
Chapter 5: Modeling External Compressible Flow

This tutorial is divided into the following sections:

- [5.1. Introduction](#)
- [5.2. Prerequisites](#)
- [5.3. Problem Description](#)
- [5.4. Setup and Solution](#)
- [5.5. Summary](#)
- [5.6. Further Improvements](#)

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 monitors 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 \(p. 123\)](#)

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.

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. 222).

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
- 5.4.2. Mesh
- 5.4.3. General Settings
- 5.4.4. Models
- 5.4.5. Materials
- 5.4.6. Boundary Conditions
- 5.4.7. Operating Conditions
- 5.4.8. Solution
- 5.4.9. Postprocessing

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 **15.0** under **Version**.
5. Select this tutorial from the list.

6. Click **Files** to download the input and solution files.
7. Unzip `external_compressible_R150.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 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. 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.

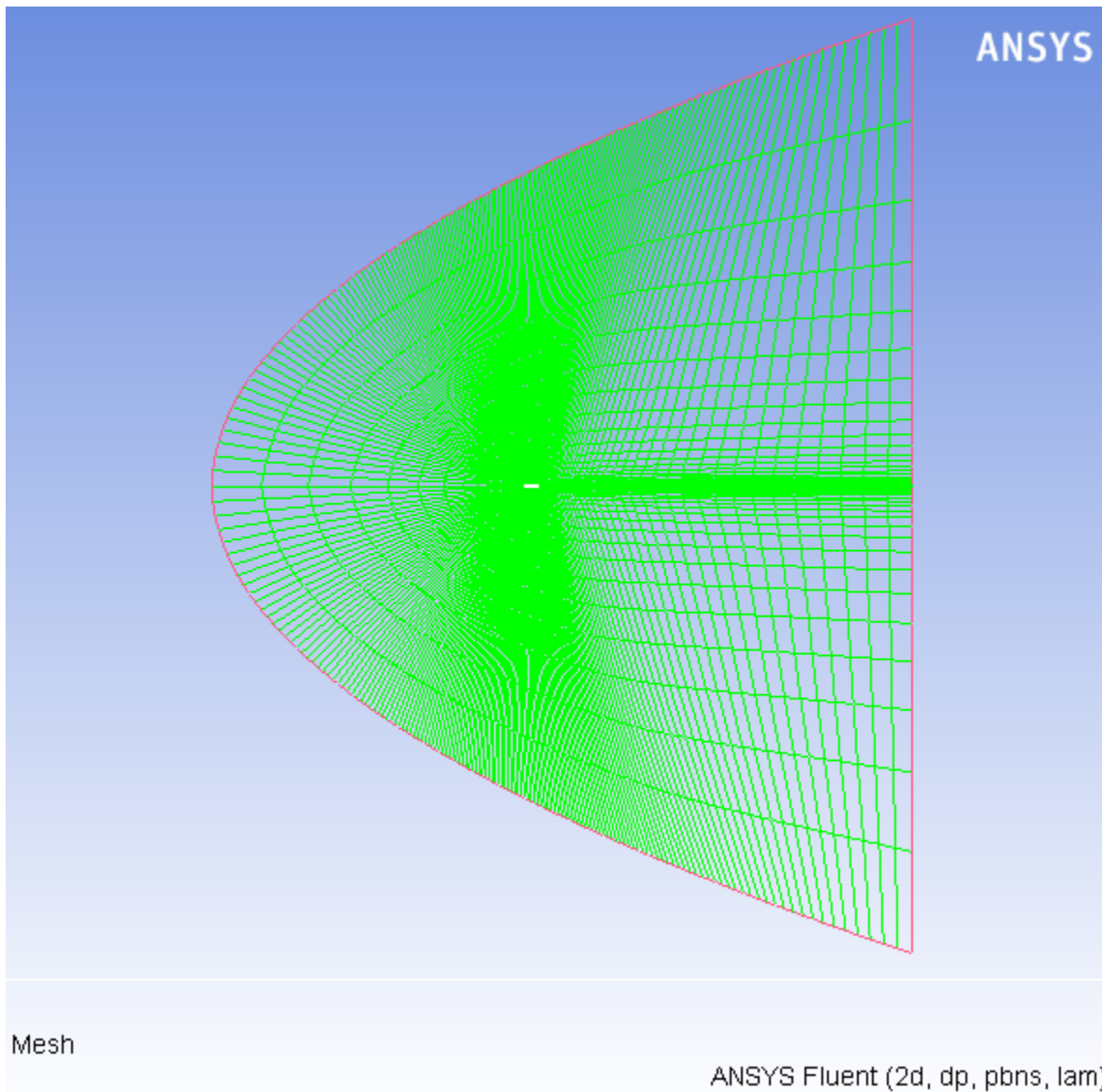
 **General** → **Check**

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.

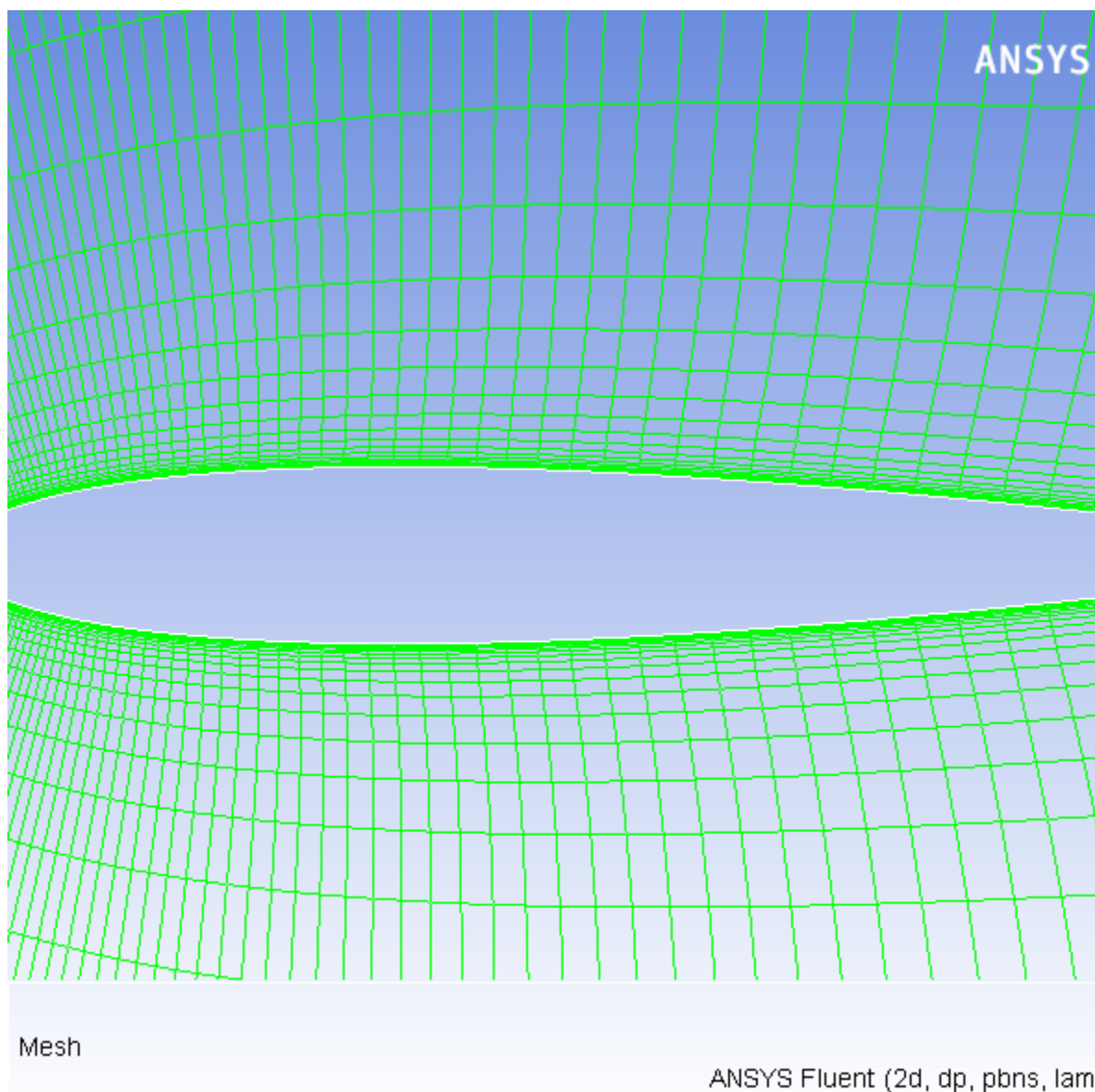
Note

ANSYS Fluent will issue a warning concerning the high aspect ratios of some cells and possible impacts on calculation of Cell Wall Distance. The warning message includes recommendations for verifying and correcting the Cell Wall Distance calculation. In this particular case the cell aspect ratio does not cause problems so no further action is required. As an optional activity, you can confirm this yourself after the solution is generated by plotting Cell Wall Distance as noted in the warning message.

3. Examine the mesh ([Figure 5.2: The Entire Mesh \(p. 224\)](#) and [Figure 5.3: Magnified View of the Mesh Around the Airfoil \(p. 225\)](#)).

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 comprised 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.

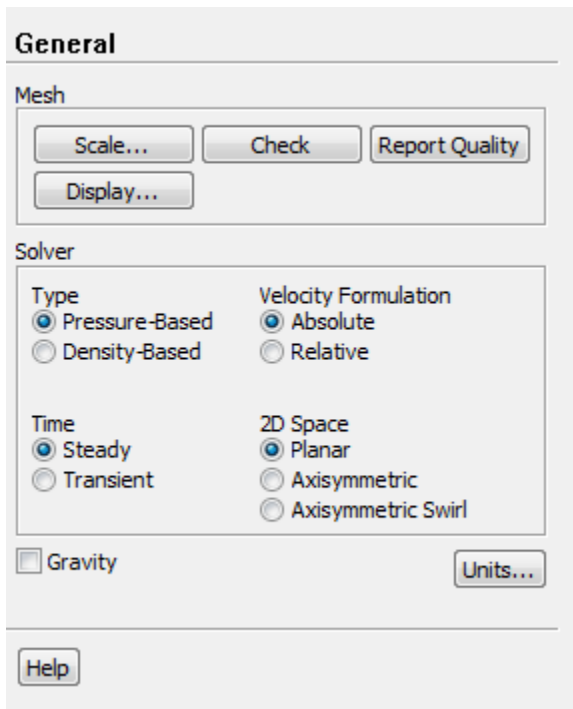
4. Reorder the mesh.

Mesh → Reorder → Domain

This is done to reduce the bandwidth of the cell neighbor number and to speed up the computations. This is especially important for large cases involving 1 million or more cells. The method used to reorder the domain is the Reverse Cuthill-McKee method.

5.4.3. General Settings

1. Set the solver settings.



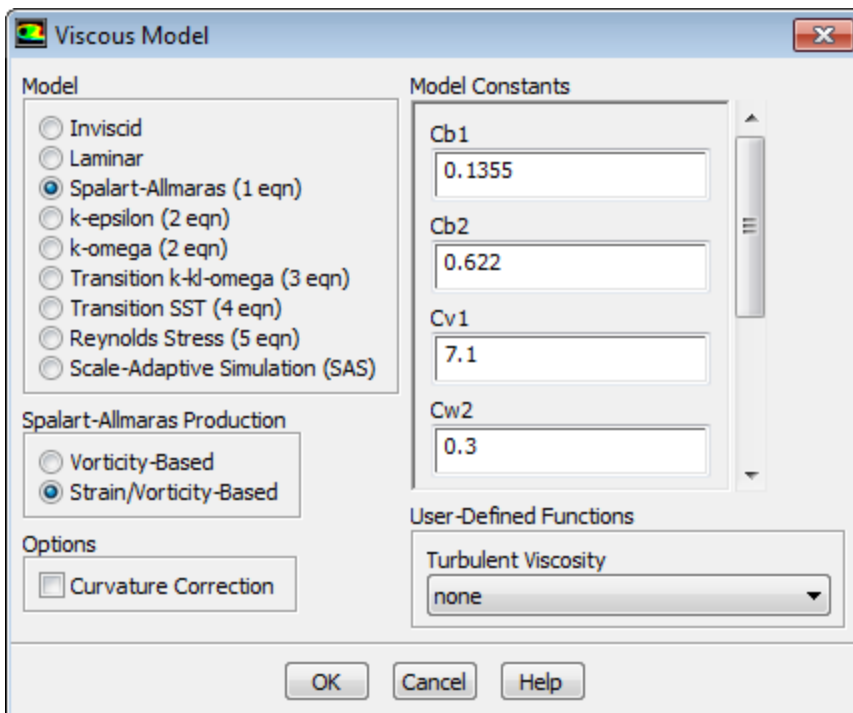
- a. Retain the default selection of **Pressure-Based** from the **Type** list.

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.

 **Models** →  **Viscous** → **Edit...**



- Select **Spalart-Allmaras (1eqn)** in the **Model** list.
- Select **Strain/Vorticity-Based** in the **Spalart-Allmaras Production** list.
- Retain the default settings in the **Model Constants** group box.
- Click **OK** to close the **Viscous Model** dialog box.

