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 will demonstrate 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 grid resolution by plotting the distribution of y^+.

Prerequisites

This tutorial assumes that you are familiar with the menu structure in FLUENT and that you have completed Tutorial 1. 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.

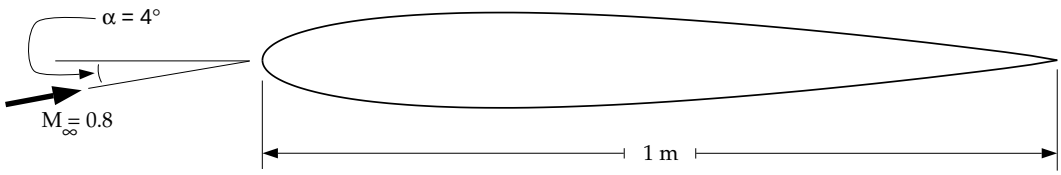


Figure 3.1: Problem Specification

Setup and Solution

Preparation

1. Download `external_compressible.zip` from the Fluent Inc. User Services Center or copy it from the FLUENT documentation CD 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. Start the 2D (2d) version of FLUENT.

© Fluent Inc. September 21, 2006

Step 1: Grid

1. Read the grid file `airfoil.msh`.

   File ⟶ Read ⟶ Case...

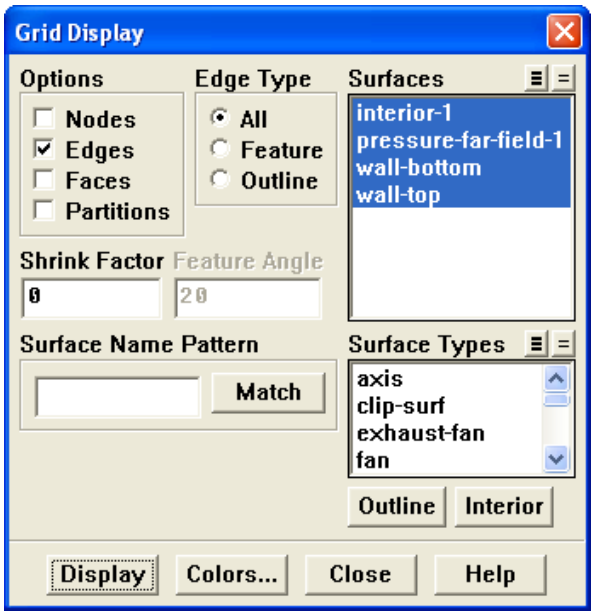
2. Check the grid.

   Grid ⟶ Check

   FLUENT *will perform various checks on the mesh and will report the progress in the console. Make sure that the minimum volume reported is a positive number.*

3. Display the grid (Figures 3.2 and 3.3).

   Display ⟶ Grid...



   (a) Retain the default settings.

   (b) Click Display and close the Grid Display panel.

   (c) Zoom in on the region around the airfoil (as shown in Figure 3.3), by using the middle mouse button in the graphics window.

   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.

   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.

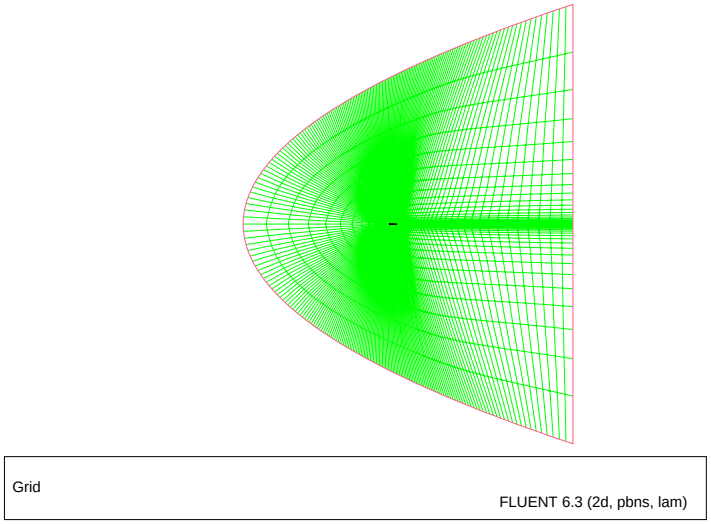


Figure 3.2: The Entire Grid

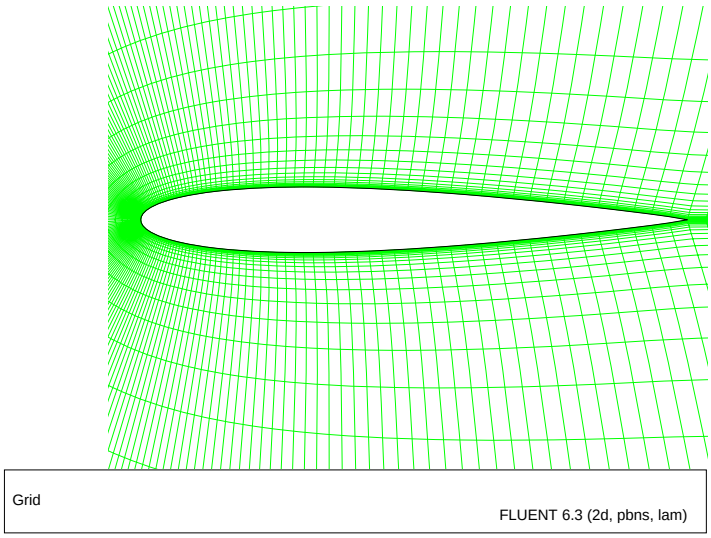


Figure 3.3: Magnified View of the Grid Around the Airfoil

© Fluent Inc. September 21, 2006

Extra: *You can use the right mouse button to probe for grid information in the graphics window. If you click the right mouse button on any node in the grid, information will be displayed in the* 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.

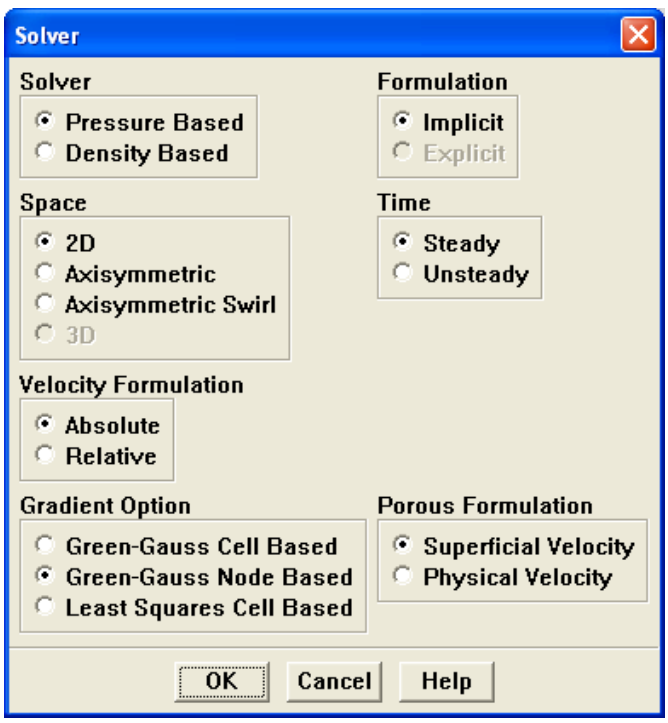
   Grid ⟶ Reorder ⟶ Domain

   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.

Step 2: Models

1. Specify the solver settings.

   Define ⟶ Models ⟶ Solver...



