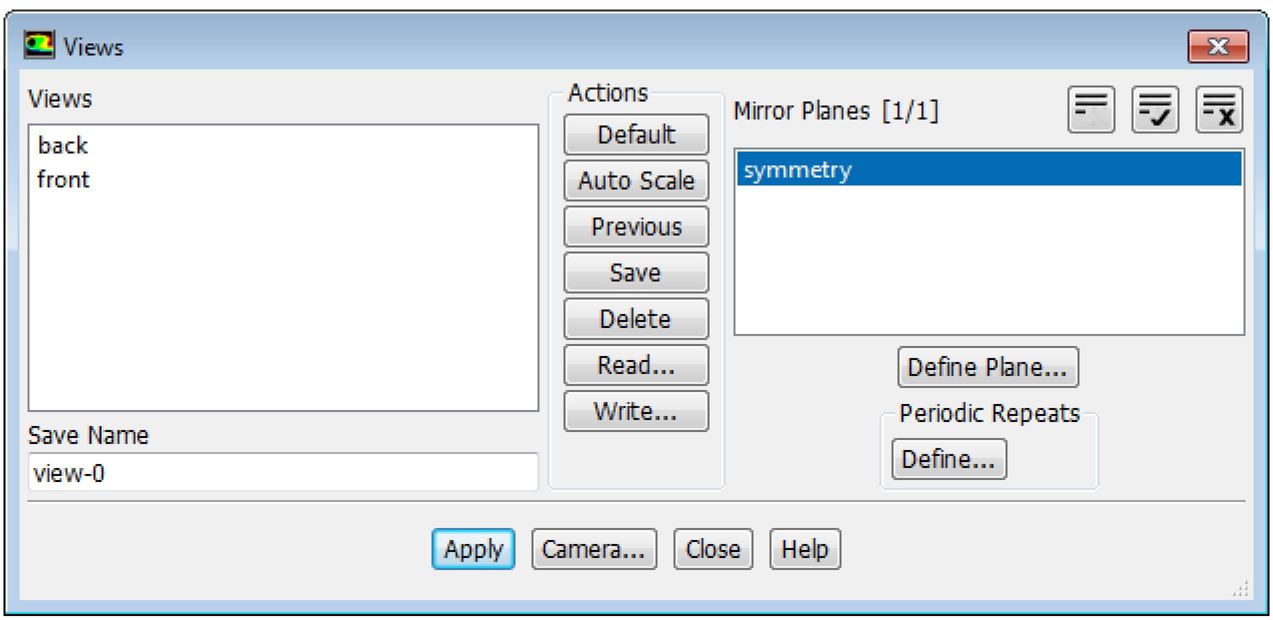
   Setting Up Domain → **Mesh** → **Scale**



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...**

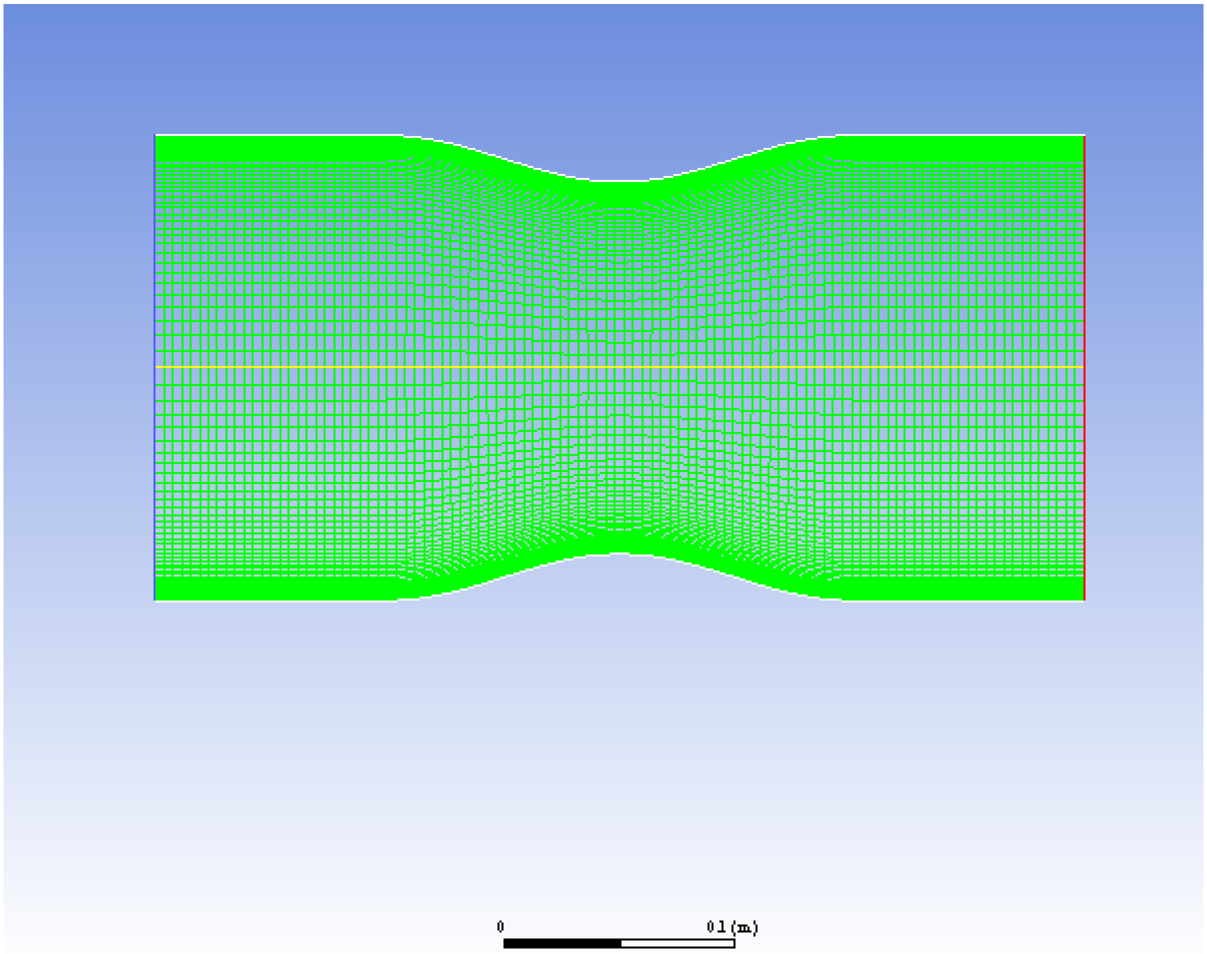


a. Select **symmetry** in the **Mirror Planes** selection list.

b. Click **Apply** to refresh the display.

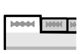
c. Close the **Views** dialog box.

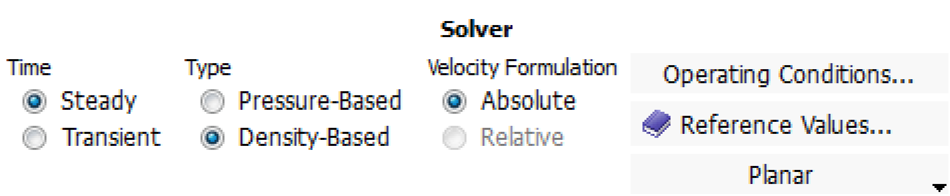
Figure 6.2: 2D Nozzle Mesh Display with Mirroring



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.

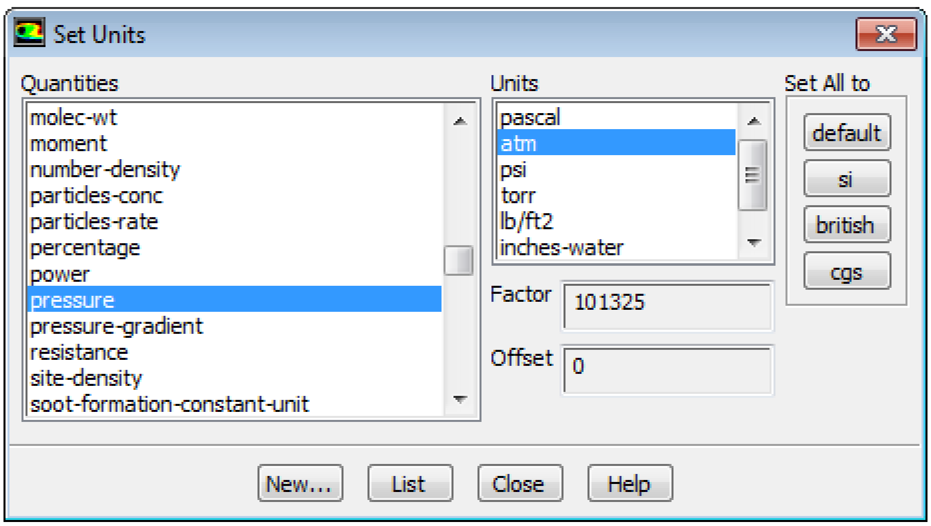
b. Retain the default selection of **Steady** from the **Time** list.

> **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**.

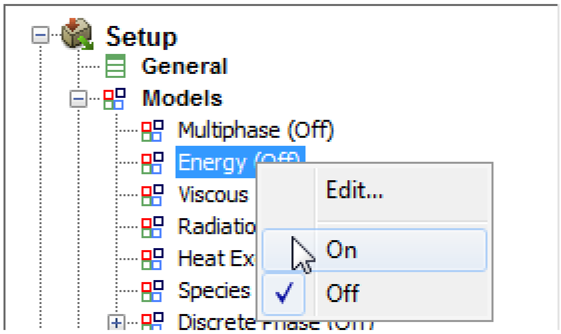
   b. Select **atm** in the **Units** selection list.

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

6.4.4. Models

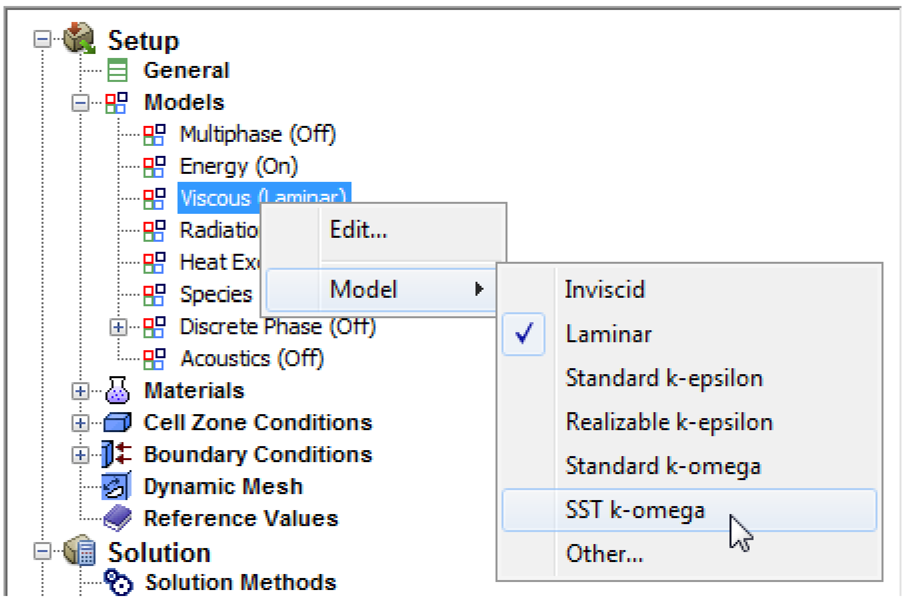
1. Enable the energy equation.

   Setup → **Models** → **Energy** **Model** → **On**



2. Select the k-omega SST turbulence model.

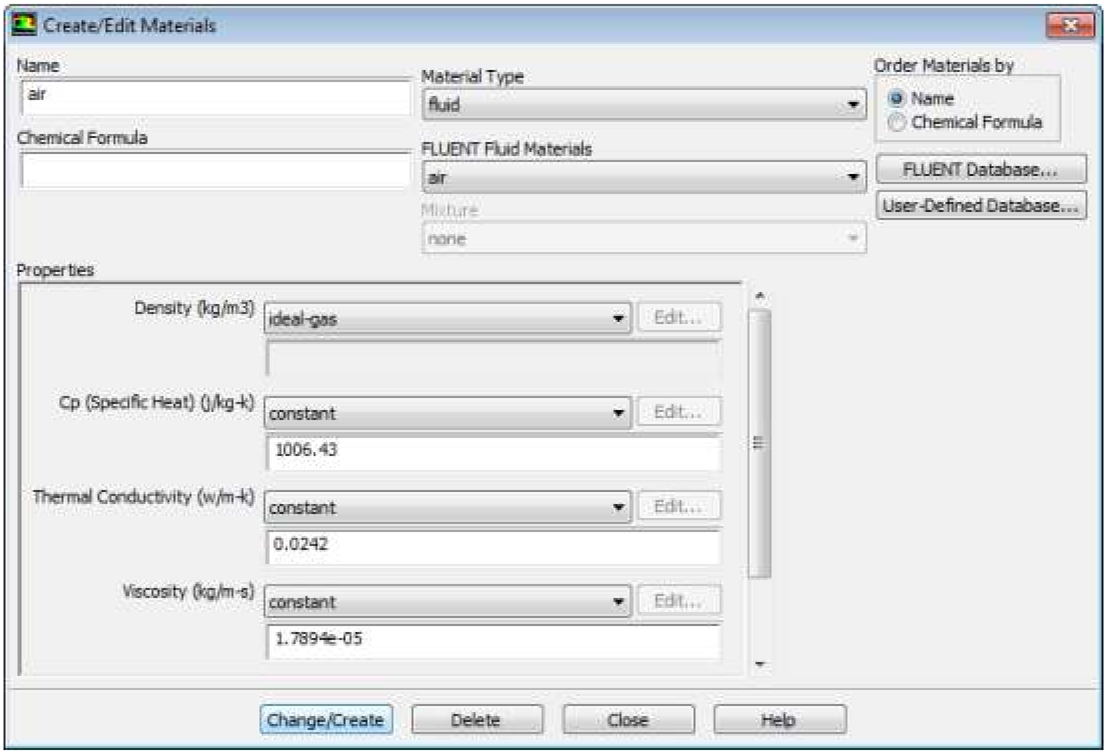
Setup → **Models** → **Viscous** **Model** → **SST k-omega**



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.

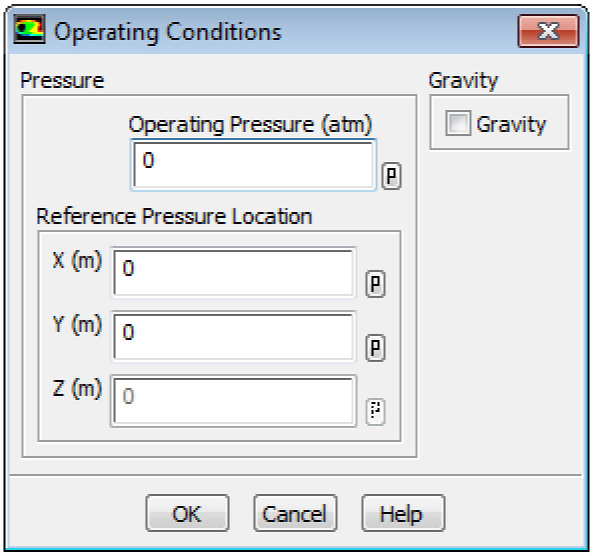
c. Click the **Change/Create** button to save your change.

