Tutorial 5.   Modeling Compressible Flow over an Airfoil

Introduction

The purpose of this tutorial is to illustrate the setup and solution of an external compressible flow.

This tutorial demonstrates how to do the following:

- Model compressible flow.

- Set boundary conditions for external aerodynamics.

- Use force monitors to judge convergence.

- Check the grid by plotting the distribution of Y^+.

- Adapt the mesh to meet Y^+ requirements.

The study of flow over an airfoil is crucial in the design of airplanes. The correct shape of airfoils has a large impact on the performance of an airplane. There are different flow regimes depending on the free stream conditions. The important parameters in the study of airfoils are the drag and lift forces on the airfoil. Every airfoil performs best in a specific operating range; selection of airfoil depends on the operating conditions.

Prerequisites

This tutorial assumes that you have little experience with FLUENT but are familiar with the interface.

Problem Description

In this tutorial, we consider the flow around a NACA2415 airfoil (Figures 5.1 and 5.2) at an angle of attack a = 10° and a free stream Mach number (M_∞) of 0.4066 and 0.8. The chord length (C) is 1 m.
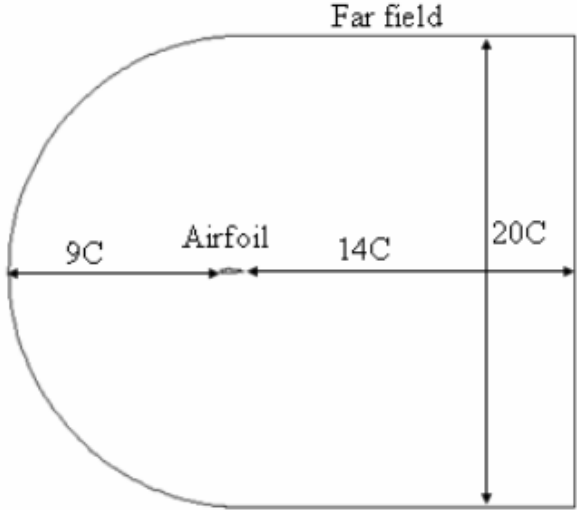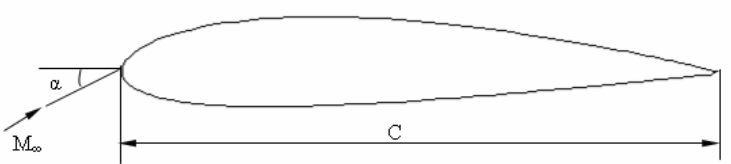
Figure 5.1: Problem Schematic

Figure 5.2: Airfoil Details

Preparation

1. Copy the mesh file, `airfoil.msh` to your working folder.

2. Start the 2D double precision (2ddp) version of FLUENT.

Setup and Solution

Step 1: Grid

1. Read the mesh file, `airfoil.msh`.

   File ⟶ Read ⟶Case...

   FLUENT *will read the mesh file and report the progress in the console window.*
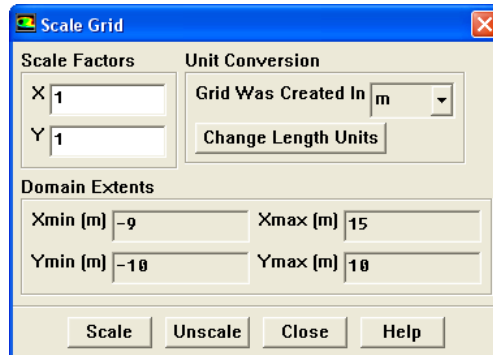
2. Check the grid.

   Grid ⟶Check

   This procedure checks the integrity of the mesh. Make sure the reported minimum volume is a positive number.
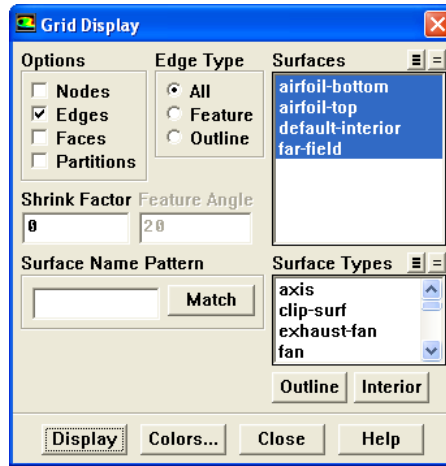
3. Scale the grid.

   Grid ⟶Scale...

Check the domain extents to see if they correspond to the actual physical dimensions. Otherwise the grid has to be scaled with proper units.

4. Display the grid (Figure 5.3).

Display —→Grid...
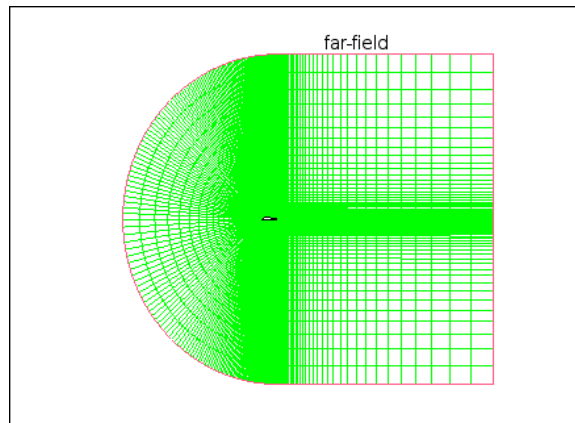

(a) Click Display and close the Grid Display panel.