Note

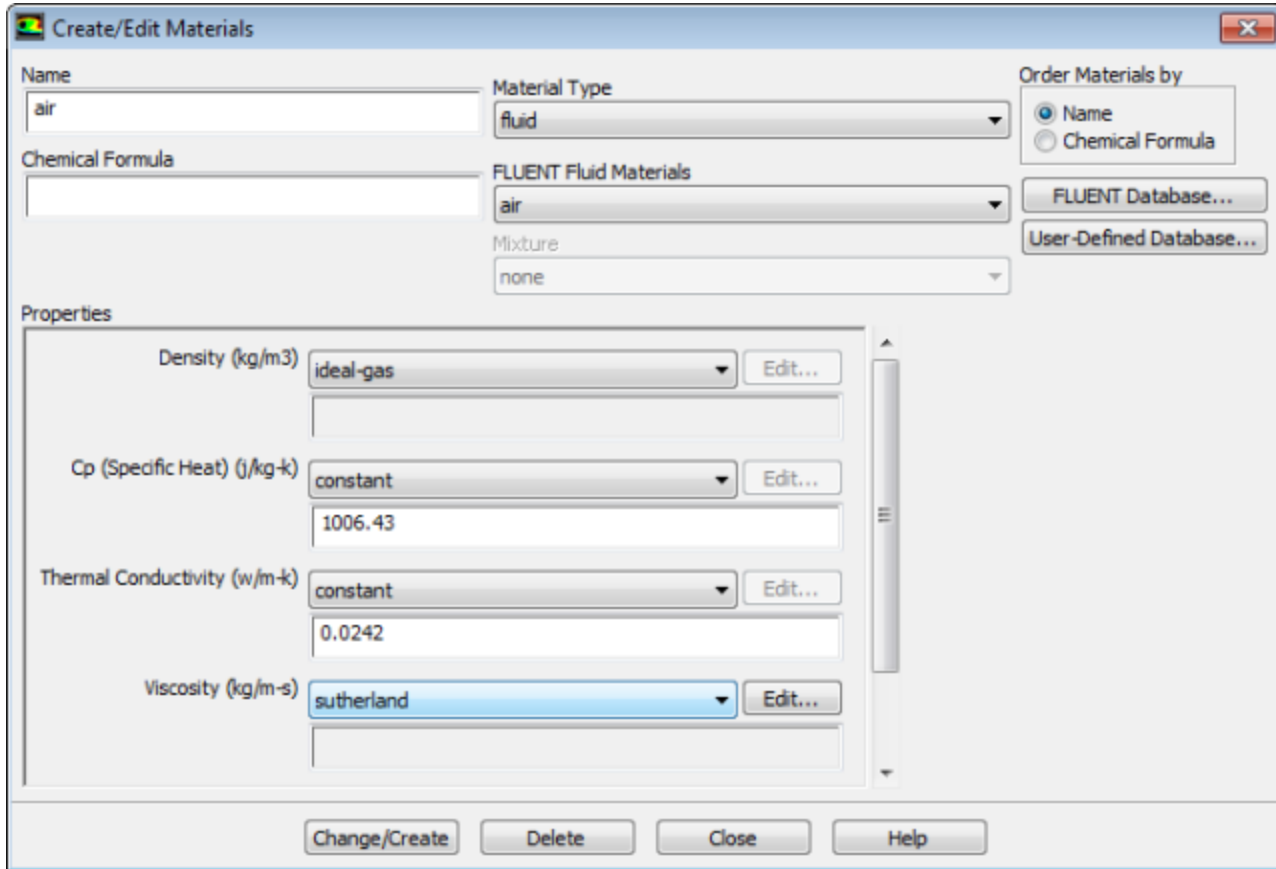
The Spalart-Allmaras model is a relatively simple one-equation model that solves a modeled transport equation for the kinematic eddy (turbulent) viscosity. This embodies a relatively new class of one-equation models in which it is not necessary to calculate a length scale related to the local shear layer 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

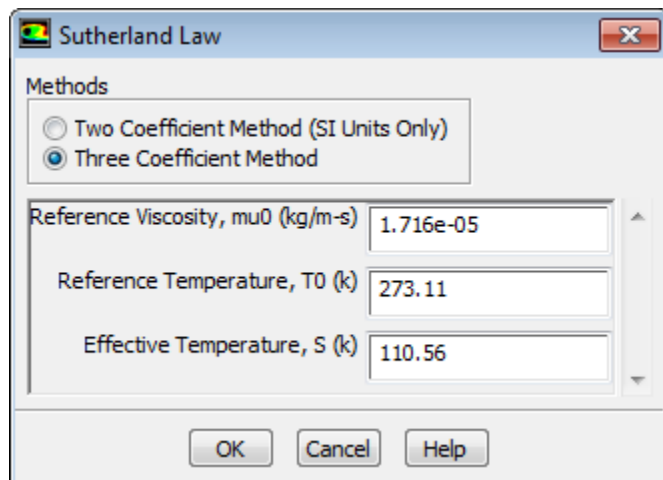
The default **Fluid Material** is **air**, which is the working fluid in this problem. The default settings need to be modified to account for compressibility and variations of the thermophysical properties with temperature.

- Set the properties for **air**, the default fluid material.

 **Materials** →  **air** → **Create/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.

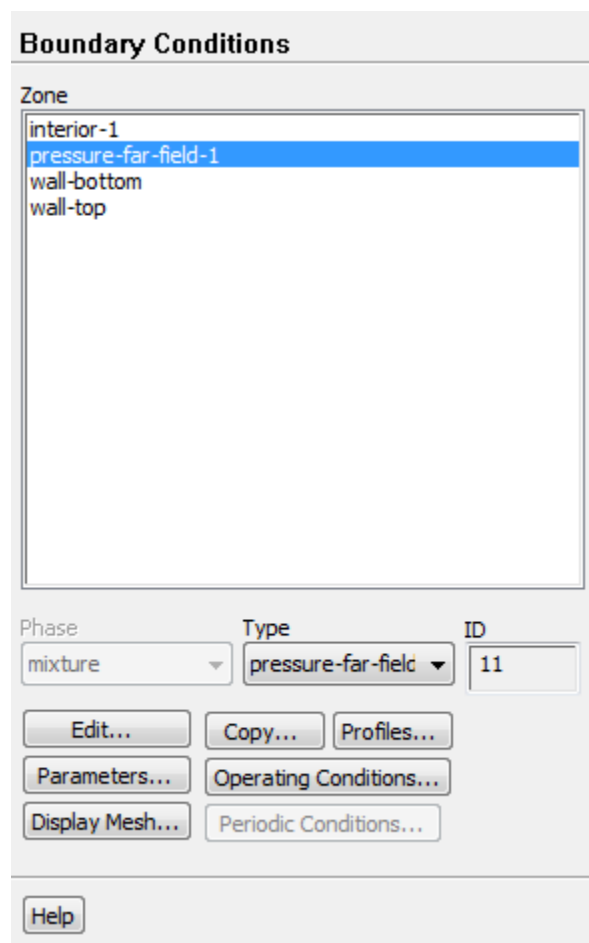
The Sutherland law for viscosity is well suited for high-speed compressible flows.

- 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.

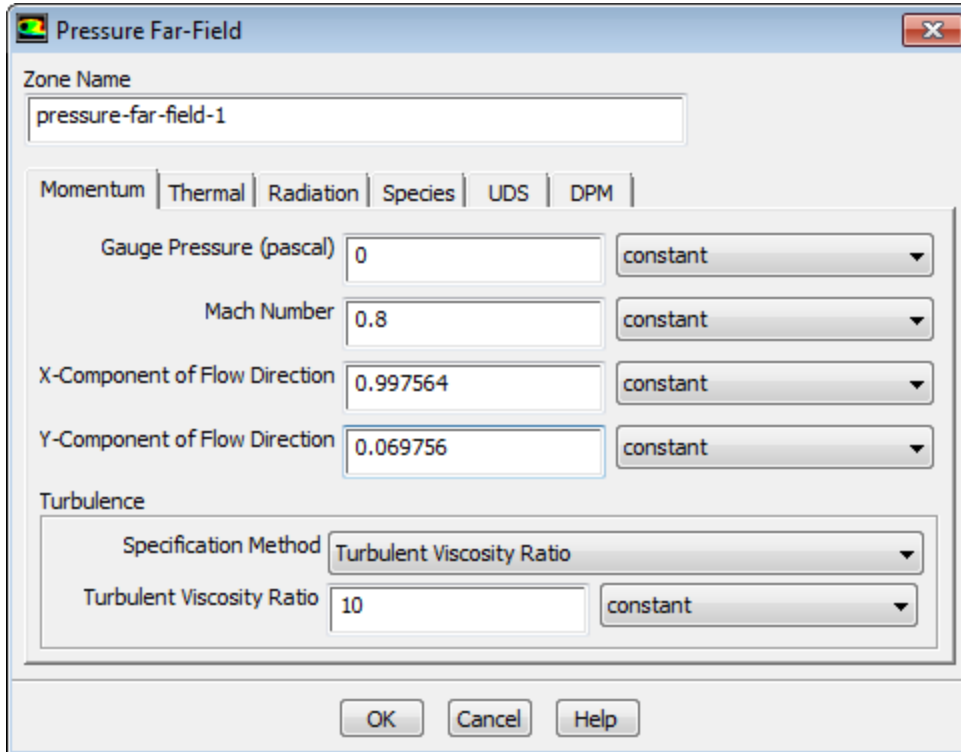
5.4.6. Boundary Conditions

Boundary Conditions



1. Set the boundary conditions for **pressure-far-field-1**.

 **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).

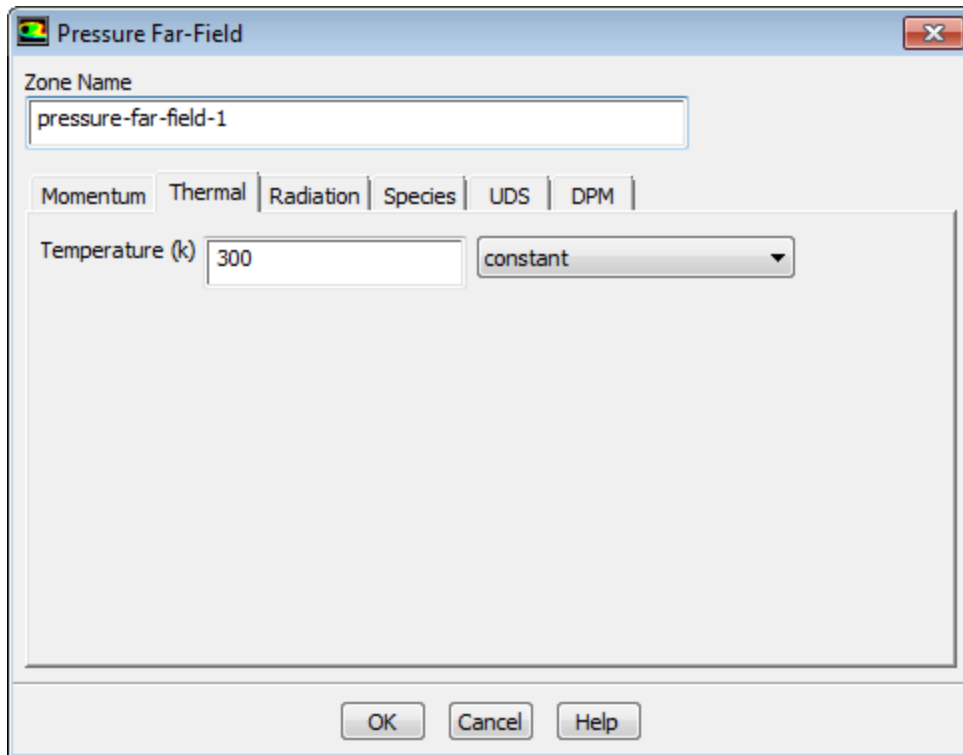
- b. Enter 0.8 for **Mach Number**.
- c. Enter 0.997564 and 0.069756 for the **X-Component of Flow Direction** and **Y-Component of Flow Direction**, respectively.

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**.

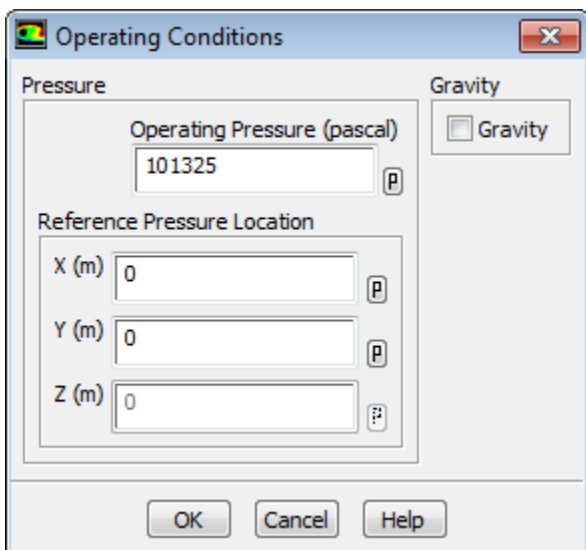


- g. Click **OK** to close the **Pressure Far-Field** dialog box.

5.4.7. Operating Conditions

1. Set the operating pressure.

◆ **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.