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

6.4.6. Operating Conditions

1. Define the operating pressure.

Setting Up Physics → Solver → Operating Conditions...



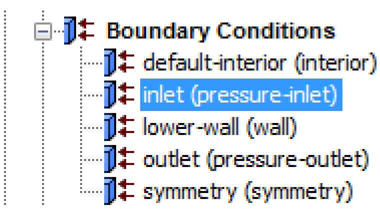
a. Enter 0 atm for **Operating Pressure**.

b. Click **OK** to close the **Operating Conditions** dialog box.

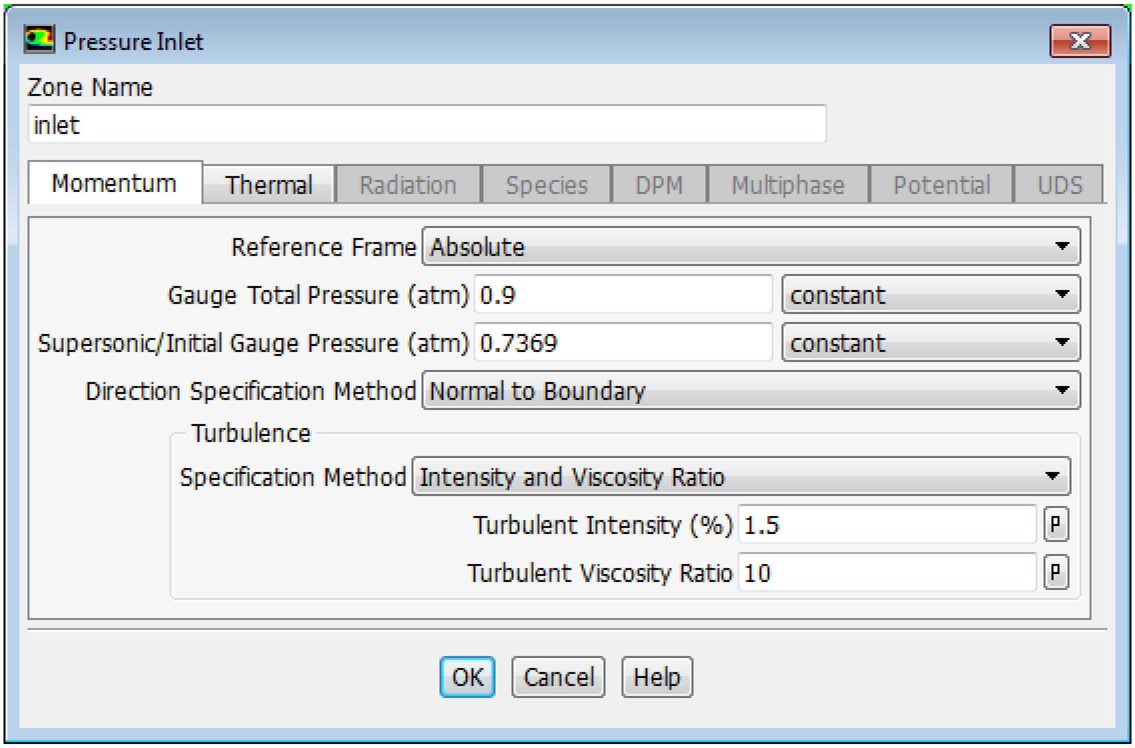
Since you have set the operating pressure to zero, you will specify the boundary condition inputs for pressure in terms of absolute pressures when you define them in the next step. Boundary condition inputs for pressure should always be relative to the value used for operating pressure.

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.

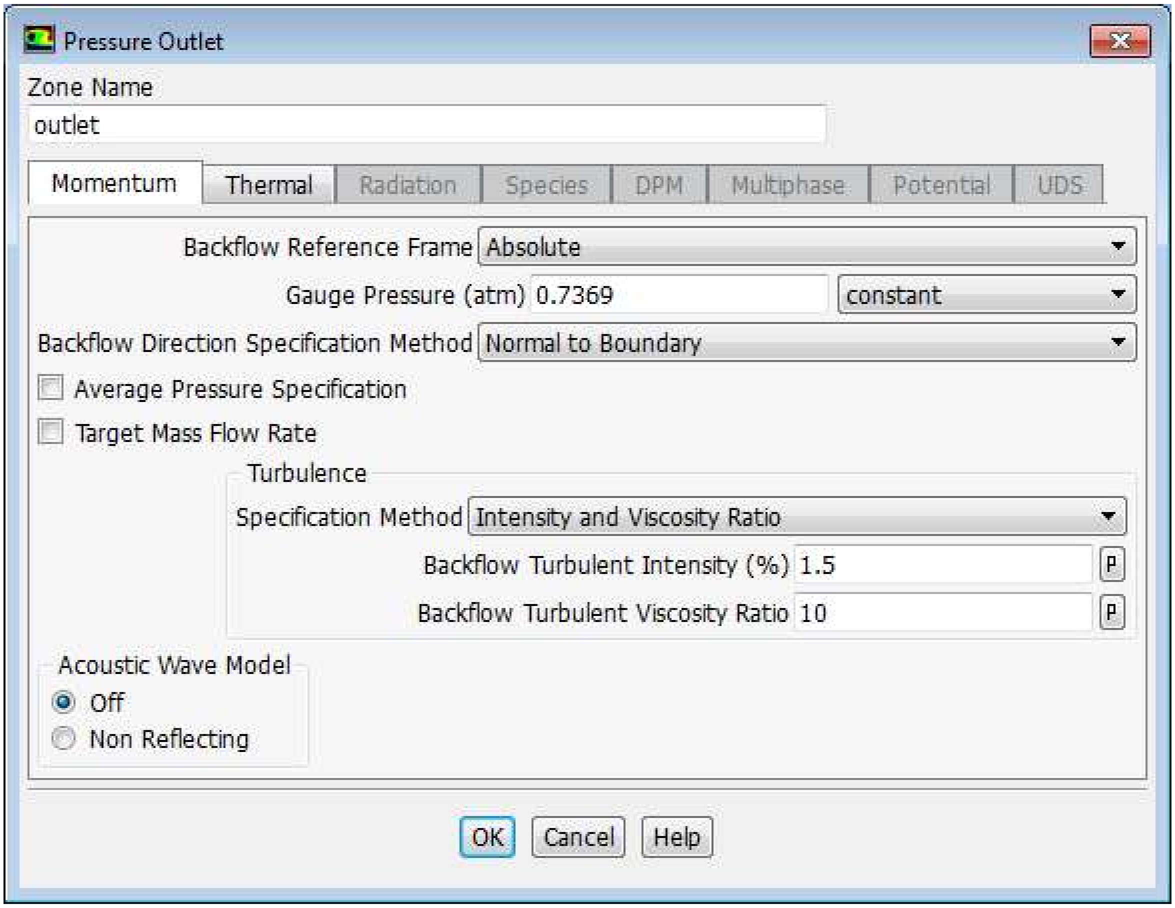
d. Enter `1.5`% for **Turbulent Intensity**.

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**.

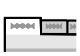
If substantial backflow occurs at the outlet, you may need to adjust the backflow values to levels close to the actual exit conditions.

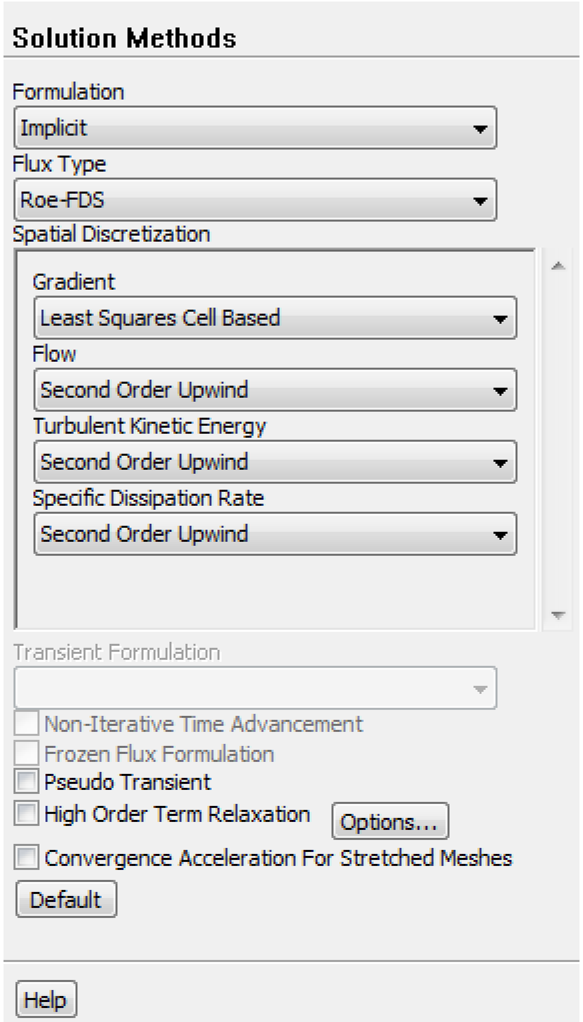
e. Click **OK** to close the **Pressure Outlet** dialog box.

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...**



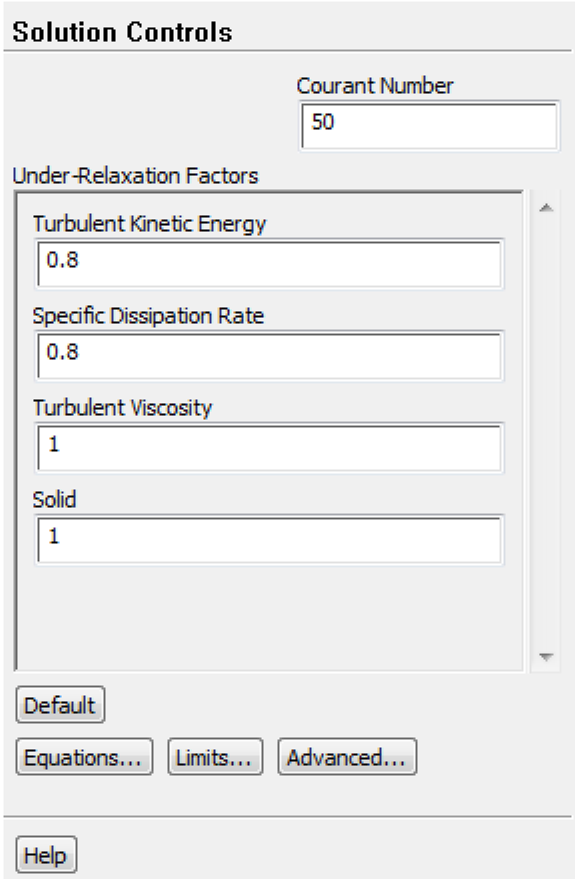
a. Retain the default selection of **Least Squares Cell Based** from the **Gradient** drop-down list in the **Spatial Discretization** group box.

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**.

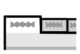
Note

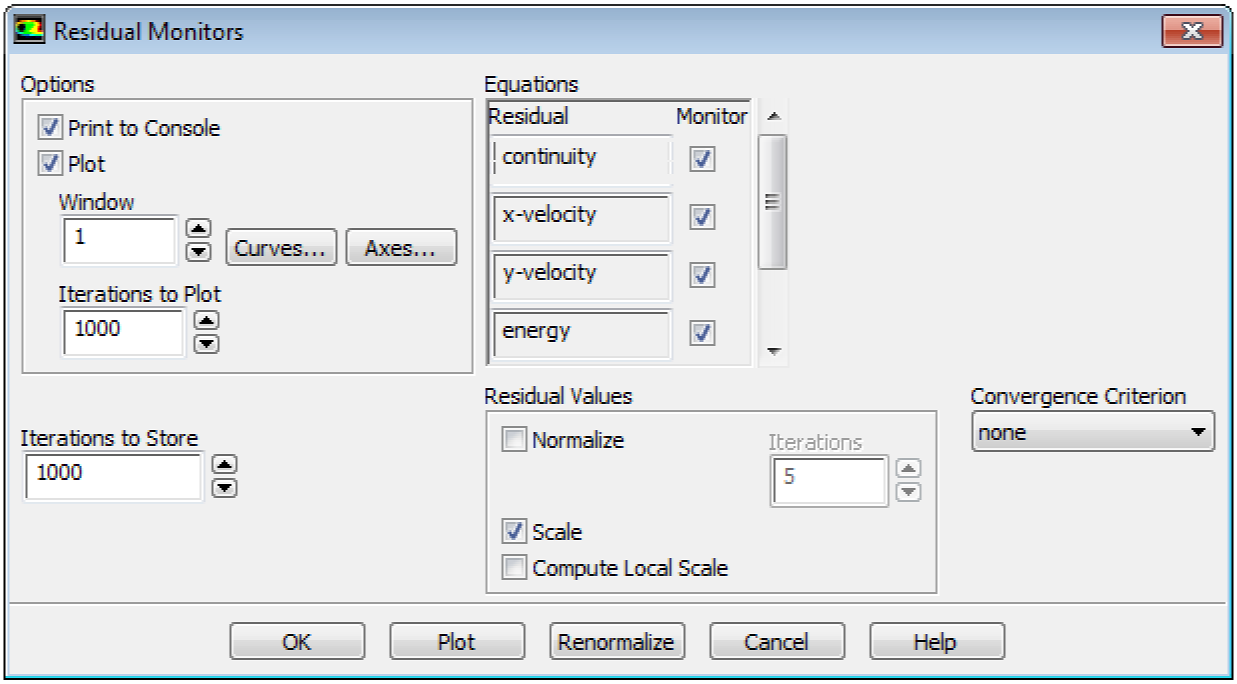
The default Courant number for the density-based implicit formulation is 5. For relatively simple problems, setting the Courant number to 10, 20, 100, or even higher value may be suitable and produce fast and stable convergence. However, if you encounter convergence difficulties at the startup of the 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...**



