Tutorial 3.　　　Modeling External Compressible Flow

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.

- Use force and surface monitors to check solution convergence.

- Check the near-wall mesh resolution by plotting the distribution of y^+.

Prerequisites

This tutorial is written with the assumption that you have completed Tutorial 1, 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.

Problem Description

The problem considers the flow around an airfoil at an angle of attack $\alpha = 4°$ 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 3.1.
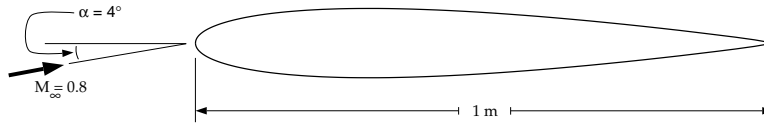

Figure 3.1: Problem Specification

Setup and Solution

Preparation

1. Download `external_compressible.zip` from the User Services Center to your working folder (as described in Tutorial 1).

2. Unzip `external_compressible.zip`.

   The file `airfoil.msh` *can be found in the* `external_compressible` *folder created after unzipping the file.*

3. Use FLUENT Launcher to start the 2D version of ANSYS FLUENT.

   For more information about FLUENT Launcher, see Section 1.1.2 in the separate User's Guide.

   Note: *The* Display Options *are enabled by default. Therefore, once you read in the mesh, it will be displayed in the embedded graphics window.*

Step 1: 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.*
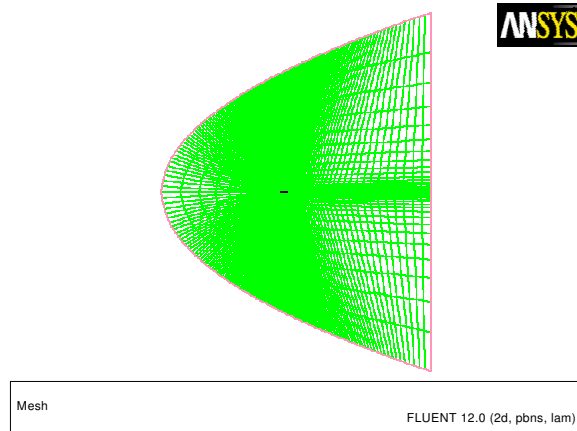
Figure 3.2: The Entire Mesh

3. Examine the mesh (Figures 3.2 and 3.3).

   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.
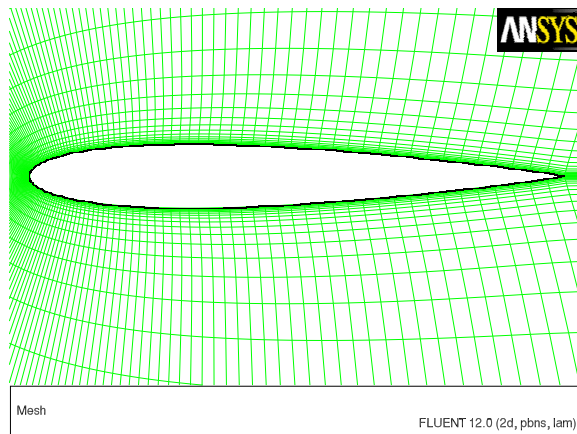

Figure 3.3: Magnified View of the Mesh Around the Airfoil

   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.*
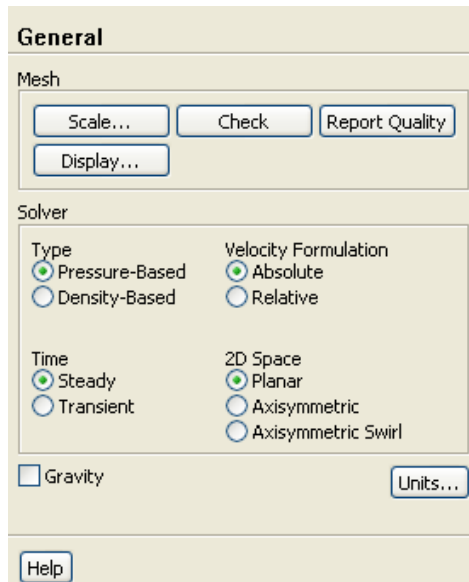
4. Reorder the mesh.

    Mesh $\longrightarrow$ Reorder $\longrightarrow$ 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.

Step 2: General Settings

⬥ General

1. Set the solver settings.

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

    The pressure-based coupled solver 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 Step 7: Solution.*

Step 3: Models

✥ Models

1. Select the Spalart-Allmaras turbulence model.

   ✥ Models ⟶ ≣ Viscous ⟶ Edit...
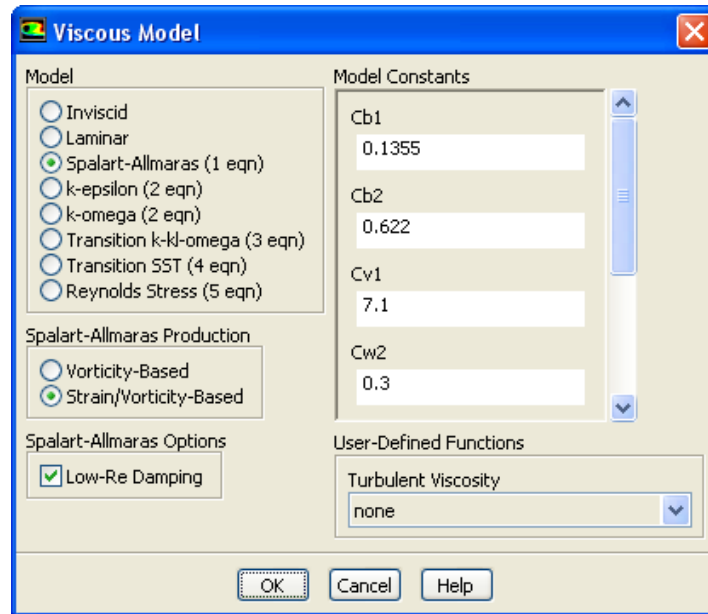
   (a) Select Spalart-Allmaras (1 eqn) in the Model list.

   (b) Select Strain/Vorticity-Based in the Spalart-Allmaras Production list.

   (c) Retain the default settings in the Model Constants group box.

   (d) Click OK to close the Viscous Model dialog box.