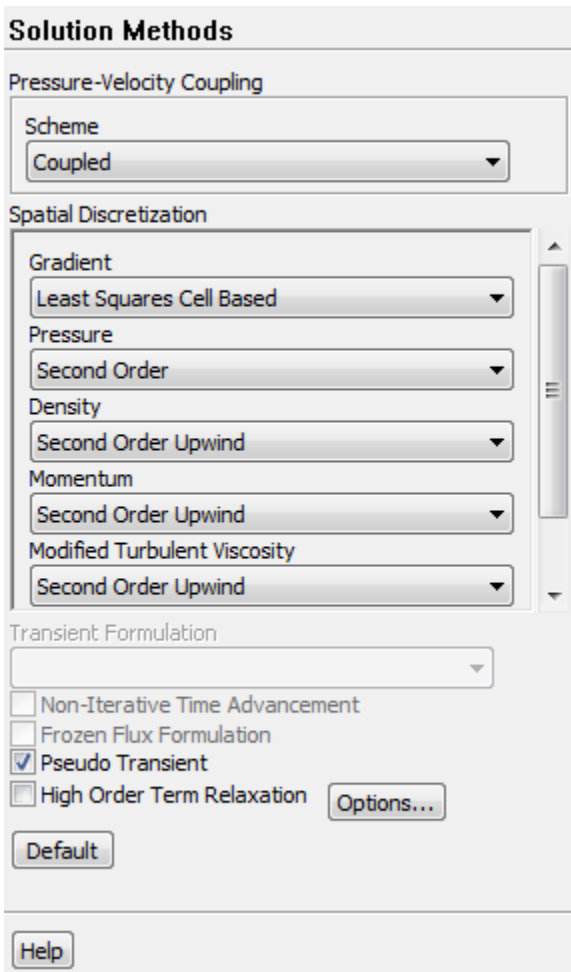
- b. Click **OK** to close the **Operating Conditions** dialog box.

For information about setting the operating pressure, see [Operating Pressure](#) in the [User's Guide](#).

5.4.8. Solution

1. Set the solution parameters.

Solution Methods



Solution Methods

Pressure-Velocity Coupling

Scheme
Coupled

Spatial Discretization

Gradient
Least Squares Cell Based

Pressure
Second Order

Density
Second Order Upwind

Momentum
Second Order Upwind

Modified Turbulent Viscosity
Second Order Upwind

Transient Formulation

Non-Iterative Time Advancement

Frozen Flux Formulation

Pseudo Transient

High Order Term Relaxation [Options...](#)

[Default](#)

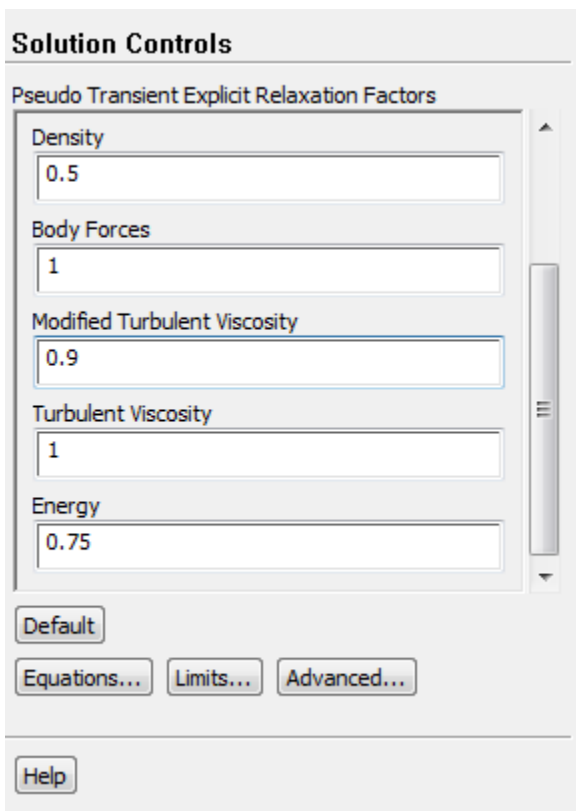
[Help](#)

- 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.

- Set the solution controls.

Solution Controls



Solution Controls

Pseudo Transient Explicit Relaxation Factors

Density
0.5

Body Forces
1

Modified Turbulent Viscosity
0.9

Turbulent Viscosity
1

Energy
0.75

Default

Equations... Limits... Advanced...

Help

- Enter 0.5 for **Density** in the **Pseudo Transient Explicit Relaxation Factors** group box.

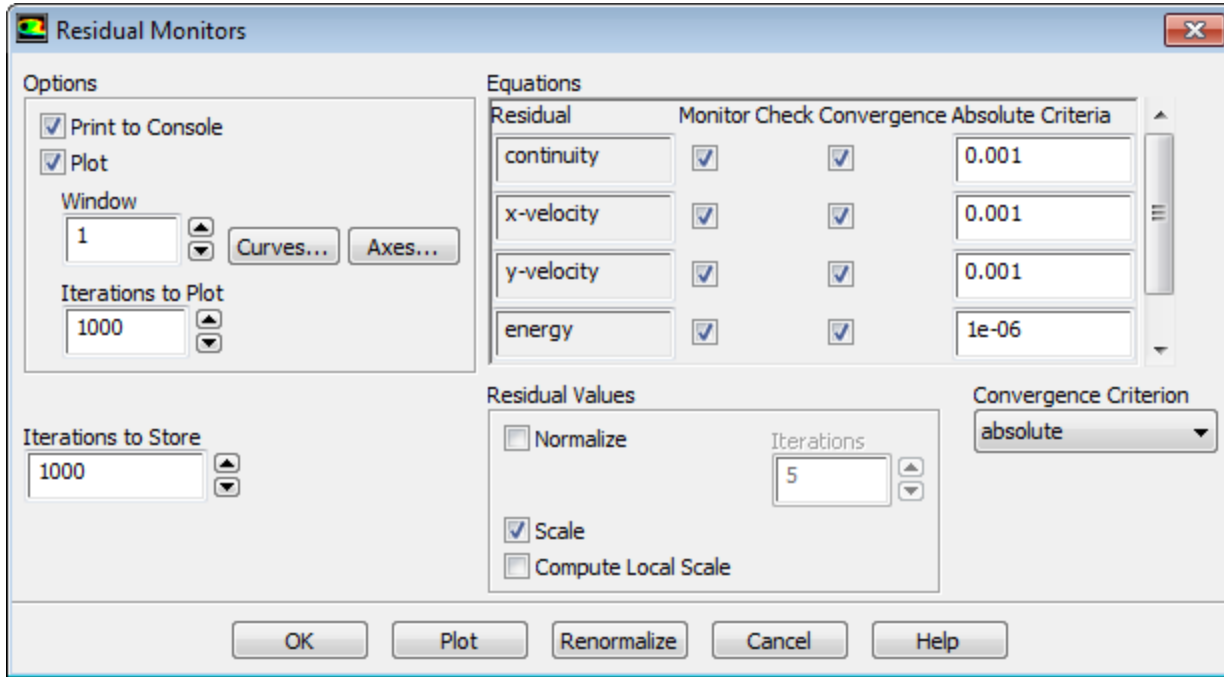
Under-relaxing the density factor is recommended for high-speed compressible flows.

- 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.

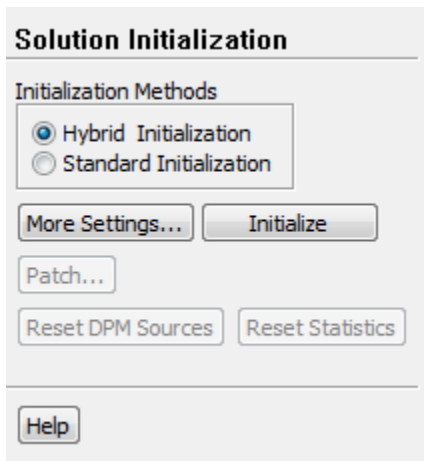
- Enable residual plotting during the calculation.

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



- 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 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]

Number of FMG (and FAS geometric multigrid) levels: 5
* FMG customization summary:
* residual reduction on level 0 [finest grid] is: 0.001
* number of cycles on level 0 is: 1
* residual reduction on level 1 is: 0.001
* number of cycles on level 1 is: 100
* residual reduction on level 2 is: 0.001
* number of cycles on level 2 is: 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 [Full Multigrid \(FMG\) Initialization](#) in the 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.

- Start the calculation by requesting 50 iterations.

Run Calculation

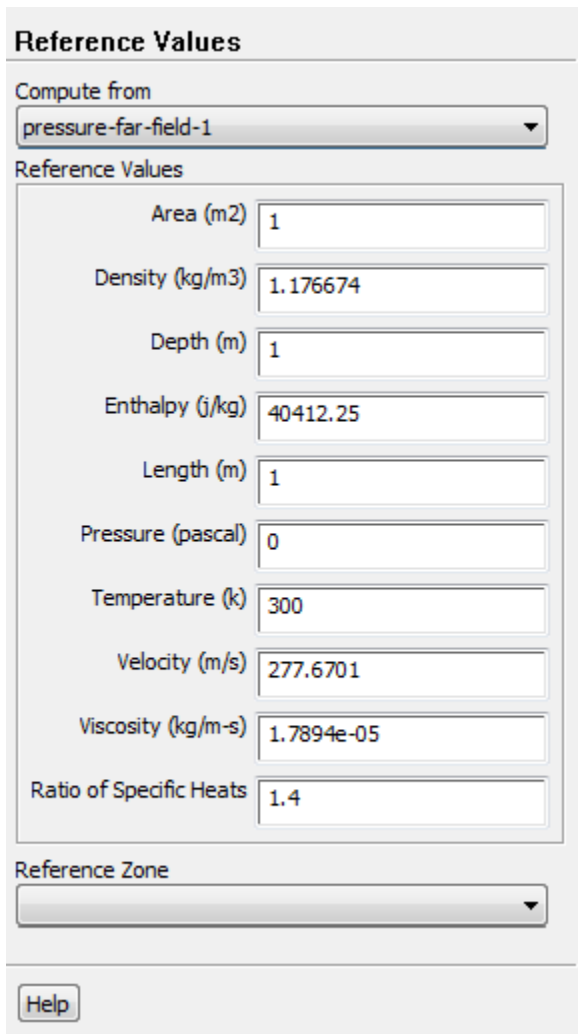
- Enter 50 for **Number of Iterations**.
- Click **Calculate**.

By performing some iterations before setting up the force monitors, you will avoid large initial transients in the monitor plots. This will reduce the axes range and make it easier to judge the convergence.

- Set the reference values that are used to compute the lift, drag, and moment coefficients.

Reference Values

The reference values are used to nondimensionalize the forces and moments acting on the airfoil. The dimensionless forces and moments are the lift, drag, and moment coefficients.



Reference Values	
Area (m ²)	1
Density (kg/m ³)	1.176674
Depth (m)	1
Enthalpy (j/kg)	40412.25
Length (m)	1
Pressure (pascal)	0
Temperature (k)	300
Velocity (m/s)	277.6701
Viscosity (kg/m-s)	1.7894e-05
Ratio of Specific Heats	1.4

Reference Zone

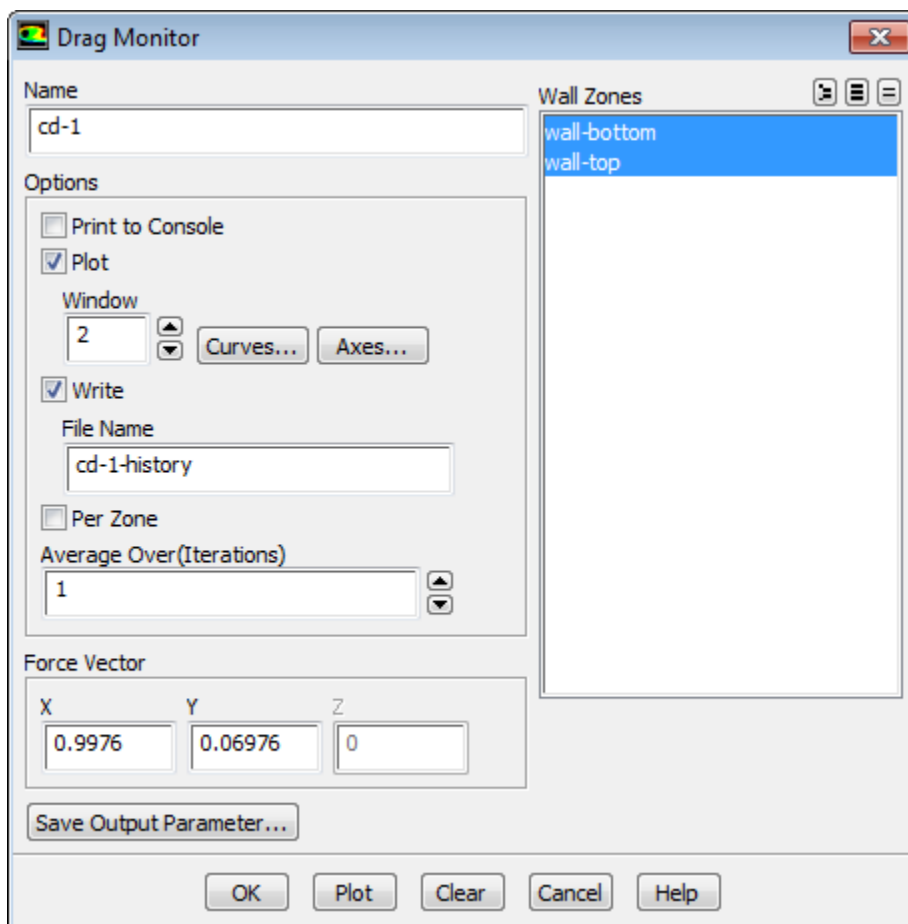
Help

- 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. Define a force monitor to plot and write the drag coefficient for the walls of the airfoil.