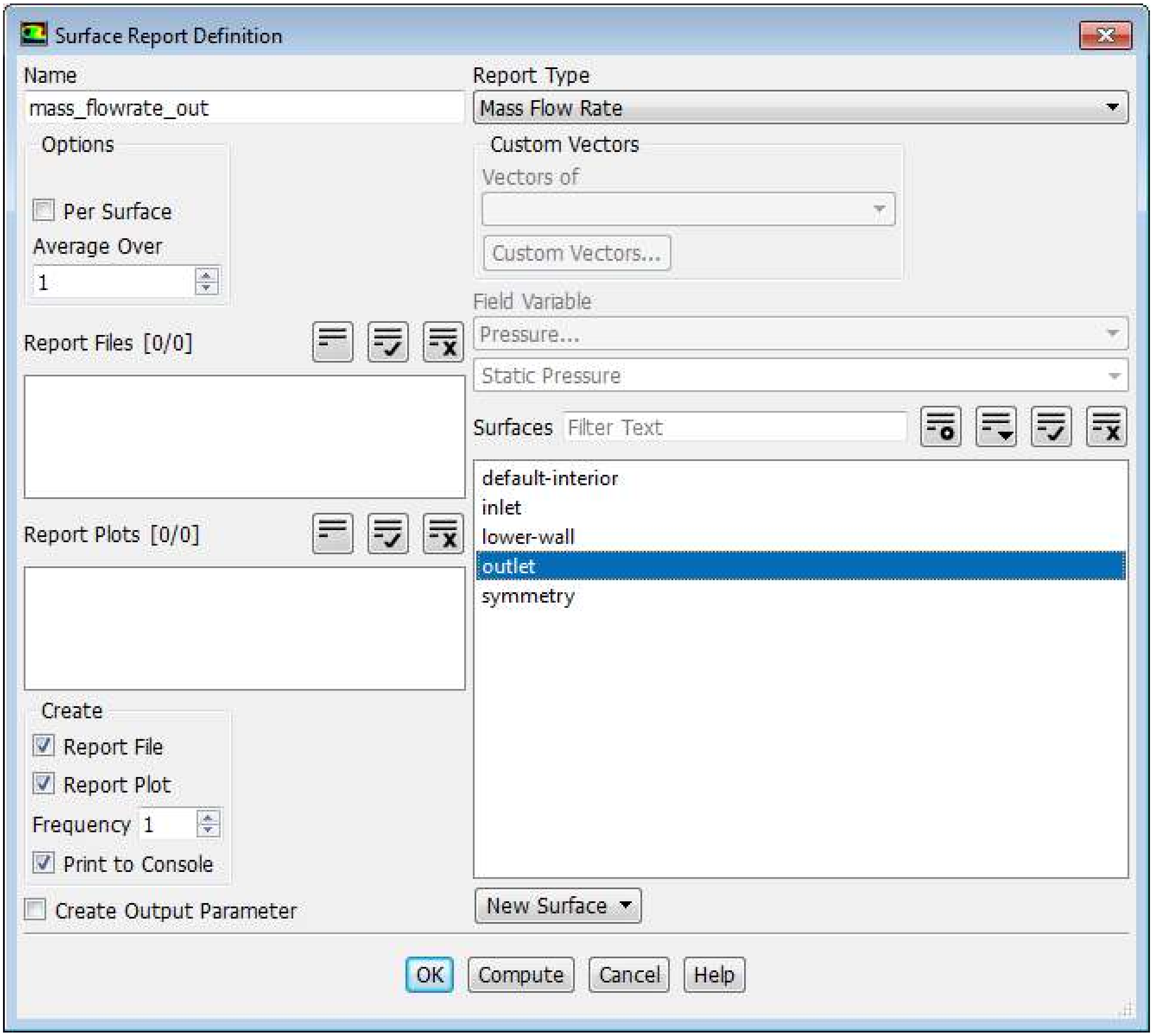
a. Ensure that **Plot** is enabled in the **Options** group box.

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.

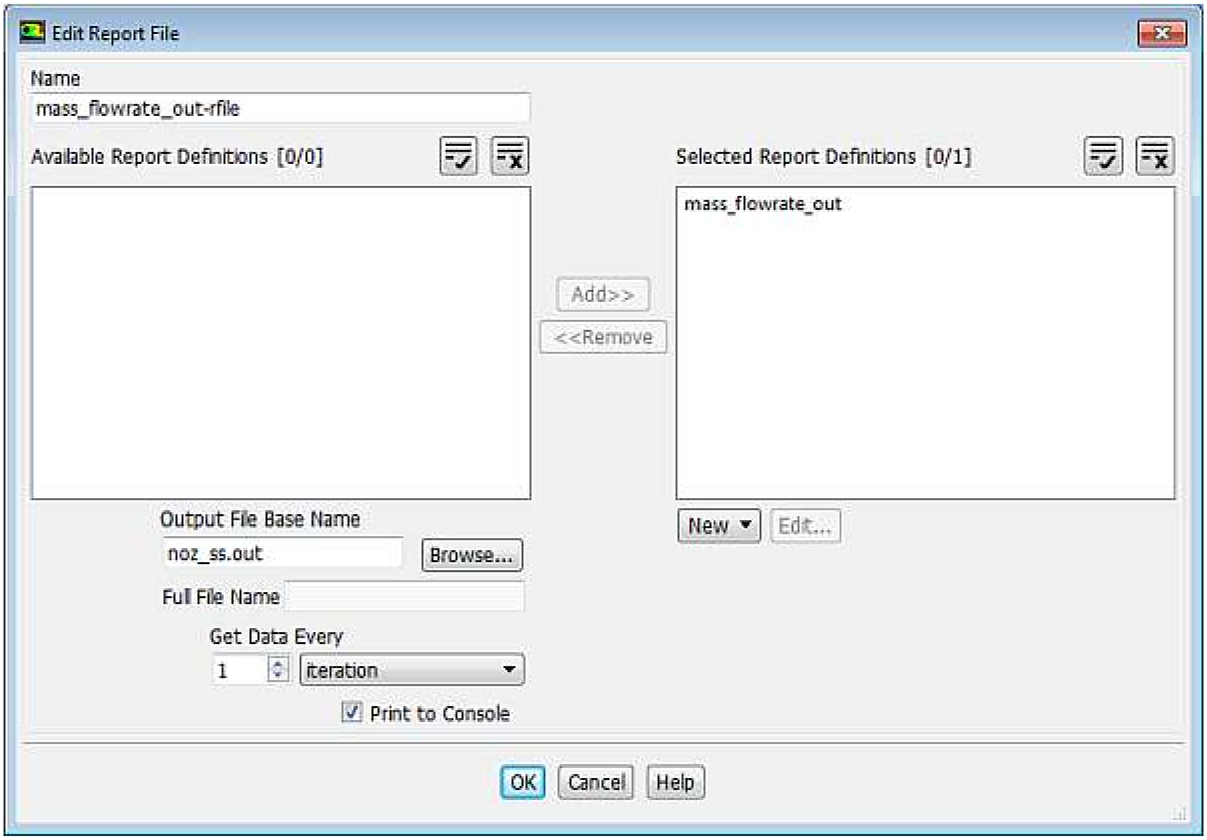
---

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**.

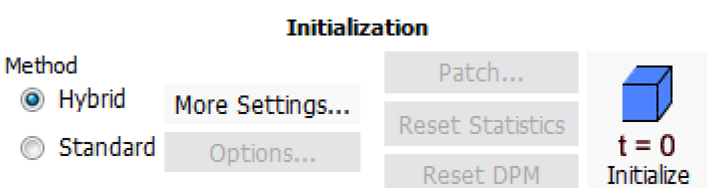
ii. Click **OK** to close the **Edit Report File** dialog box.

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

File → **Write** → **Case...**

6. Initialize the solution.

Solving → **Initialization**



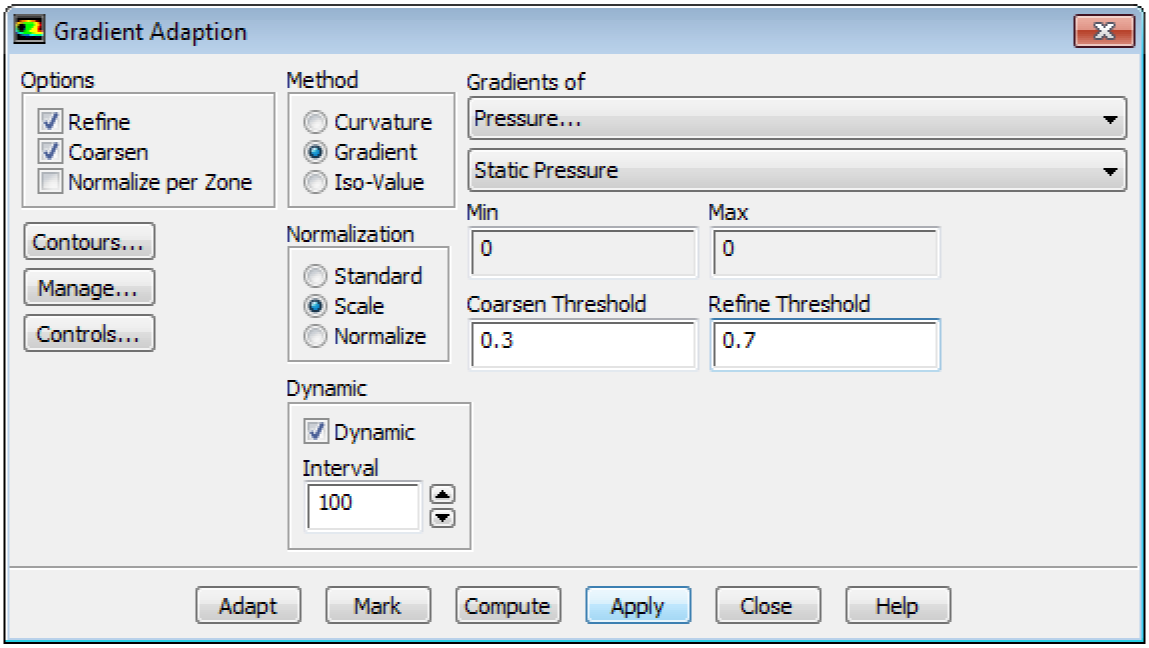
a. Keep the **Method** at the default of **Hybrid**.

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.

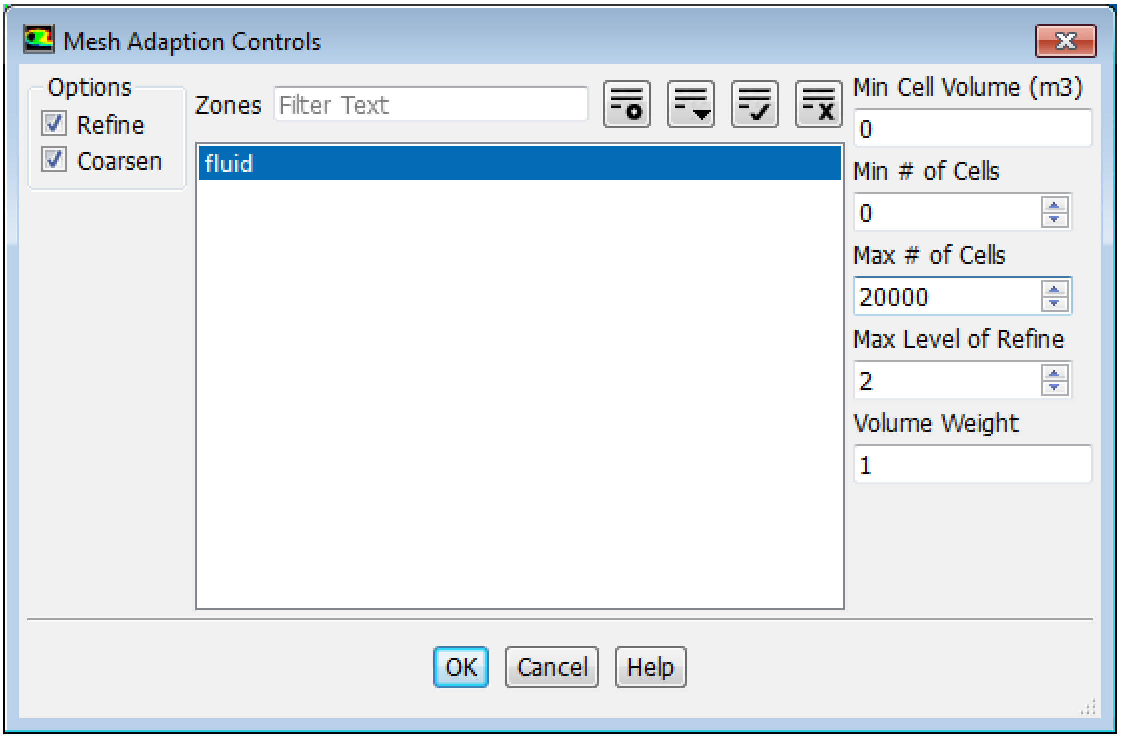
f. Enter `0.3` for **Coarsen Threshold**.

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.

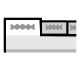
ii. Enter `20000` for **Max # of Cells**.

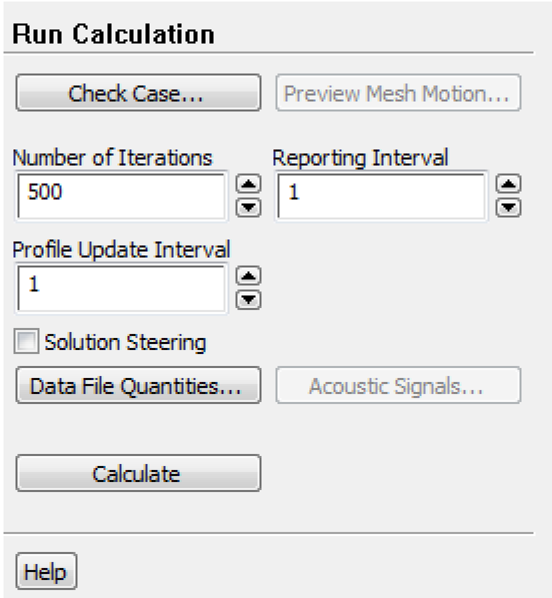
To restrict the mesh adaption, the maximum number of cells can be limited. If this limit is violated during the adaption, the coarsen and refine thresholds are adjusted to respect the maximum number of cells. Additional restrictions can be placed on the minimum cell volume, minimum number of cells, and 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.