Note: *The Spalart-Allmaras model is a relatively simple one-equation model that solves a modeled transport equation for the kinematic eddy (turbulent) viscosity. This embodies a relatively new class of one-equation models in which it is not necessary to calculate a length scale related to the local shear layer thickness. The Spalart-Allmaras model was designed specifically for aerospace applications involving wall-bounded flows and has been shown to give good results for boundary layers subjected to adverse pressure gradients.*

Step 4: Materials

✛ Materials

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

1. Set the properties for air, the default fluid material.

   ✛ Materials ⟶ ☰ air ⟶ Create/Edit...
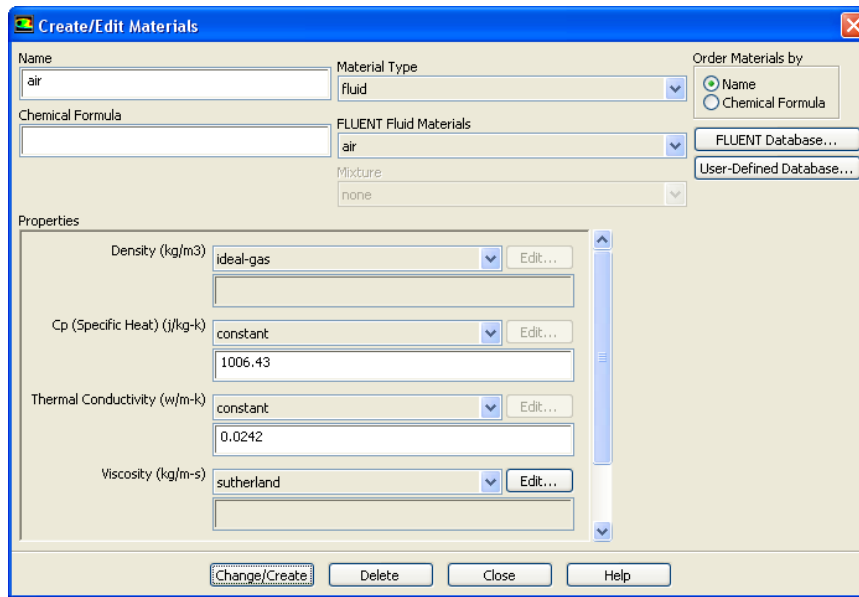
   (a) Select ideal-gas from the Density drop-down list.

      The Energy Equation will be enabled.

   (b) Select sutherland from the Viscosity drop-down list to open the Sutherland Law dialog box.
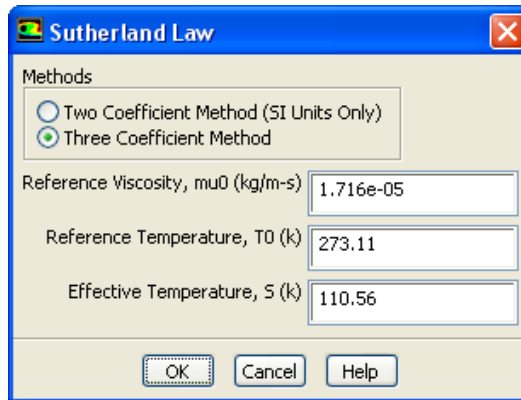
Scroll down the Viscosity *drop-down list to find* sutherland.

   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 *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 *are assumed to be constant in this tutorial.*

Step 5: Boundary Conditions

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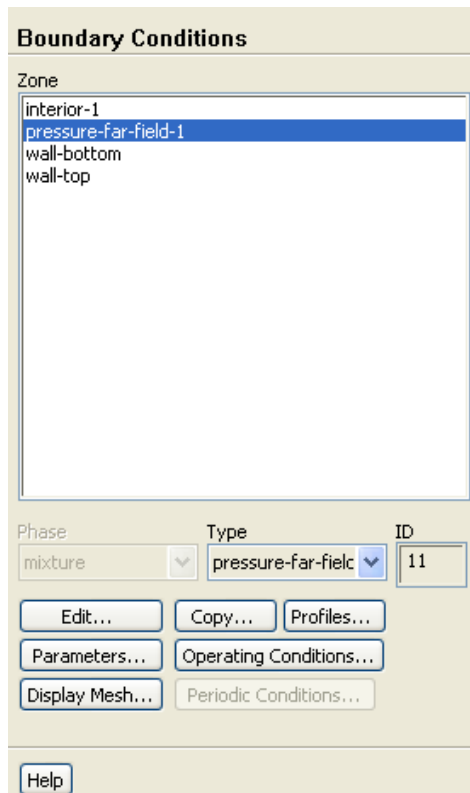
1. Set the boundary conditions for **pressure-far-field-1**.

    ⬥ Boundary Conditions ⟶ ☰ pressure-far-field-1 ⟶ Edit...
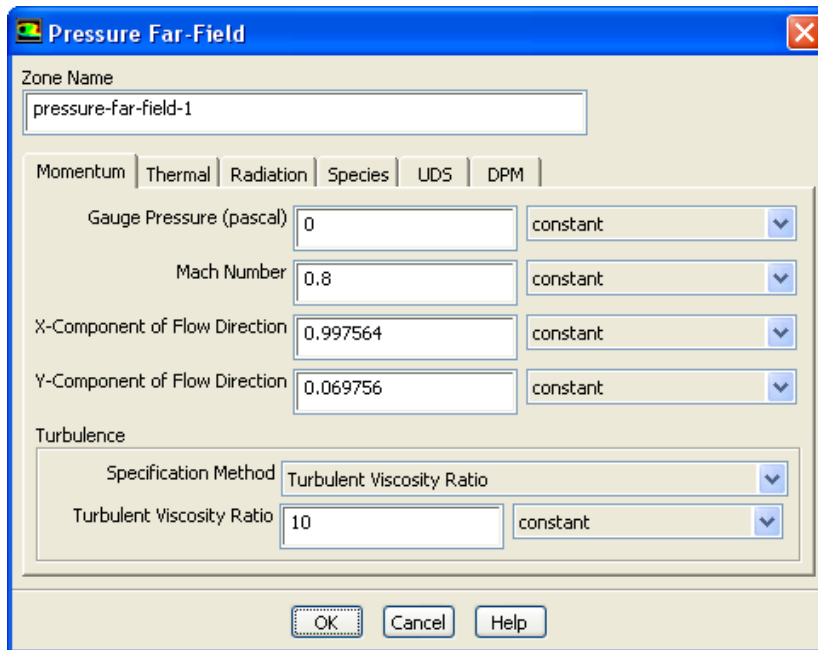
   (a) Retain the default value of 0 Pa for Gauge Pressure.

   (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: $\cos 4° \approx 0.997564$ *and*
   $\sin 4° \approx 0.069756$.