📌 **Monitors (Residuals, Statistic and Force Monitors)** → **Create** → 📄 **Drag...**



- Enable **Plot** in the **Options** group box.
- Enable **Write** to save the monitor history to a file.

Note

If you do not enable the **Write** option, the history information will be lost when you exit ANSYS Fluent.

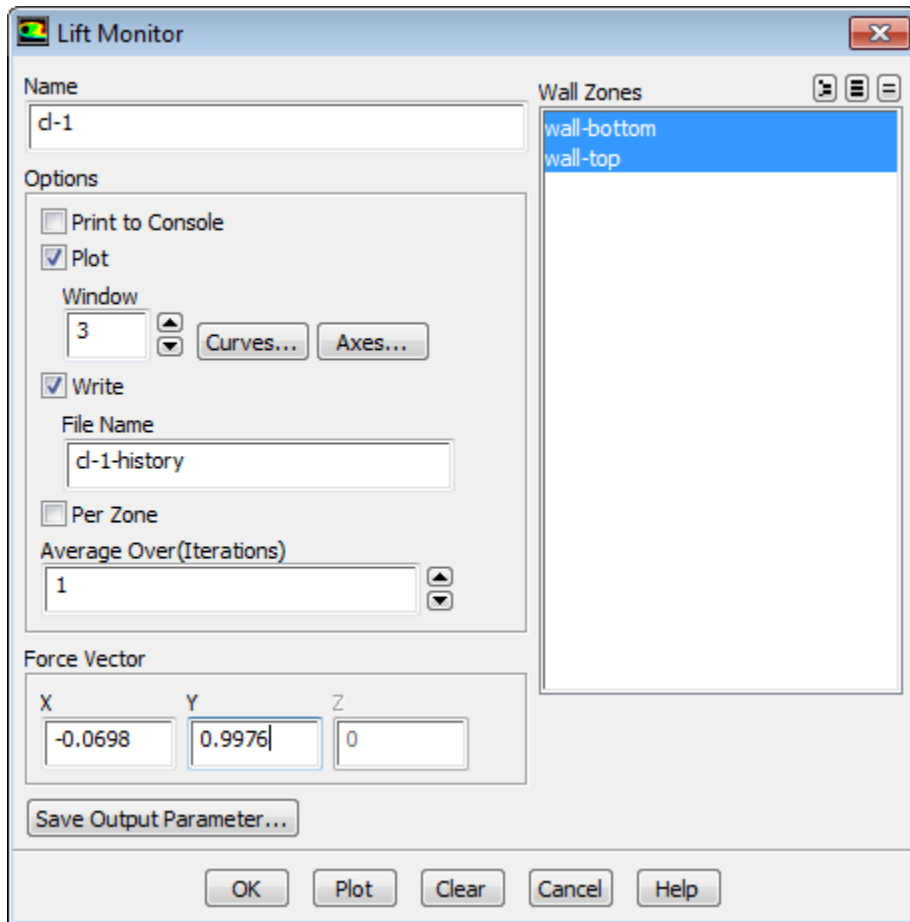
- Retain the default entry of `cd-1-history` for **File Name**.
- Select **wall-bottom** and **wall-top** in the **Wall Zones** selection list.
- Enter 0.9976 for **X** and 0.06976 for **Y** in the **Force Vector** group box.

*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.*

- Click **OK** to close the **Drag Monitor** dialog box.

9. Similarly, define a force monitor for the lift coefficient.

📏 **Monitors** → **Create** → 📄 **Lift...**

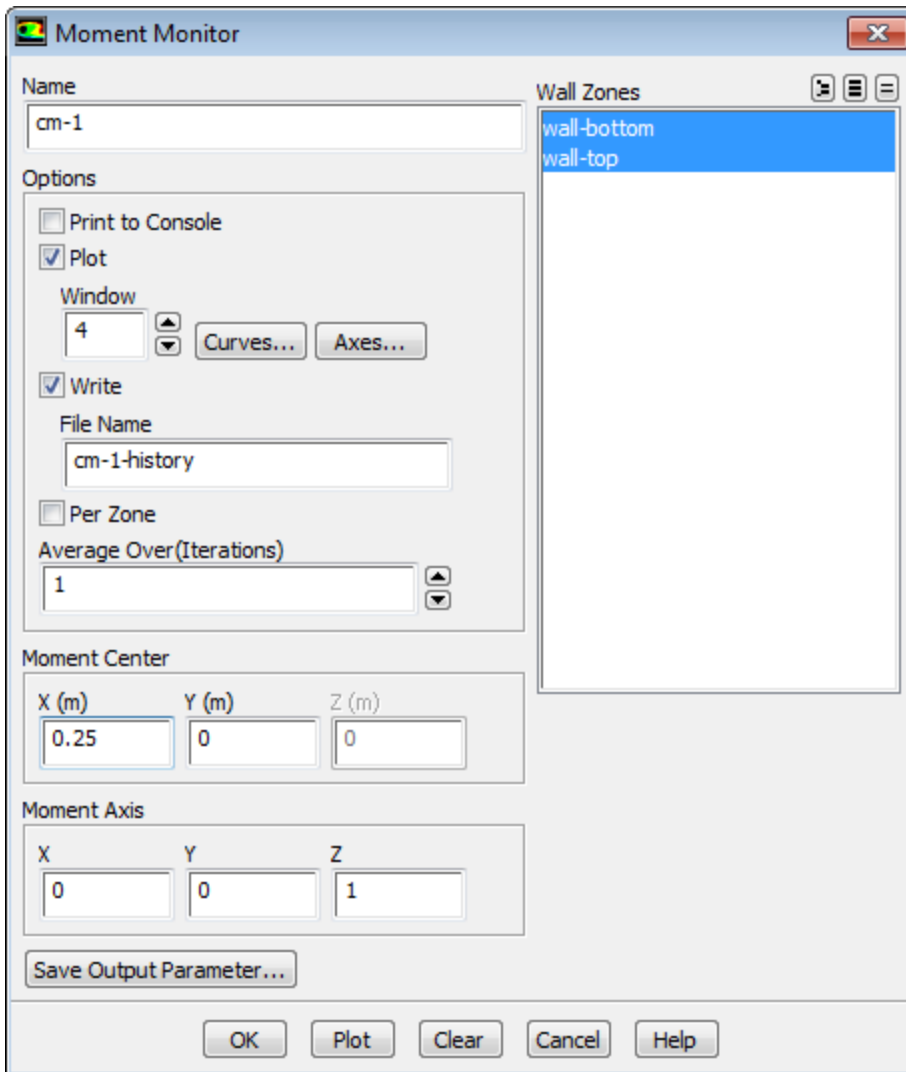


Enter the values for **X** and **Y** shown in the **Lift Monitor** dialog box.

*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, define a force monitor for the moment coefficient.

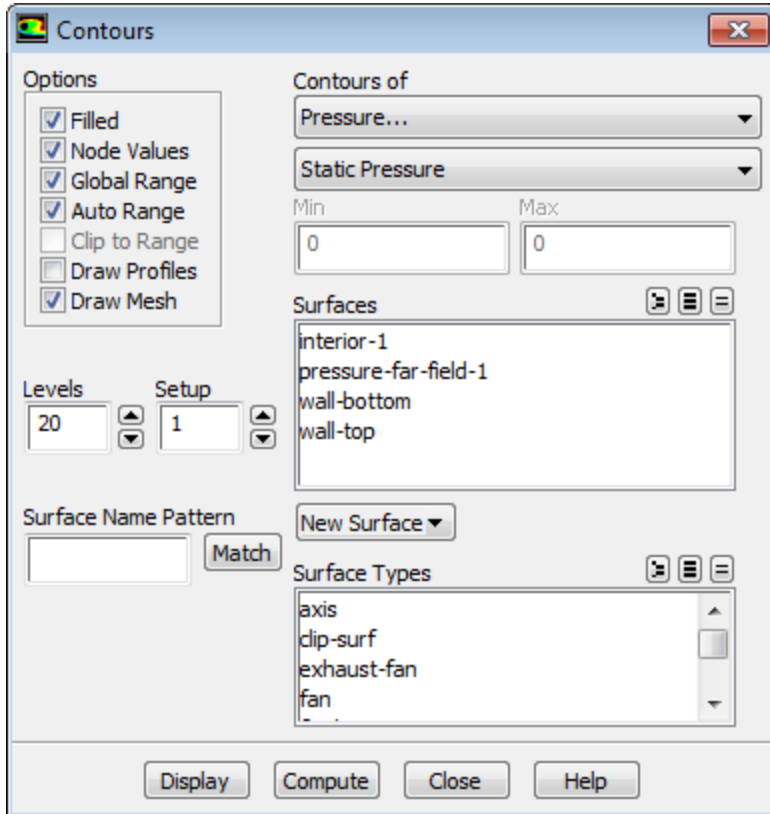
📏 **Monitors** → **Create** → 📄 **Moment...**



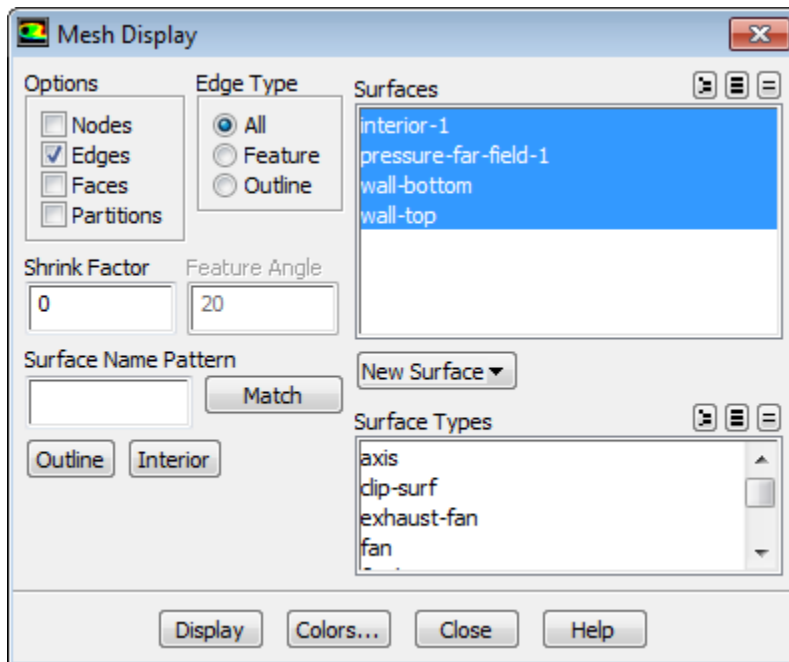
Enter the values for the **Moment Center** and **Moment Axis** shown in the **Moment Monitor** dialog box.

11. Display filled contours of pressure overlaid with the mesh in preparation for defining a surface monitor (Figure 5.4: Pressure Contours After 50 Iterations (p. 241) and Figure 5.5: Magnified View of Pressure Contours Showing Wall-Adjacent Cells (p. 242)).

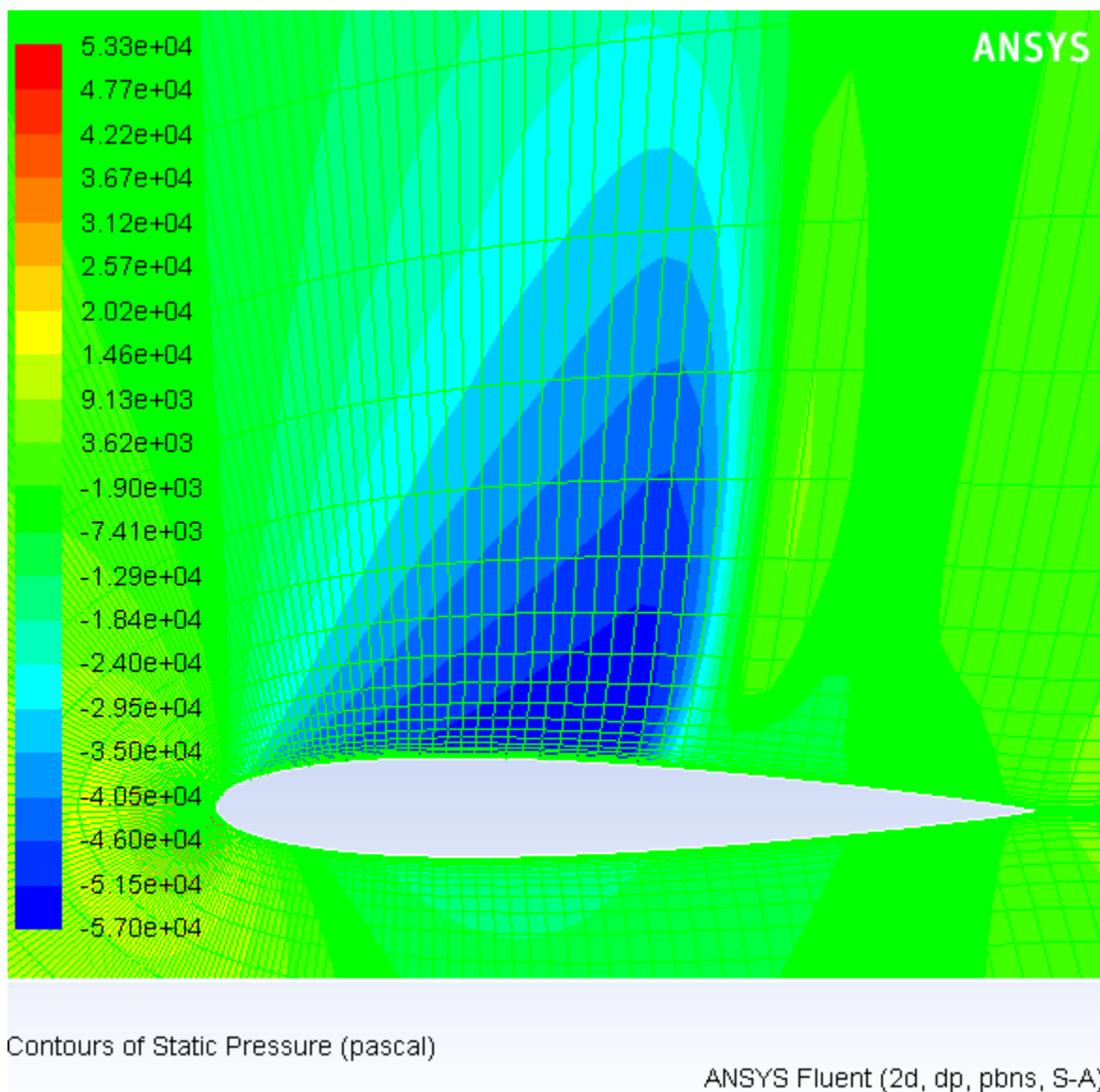
◆ **Graphics and Animations** →  **Contours** → **Set Up...**



- 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.