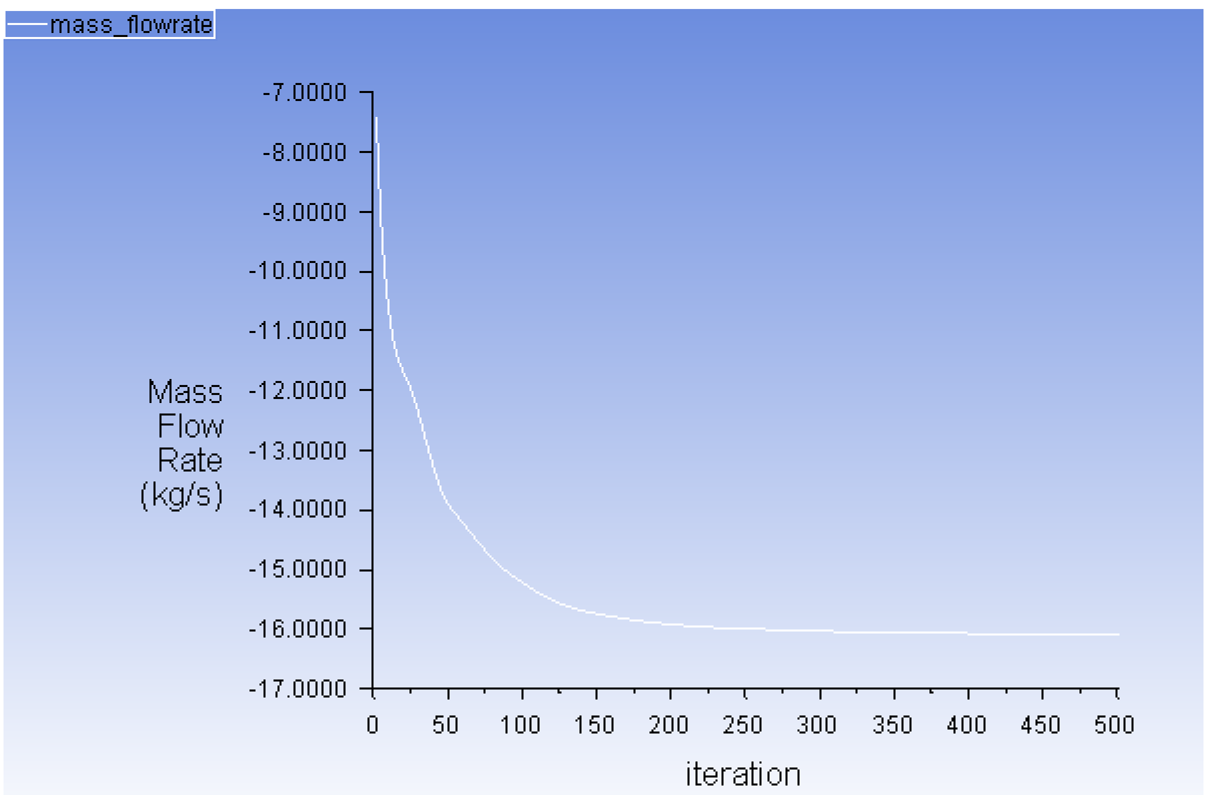
 **Solving → Run Calculation → Advanced...**



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

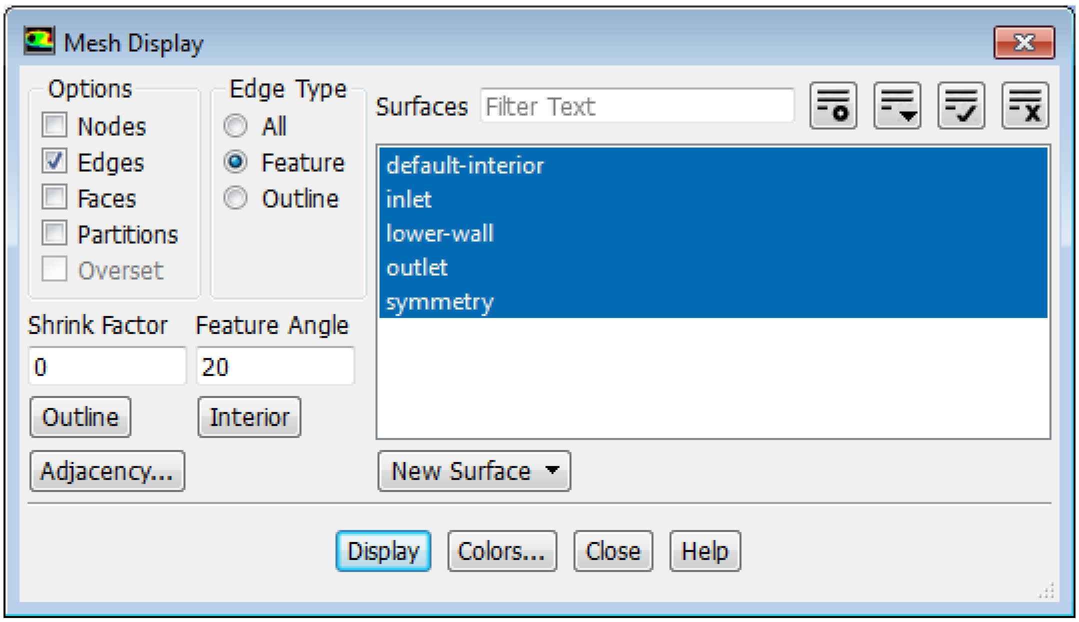
When you write the case and data files at the same time, it does not matter whether you specify the file name with a `.cas` or `.dat` extension, as both will be saved.

---

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.

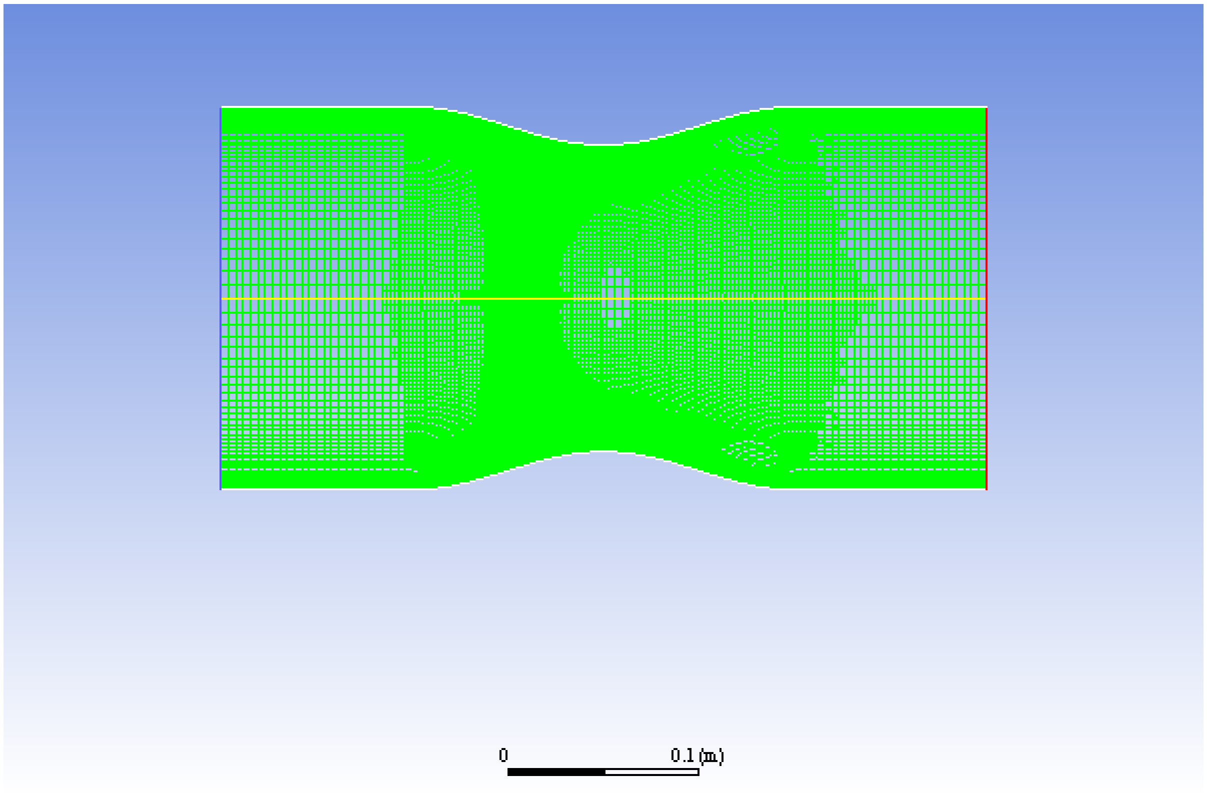
b. Select **Feature** from the **Edge Type** list.

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))

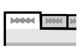
Figure 6.4: 2D Nozzle Mesh after Adaption

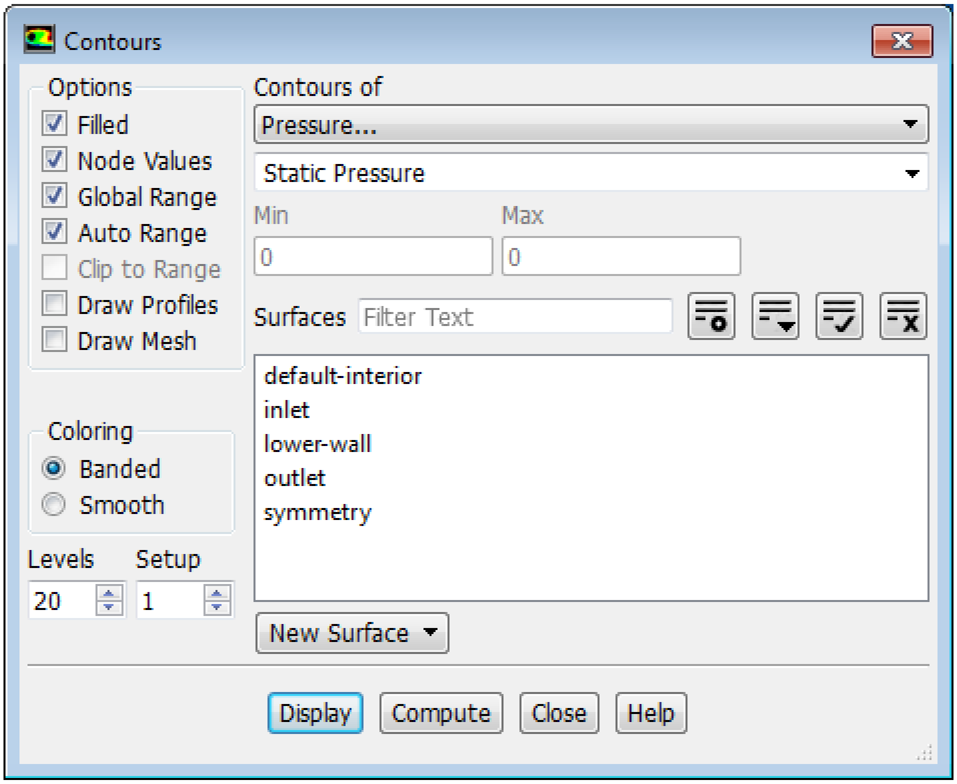


e. Zoom in using the middle mouse button to view aspects of your mesh.

   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)).

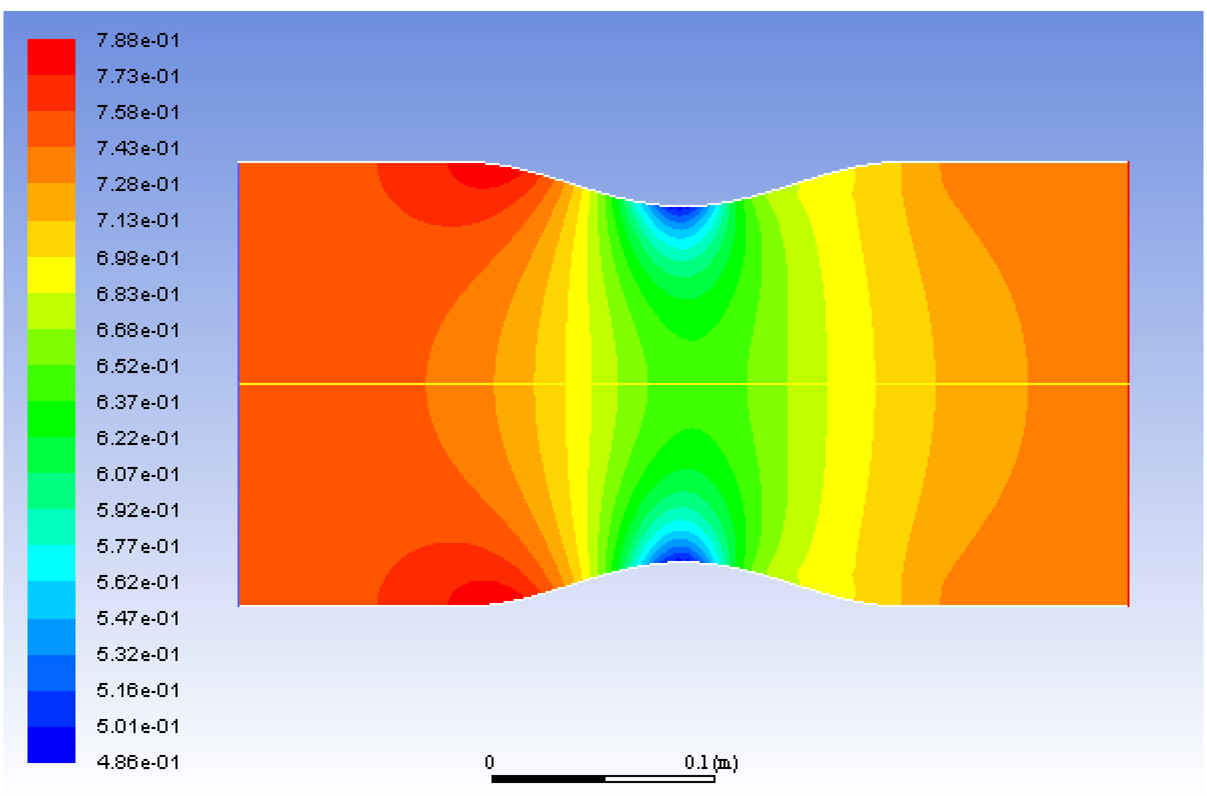
     **Postprocessing** → **Graphics** → **Contours** → **Edit...**



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

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

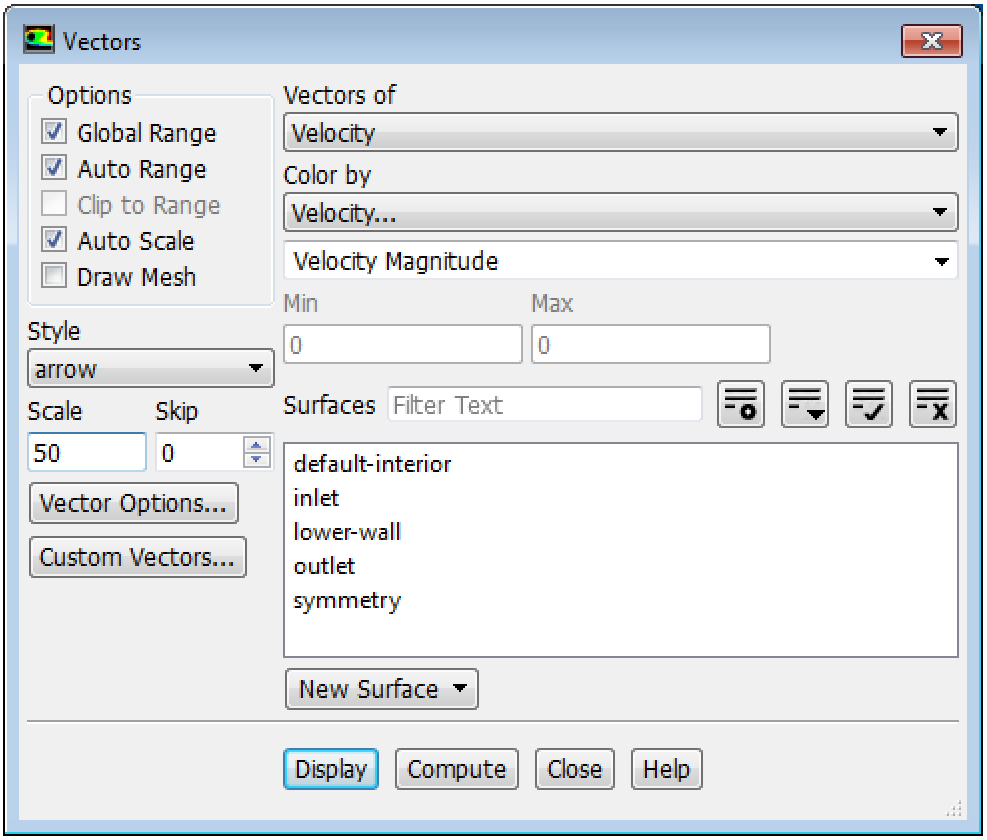
Figure 6.5: Contours of Static Pressure (Steady Flow)