Figure 5.3: Grid Display

Use the middle mouse button to zoom in on the image so that the mesh near airfoil can be viewed.
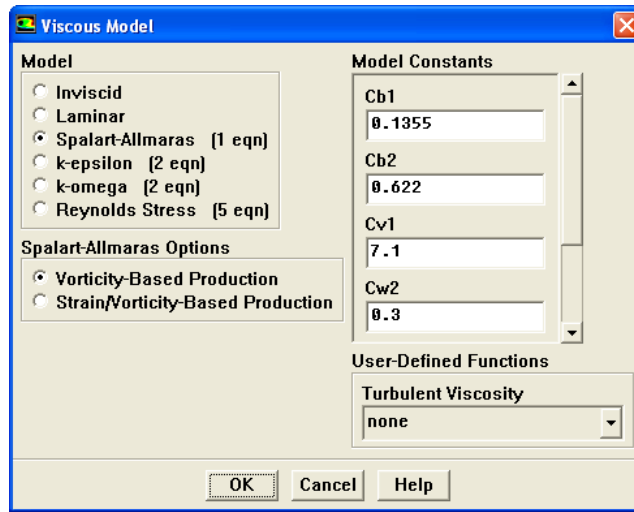
Step 2: Models

1. Set the solver settings.

Define —→ Models —→Solver...

(a) Click OK to retain the default settings.

2. Enable the Spalart-Allmaras turbulence model.

Define ⟶ Models ⟶Viscous...

(a) Enable Spalart-Allmaras (1 eqn) in the Model list.

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

The Spalart-Allamaras model is a simple one-equation model that solves a modeled transport equation for the kinematic eddy (turbulent) viscosity. The Spalart-Allmaras model was designed for aerospace applications involving wall-bounded flows and has shown to give good results for boundary layers subjected to adverse pressure gradients.
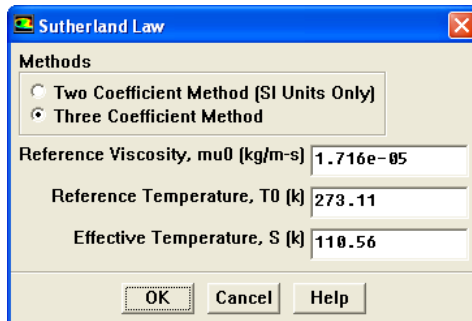
Step 3: Materials

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

Define ⟶Materials...
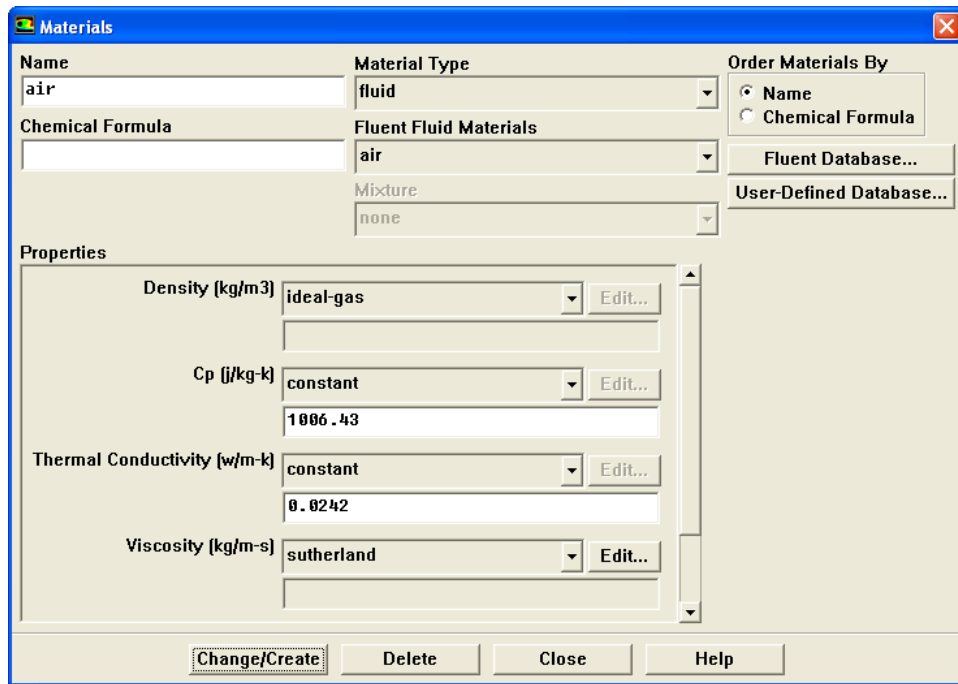
1. Select ideal-gas from the Density drop-down list.

2. Select sutherland from the Viscosity drop-down list.

   (a) Click OK to accept the default Three Coefficient Method and other parameters.

3. Click Change/Create and close the Materials panel.

   Density *and* Viscosity *have been made temperature-dependent. For high-speed com-pressible flows, thermal dependency of the physical properties is recommended. But for simplicity,* Thermal Conductivity *and* Cp *are assumed to be constant in this case.*

Step 4: Operating Conditions
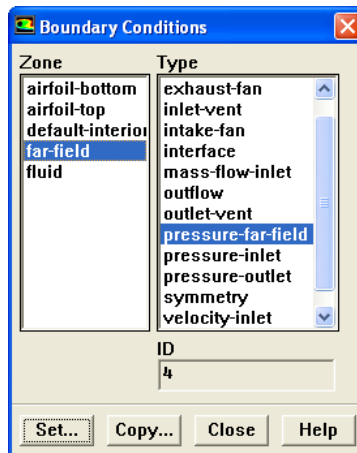
Define ⟶ Operating Conditions...
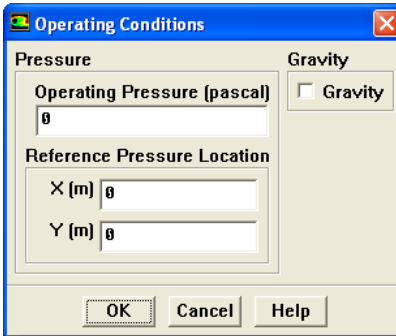
1. Set the Operating Pressure to 0 pascal.