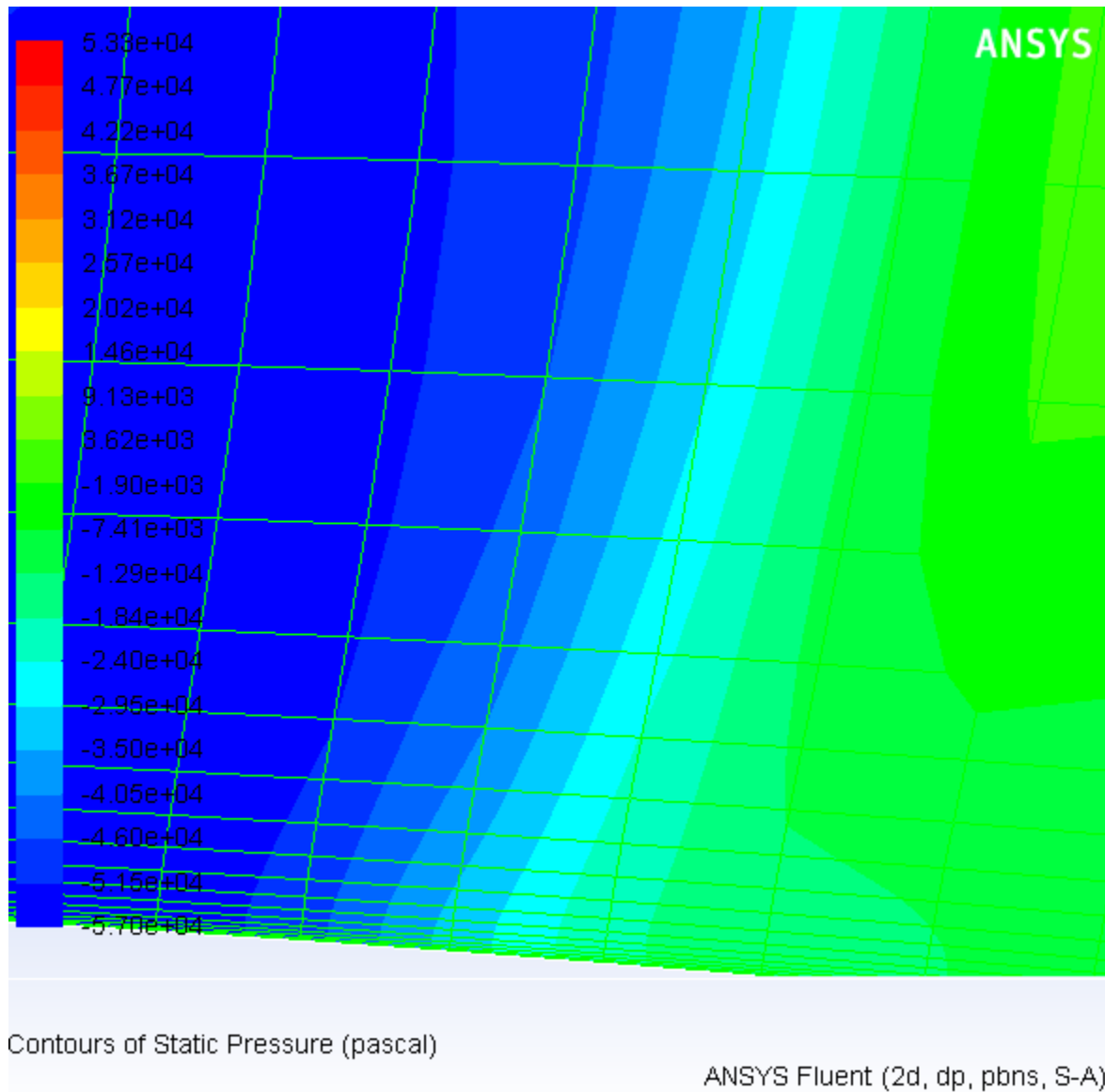
Figure 5.4: Pressure Contours After 50 Iterations

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**.

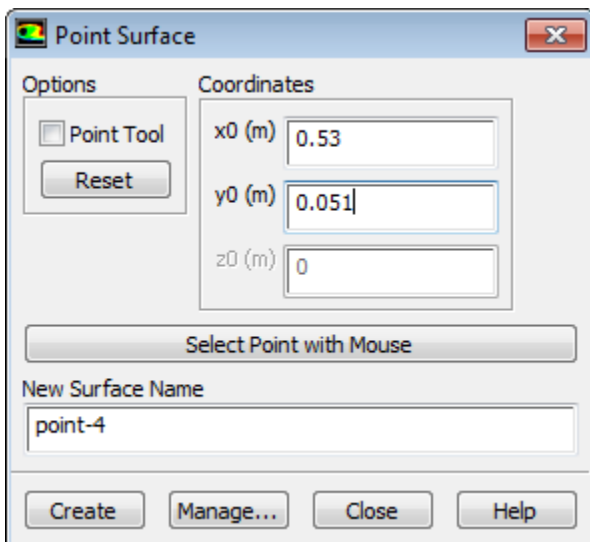
- d. Zoom in on the shock wave, until individual cells adjacent to the upper surface (**wall-top** boundary) are visible, as shown in Figure 5.5: Magnified View of Pressure Contours Showing Wall-Adjacent Cells (p. 242).

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 define a surface monitor.

12. Create a point surface just downstream of the shock wave.

Surface → Point...



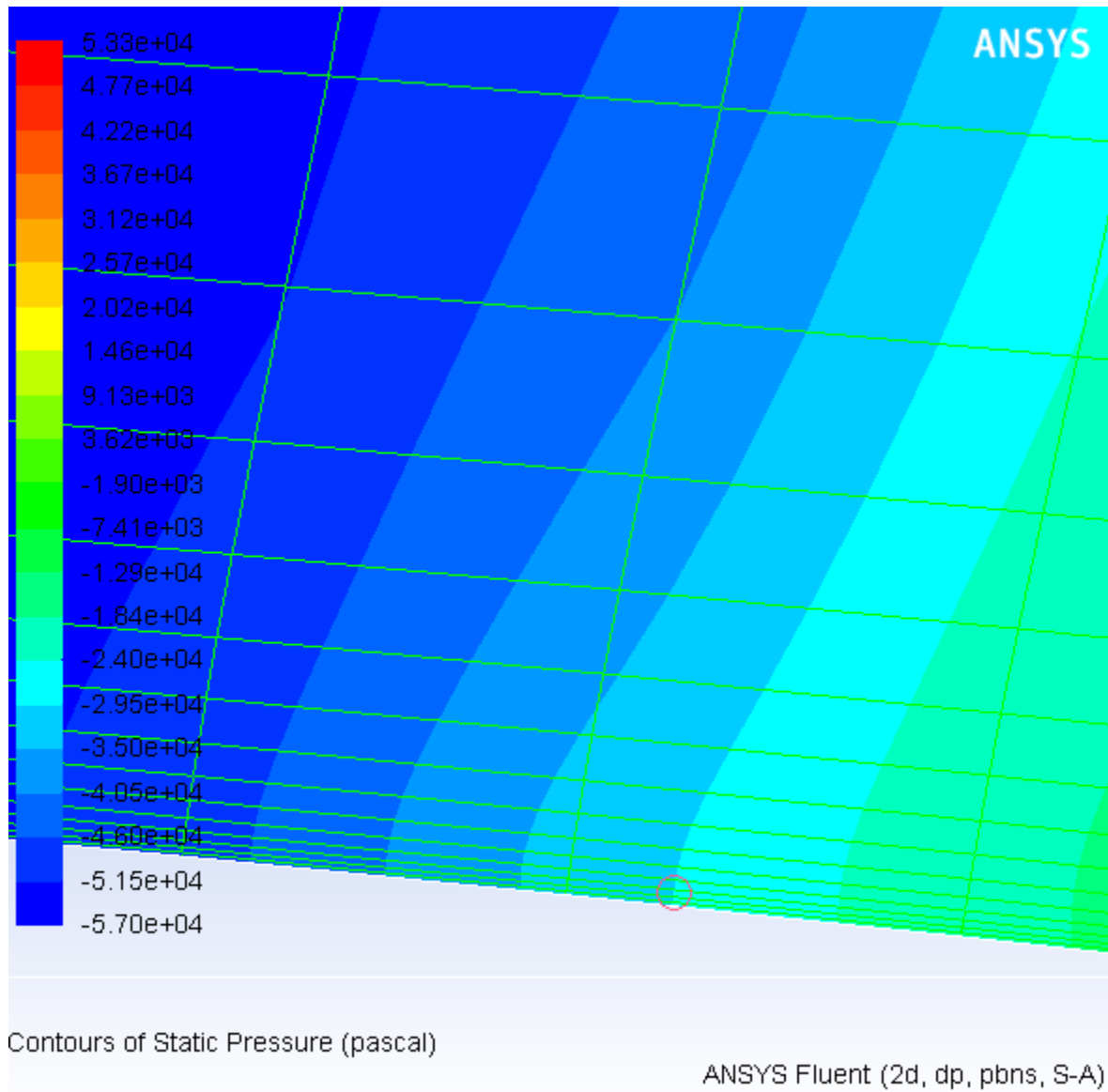
- Enter 0.53 m for **x0** and 0.051 m for **y0** in the **Coordinates** group box.
- Retain the default entry of `point-4` for **New Surface Name**.
- Click **Create** and close the **Point Surface** dialog box.

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:

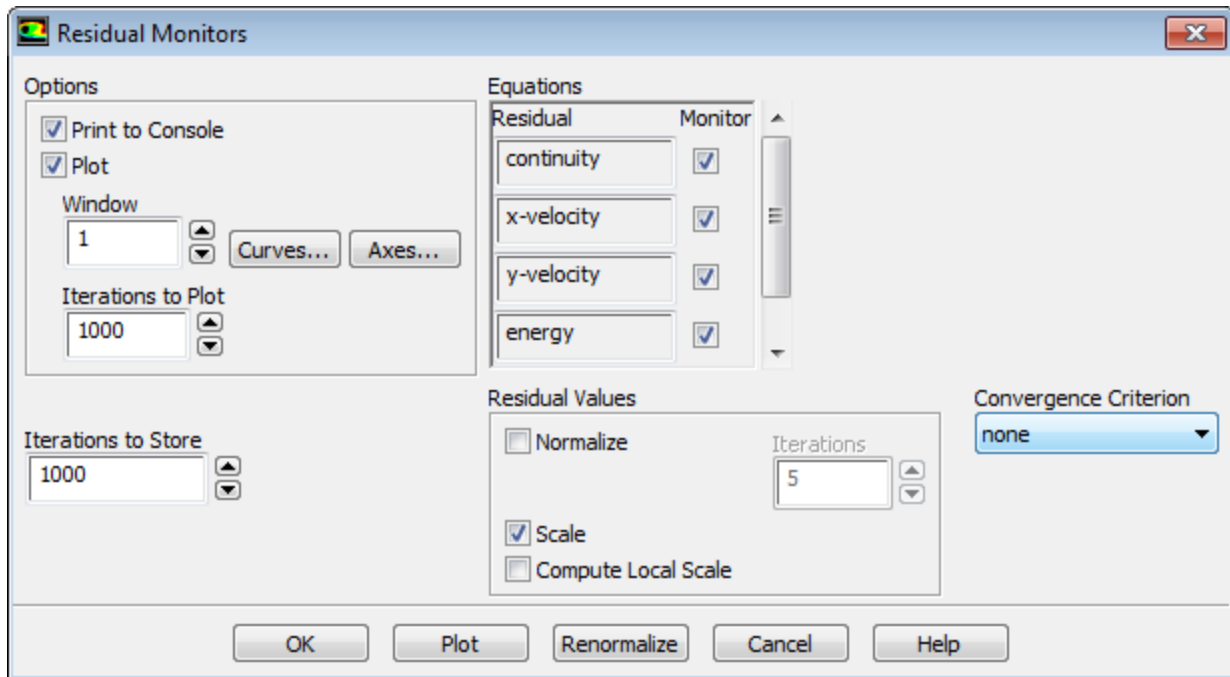
- In the **Point Surface** dialog box, click the **Select Point with Mouse** button. A **Working** dialog box will open telling you to "Click on a location in the graphics window with the MOUSE-PROBE mouse button."
 - Position the mouse pointer at a point located inside one of the cells adjacent to the upper surface (**wall-top** boundary), downstream of the shock (see [Figure 5.6: Pressure Contours after Creating a Point with the Mouse \(p. 244\)](#)).
 - Click the right mouse button.
 - 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.