   (d) Select 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.
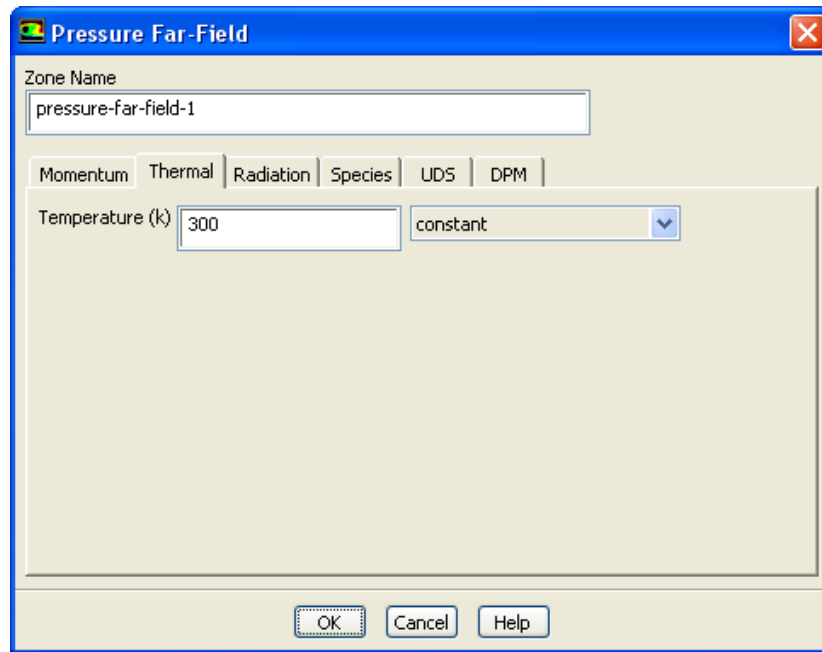
(g) Click OK to close the Pressure Far-Field dialog box.

Step 6: Operating Conditions

1. Set the operating pressure.

   ✧ Boundary Conditions ⟶ Operating Conditions...
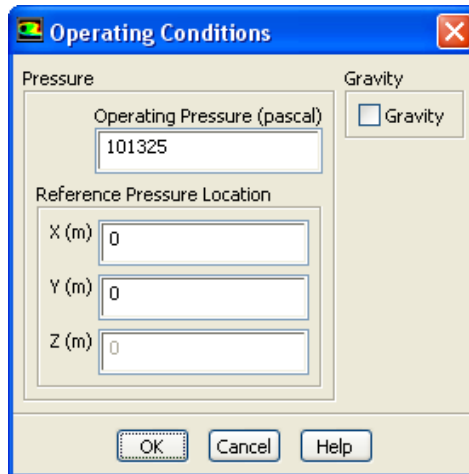
The Operating Conditions *dialog box can also be accessed from the* Cell Zone Conditions *task page.*

(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.

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

For information about setting the operating pressure, see Section 8.14 in the separate User's Guide.

Step 7: Solution

⬧ Solution

1. Set the solution parameters.

⬧ Solution Methods

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

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

(c) Retain the default selection of **Standard** from the **Pressure** drop-down list.

(d) Select **Second Order Upwind** from the **Density**, **Momentum**, **Modified Turbulent Viscosity**, and **Energy** drop-down lists.

Scroll down the **Spatial Discretization** *group box to find the* **Energy** *drop-down list.*

The second-order scheme will resolve the boundary layer and shock more accurately than the first-order scheme.

2. Set the solution controls.

✧ Solution Controls

(a) Retain the default value of 200 for Courant Number.

(b) Enter 0.5 for Momentum and Pressure in the Explicit Relaxation Factors group box.

Under-relaxing the momentum and pressure factors is recommended for higher-order discretization schemes.

(c) Enter 0.5 for Density in the Under-Relaxation Factors group box.

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

(d) Enter 0.9 for Modified Turbulent Viscosity.

Larger under-relaxation factors (i.e., closer to 1) will generally result in faster convergence. However, instability can arise that may need to be eliminated by decreasing the under-relaxation factors.
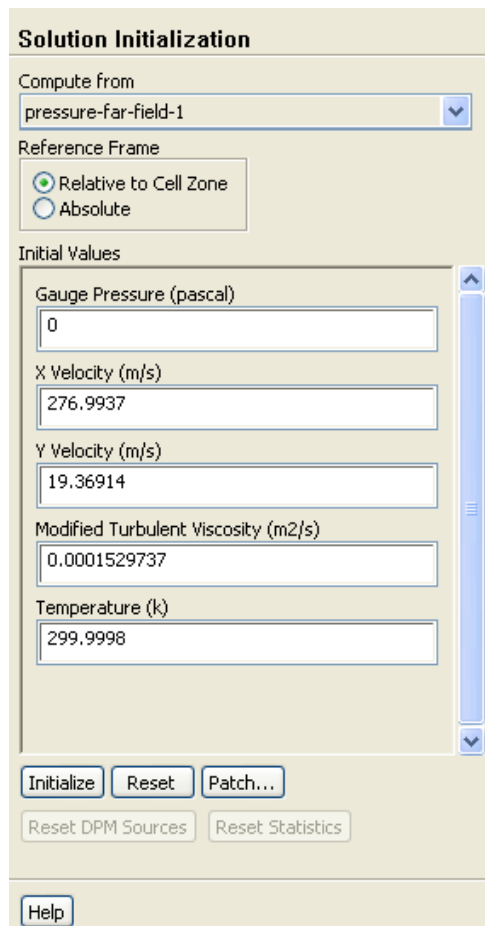
3. Enable residual plotting during the calculation.

❖ Monitors ⟶ ≣ Residuals ⟶ Edit...
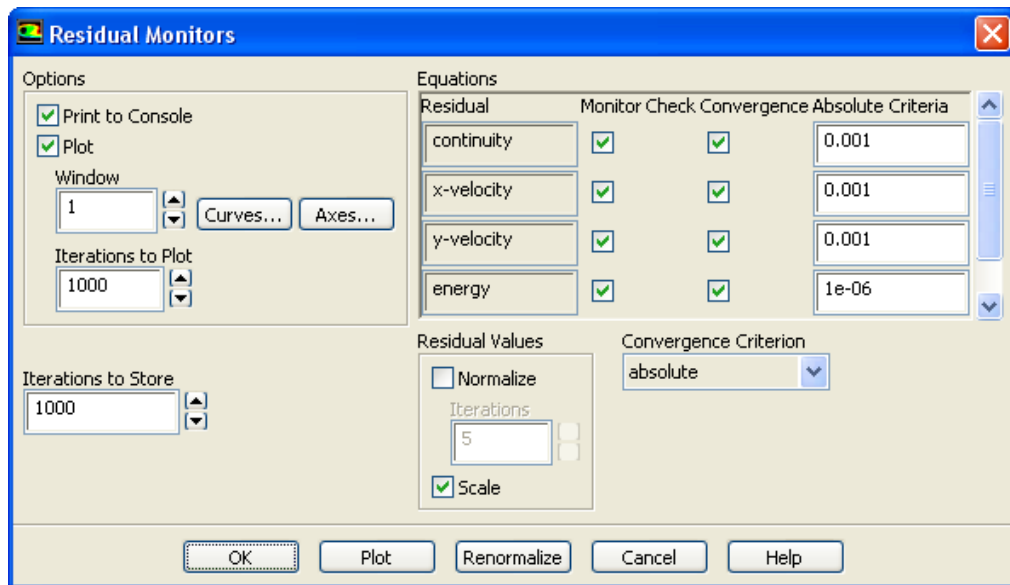
    (a) Make sure that Plot is enabled in the Options group box and click OK to close the Residual Monitors dialog box.