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

   The pressure-based coupled solver is a good alternative to FLUENT*'s density-based solvers when dealing with applications involving high-speed aerodynamics with shocks. Selection of the coupled algorithm is made in the* Solution Controls *panel in Step 6: Solution.*

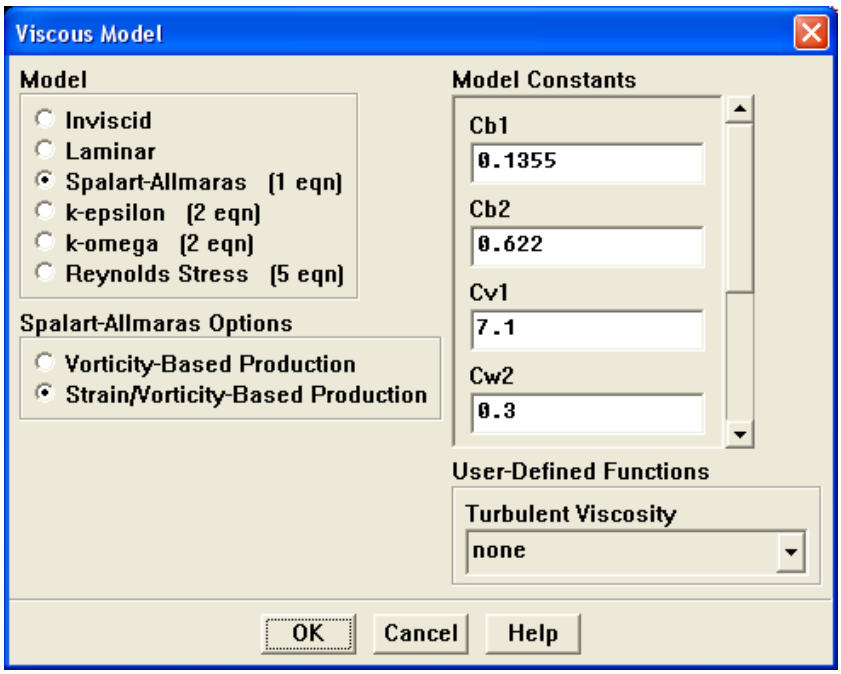
(b) Select Green-Gauss Node Based from the Gradient Option list.

This option uses better numerics, in particular on unstructured meshes. It also tends to predict drag more accurately.

(c) Click OK to close the Solver panel.

2. Enable the Spalart-Allmaras turbulence model.

Define ⟶ Models ⟶ Viscous...



(a) Select the Spalart-Allmaras in the Model list.

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

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

(d) Click OK to close the Viscous Model panel.

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.*

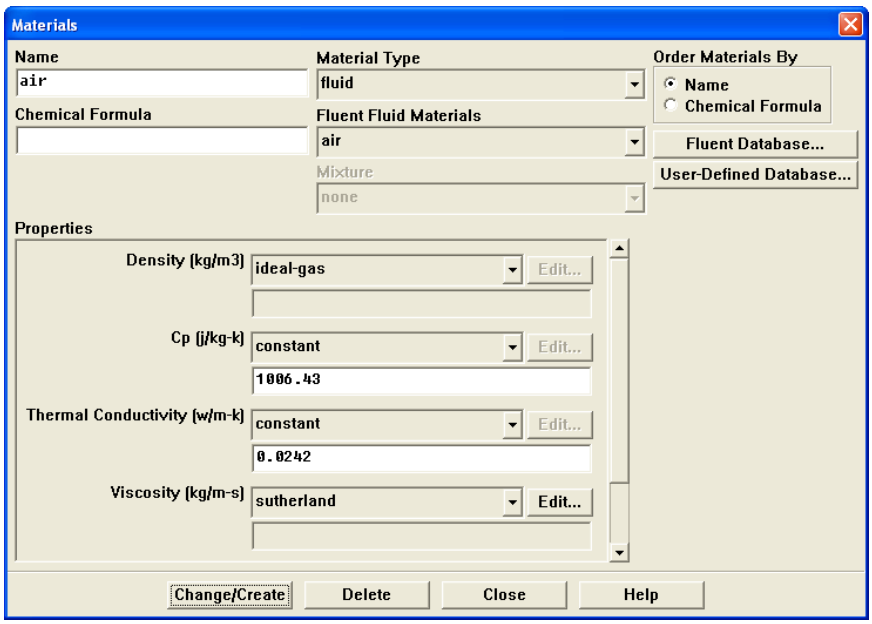
© Fluent Inc. September 21, 2006

Step 3: 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 thermo-physical properties with temperature.*

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

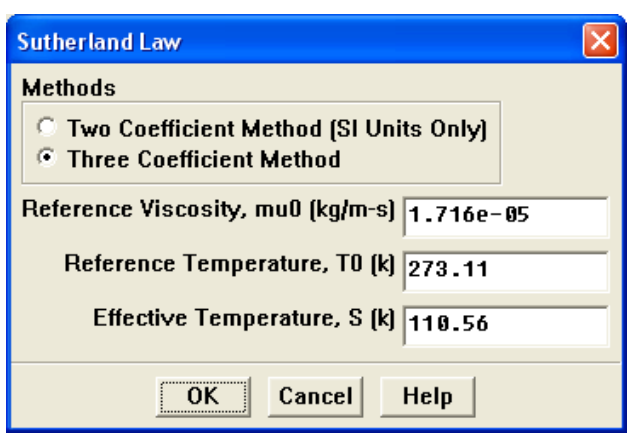
   Define ⟶ Materials...



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

   (b) Select sutherland from the Viscosity drop-down list to open the Sutherland Law panel.

   Scroll down the list to find sutherland.



i. Retain the default selection of Three Coefficient Method from the Methods list.

ii. Click OK to close the Sutherland Law panel.

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

(c) Click Change/Create to save these settings.

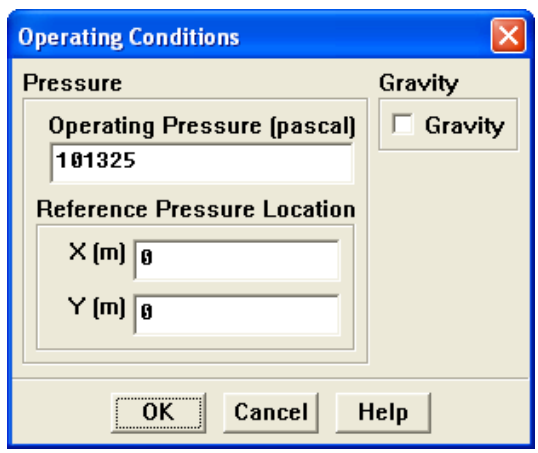
(d) Close the Materials panel.

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 4: Operating Conditions

1. Set the operating pressure.

   Define ⟶ Operating Conditions...



(a) Retain the default entry of 101325 Pa for Operating Pressure.

The operating pressure should be set to a meaningful mean value in order to avoid round-off errors.

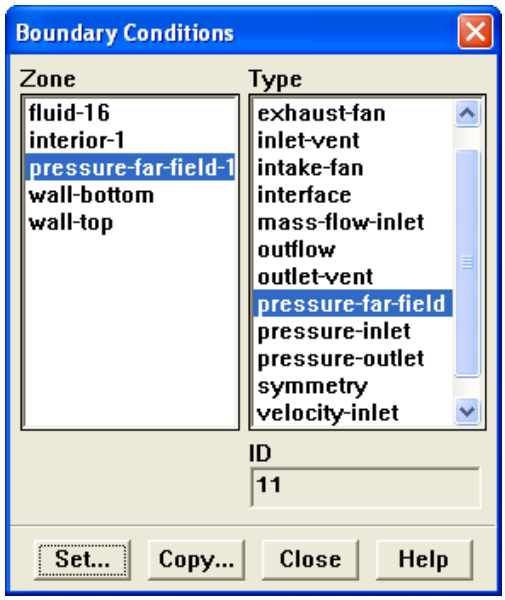
(b) Click OK to close the Operating Conditions panel.

See Section 8.14 of the User's Guide for more information on how to set the operating pressure.