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...**



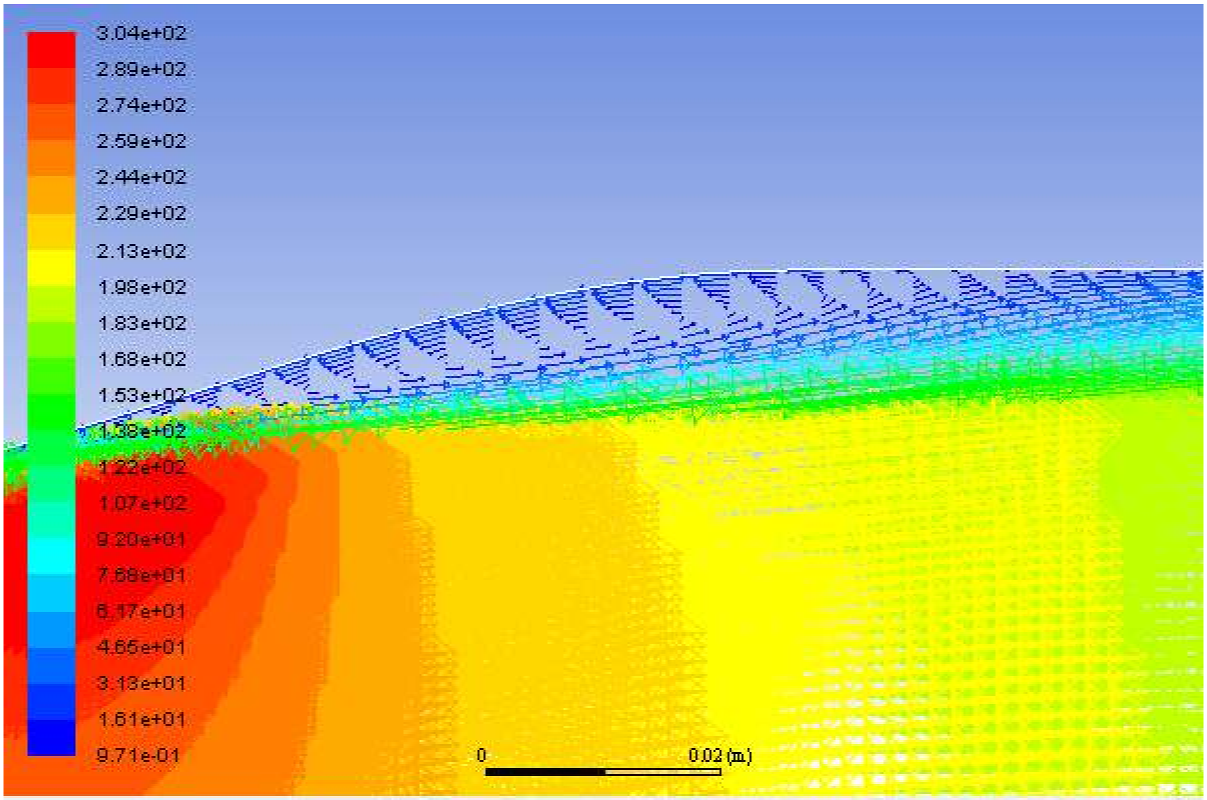
a. Enter `50` under **Scale**.

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) .

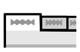
Figure 6.6: Velocity Vectors Showing Recirculation (Steady Flow)

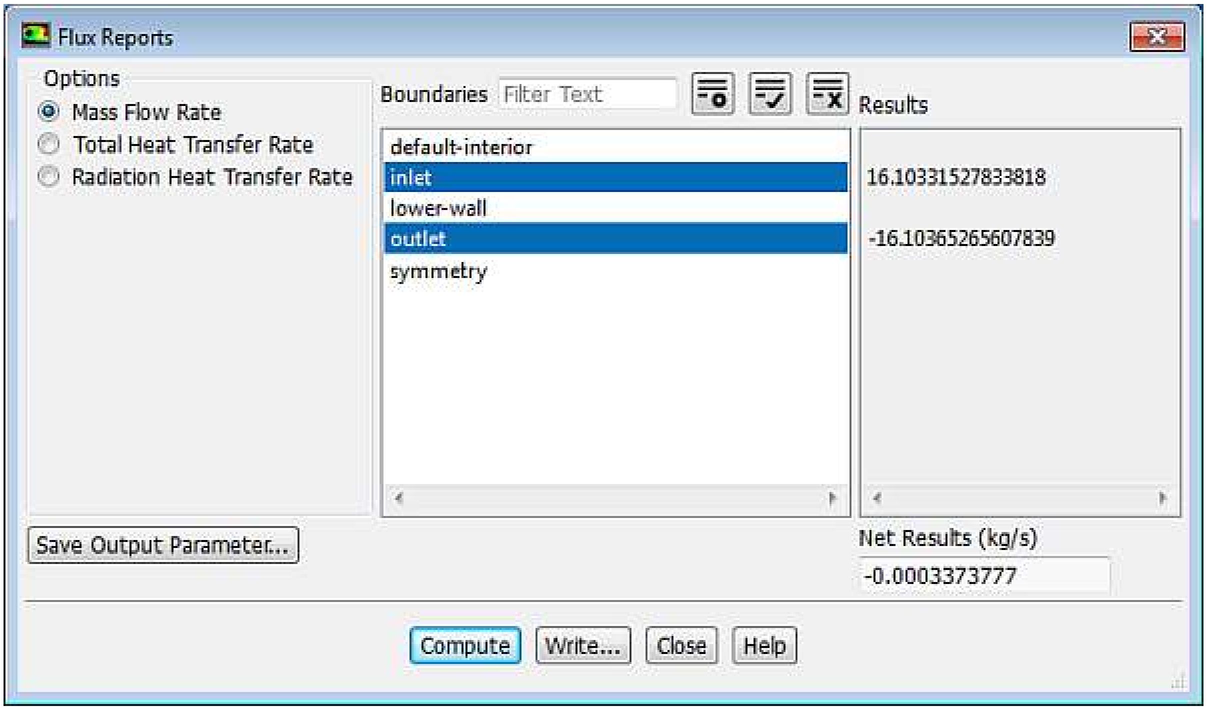


14. Check the mass flux balance.

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)

$\bar{p}_{exit}$ = mean exit pressure (atm)

In this case, ω=2200 *rad/s, and* $\bar{p}_{exit}$= 0.7369 *atm.*

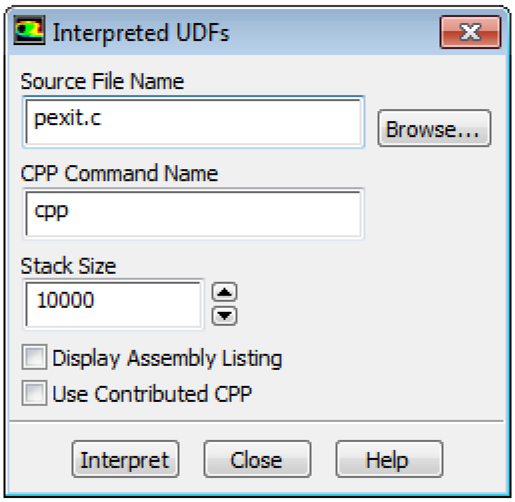
A user-defined function (`pexit.c` *) has been written to define the equation (Equation 6.1 (p. 291)) required for the pressure profile.*

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.*

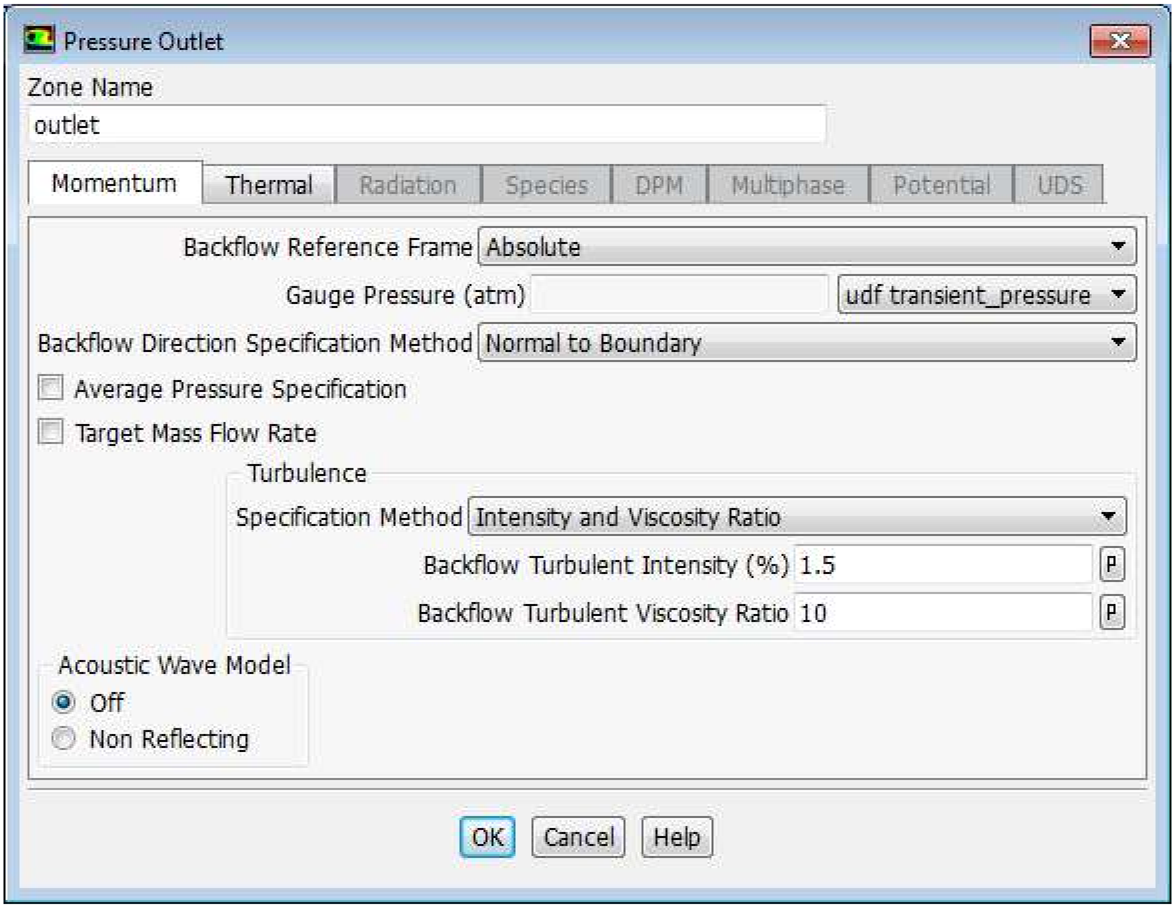
b. Click **Interpret**.

   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).

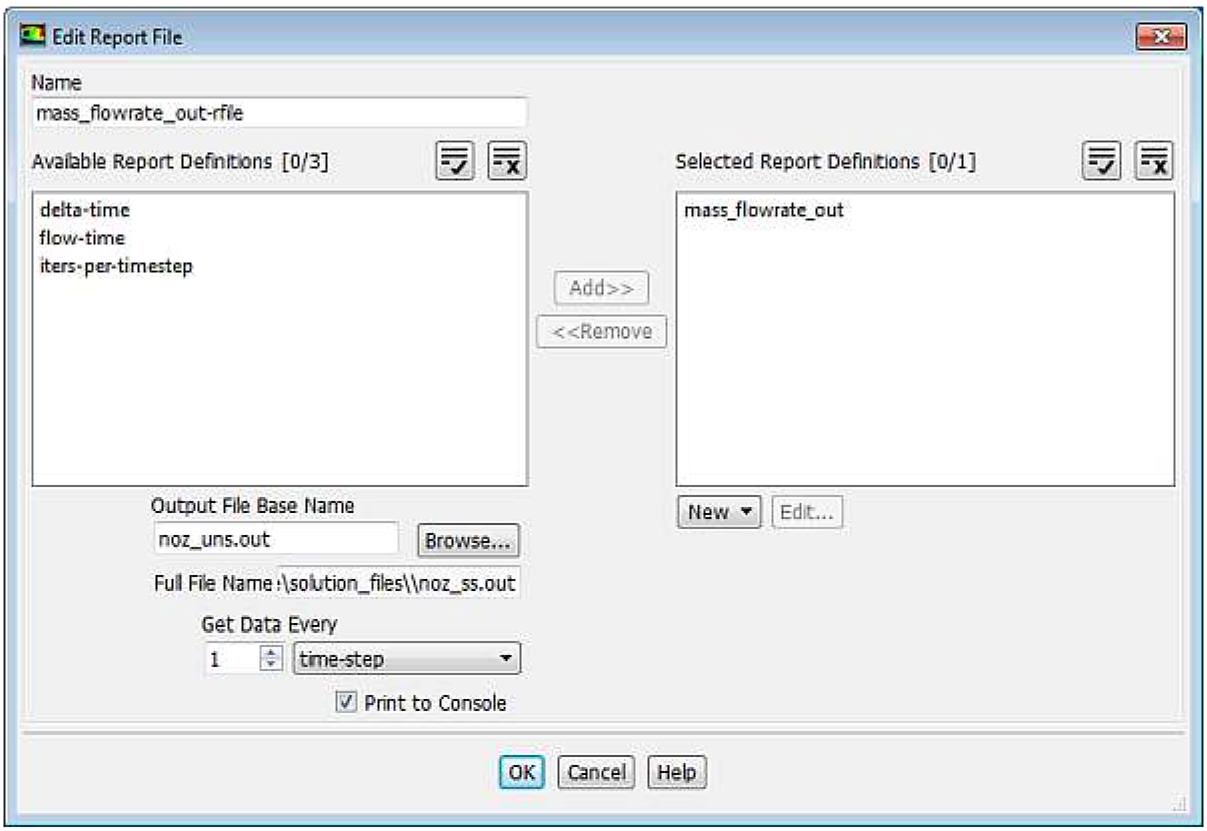
iii. Click **OK** to close the **Mesh Adaption Controls** dialog box.