4. Initialize the solution.

   ⟡ Solution Initialization

   (a) Select pressure-far-field-1 from the Compute from drop-down list.

   (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 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 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 Section 26.10 in the separate User's Guide.*

5. Save the case and data files (`airfoil.cas` and `airfoil.dat`).

    File ⟶ Write ⟶Case & Data...

    It is good practice to save the case and data files during several stages of your case setup.

6. Start the calculation by requesting 50 iterations.

    ✧ Run Calculation

    (a) Enter 50 for Number of Iterations.

    (b) 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.

7. 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.
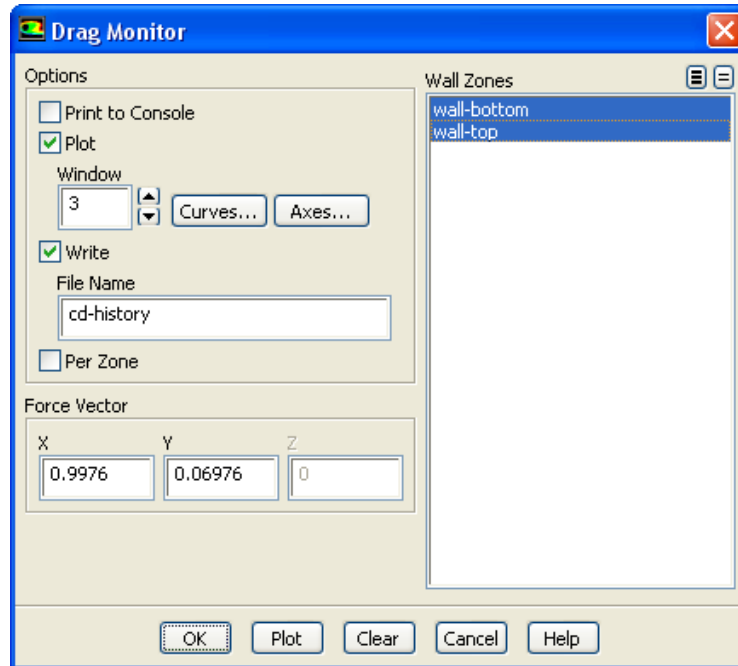

   (a) Select **pressure-far-field-1** from the **Compute from** drop-down list.

   ANSYS FLUENT *will update the* Reference Values *based on the boundary conditions at the far-field boundary.*

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

⟡ Monitors ⟶ ☰ Drag ⟶ Edit...

(a) Enable Plot in the Options group box.

(b) Set Window to 3.

(c) 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.

(d) Retain the default entry of cd-history for File Name.

(e) Select wall-bottom and wall-top in the Wall Zones selection list.

(f) Enter 0.9976 for X and 0.06976 for Y in the Force Vector group box.

   These X *and* Y *values ensure that the drag coefficient is calculated parallel to the free-stream flow, which is 4° off of the global coordinates.*
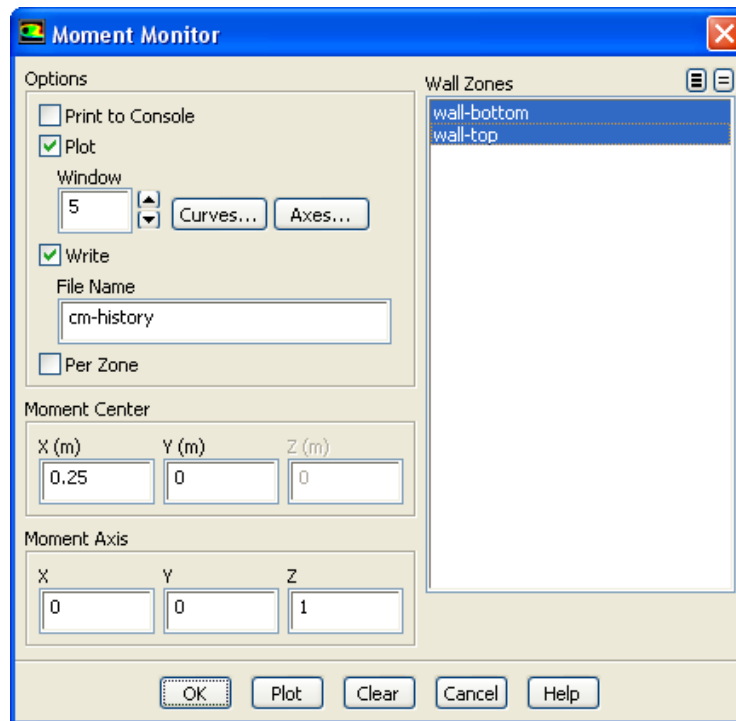
(g) Click OK to close the Drag Monitor dialog box.

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

✛ Monitors ⟶ ☰ Lift ⟶ Edit...
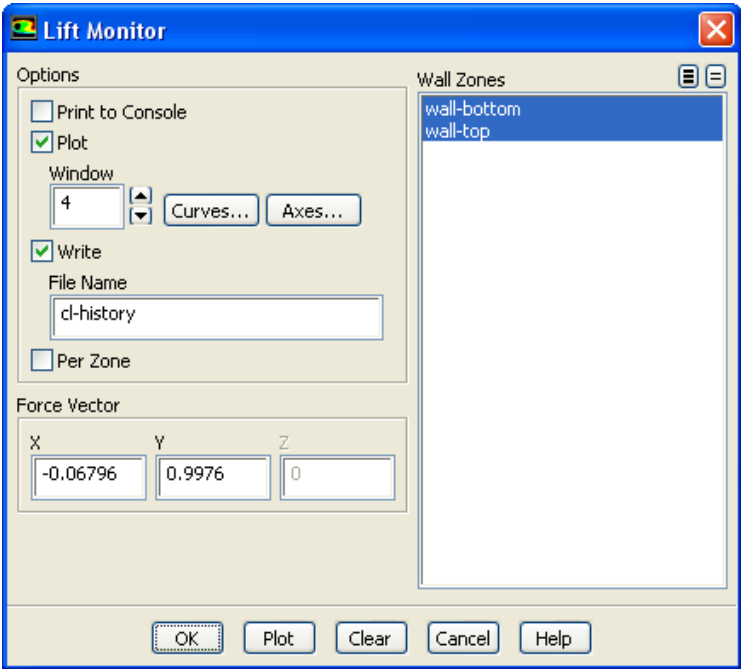
The X and Y values shown ensure that the lift coefficient is calculated normal to the free-stream flow, which is 4° off of the global coordinates.
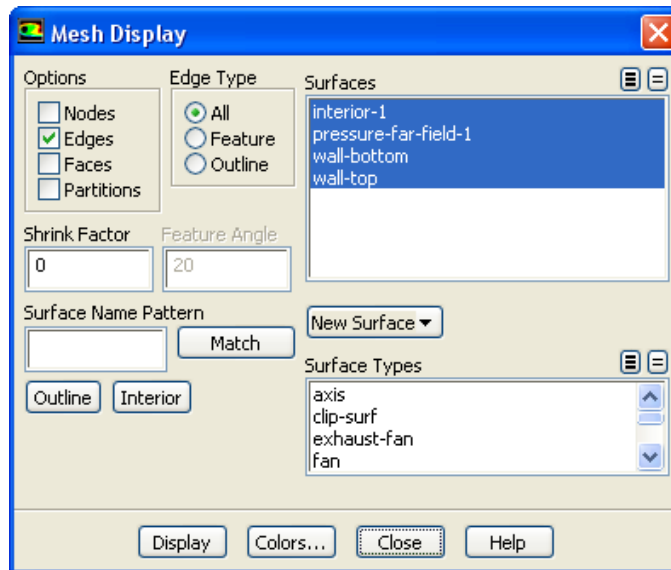
10. In a similar manner, define a force monitor for the moment coefficient.