Note: *For compressible flows it is recommended to set the operating pressure to zero, to minimize the errors due to pressure fluctuations.*
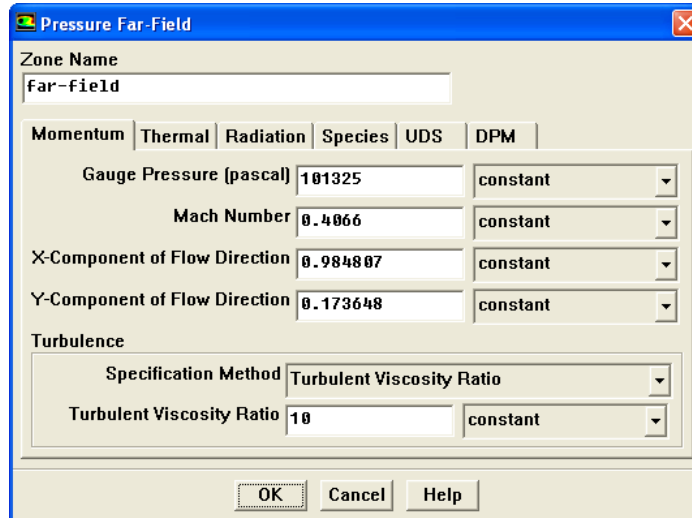
Step 5: Boundary Conditions

Define ⟶Boundary Conditions...

1. Set the boundary condition for pressure-far-field (far-field).

(a) Select far-field from the Zone list.

This Type *will be reported as* pressure-far-field.

(b) Click the Set... button.

The flow Reynolds number is 1e+5 and the velocity can be calculated using the expression: $Re = \dfrac{\rho\, U H}{\mu}$

   i. Enter 101325 pascal for the Gauge Pressure.

   ii. Enter 0.4066 for the Mach Number.

   iii. Enter 0.984807 for X-component of Flow Direction and 0.173648 for Y-component of Flow Direction.

       As the angle of attack is 10°, X component of flow direction $= 1.0 \times cos(10°)$ *and Y component of flow direction* $= 1.0 \times sin(10°)$

   iv. Select Turbulent Viscosity Ratio from the Specification Method drop-down list.

   v. Retain the default value of 10 for Turbulent Viscosity Ratio.

       For external flows, Turbulent Viscosity Ratio *should be set between 1 and 10.*

   vi. Click OK to close the Pressure Far Field panel.
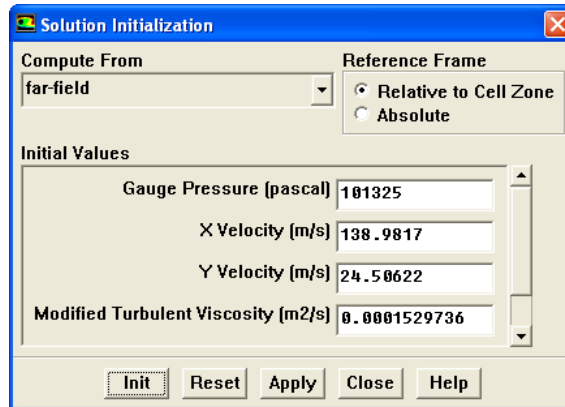
2. Close the Boundary Conditions panel.

Step 6: Solution

1. Retain default solver settings for initial solution...

   Solve ⟶ Controls ⟶ Solution...

2. Initialize the flow.

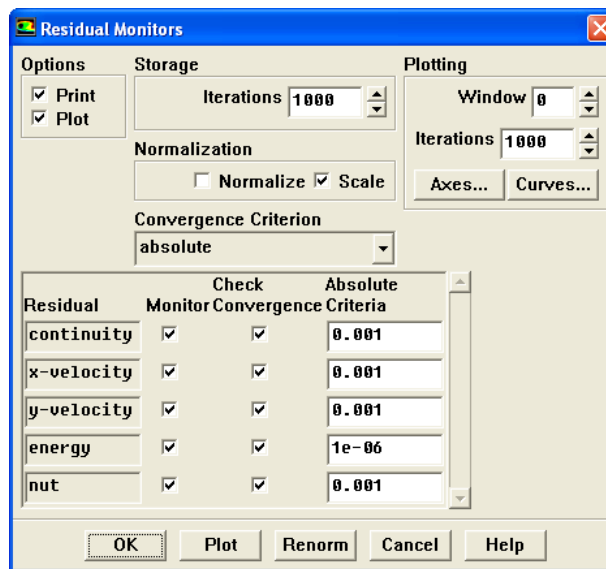   Solve ⟶ Initialize ⟶ Initialize...