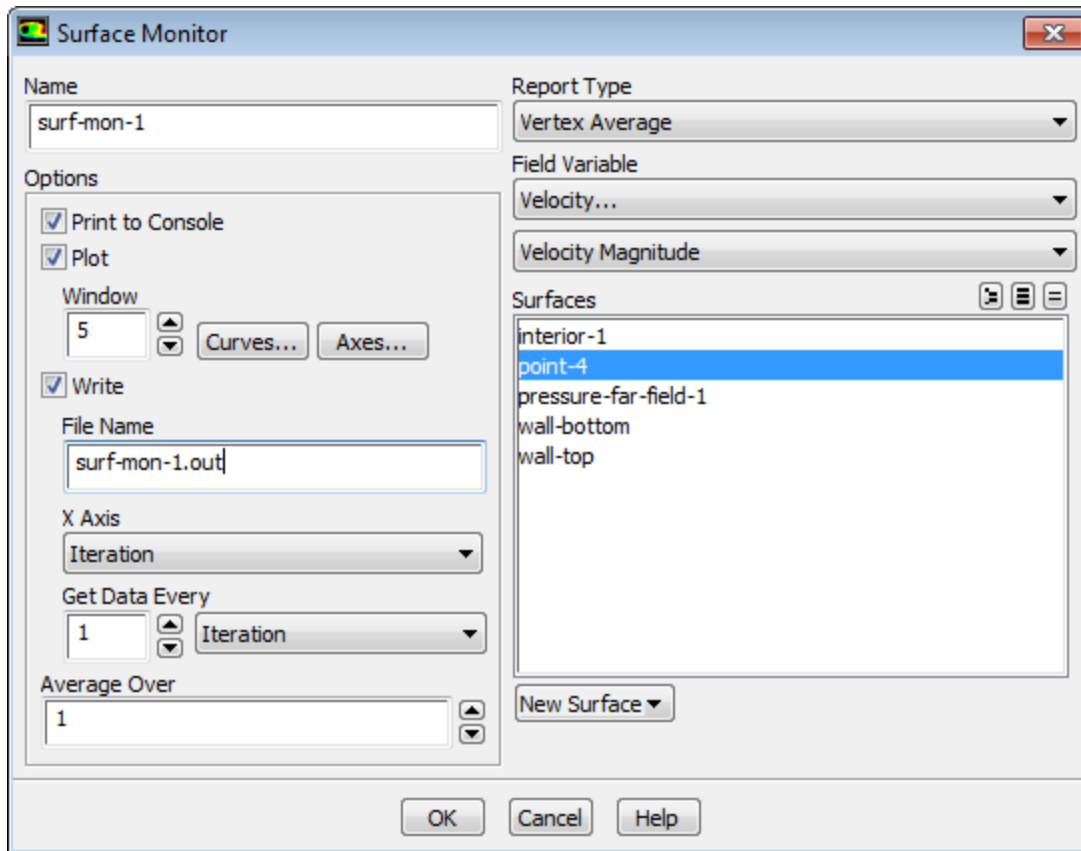
 **Monitors** →  **Residuals** → **Edit...**



- 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. Define a surface monitor 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, define a monitor at a point (just downstream of the shock) where there is likely to be significant variation, and monitor the value of the velocity magnitude.

◆ **Monitors (Surface Monitors) → Create...**



- a. Enable **Plot** and **Write**.
- b. Select **Vertex Average** from the **Report Type** drop-down list.
*Scroll down the **Report Type** drop-down list to find **Vertex Average**.*
- c. Select **Velocity...** and **Velocity Magnitude** from the **Field Variable** drop-down list.
- d. Select **point-4** in the **Surfaces** selection list.
- e. Click **OK** to close the **Surface Monitor** 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.

Run Calculation

The force monitors (*Figure 5.8: Drag Coefficient Convergence History (p. 247)* and *Figure 5.9: Lift Coefficient Convergence History (p. 248)*) show that the case is converged after approximately 200 iterations.

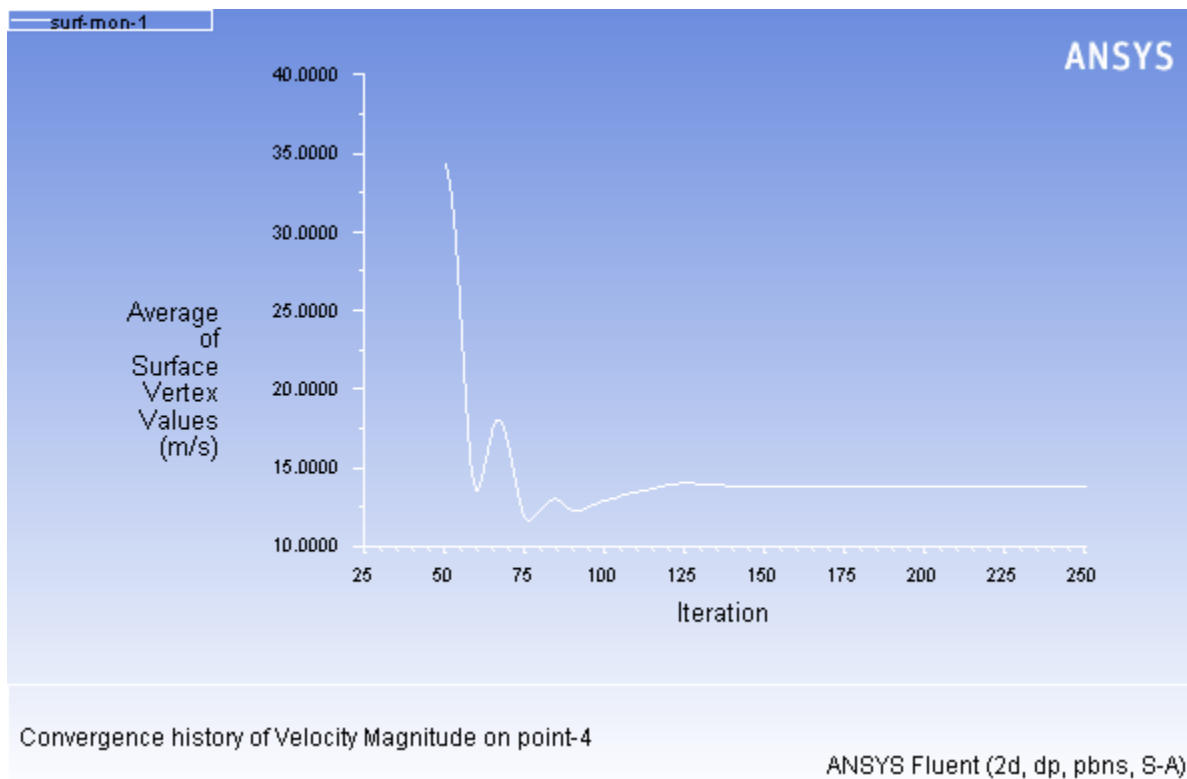
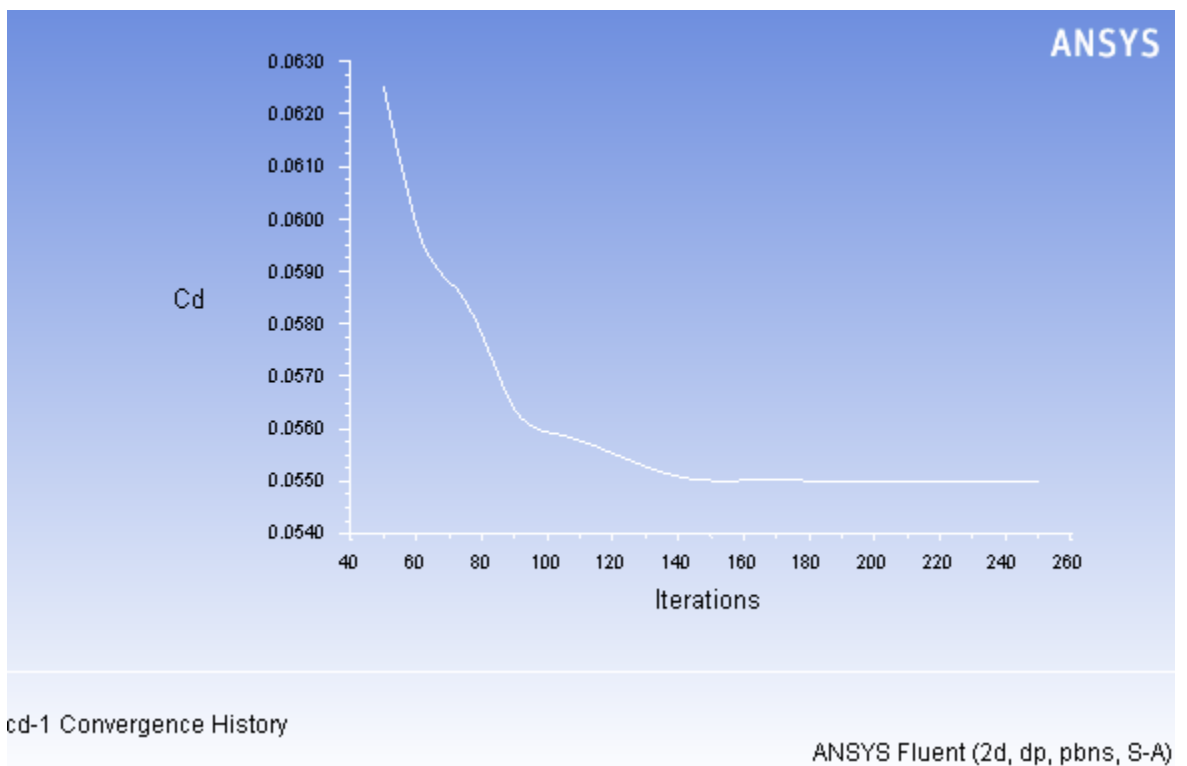
Figure 5.7: Velocity Magnitude History**Figure 5.8: Drag Coefficient Convergence History**

Figure 5.9: Lift Coefficient Convergence History

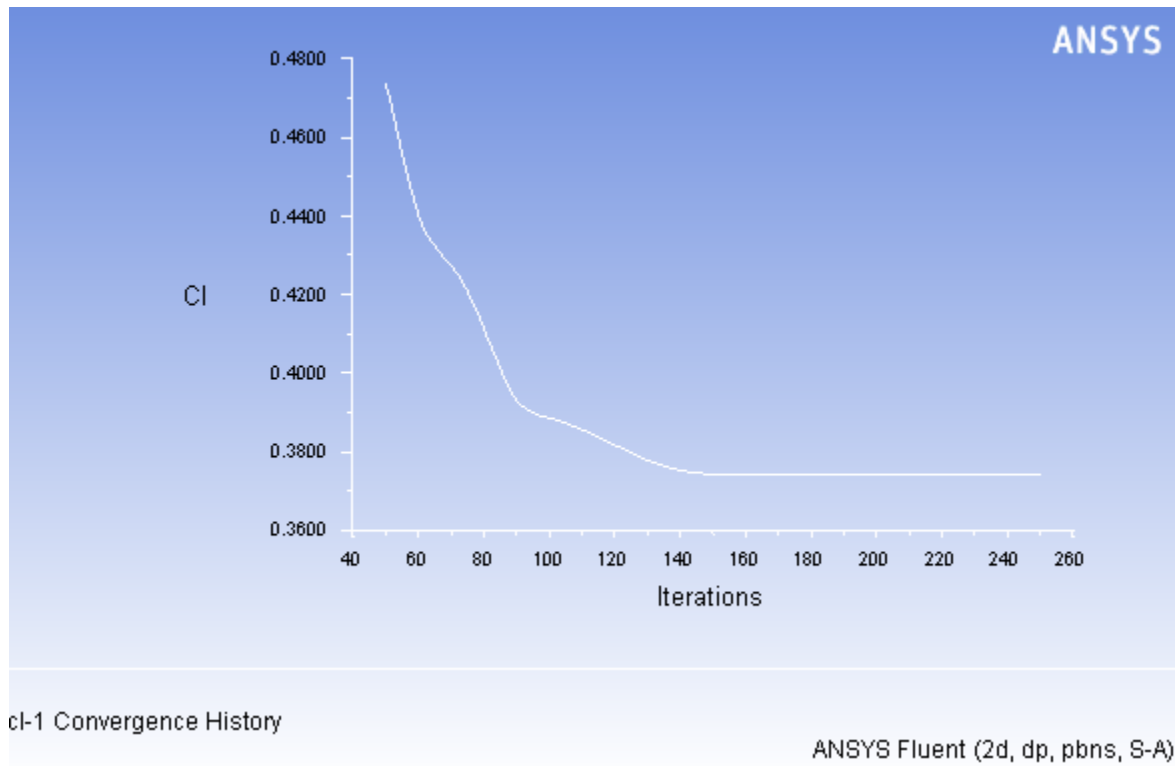
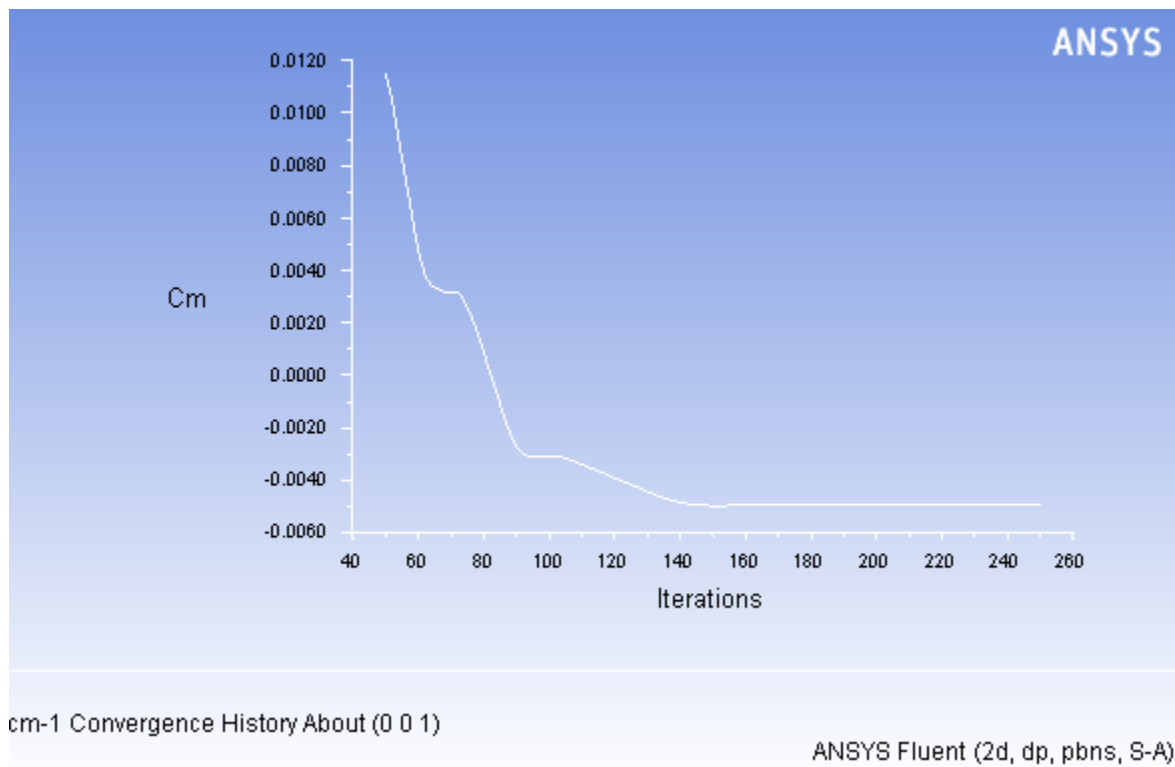