    ✛ Monitors ⟶ ☰ Moment ⟶ Edit...

11. Display filled contours of pressure overlaid with the mesh in preparation for defining a surface monitor (Figures 3.4 and 3.5).

✛ Graphics and Animations ⟶ ☰ Contours ⟶ Set Up...
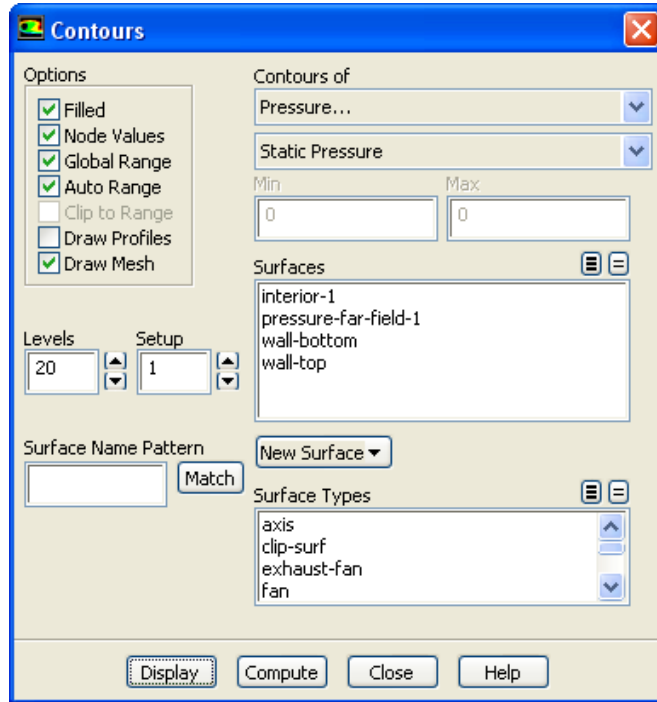
(a) Enable Filled in the Options group box.

(b) Enable Draw Mesh to open the Mesh Display dialog box.

i. Retain the default settings.

ii. Close the Mesh Display dialog box.

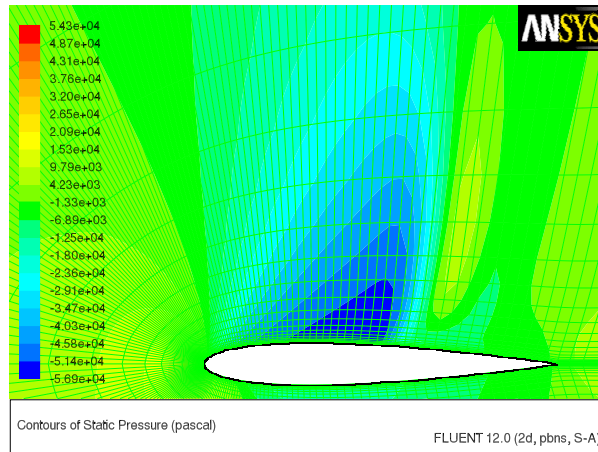(c) Click Display and close the Contours dialog box.

Figure 3.4: 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 3.5.
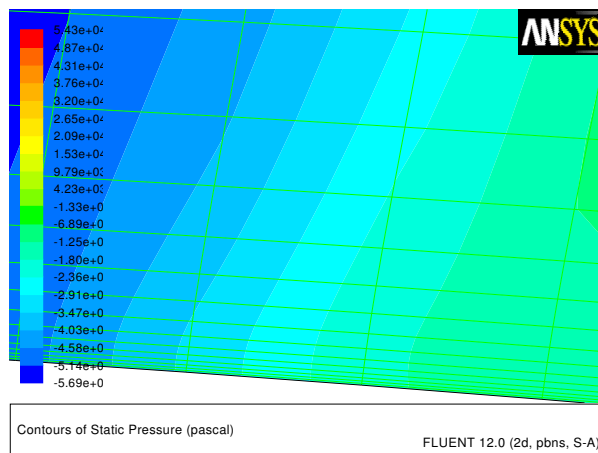

Figure 3.5: Magnified View of Pressure Contours Showing Wall-Adjacent Cells

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 $\longrightarrow$Point...
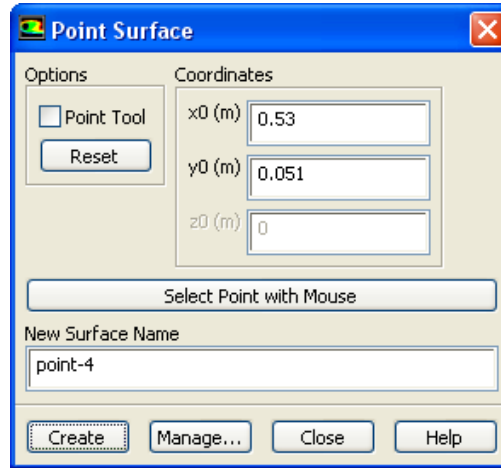
(a) Enter `0.53` m for x0 and `0.051` m for y0 in the Coordinates group box.

(b) Retain the default entry of `point-4` for New Surface Name.

(c) Click Create and close the Point Surface dialog box.

Note: *You have entered the exact coordinates of the point surface so that your convergence history will match the plots and description in this tutorial. In general, however, you will not know the exact coordinates in advance, so you will need to select the desired location in the graphics window as follows:*

    (a) Click the Select Point with Mouse *button.*

    *(b) Position the mouse pointer to a point located inside one of the cells adjacent to the upper surface (*wall-top *boundary), downstream of the shock (see Figure 3.6).*

    (c) Click the right mouse button.