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...**



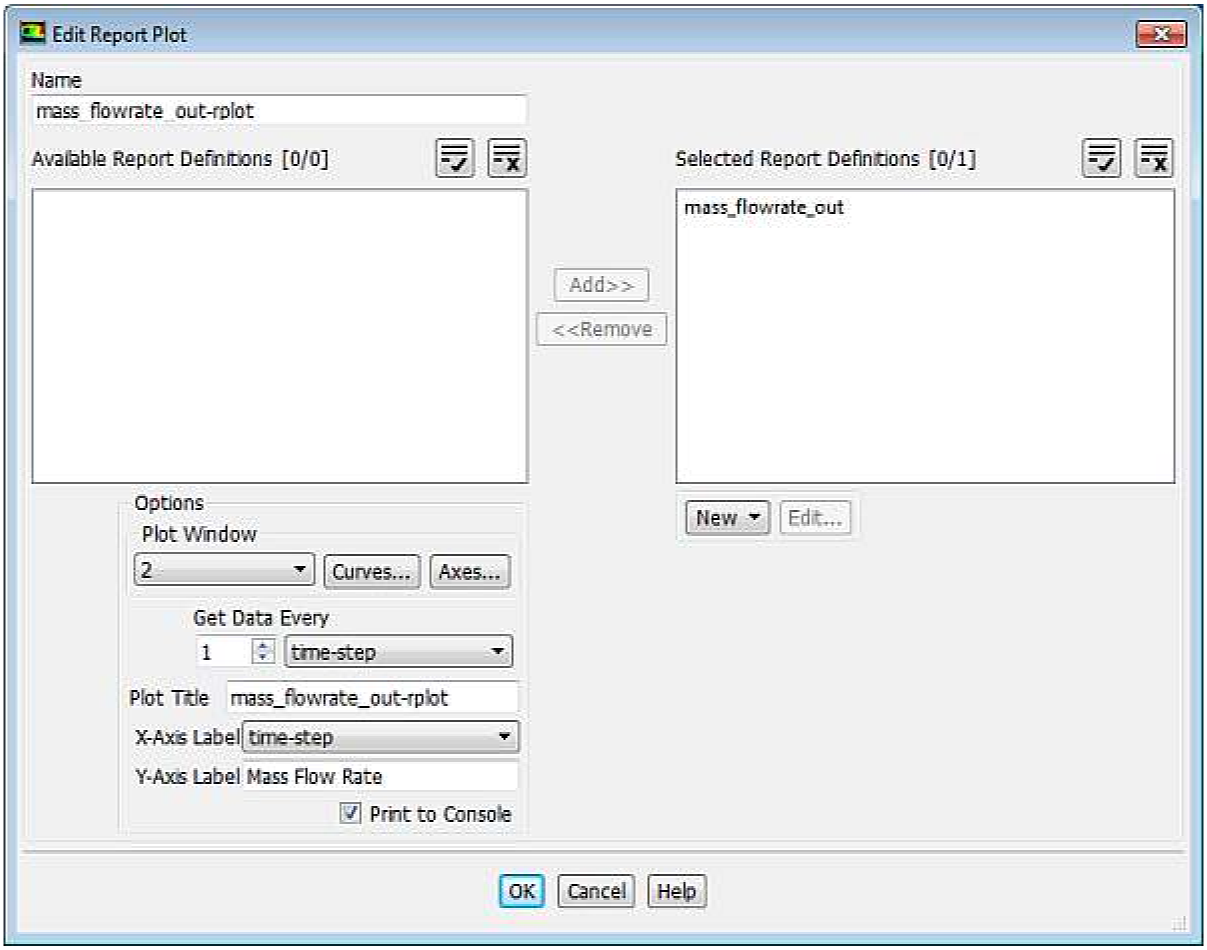
a. Enter `noz_uns.out` for **Output File Base Name**.

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.

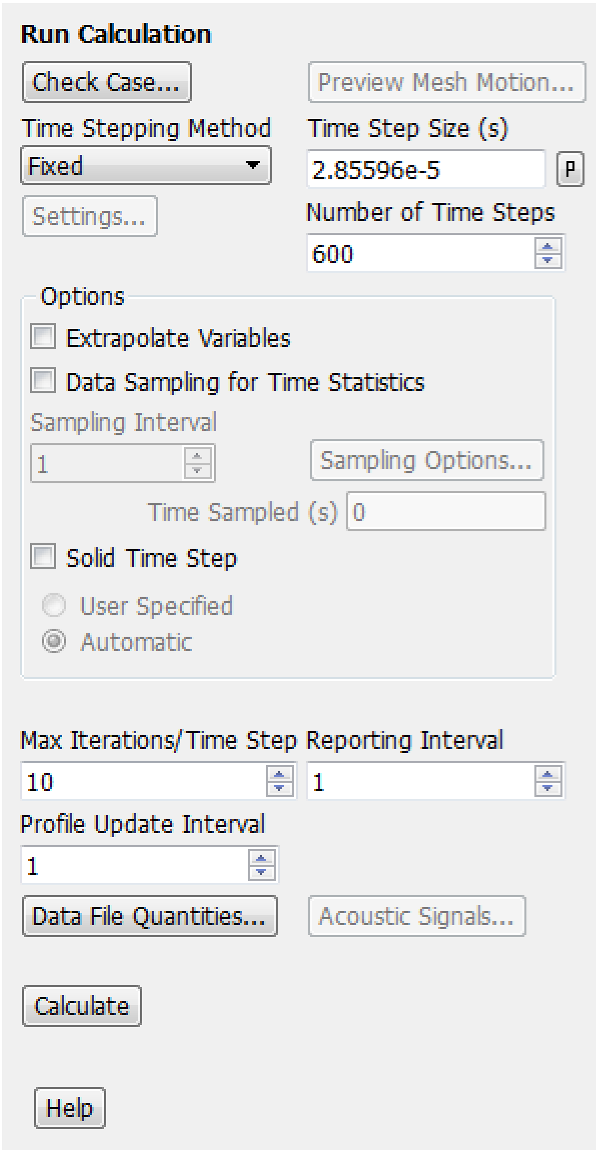
c. Set the **Iterations to Plot** to `100`.

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**.

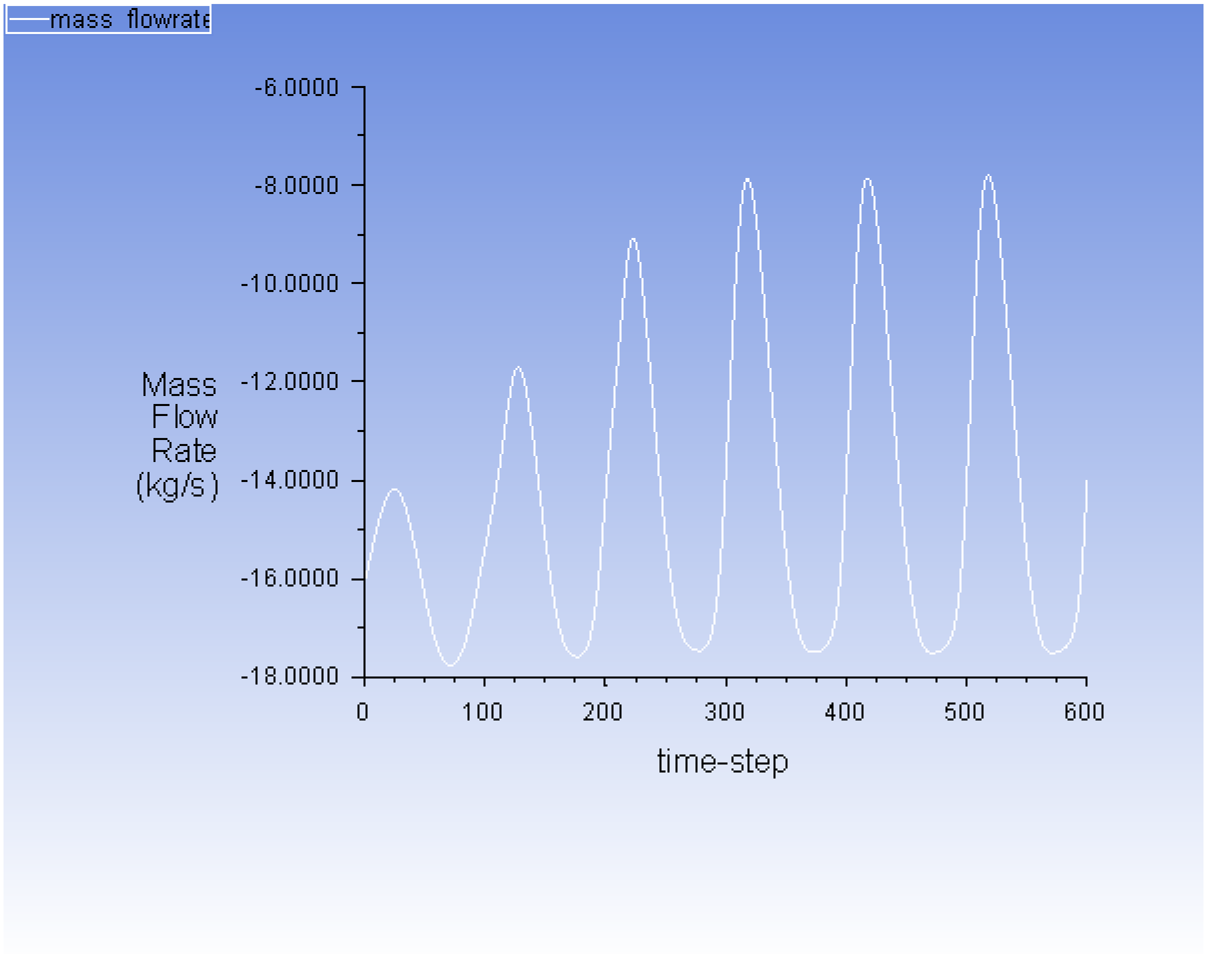
b. Enter `600` for **Number of Time Steps**.

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`).

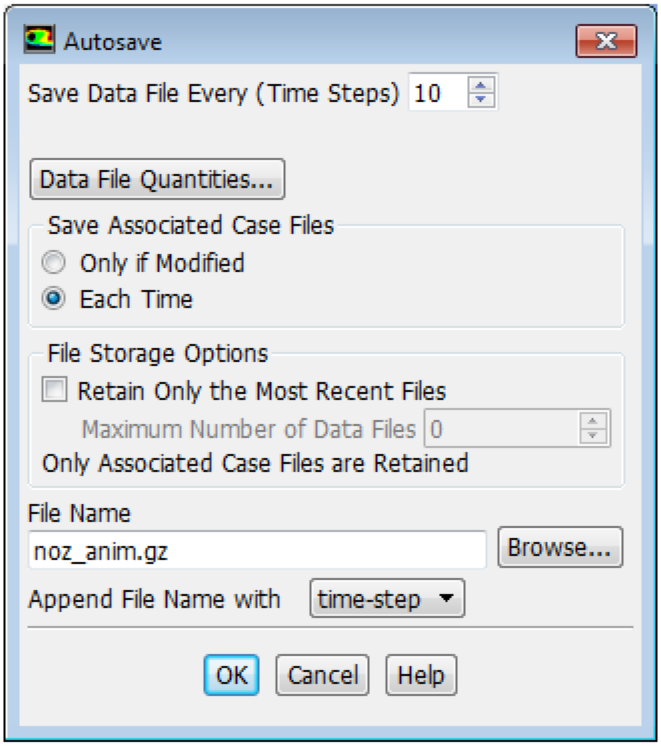
   File → **Write** → **Case & Data...**

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.

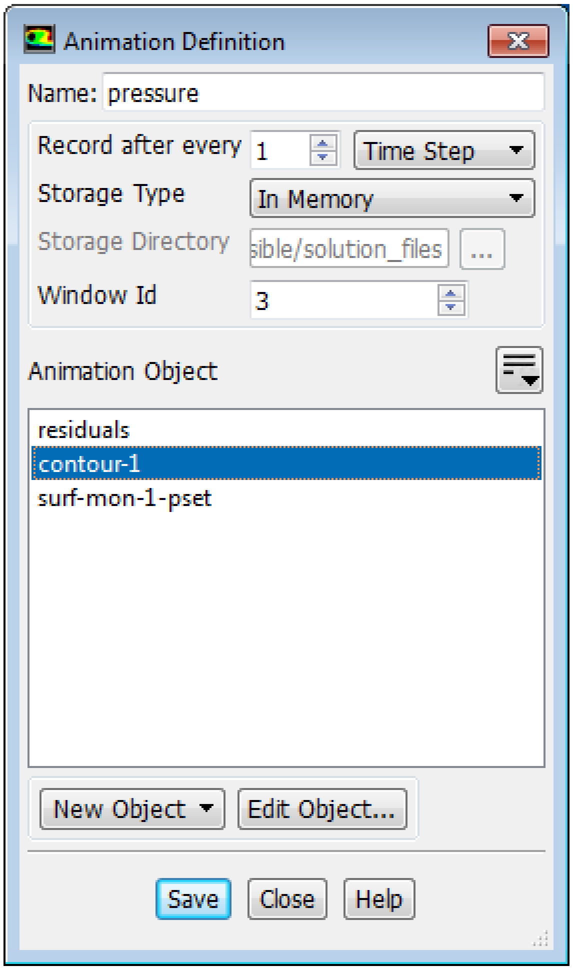
Tip

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**.

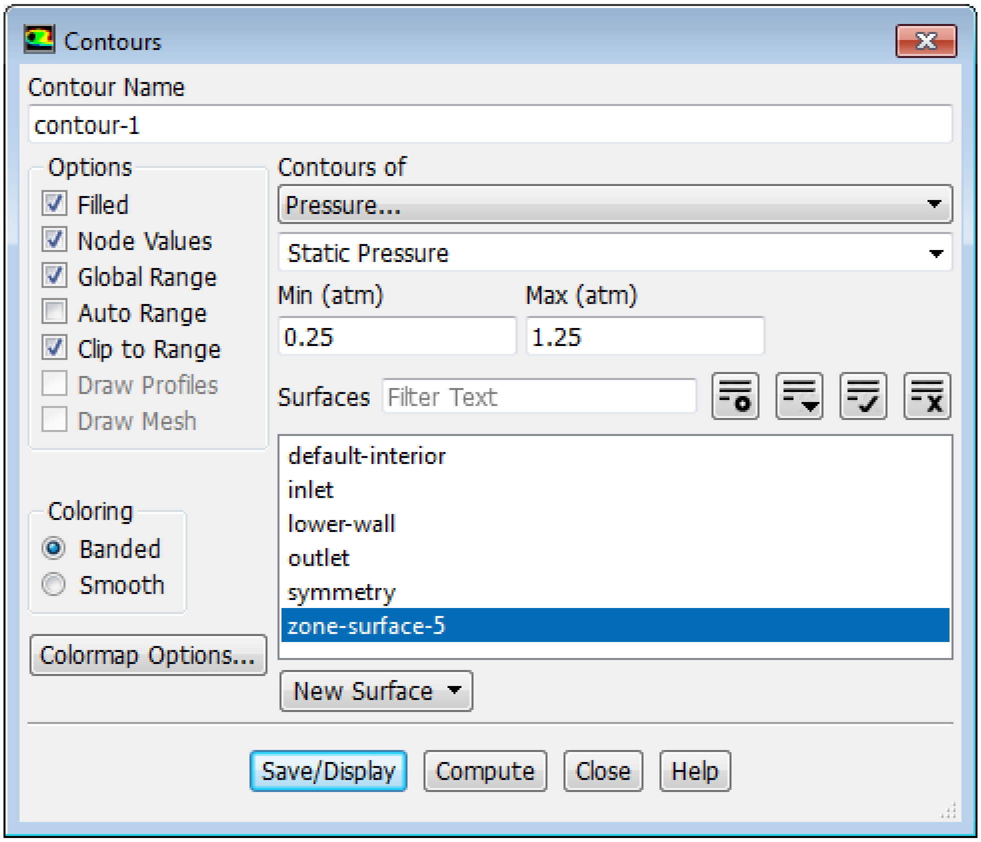
   The default value of `1` *in the integer number entry box instructs ANSYS Fluent to update the animation sequence at every time step.*

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.