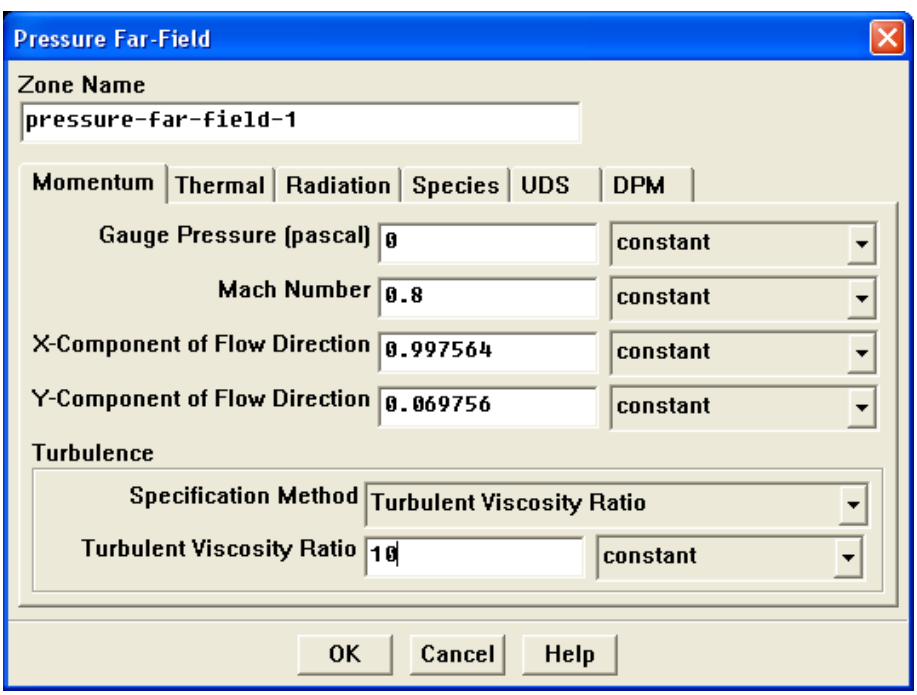
© Fluent Inc. September 21, 2006

Step 5: Boundary Conditions

Define ⟶ Boundary Conditions...



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



(a) Retain the default entry 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.

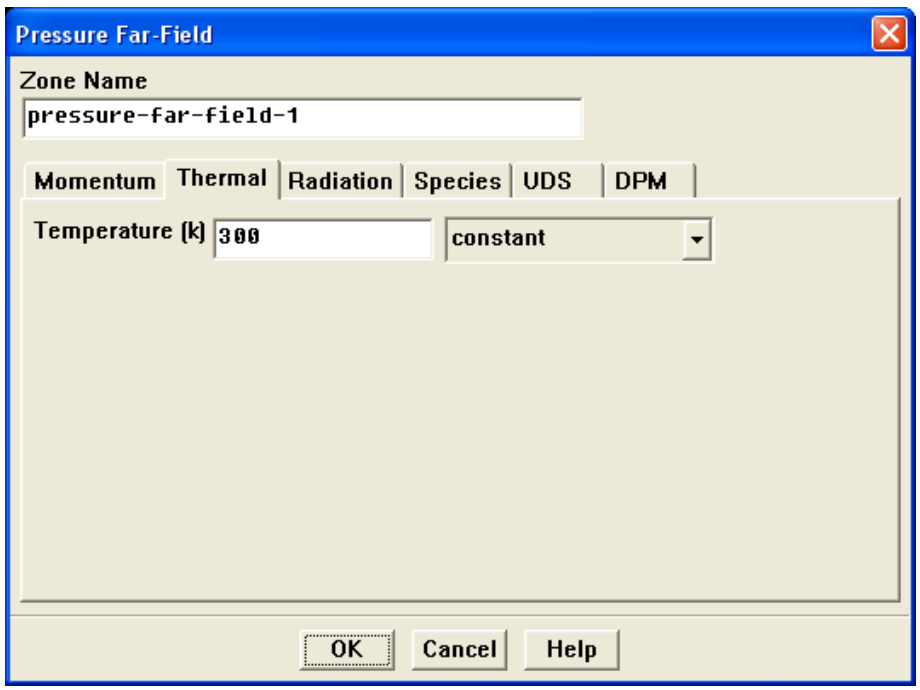
These values are determined by the 4° *angle of attack:* $\cos 4^\circ \approx 0.997564$ *and* $\sin 4^\circ \approx 0.069756$.

(d) Select Turbulent Viscosity Ratio from the Specification Method drop-down list in the Turbulence group box.

(e) Retain the default entry 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 entry of 300 K for Temperature.



(g) Click OK to close the Pressure Far-Field panel.

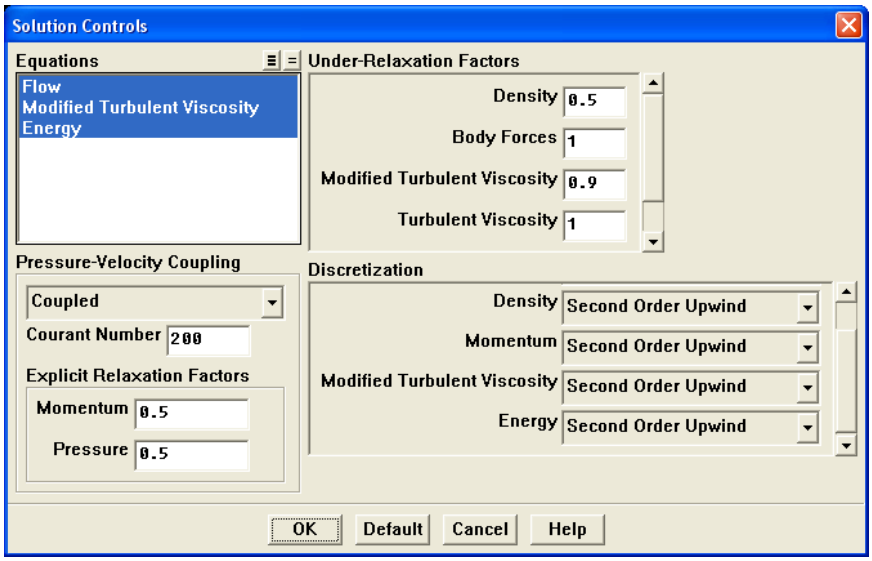
2. Close the Boundary Conditions panel.

© Fluent Inc. September 21, 2006

Step 6: Solution

1. Set the solution controls.

   Solve ⟶ Controls ⟶ Solution...



   (a) Select Coupled from the Pressure-Velocity Coupling drop-down list.

   (b) Retain the default entry of 200 for the Courant Number.

   (c) 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.

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

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

   (e) 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.

   (f) Retain the default selection of Standard from the Pressure drop-down list in the Discretization group box.

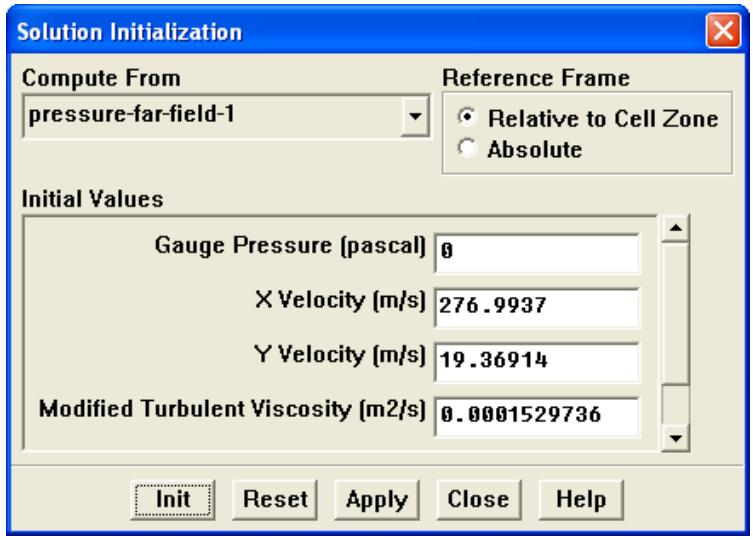
   (g) Select Second Order Upwind from the Density, Momentum, Modified Turbulent Viscosity, and Energy drop-down lists.