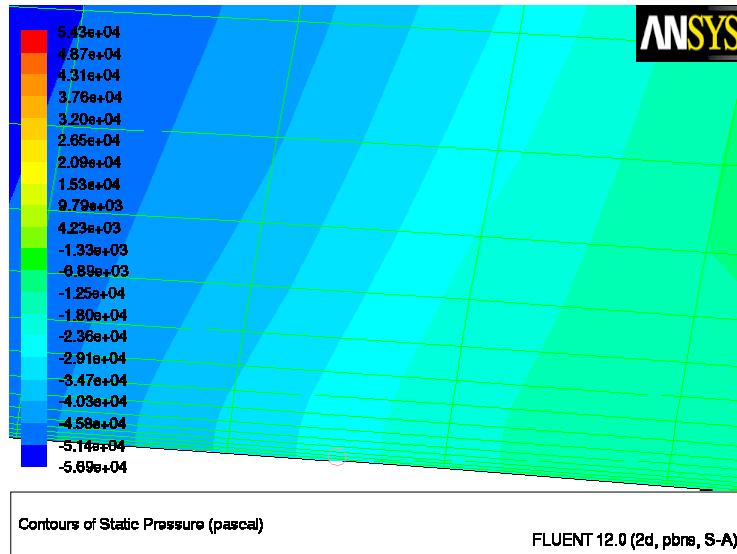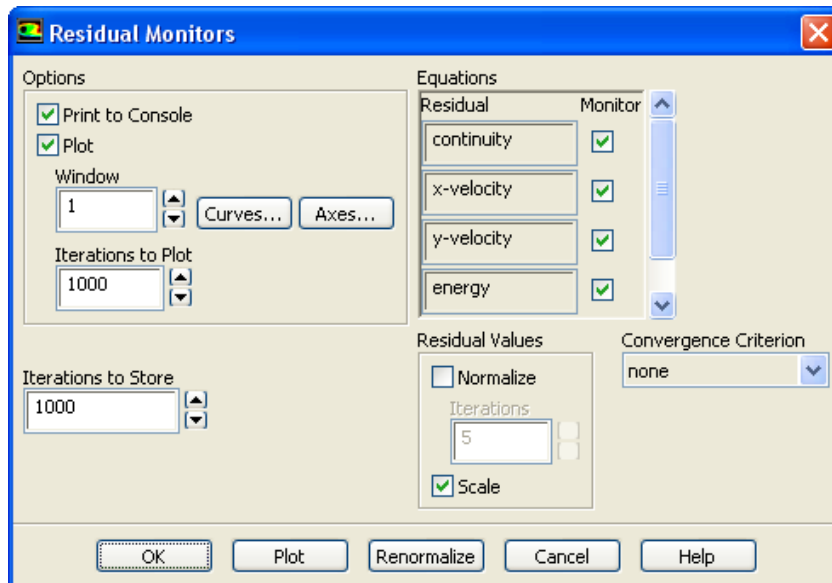   *(d) Click* Create *to create the point surface and close the* Point Surface *dialog box.*

Figure 3.6: Pressure Contours after Creating a Point with the Mouse

13. Enable residual plotting during the calculation.

◈ Monitors ⟶ ☰ Residuals ⟶ Edit...

(a) Make sure 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...
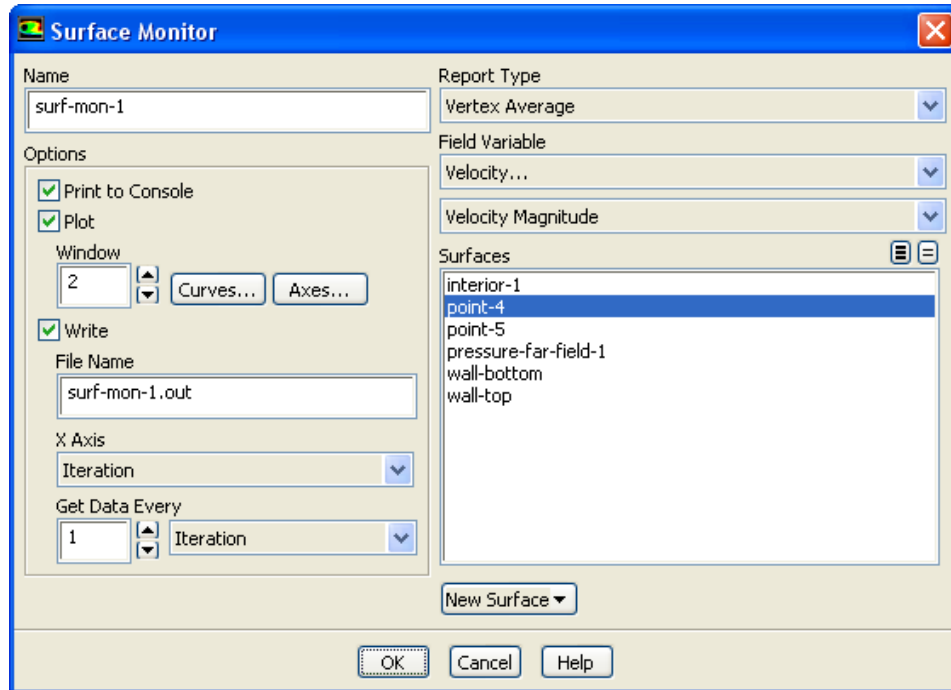
    (a) Enable Plot and Write.

    (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` and `airfoil-1.dat`).

    File ⟶ Write ⟶Case & Data...