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

       For airfoil simulations, initialize the flow field using the pressure-far-field *boundary as it helps in faster convergence.*

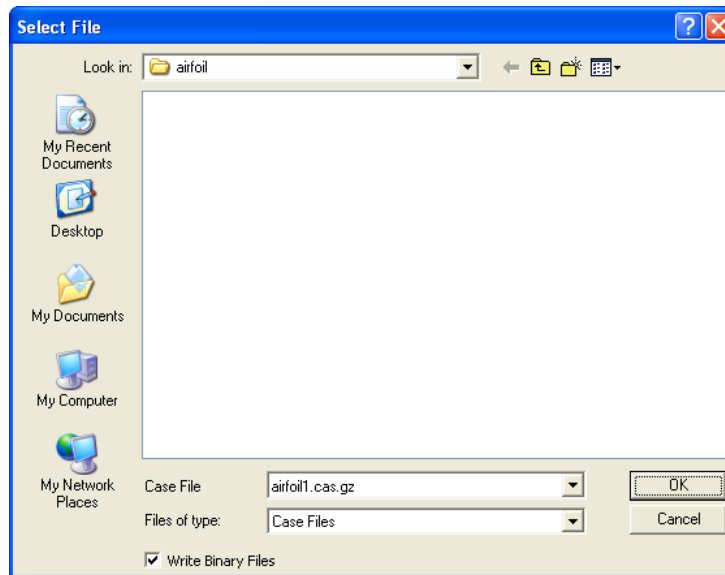   (b) Click Init and close the Solution Initialization panel.

3. Enable the plotting of residuals during the calculation.

   Solve ⟶ Monitors ⟶ Residual...

(a) Enable Plot from the Options list.

(b) Click OK to close the Residual Monitors panel.
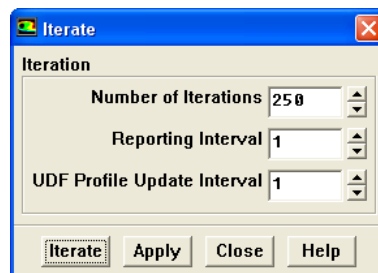
4. Save the case file (`airfoil1.cas.gz`).

File ⟶ Write ⟶ Case...

Retain the default Write Binary Files *option so that you can write a binary file. The* `.gz` *extension will save compressed files on both,* Windows *and* **UNIX** *platforms.*

5. Start the calculation by requesting 250 iterations.

Solve ⟶ Iterate...

(a) Set the Number of Iterations to 250.

(b) Click Iterate.

(c) Close the Iterate panel after the completion of iterations.

Iterate till the solution converges for default convergence criteria. The solution will converge in about 176 iterations. The residuals plot is shown in Figure 5.4.
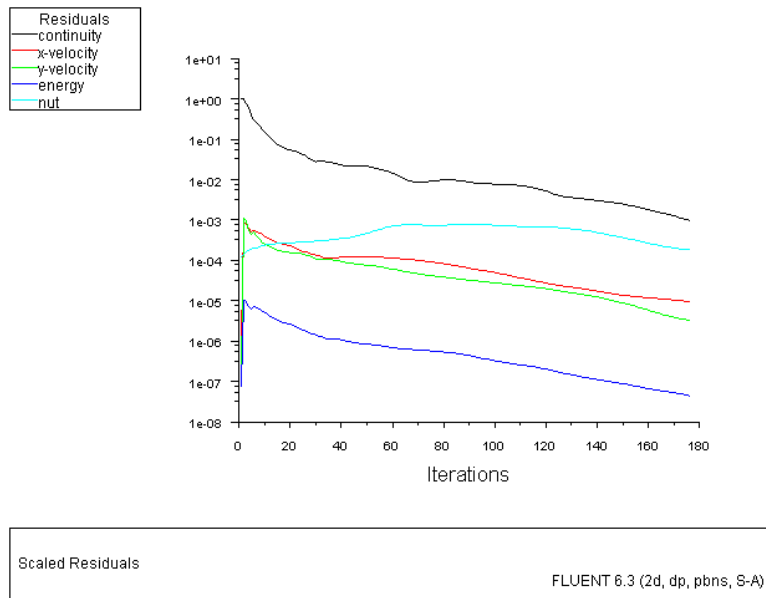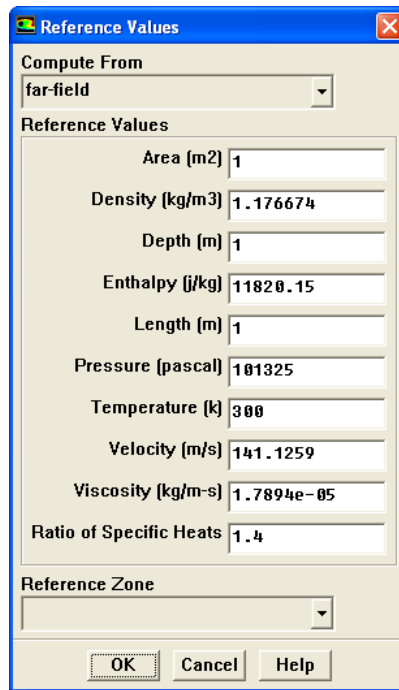
Figure 5.4: Scaled Residuals

6. Set the reference values used to compute the lift, drag and moment coefficients.

   The reference values are used to non-dimensionalize the forces and moments acting on the airfoil.
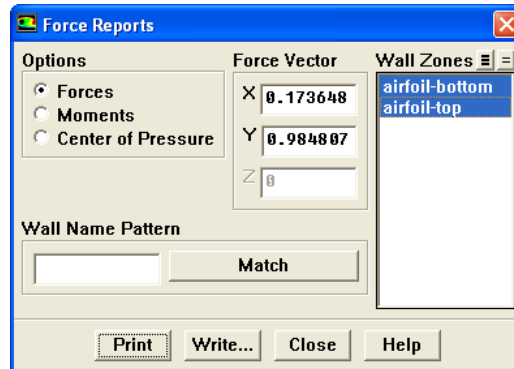
   Report ⟶Reference Values...

(a) Select far-field 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.

7. Check the lift coefficient.

Report ⟶Forces...

(a) Select airfoil-bottom and airfoil-top from the Wall Zones list.

(b) Enter 0.173648 for X and 0.984807 for Y in the Force Vector list.