Figure 5.10: Moment Coefficient Convergence History



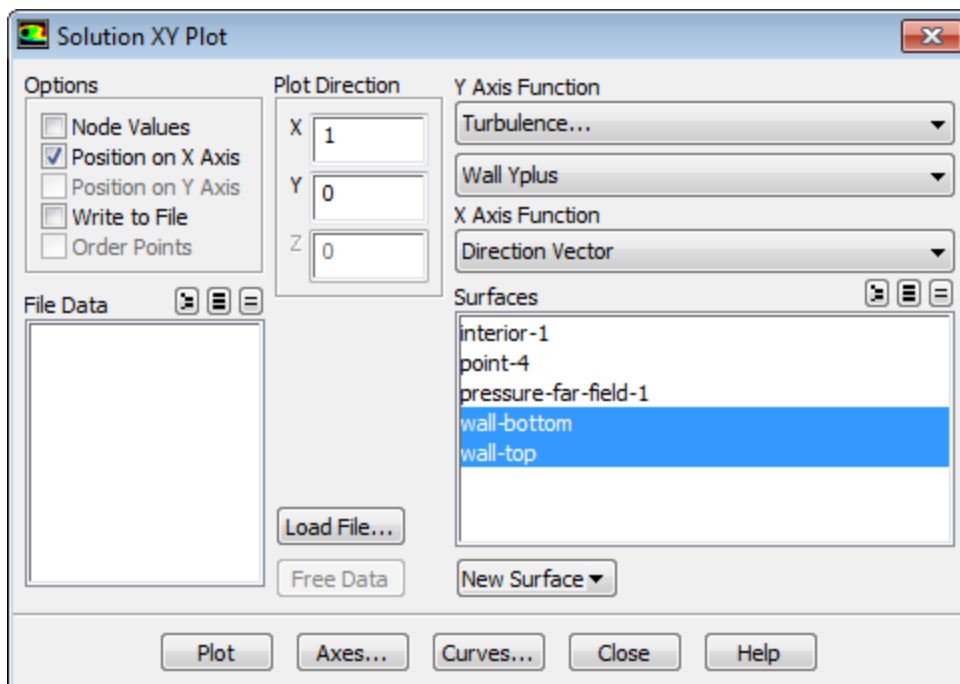
17. Save the case and data files (airfoil-2.cas.gz and airfoil-2.dat.gz).

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. 250)).

Plots → XY Plot → Set Up...



- 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.

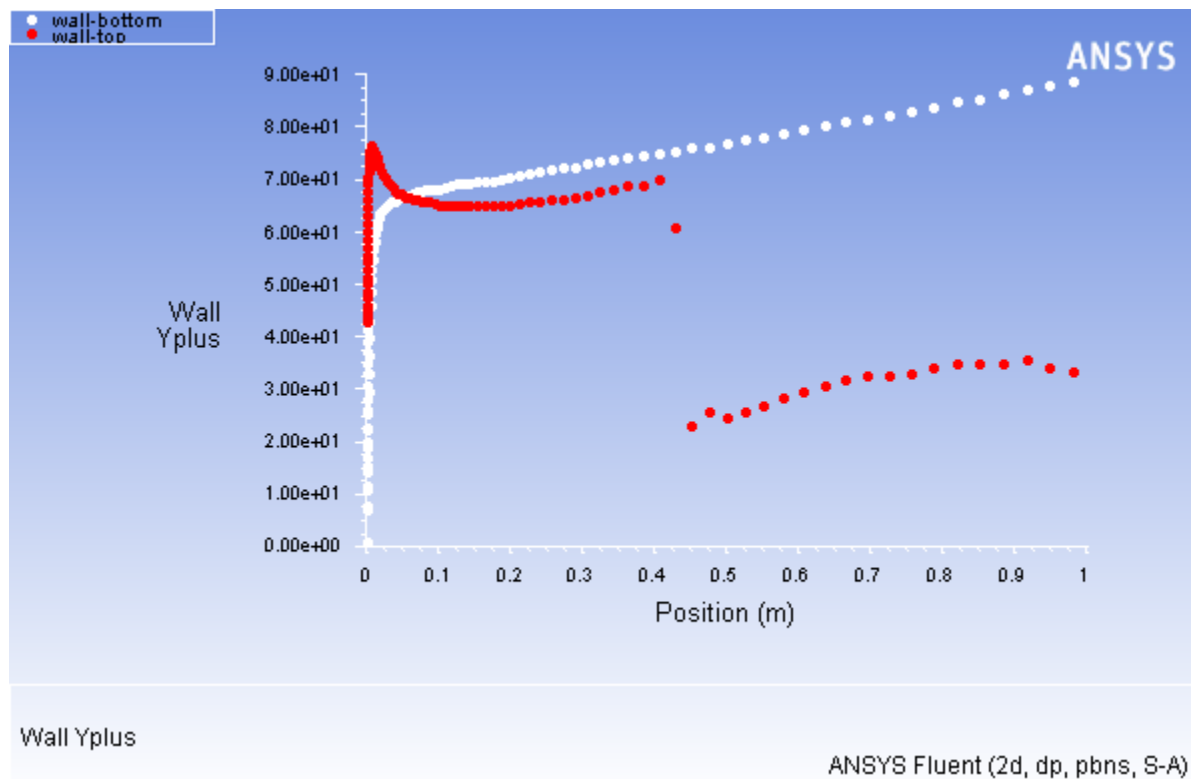
The equation for y^+ is

$$y^+ = \frac{y}{\mu} \sqrt{\rho \tau_w} \quad (5.1)$$

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. 250) 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.

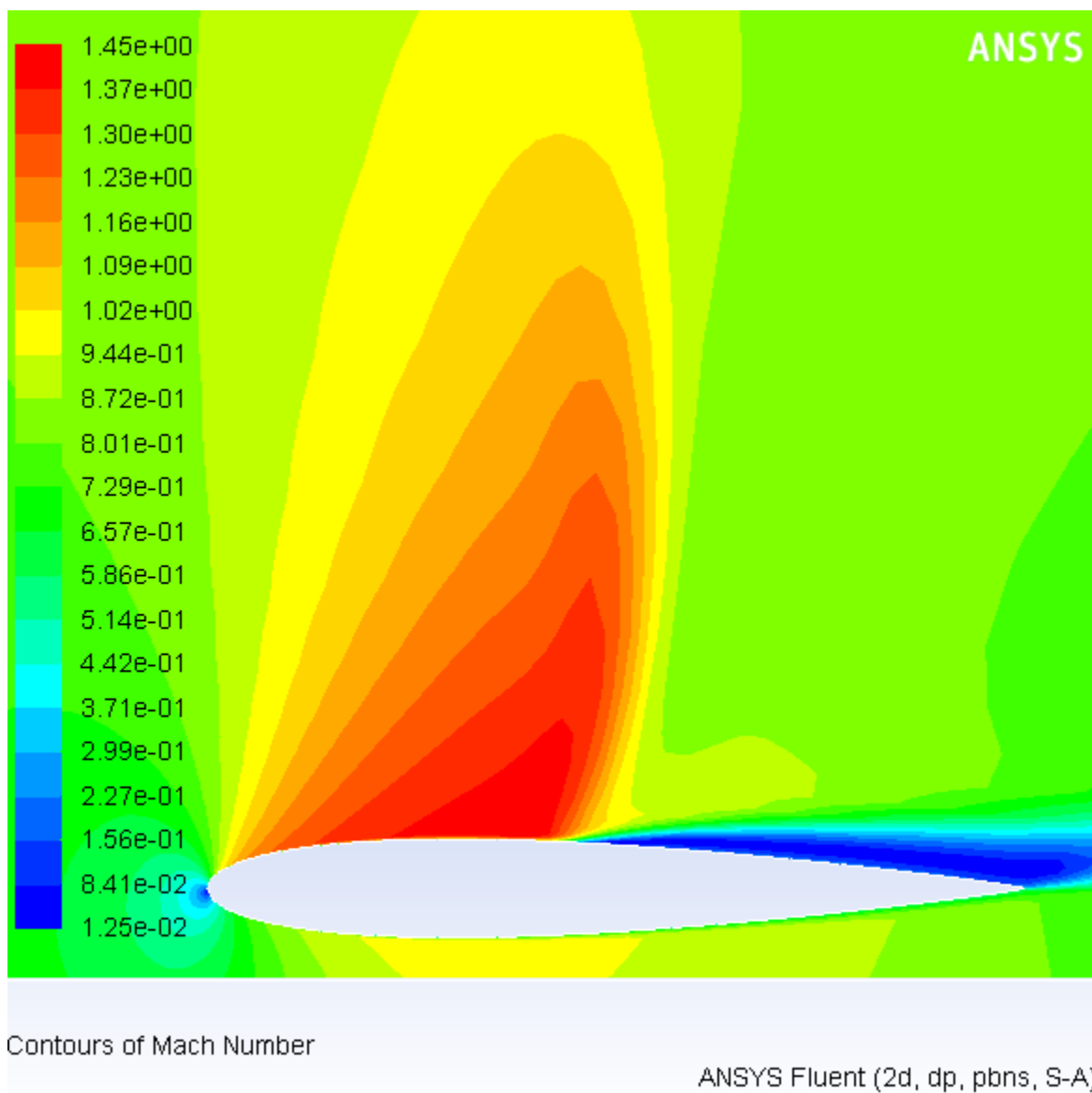
Figure 5.11: XY Plot of y^+ Distribution



2. Display filled contours of Mach number (Figure 5.12: Contour Plot of Mach Number (p. 251)).

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

- 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. 251).