   Scroll down 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.*

(h) Click OK to accept the settings and close the Solution Controls panel.

2. Initialize the solution.

   Solve ⟶ Initialize ⟶ Initialize...



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

   (b) Click Init to initialize the solution.

   (c) Close the Solution Initialization panel.

3. 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.

   (a) Press <Enter> in the console to get the command prompt (>).

   (b) Enter the text commands and input responses outlined in boxes as shown, accepting 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
```


© Fluent Inc. September 21, 2006

```
   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: *The FMG initialized flow field can be inspected using* FLUENT*'s postprocessing tools.*

4. 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.

5. Start the calculation by requesting 50 iterations.

   Solve ⟶ Iterate...

(a) Enter 50 for Number of Iterations.

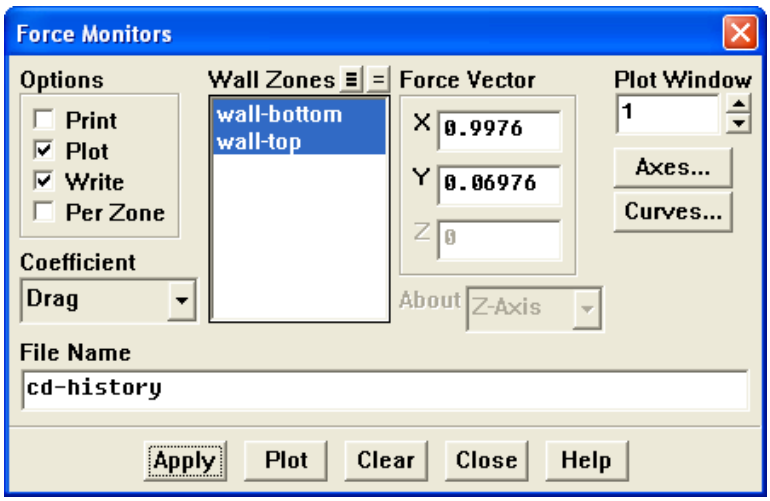
(b) Click Iterate.

(c) Close the Iterate panel when the calculation is complete.

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.

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

Solve ⟶ Monitors ⟶ Force...



(a) Enable Plot in the Options group box.

(b) Enable the Write option to save the monitor history to a file.

Note: *If you do not select the* Write *option, the history information will be lost when you exit* FLUENT.

(c) Retain the default selection of Drag from the Coefficient drop-down list.

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

(e) Select wall-bottom and wall-top from 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.*

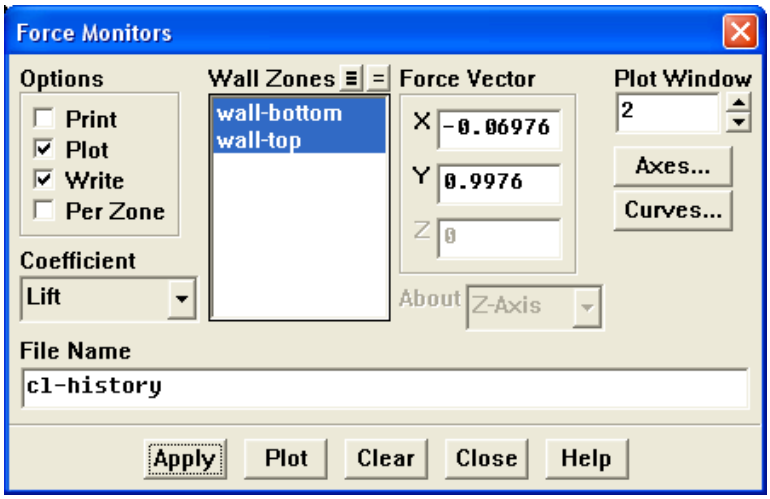
(g) Set the Plot Window to `1`.

(h) Click Apply.

© Fluent Inc. September 21, 2006

7. In a similar manner, define a force monitor for the lift coefficient, as shown in the following panel. Click Apply after the settings are complete.

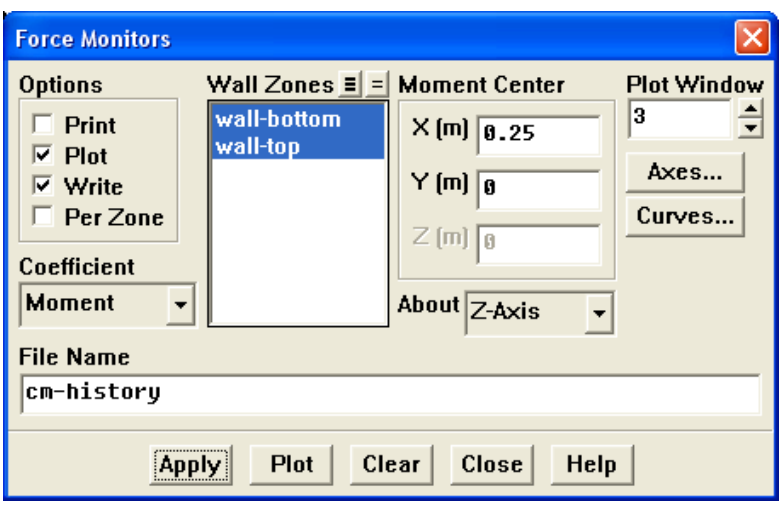
   Solve ⟶ Monitors ⟶ Force...



   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.*

8. In a similar manner, define a force monitor for the moment coefficient, as shown in the following panel. Click Apply after the settings are complete and close the Force Monitors panel.

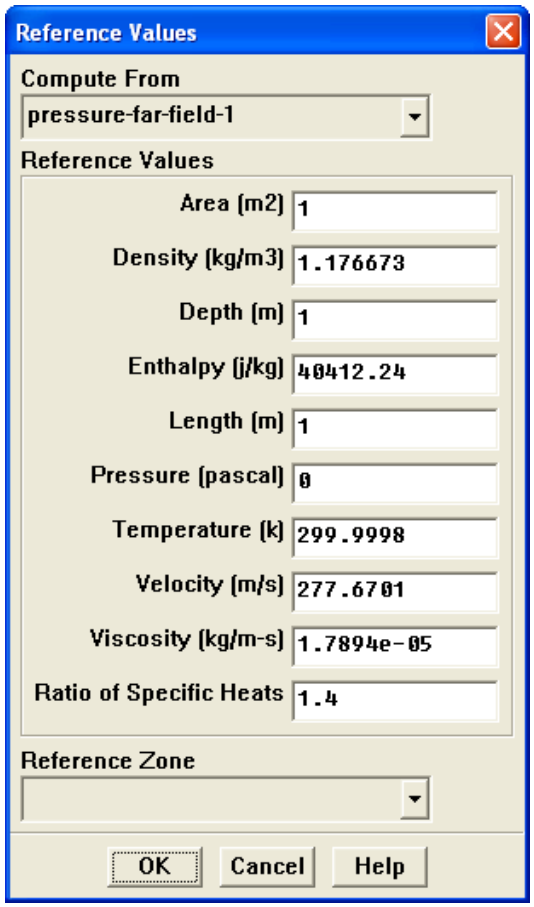
   Solve ⟶ Monitors ⟶ Force...



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

   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.

   Report ⟶Reference Values...