This is because the lift force is in the direction normal to the flow. For the flow incident at 10°, the normal force components will be $X = 1.0 \times sin(10°)$ and $Y = 1.0 \times cos(10°)$.
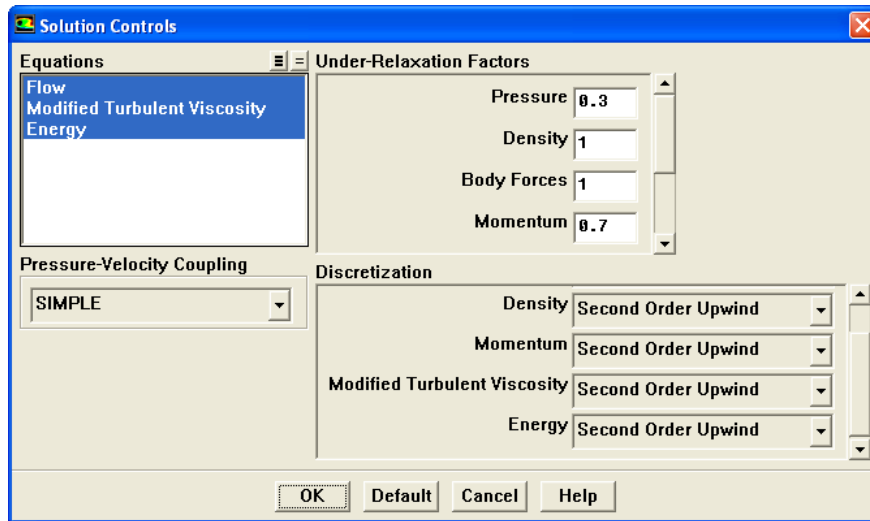
(c) Click Print.

FLUENT *prints the force components and coefficient components in the console window. In the extreme right total coefficient is displayed. It reports a value of 0.9726. The expected lift coefficient for NACA-2415 for current operating conditions is about 1.25* [1]*.*

8. Set the solution controls.

$\boxed{\text{Solve}} \longrightarrow \boxed{\text{Controls}} \longrightarrow \text{Solution...}$

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

(b) Click OK to close the Solution Controls panel.

To predict the correct drag and lift coefficients, it is important to solve with higher order schemes for key parameters of interest. However, from stability perspective, density was selected to start with.
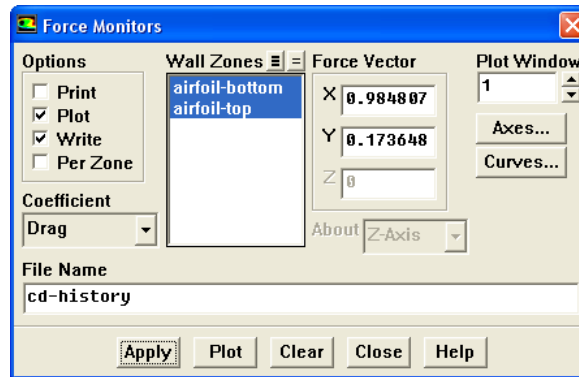
9. Specify Convergence Criteria for all the equations.

$\boxed{\text{Solve}} \longrightarrow \boxed{\text{Monitors}} \longrightarrow \text{Residual...}$
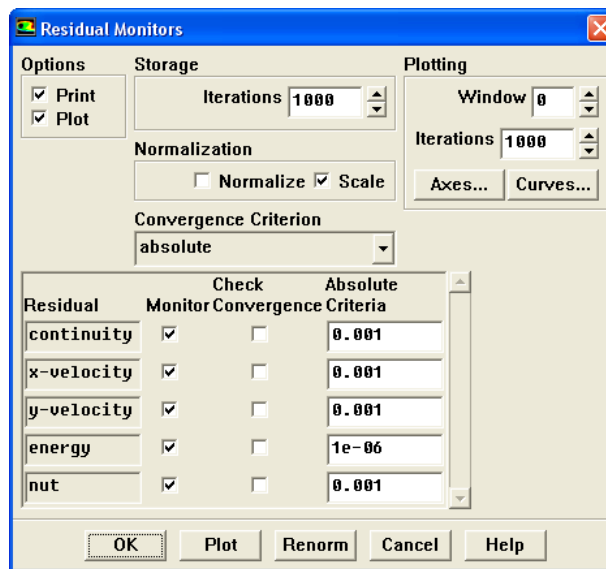
(a) Deselect Check Convergence for all the equations.

(b) Click OK to close the Residual Monitors panel.

10. Set the monitors for lift and drag coefficients.

Solve $\longrightarrow$ Monitors $\longrightarrow$Force...

(a) Select airfoil-bottom and airfoil-top from the Wall Zones list.

(b) Enter 0.984807 for X and 0.173648 for Y in the Force Vector group box.

These magnitudes ensure that the lift and drag coefficients are calculated normal and parallel respectively to the flow.

(c) Select Drag from the Coefficient drop-down list.

(d) Enable Plot and Write in the Options list.

(e) Set Plot Window to 1.

(f) Enter cd-history for the File Name.

(g) Click Apply.

(h) Similarly, set the monitor for Lift.

Note: *For* Lift, *under* Force Vector, *use the values 0.173648 and 0.984807 for* X *and* Y *respectively and set* Plot Window *to 2. The filename is* cl-history.