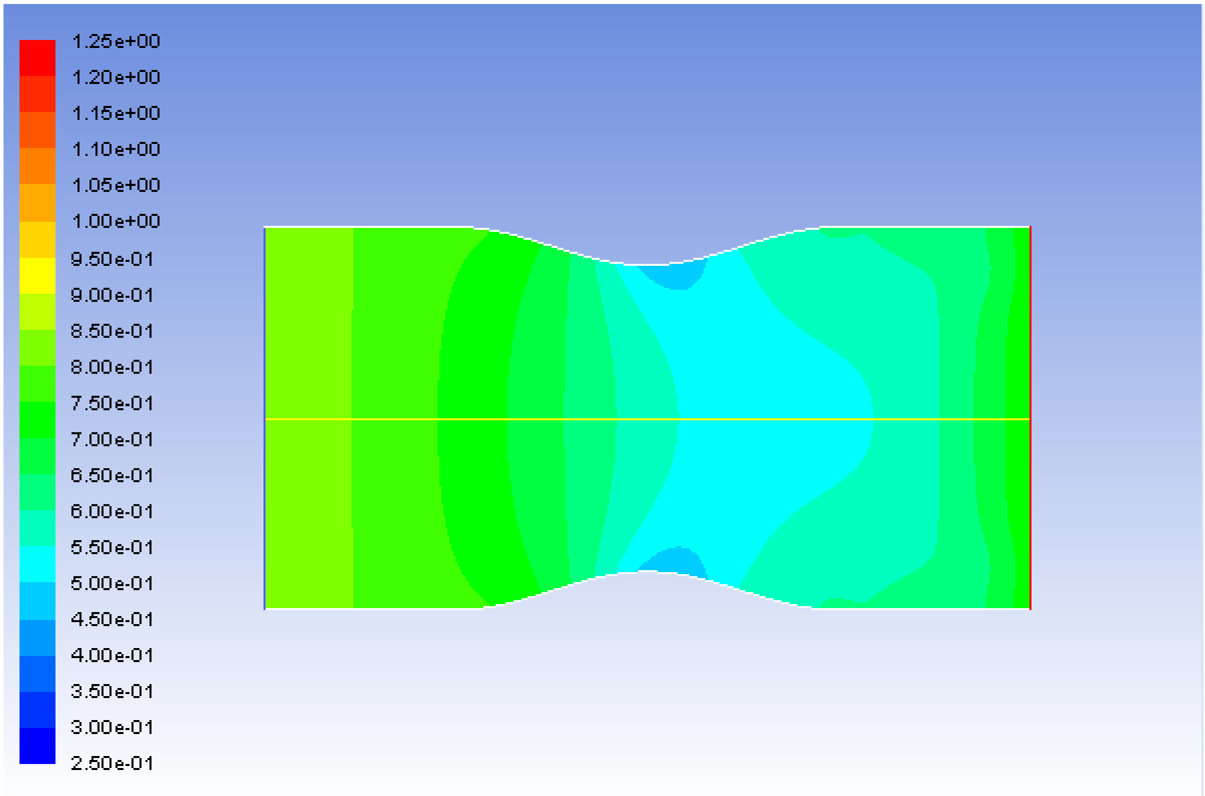
ii. Ensure that **Filled** is enabled in the **Options** group box.

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.

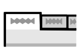
vi. **Figure 6.8: Pressure Contours at t=0.017136 s**

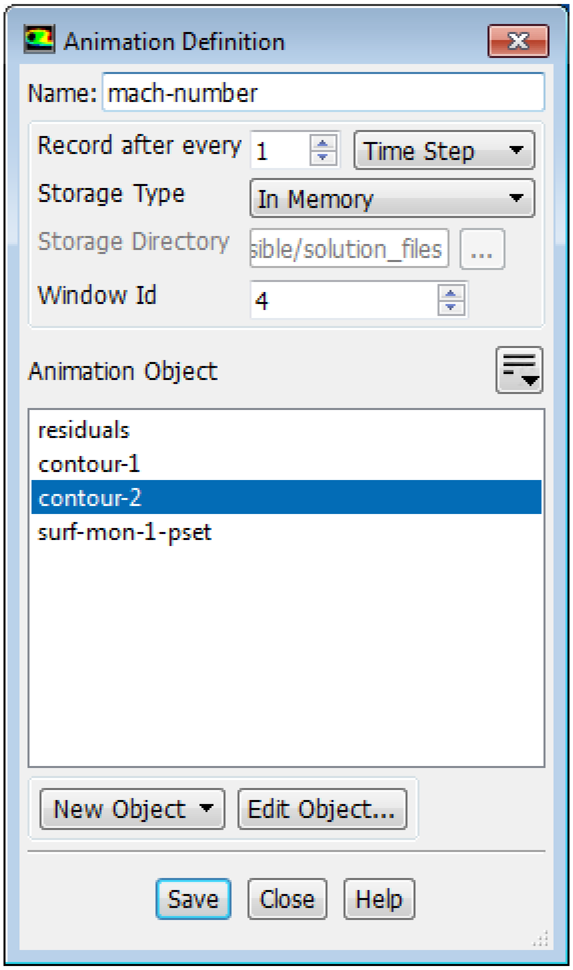


f. Ensure **contour-1** is selected in the **Animation Object** group box.

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.

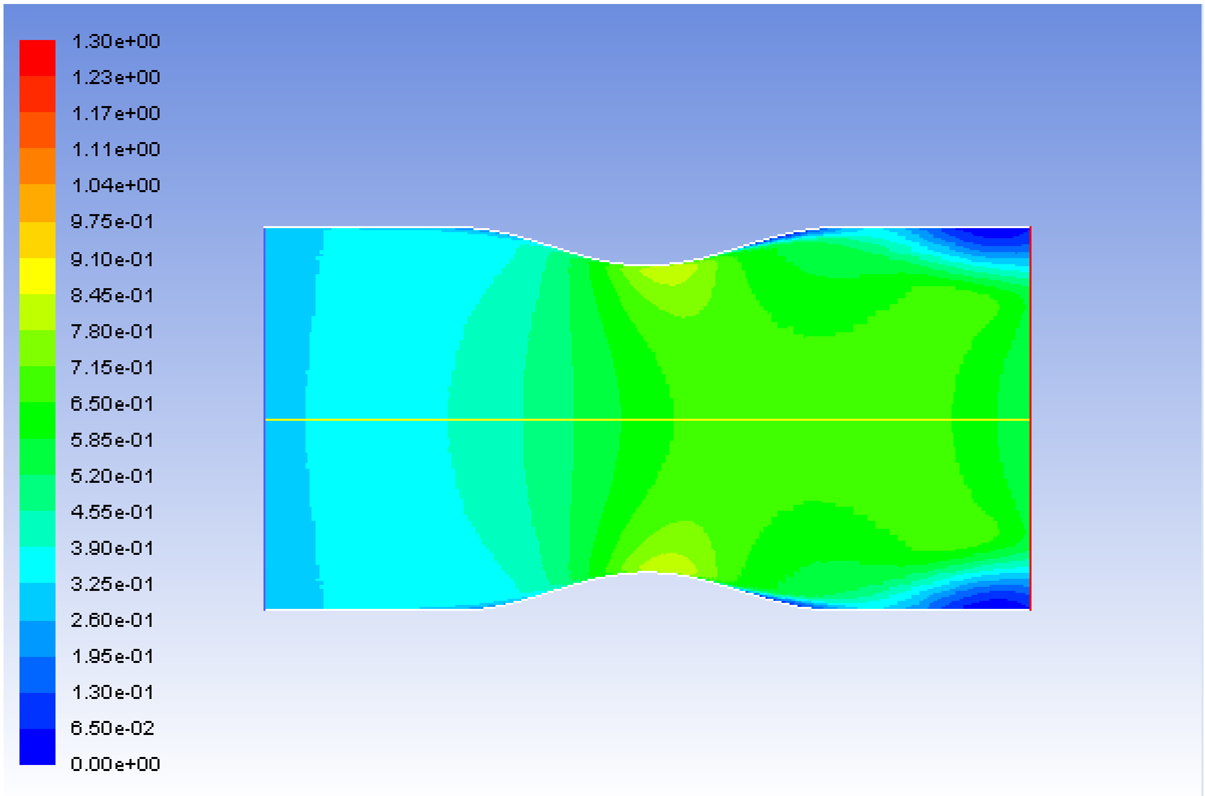
   ii. Ensure that **Filled** is enabled in the **Options** group box.

   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.

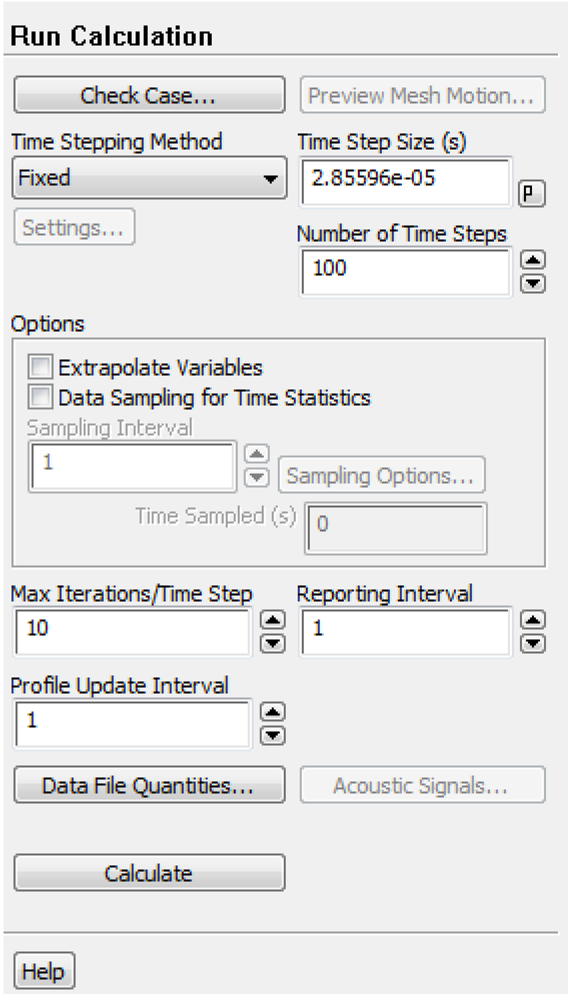
g. Click **Save** and close the **Animation Definition** dialog box.

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...

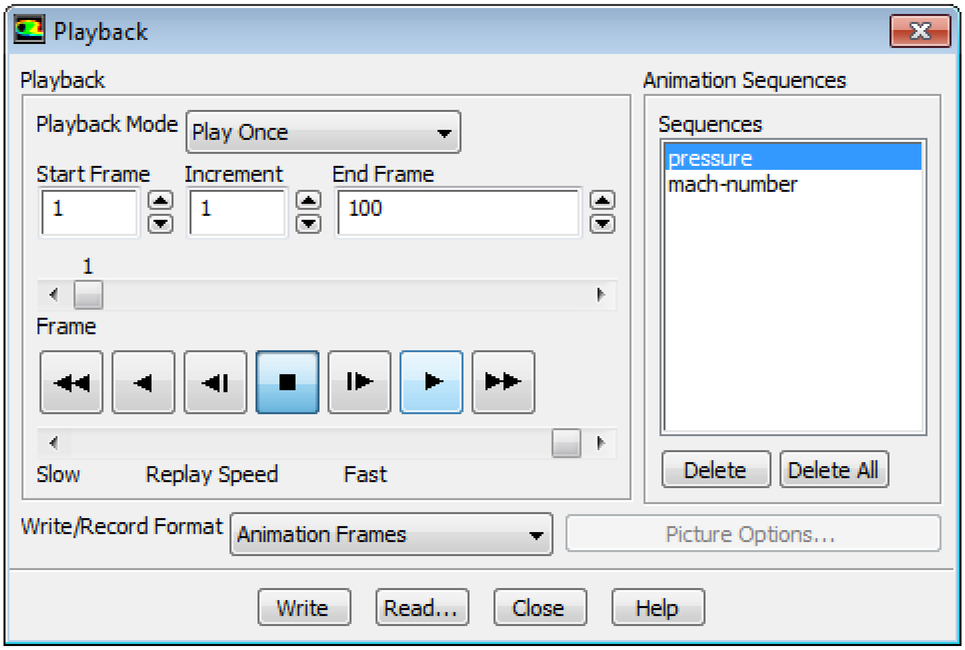


Enter `100` for **Number of Time Steps** and click **Calculate**.

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

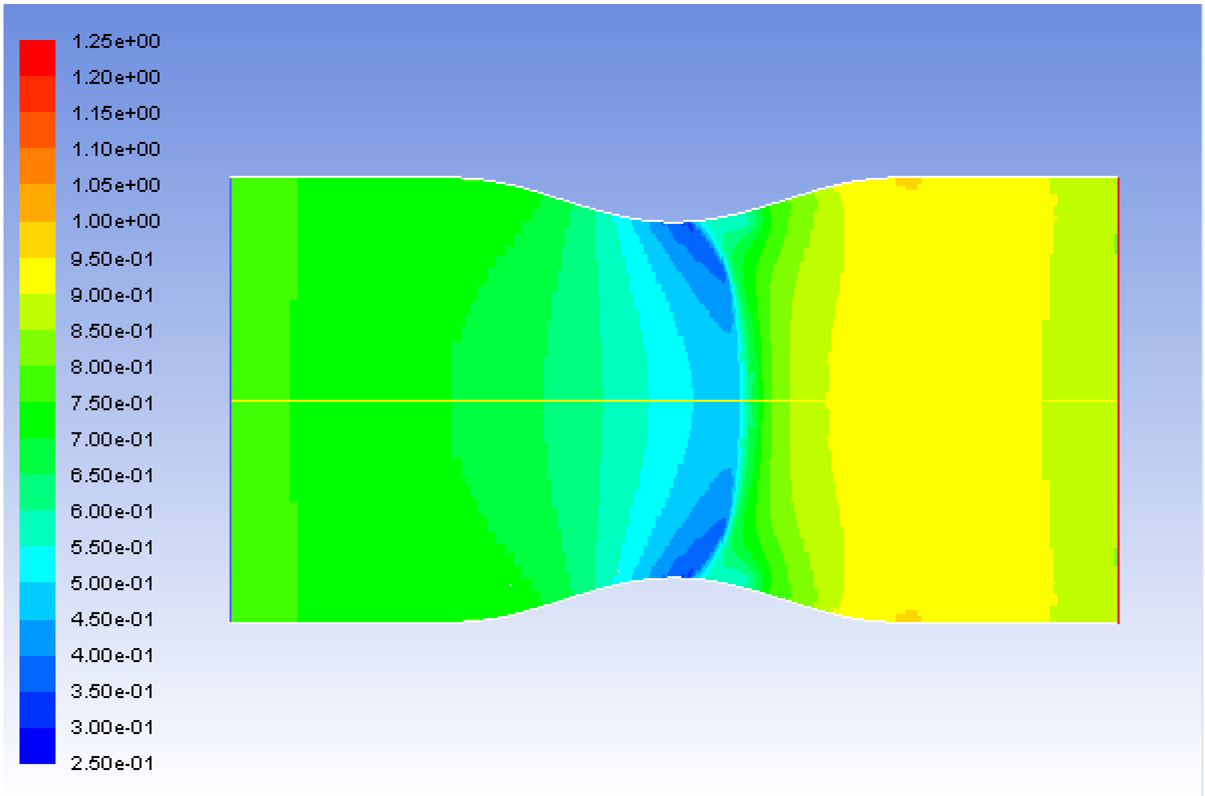
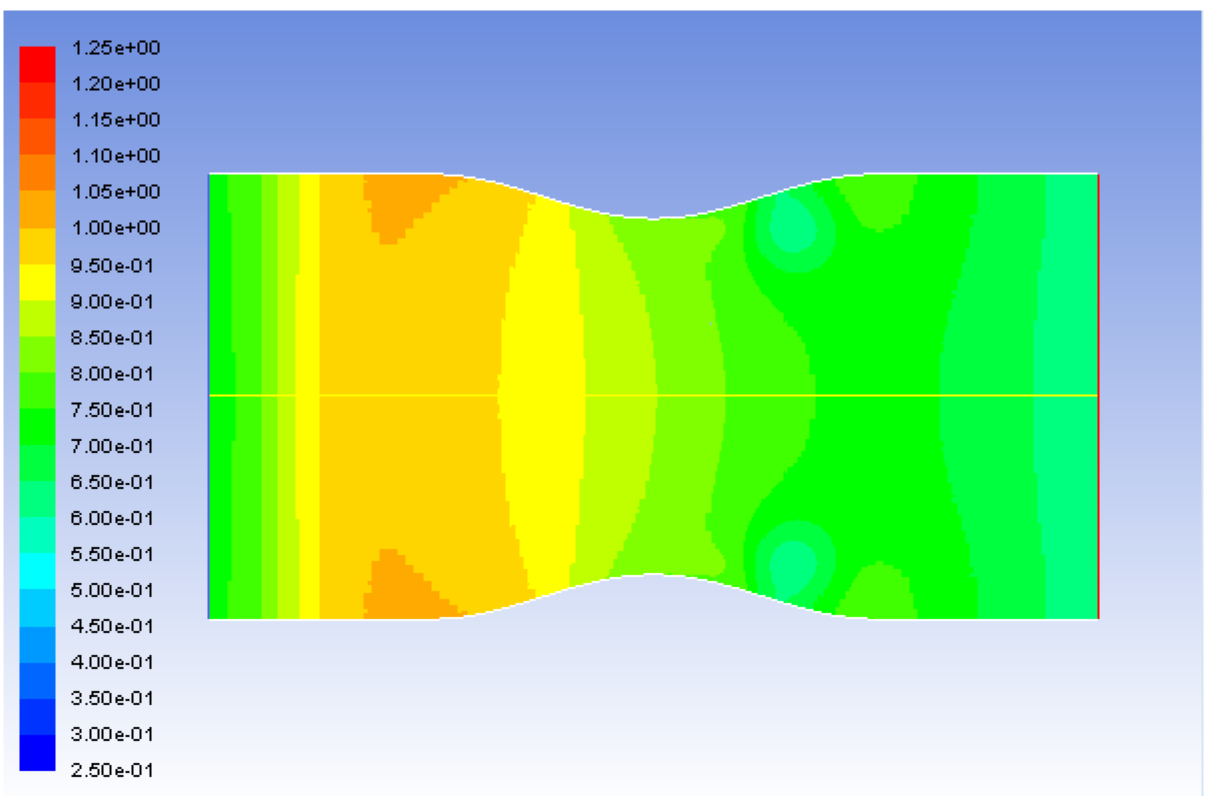