16. Continue the calculation for 200 more iterations.

    ✛ Run Calculation

    The force monitors (Figures 3.8 and 3.9) show that the case is converged after approximately 200 iterations.
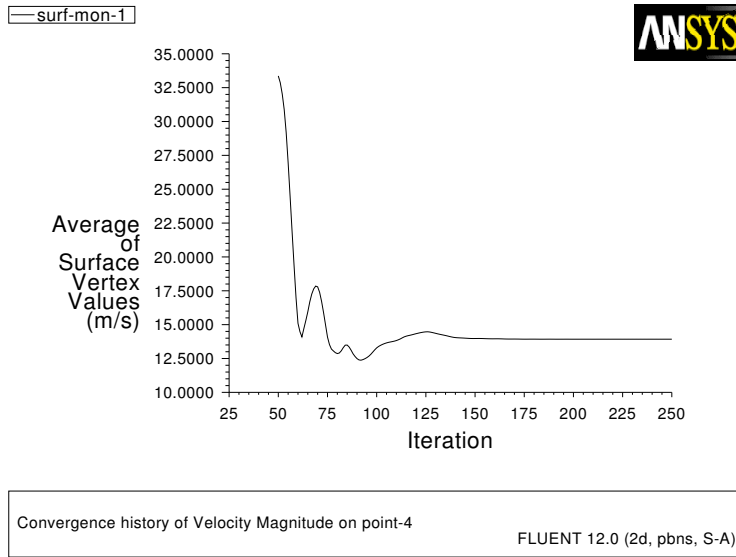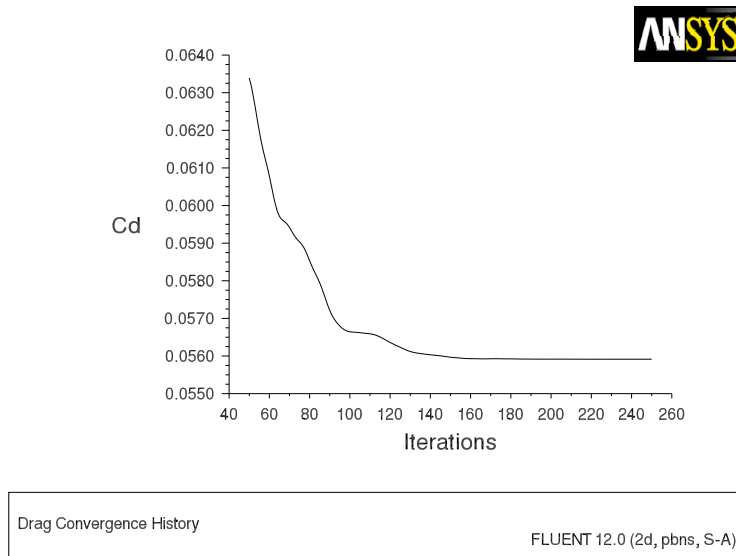
Figure 3.7: Velocity Magnitude History

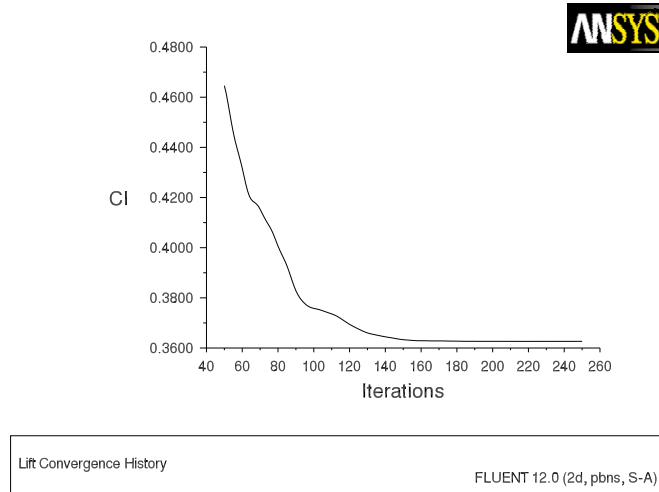Figure 3.8: Drag Coefficient Convergence History

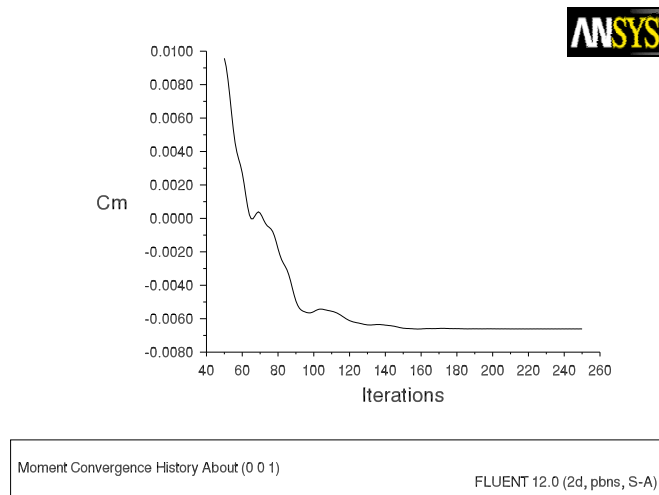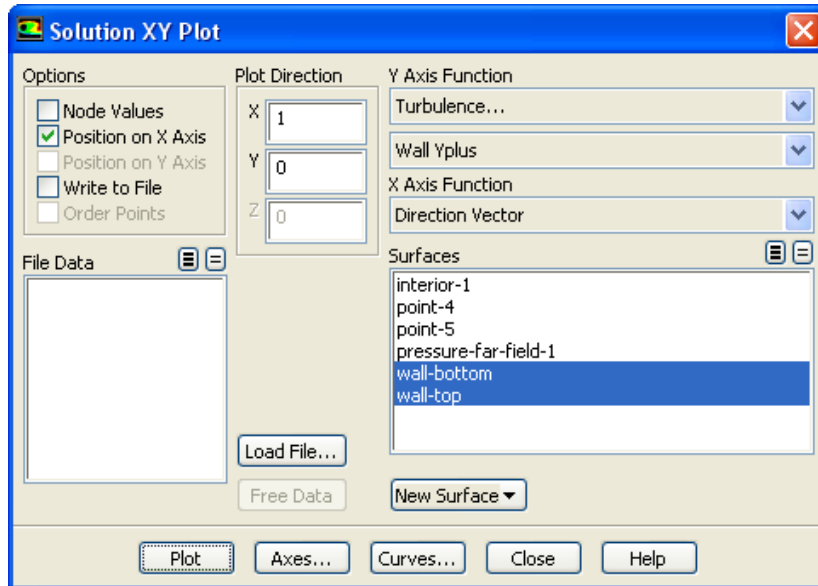Figure 3.9: Lift Coefficient Convergence History

Figure 3.10: Moment Coefficient Convergence History

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

    File ⟶ Write ⟶ Case & Data...

Step 8: Postprocessing

1. Plot the y^+ distribution on the airfoil (Figure 3.11).

   ✧ 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

$$y^+ = \frac{y}{\mu}\sqrt{\rho\tau_w}$$

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 3.11 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.
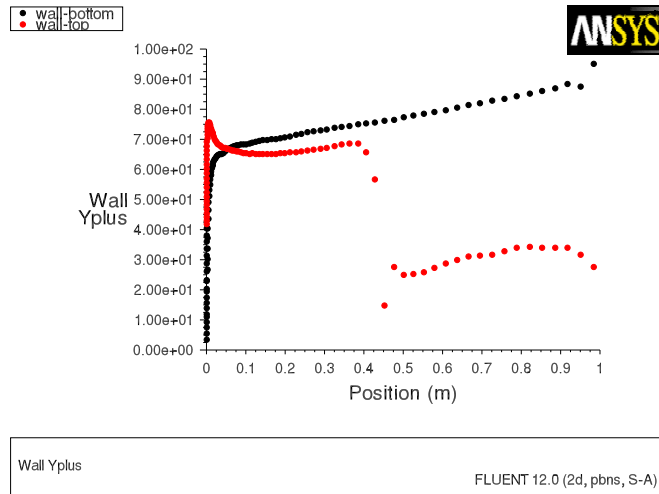
Figure 3.11: XY Plot of y^+ Distribution

2. Display filled contours of Mach number (Figure 3.12).

   ✧ Graphics and Animations ⟶ ▤ Contours ⟶ Set Up...