11. Iterate the solution till the drag and lift coefficients have reached a constant value as shown in Figures 5.5 and 5.6.
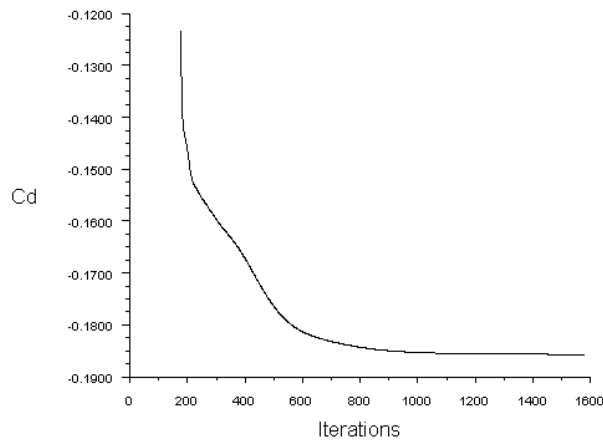
This is observed at about 1400 iterations.

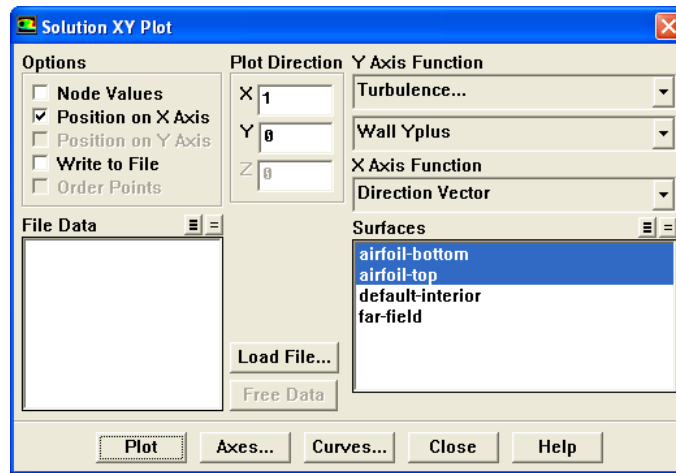12. Check the Y$^+$ values around airfoil (Figure 5.7).

Plot $\longrightarrow$XY...
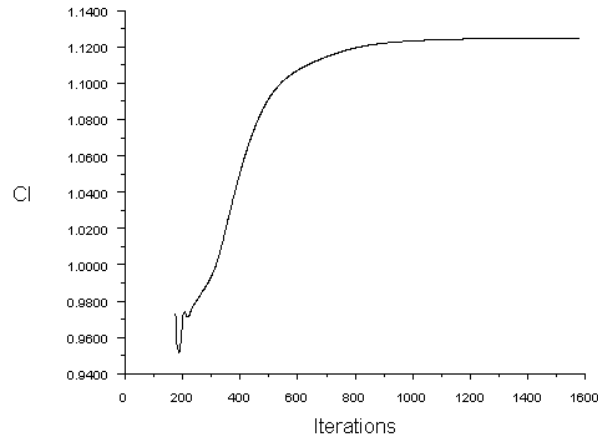
Figure 5.5: Lift Coefficient Plot

Figure 5.6: Drag Coefficient Plot

(a) Select Turbulence... and Wall Yplus from the Y Axis Function lists.

(b) Select airfoil-bottom and airfoil-top in the Surfaces list.

(c) Deselect Node Values.

Wall Yplus *is available only for cell values.*
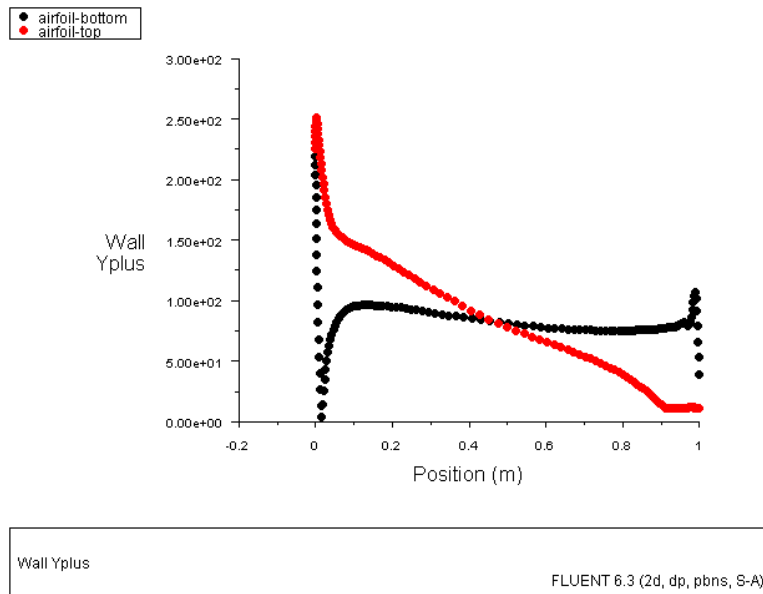
(d) Click Plot.

Figure 5.7: Wall Yplus Plot

The values of Y^+ are dependent on the grid and the Reynolds number of the flow, and are meaningful only in boundary layers. The value of Y^+ in the wall-adjacent cells dictates how wall shear stress is calculated. For Spalart-Allmaras, Y^+ should either be very small (of the order of $Y^+ =1$) or greater than 30.

In this case, the Y^+ values are slightly high and maximum is about 250. The lift coefficient can be better predicted if the Y^+ value is kept in the range of 30-150. In order to meet this condition, refine the cells adjacent to wall using Adapt.

13. Save the case and data files (`airfoil2.cas.gz` and `airfoil2.dat.gz`).

    File ⟶ Write ⟶Case & Data...