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

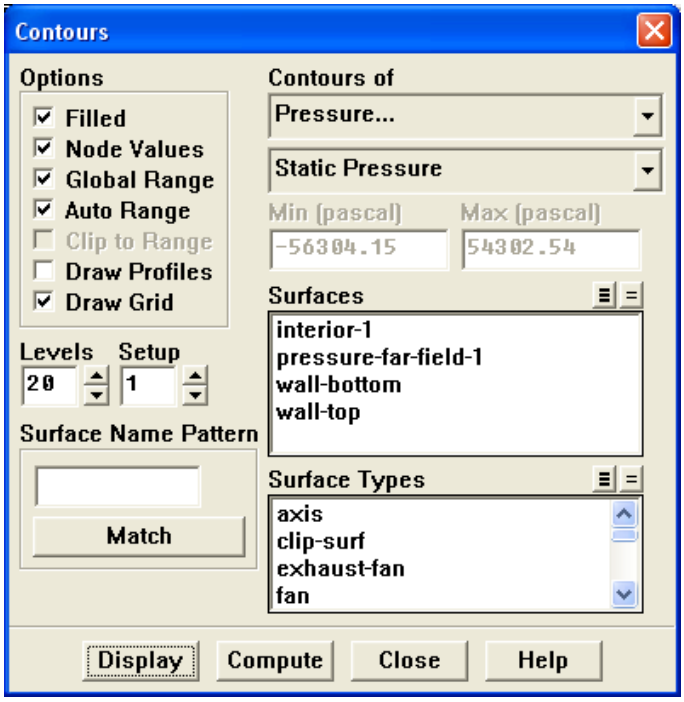
   FLUENT *will update the* Reference Values *based on the boundary conditions at the far-field boundary.*

   (b) Click OK to close the Reference Values panel.

© Fluent Inc. September 21, 2006

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

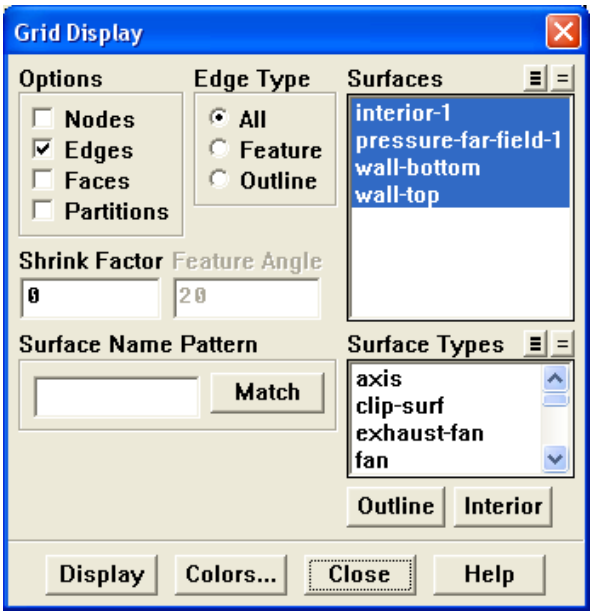
Display ⟶Contours...



(a) Enable Filled in the Options group box.

(b) Enable the Draw Grid option.

The Grid Display *panel will open.*



i. Retain the default settings.

ii. Close the Grid Display panel.

(c) Click Display and close the Contours panel.

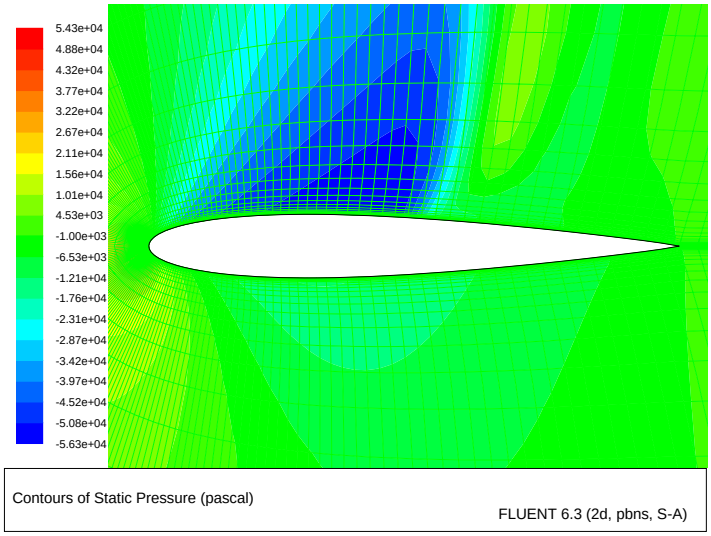


Figure 3.4: Pressure Contours After 50 Iterations

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 deselect the* Draw Grid *option on the* Contours *panel 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.

© Fluent Inc. September 21, 2006

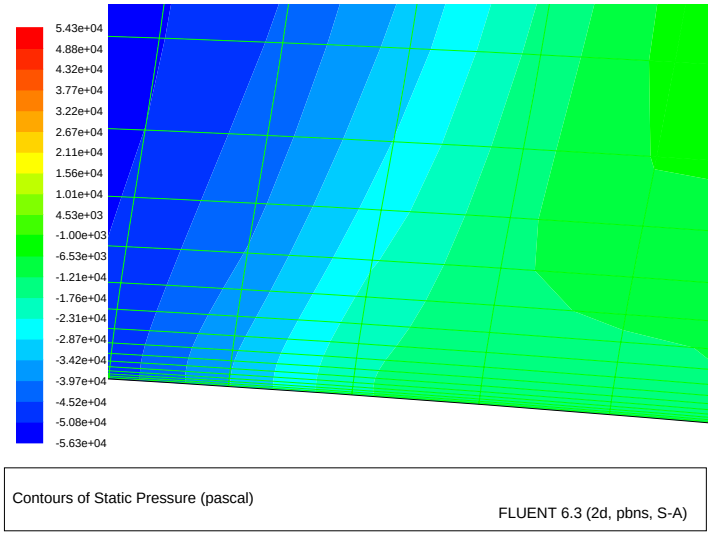
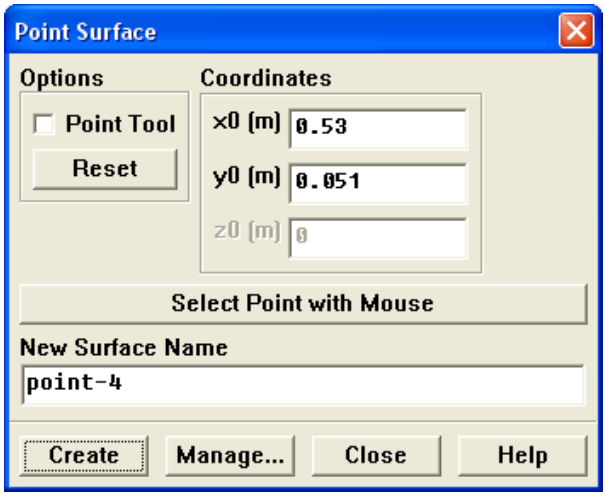


Figure 3.5: Magnified View of Pressure Contours Showing Wall-Adjacent Cells

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.

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

   Surface ⟶ 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 panel.