Figure 6.11: Pressure Contours at t=0.019135 s



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

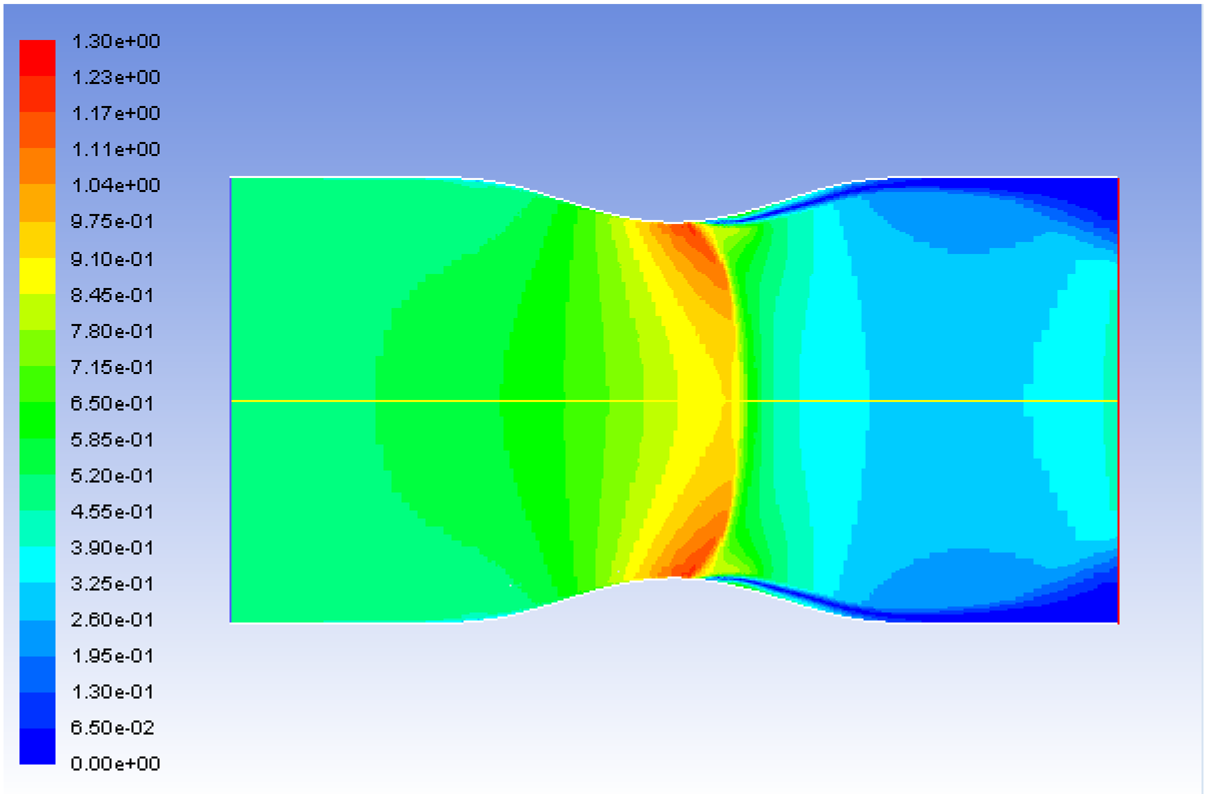
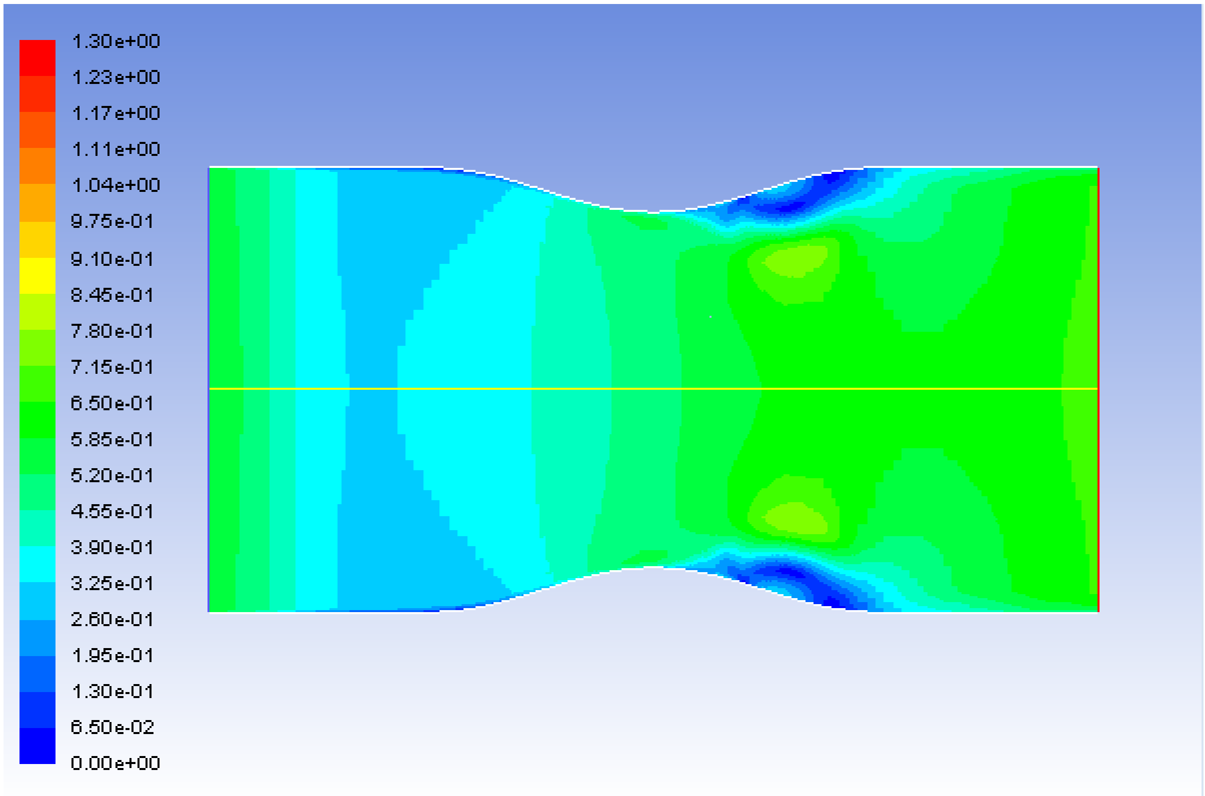


Figure 6.13: Mach Number Contours at t=0.019135 s



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**.

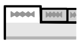
Warning

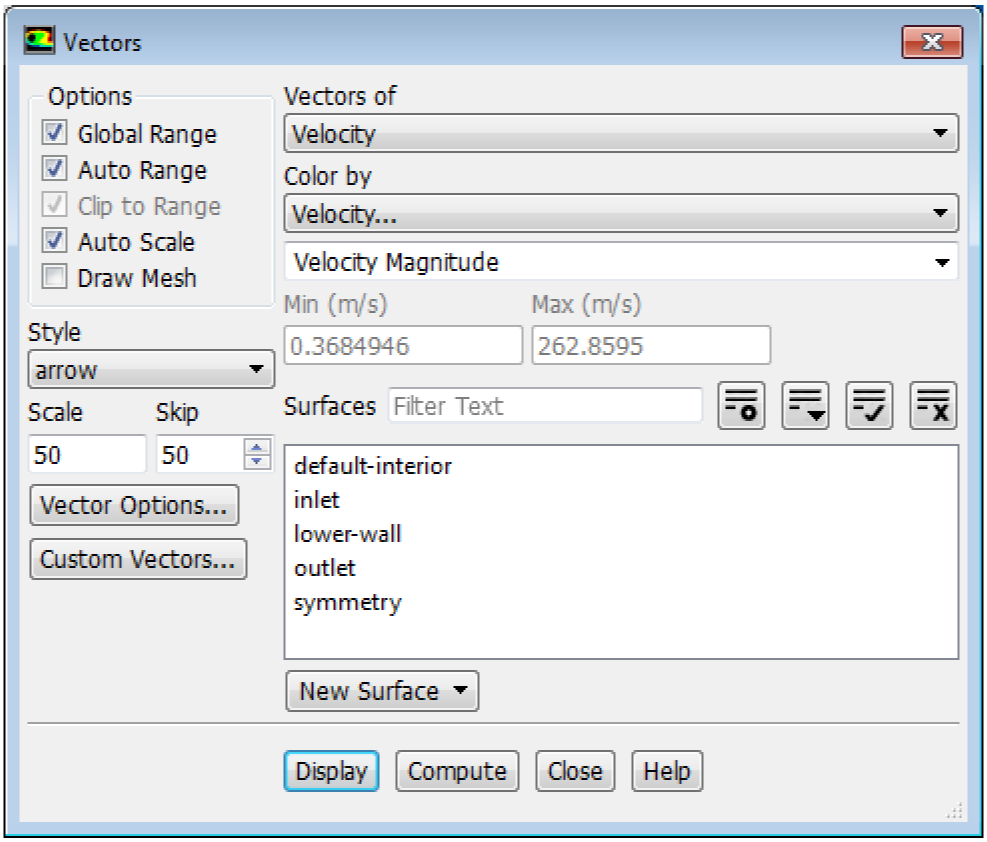
Because the solution animation was stored in memory, it will be lost if you exit ANSYS Fluent without saving it in one of the formats described previously. Note

> 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...**



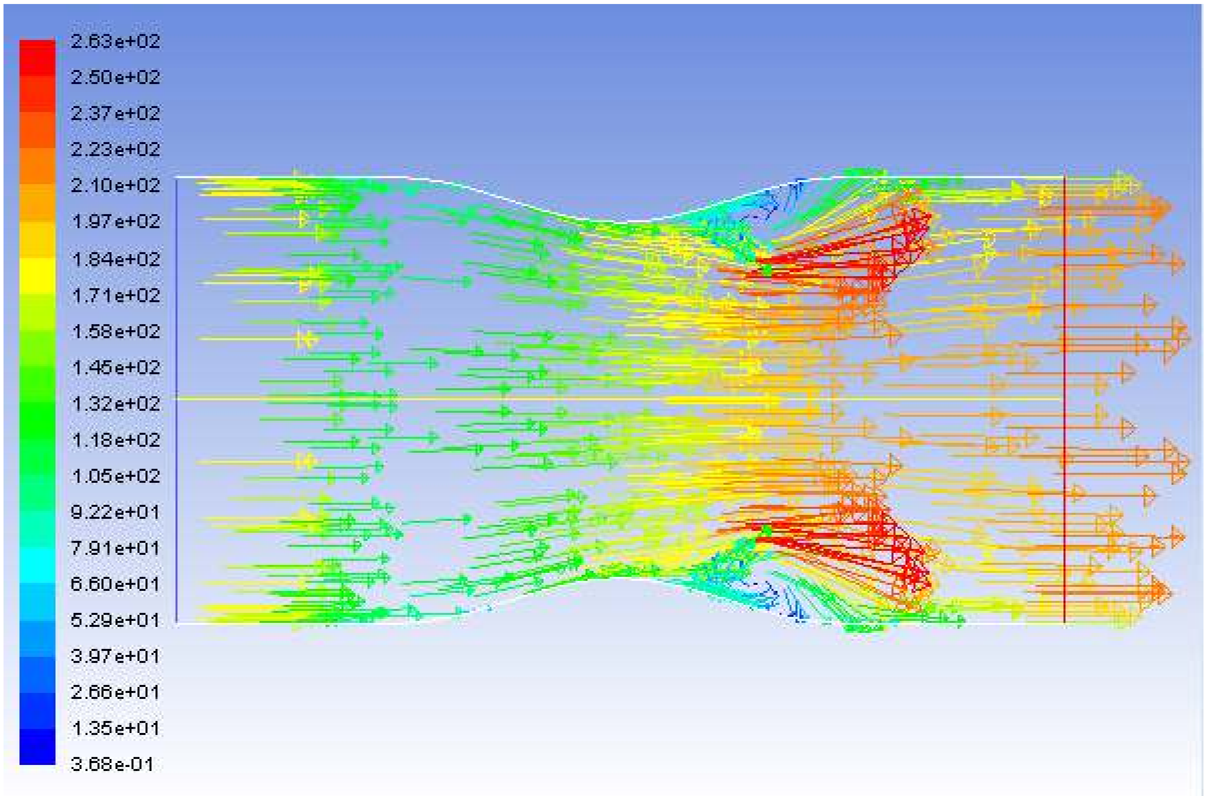
   a. Ensure **Auto Scale** is enabled under **Options**.

   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).