14. Adapt a single layer of cells adjacent to airfoil.

    Adapt ⟶Boundary...
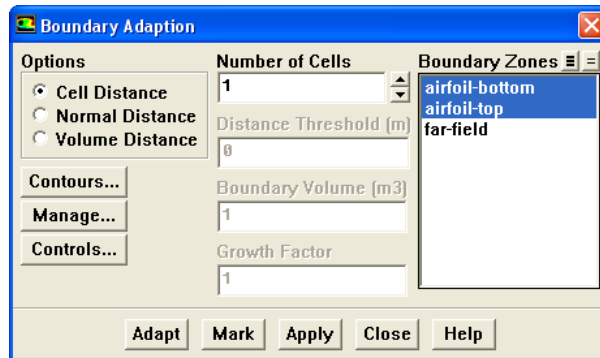
(a) Select airfoil-bottom and airfoil-top in the Boundary Zones list.

(b) Click Mark.

(c) Click Adapt.

(d) Click Yes in the Question dialog box that opens.

(e) Close the Boundary Adaption panel.

   FLUENT *will break each cell into four parts such that the first cell is at half the distance as compared to the original mesh, as shown in Figure 5.8. It can be seen that a non-conformality exists in the mesh after adaption.*

15. Iterate the solution till the monitors for lift and drag coefficients reach a constant value.

   The coefficients reach a constant value at about 3000 iterations.

   The final lift coefficient obtained was about 1.146.

16. Save the case and data files (`airfoil3.cas.gz` and `airfoil3.dat.gz`).
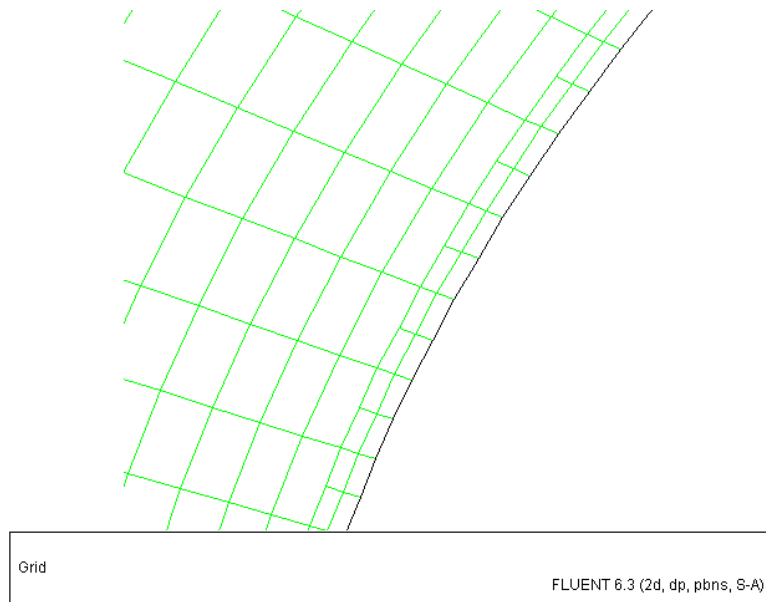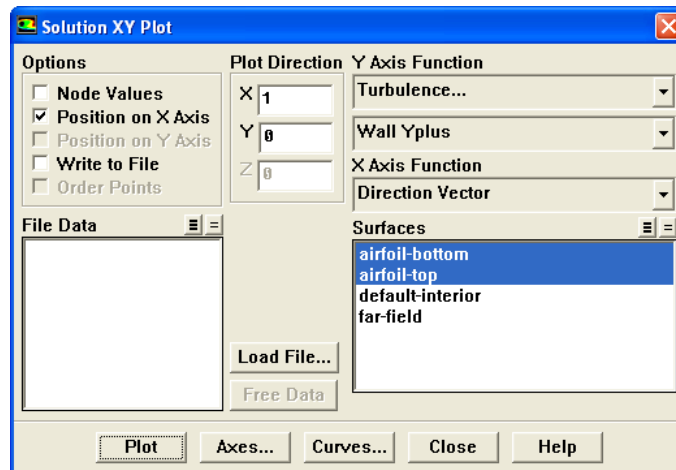
    File ⟶ Write ⟶Case & Data...

Figure 5.8: Adapted Grid

Step 7: Postprocessing

1. Check the Y^+ values (Figure 5.9).

   Plot ⟶XY Plot...

(a) Select Turbulence... and Wall Yplus in the Y Axis Function lists.

(b) Select airfoil-bottom and airfoil-top in the Surfaces list.

(c) Deselect Node Values.

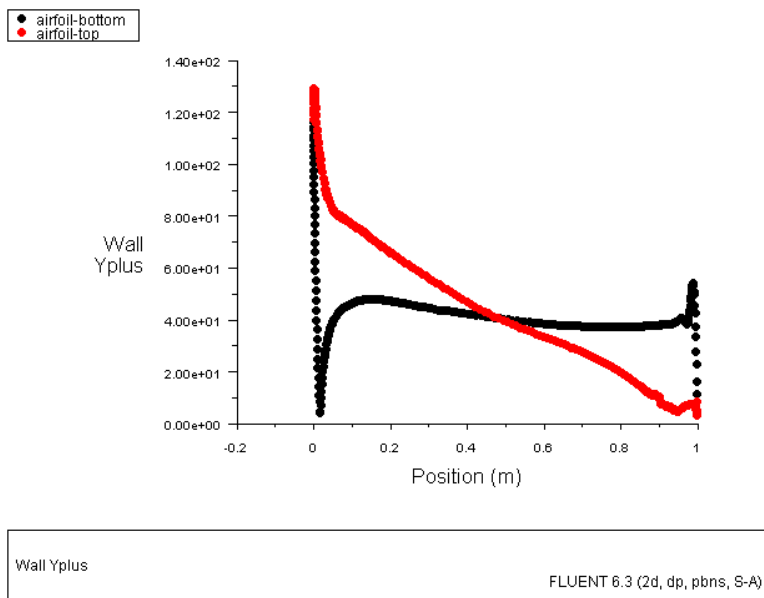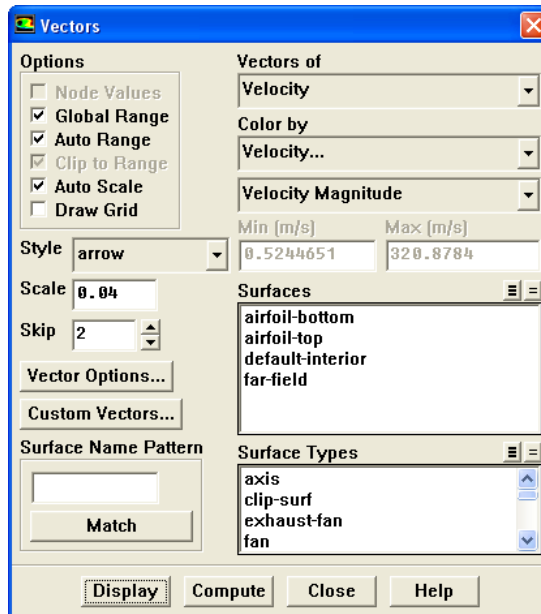(d) Click Plot and close the Solution XY Plot panel.