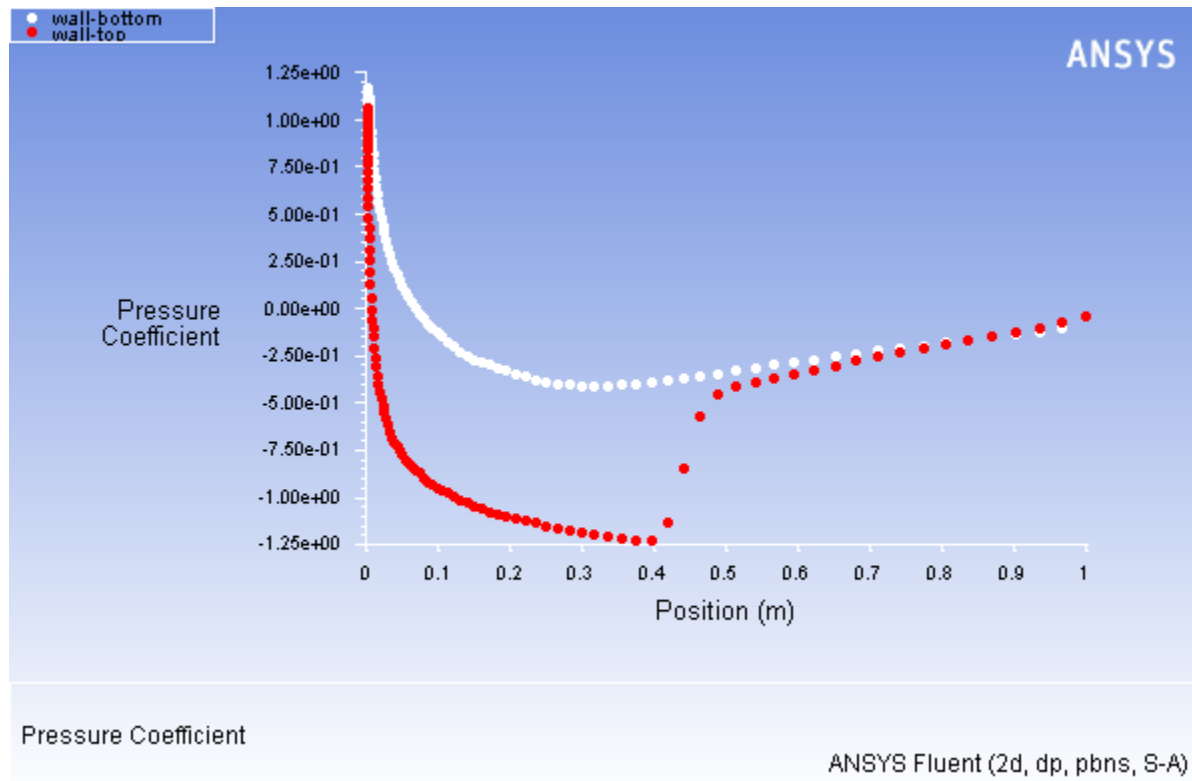
Figure 5.12: Contour Plot of Mach Number

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. 251) at about $x/c \approx 0.45$.

- Plot the pressure distribution on the airfoil (Figure 5.13: XY Plot of Pressure (p. 252)).

Plots → XY Plot → Set Up...

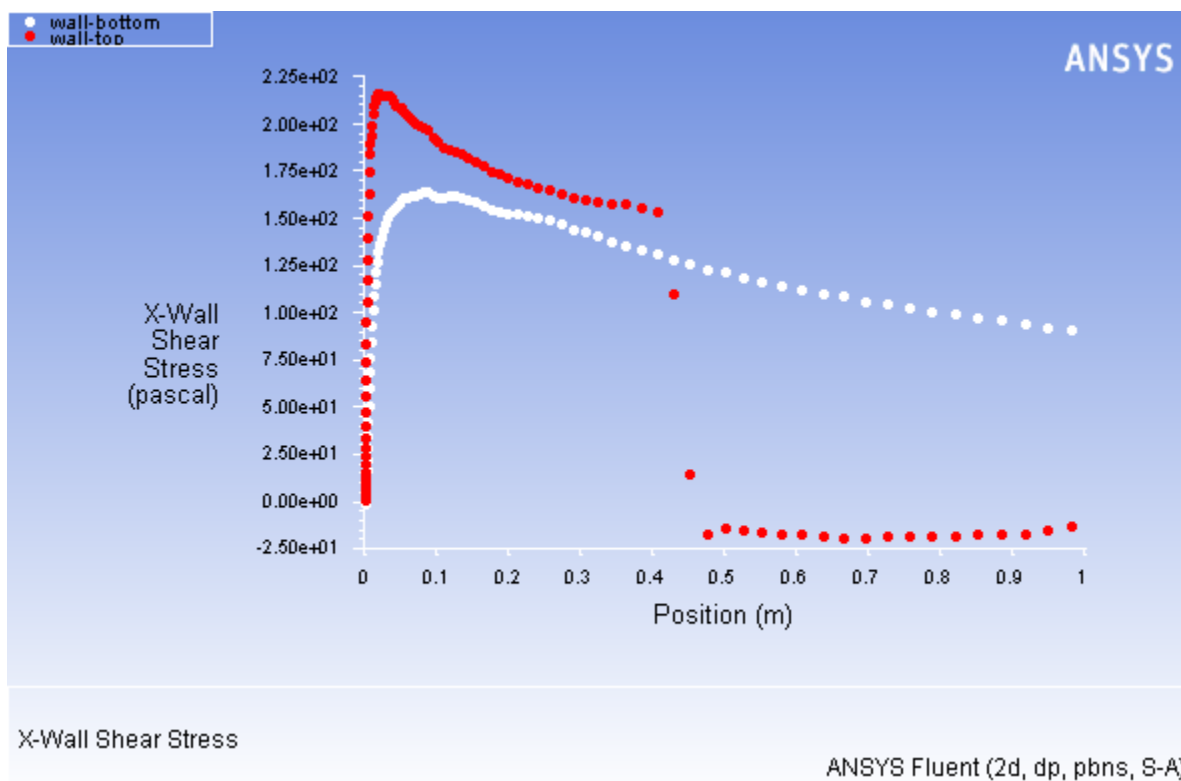
- Enable **Node Values**.
- Select **Pressure...** and **Pressure Coefficient** from the **Y Axis Function** drop-down lists.
- Click **Plot**.

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 \(p. 252\)](#).

4. Plot the x component of wall shear stress on the airfoil surface ([Figure 5.14: XY Plot of \$x\$ Wall Shear Stress \(p. 253\)](#)).
 - 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 \(p. 253\)](#), 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. 254)).

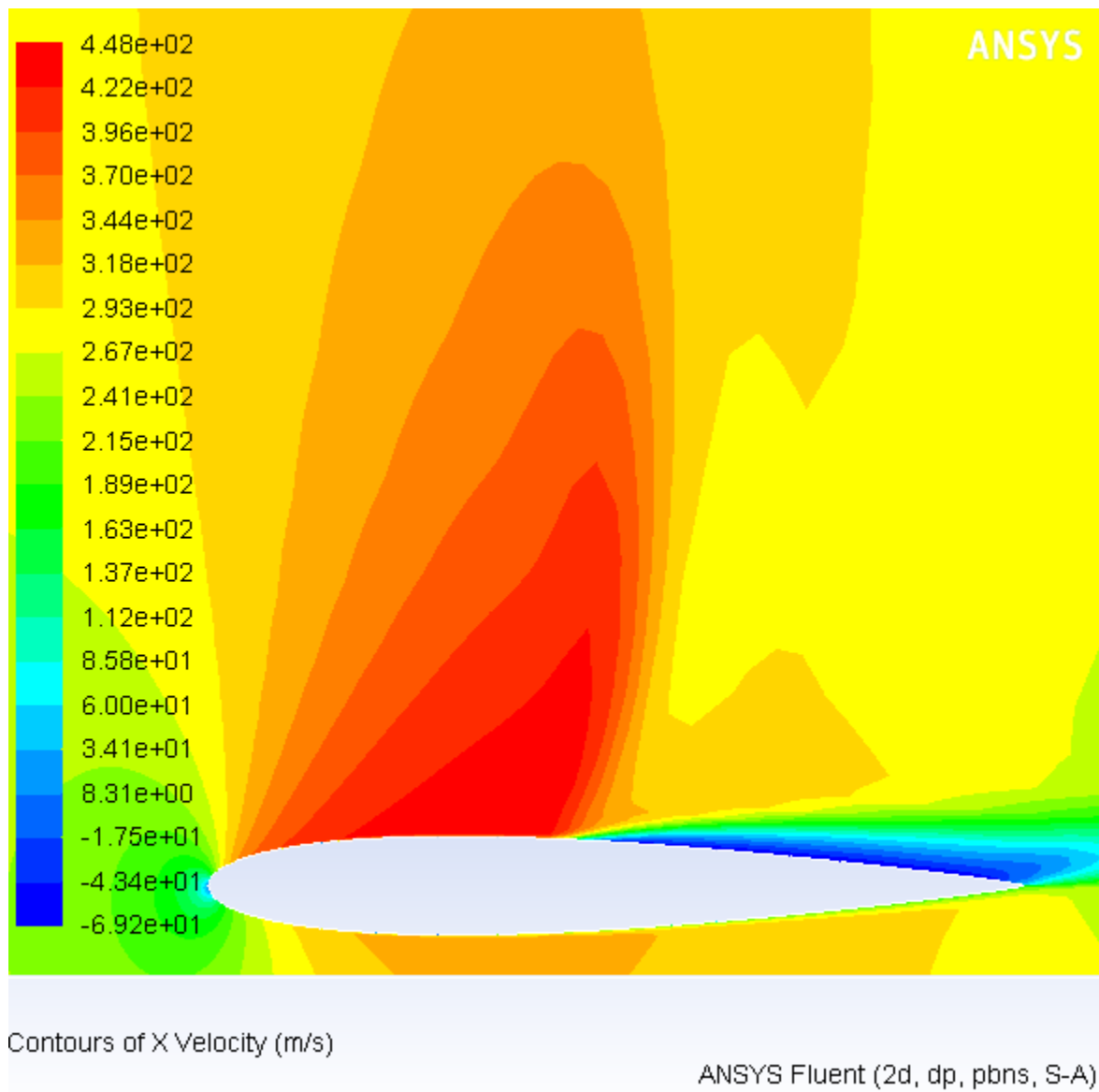
◆ **Graphics and Animations** → **Contours** → **Set Up...**

- a. Ensure **Filled** is enabled in the **Options** group box.
- b. Select **Velocity...** and **X Velocity** from the **Contours of** drop-down lists.

*Scroll up in the **Contours of** drop-down list to find **X Velocity**.*

- c. Click **Display** and close the **Contours** dialog box.

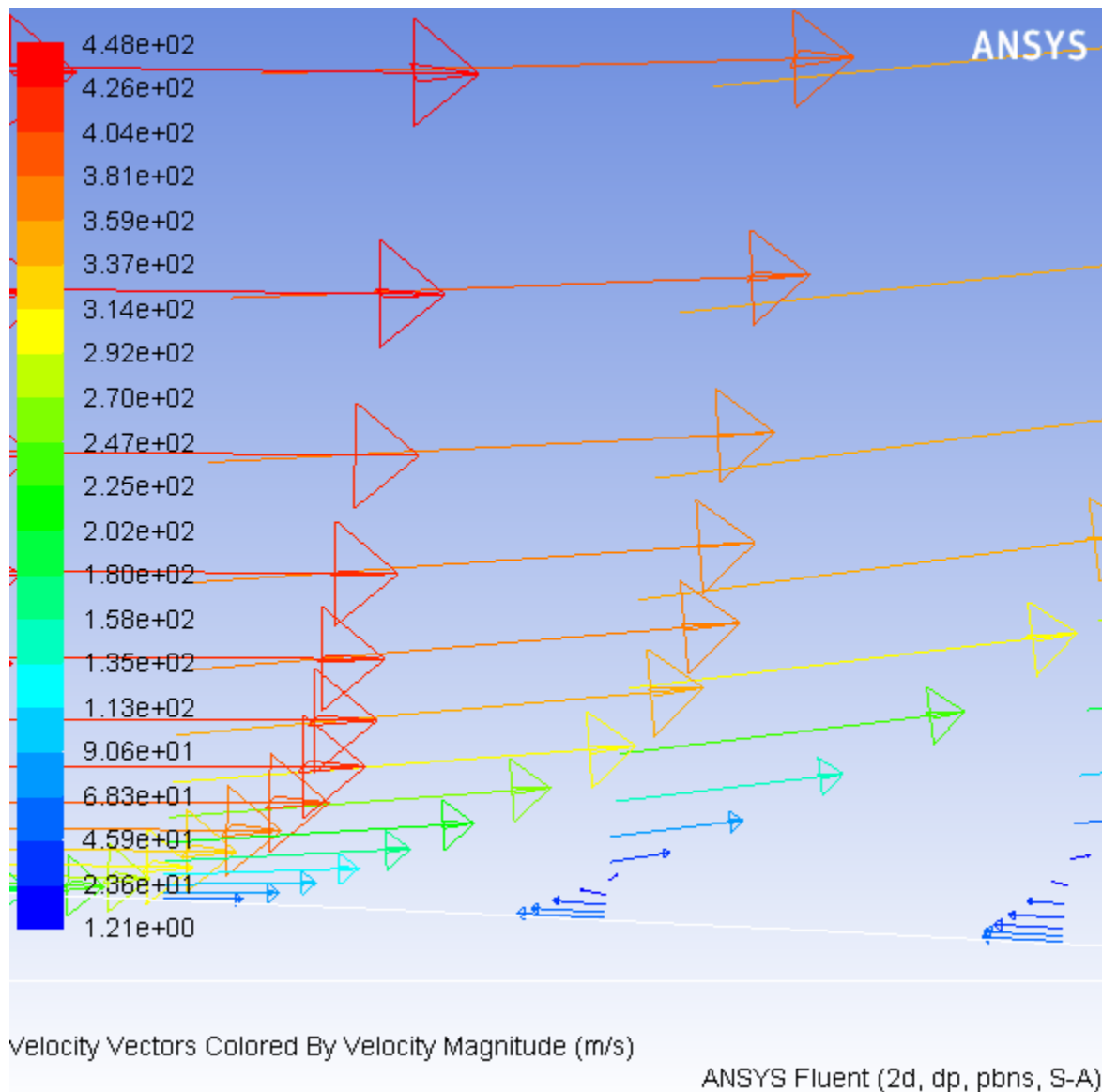
Note the flow reversal downstream of the shock in Figure 5.15: Contour Plot of x Component of Velocity (p. 254).

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. 255)).

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

- Enter 15 for **Scale**.
- Click **Display** and close the **Vectors** dialog box.
- 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. 255).

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 \(p. 255\)](#).

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 monitors, 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 \(p. 123\)](#).