   (a) Enable Filled 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 3.12.
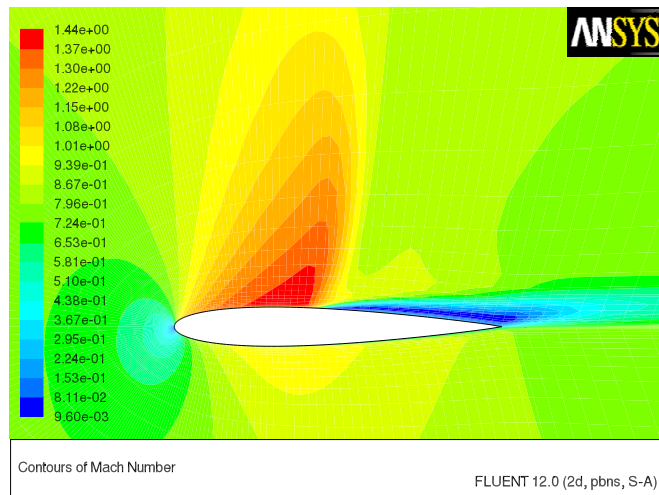

Figure 3.12: Contour Plot of Mach Number

Note the discontinuity, in this case a shock, on the upper surface of the airfoil in Figure 3.12 at about $x/c \approx 0.45$.

3. Plot the pressure distribution on the airfoil (Figure 3.13).

   ✧ Plots ⟶ XY Plot ⟶ Set Up...

   (a) Enable Node Values.

   (b) Select Pressure... and Pressure Coefficient from the Y Axis Function drop-down lists.
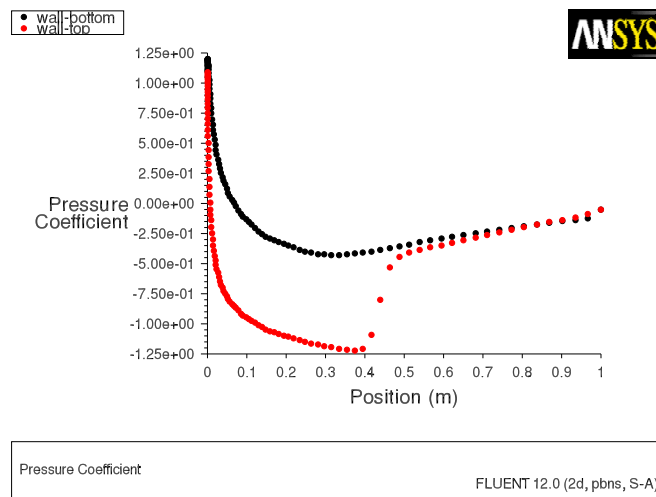
   (c) Click Plot.

Figure 3.13: XY Plot of Pressure

Notice the effect of the shock wave on the upper surface in Figure 3.13.

4. Plot the x component of wall shear stress on the airfoil surface (Figure 3.14).

   (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 3.14, 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.
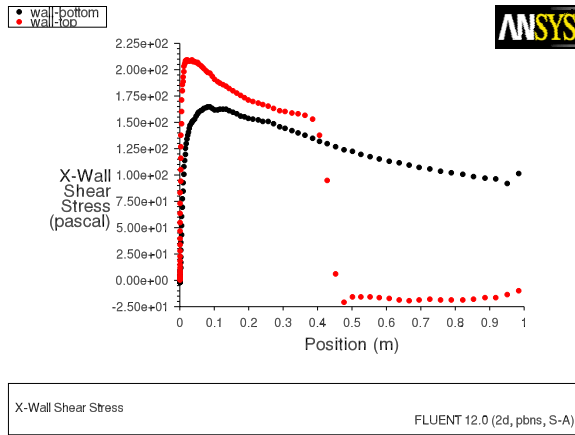
Figure 3.14: XY Plot of x Wall Shear Stress

5. Display filled contours of the x component of velocity (Figure 3.15).

◈ **Graphics and Animations** ⟶ ☰ **Contours** ⟶ **Set Up...**

(a) Enable **Filled**.

(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.

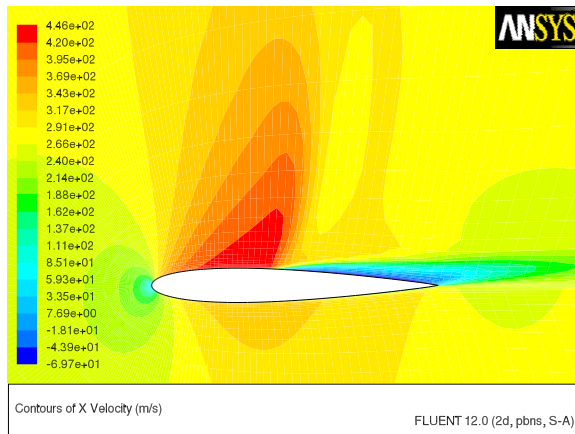*Note the flow reversal downstream of the shock in Figure 3.15.*

Figure 3.15: Contour Plot of x Component of Velocity

6. Plot velocity vectors (Figure 3.16).

✧ Graphics and Animations ⟶ ☰ Vectors ⟶ Set Up...

(a) Enter 15 for Scale.

(b) Click Display and close the Vectors dialog box.

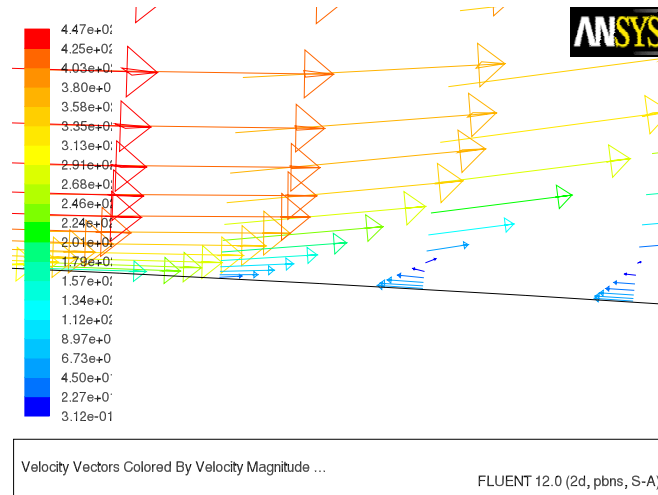(c) Zoom in on the flow above the upper surface at a point downstream of the shock, as shown in Figure 3.16.

Figure 3.16: Plot of Velocity Vectors Downstream of the Shock

Flow reversal is clearly visible in Figure 3.16.

Summary

This tutorial demonstrated how to set up and solve an external aerodynamics problem using the pressure-based coupled solver 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.

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 Tutorial 1.