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 *panel.*

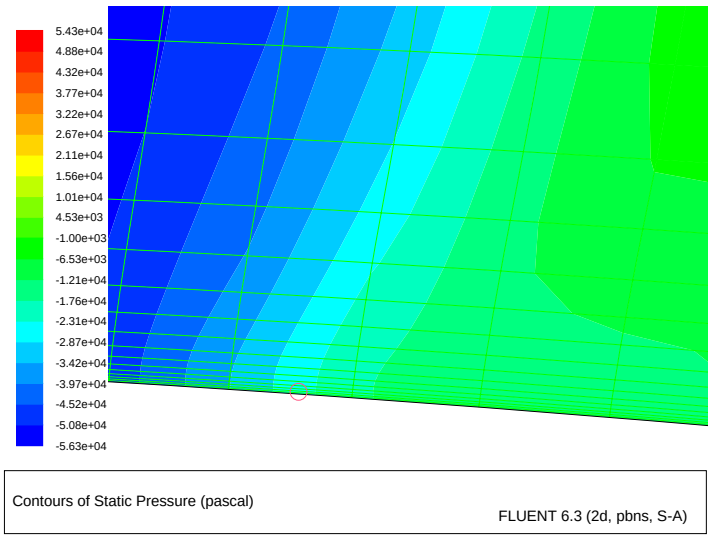


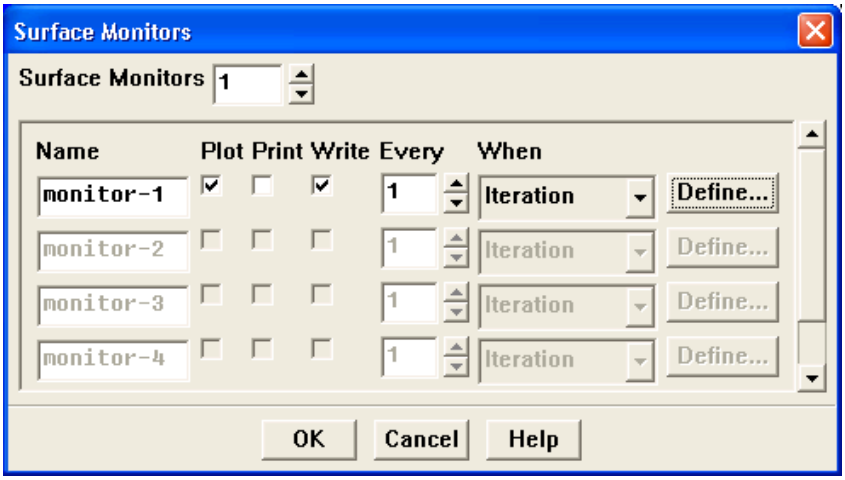
Figure 3.6: Pressure Contours after Creating a Point with the Mouse

© Fluent Inc. September 21, 2006

12. Define a 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, you will 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.

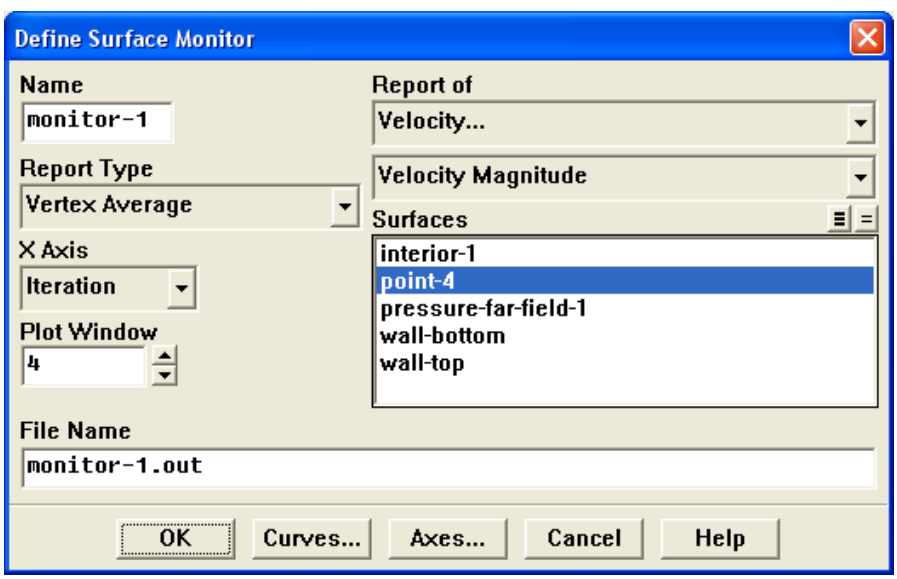
    Solve ⟶ Monitors ⟶ Surface...



    (a) Set the Surface Monitors to 1.

    (b) Enable the Plot and Write options for monitor-1.

    (c) Click the Define... button for monitor-1 to open the Define Surface Monitor panel.



i. Select Vertex Average from the Report Type drop-down list.

Scroll down to find Vertex Average.

ii. Set the Plot Window to 4.

iii. Select Velocity... and Velocity Magnitude from the Report of drop-down list.

iv. Select point-4 in the Surfaces selection list.

v. Click OK to close the Define Surface Monitor panel.

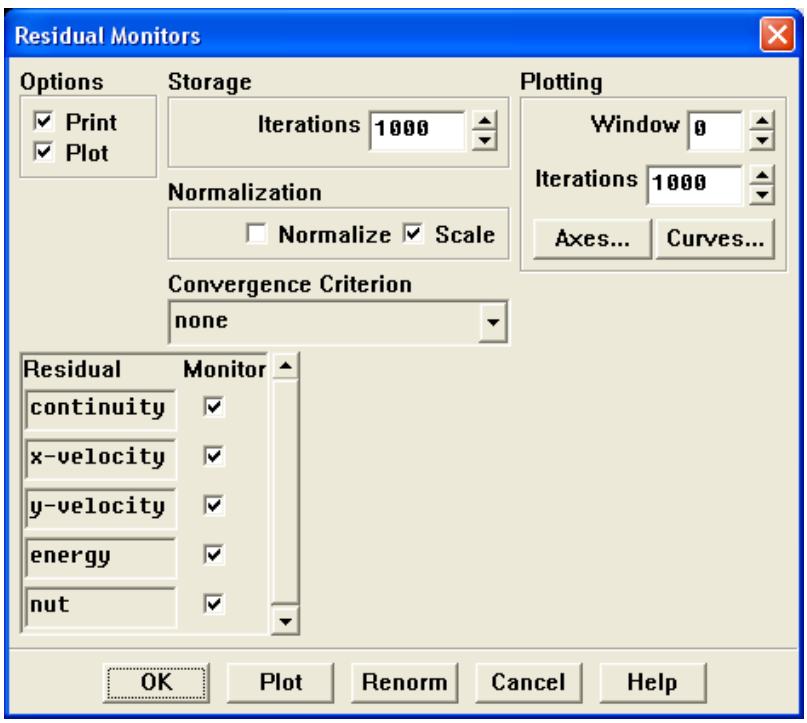
(d) Click OK to close the Surface Monitors panel.

13. Save the case and data files (`airfoil-1.cas` and `airfoil-1.dat`).

File ⟶ Write ⟶ Case & Data...

14. Enable residual plotting during the calculation.

Solve ⟶ Monitors ⟶ Residual...



(a) Enable Plot 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 panel.

© Fluent Inc. September 21, 2006

15. Continue the calculation for 200 more iterations.

    Solve ⟶ Iterate...

    The force monitors (Figures 3.8 and 3.9) show that the case is converged after about 200 iterations.