Figure 5.9: Wall Yplus Plot After Adaption

After adaption, the Y^+ values have reduced and most of the cells are below 130. This shows that the grid resolution is well within the recommended range.

2. Display velocity vectors (Figure 5.10).

$\boxed{\text{Display}}$ $\longrightarrow$ Vectors...

(a) Click Vector Options....

      i. Select Fixed Length.

        Set the Fixed Length *option to display all the vectors with the same length.*

     ii. Click Apply and close the Vector Options panel.

(b) Enter 0.04 for the Scale and 2 for Skip.

Increase Skip *to reduce the number of vectors, so that the display does not appear cluttered due to large number of vectors.*
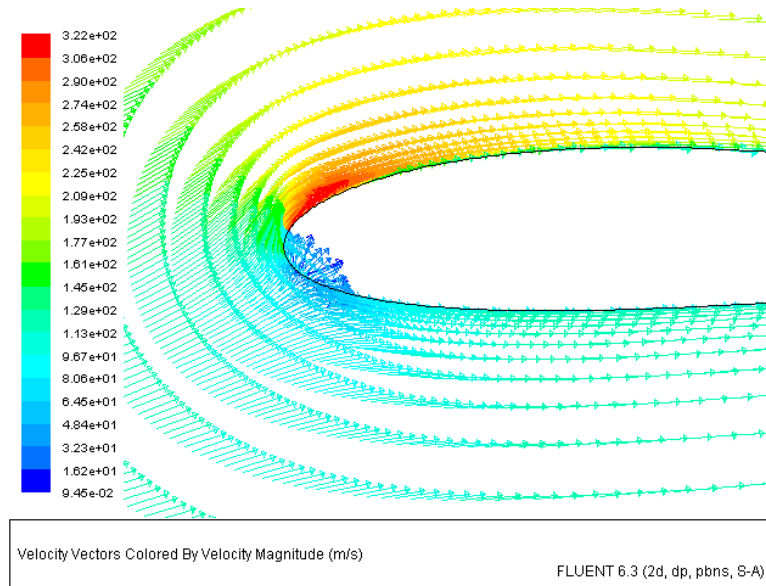
(c) Click Display and close the Vectors panel.

Figure 5.10: Velocity Vectors

3. Display pressure distribution (Figure 5.11).

Display ⟶ Contours...

(a) Select Pressure... and Static Pressure from the Contours of drop-down lists.

(b) Enable Filled in the Options list.

(c) Click Display.

An additional simulation was performed with free stream Mach number of 0.8 at same angle of attack (10). The Mach Number Contour and Velocity Vector for this simulation are shown in Figures 5.12 and 5.13 respectively.
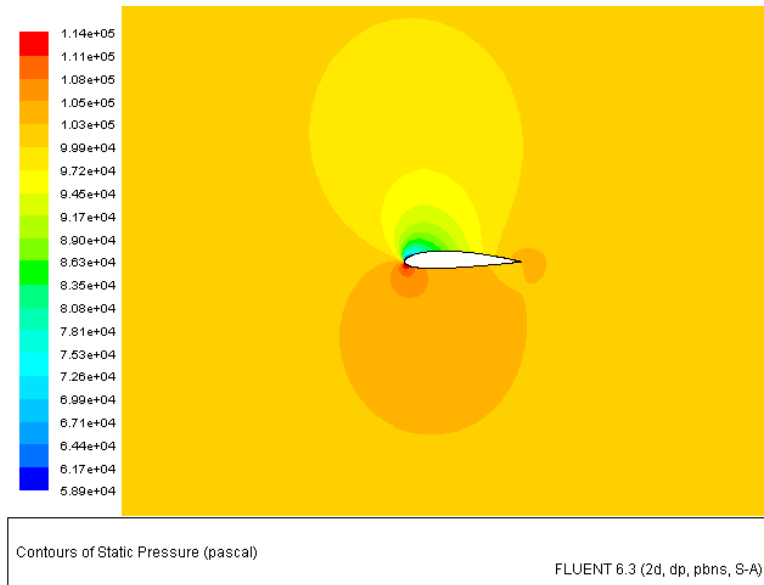
Figure 5.11: Contours of Static Pressure

The upper surface of airfoil clearly shows the shock and flow reversal. Shock reduces the lift coefficient on the airfoil due to the flow reversal and hence is undesirable (Figure 5.13). The lift coefficient with these conditions was 0.7638.