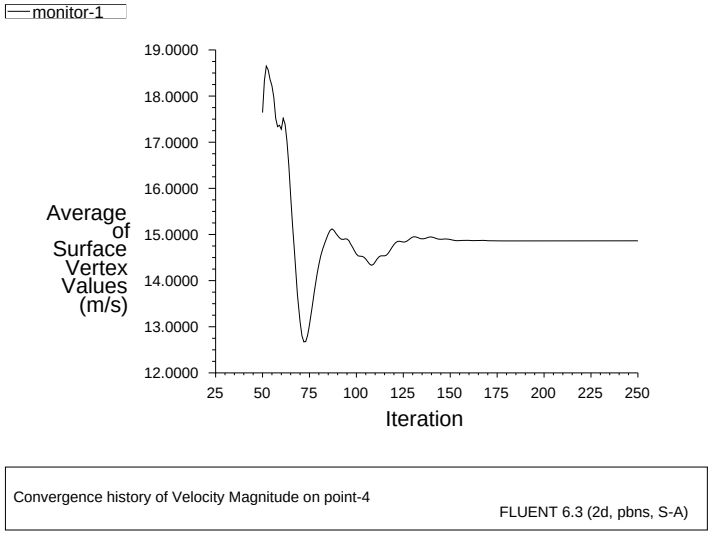


Figure 3.7: Velocity Magnitude History

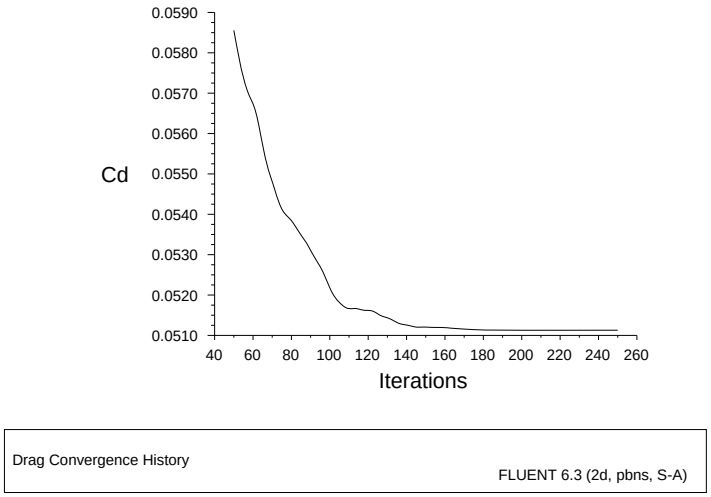


Figure 3.8: Drag Coefficient Convergence History

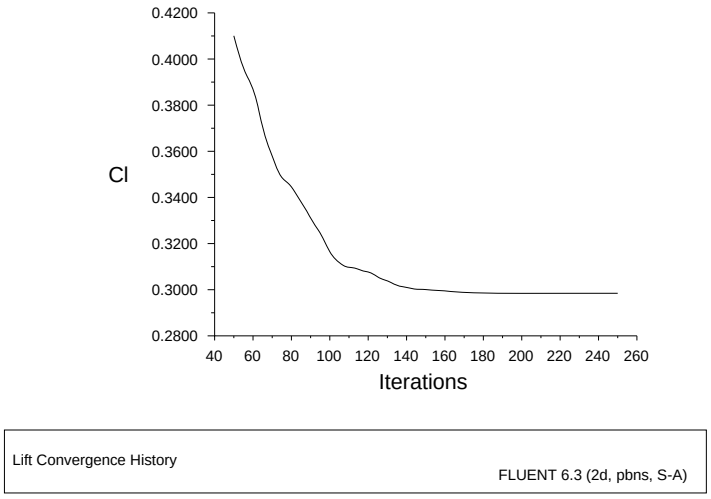


Figure 3.9: Lift Coefficient Convergence History

© Fluent Inc. September 21, 2006

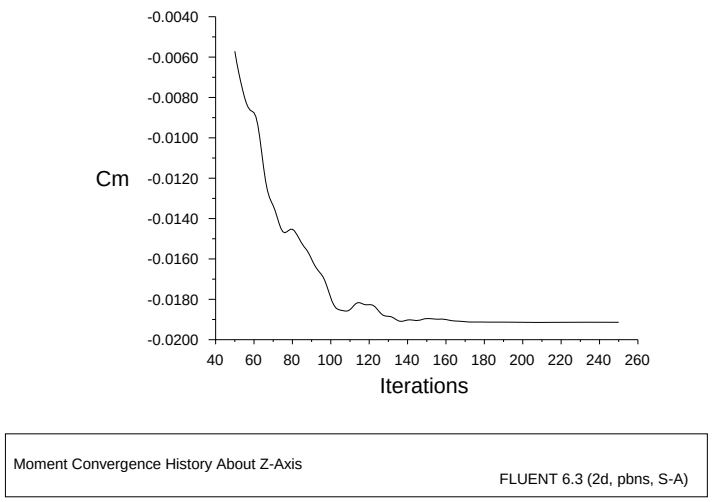


Figure 3.10: Moment Coefficient Convergence History

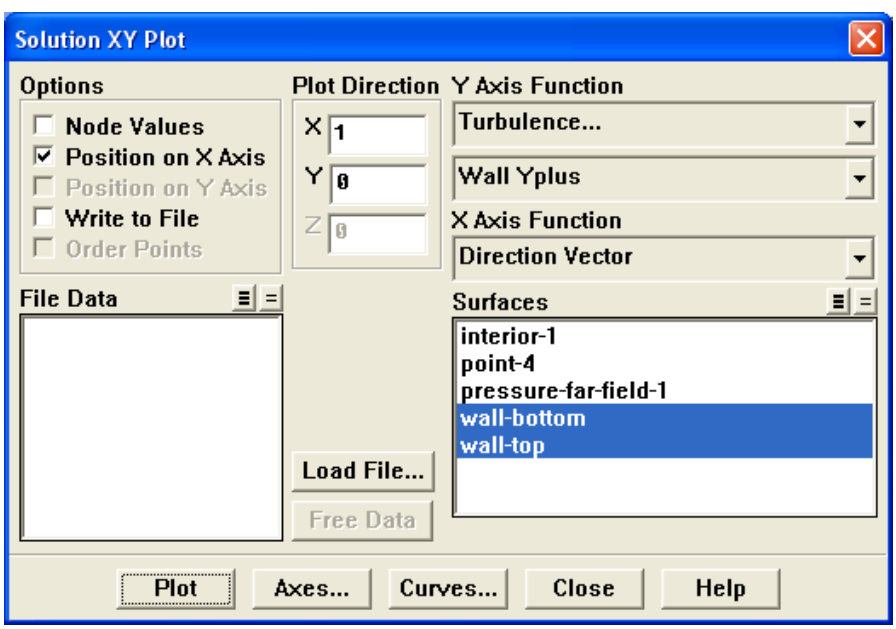
16. Save the case and data files (`airfoil-2.cas` and `airfoil-2.dat`).

    File ⟶ Write ⟶ Case & Data...

Step 7: Postprocessing

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

   Plot ⟶ XY Plot...



(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 from the Surfaces selection list.

(d) Click Plot and close the Solution XY Plot panel.

Note: *The values of y^+ are dependent on the resolution of the grid 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 grid.*

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 grid resolution is acceptable.

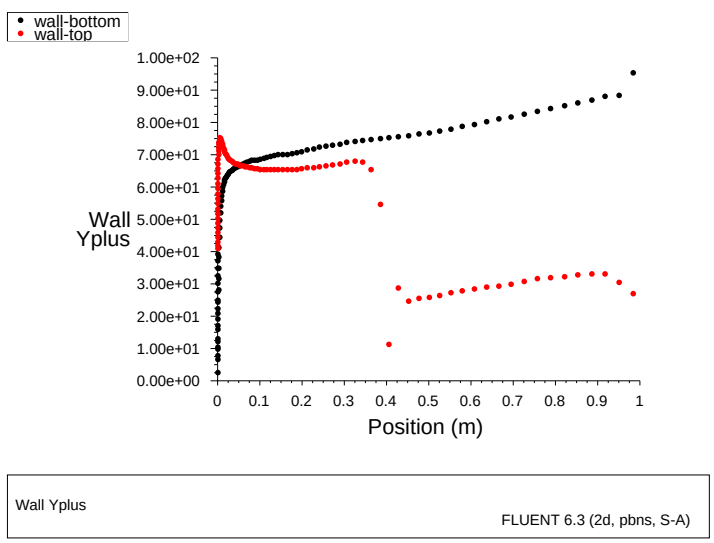


Figure 3.11: XY Plot of y^+ Distribution

© Fluent Inc. September 21, 2006

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

   Display ⟶ Contours...

   (a) Disable the Draw Grid option.

   (b) Select Velocity... and Mach Number from the Contours of drop-down list.

   (c) Click Display and close the Contours panel.

   (d) Zoom in on the region around the airfoil, as shown in Figure 3.12.

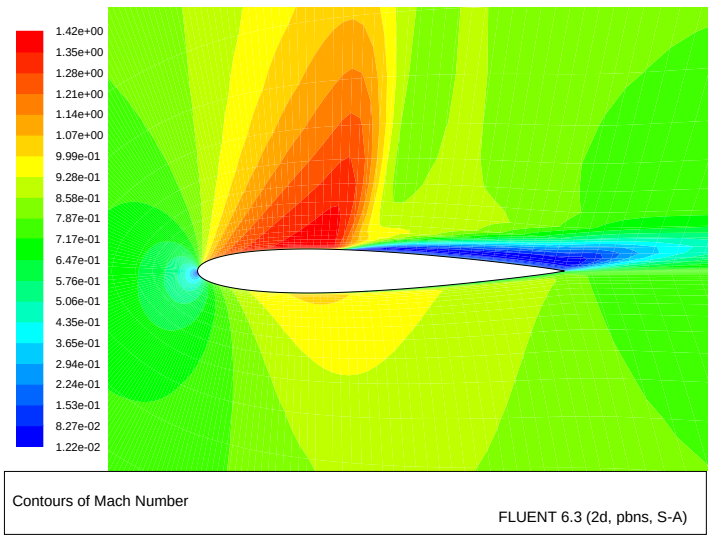


Figure 3.12: Contour Plot of Mach Number

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).

   Plot ⟶ XY Plot...

   (a) Enable the Node Values option.

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

   (c) Click Plot.

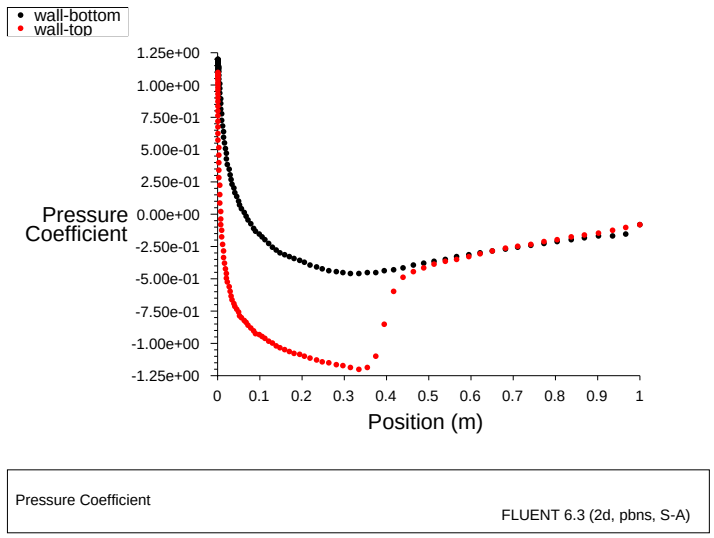


Figure 3.13: XY Plot of Pressure

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).

   Plot ⟶XY Plot...

   (a) Disable the Node Values option.

   (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 panel.

   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.

© Fluent Inc. September 21, 2006

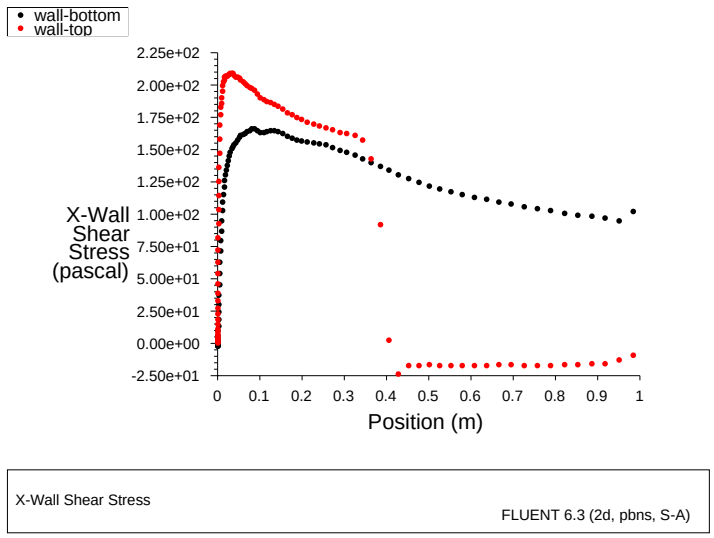


Figure 3.14: XY Plot of x Wall Shear Stress

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

   Display ⟶ Contours...

   (a) Select Velocity... and X Velocity from the Contours of drop-down lists.

   Scroll up to find X Velocity.

   (b) Click Display and close the Contours panel.

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

6. Plot velocity vectors (Figure 3.16).

   Display ⟶ Vectors...

   (a) Enter 15 for Scale.

   (b) Click Display and close the Vectors panel.

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

   Flow reversal is clearly visible in Figure 3.16.

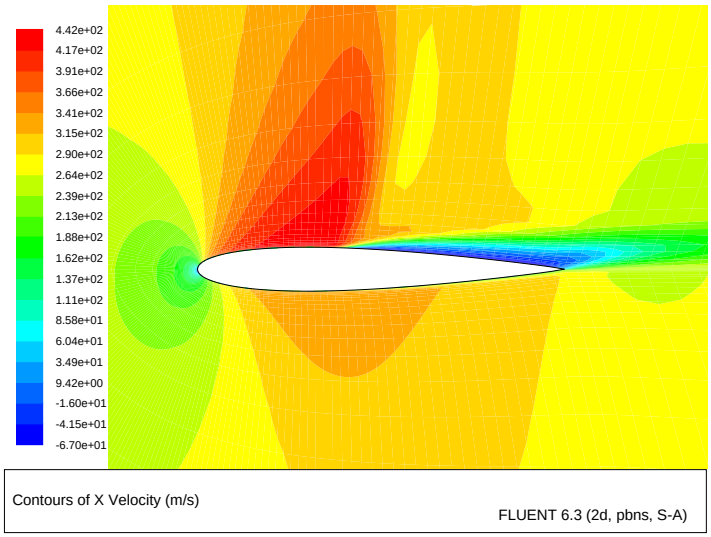


Figure 3.15: Contour Plot of x Component of Velocity

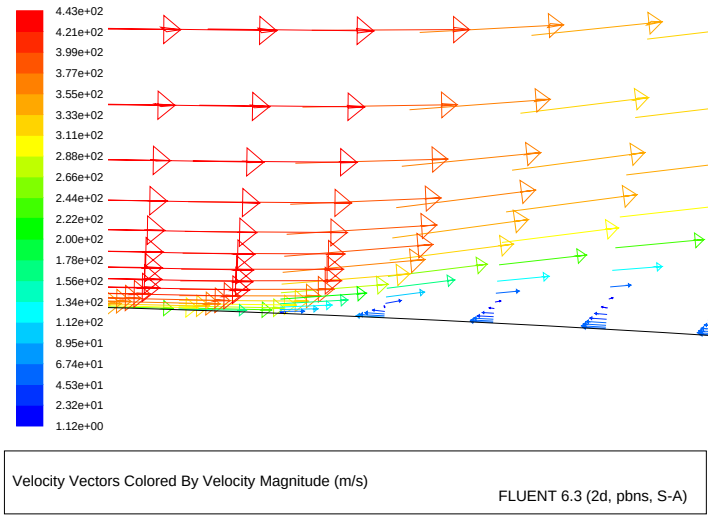


Figure 3.16: Plot of Velocity Vectors Downstream of the Shock

© Fluent Inc. September 21, 2006

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 grid. Grid adaption can also ensure that the solution is independent of the grid. These steps are demonstrated in Tutorial 1.

© Fluent Inc. September 21, 2006