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.

In this tutorial you will learn how to:

- 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.

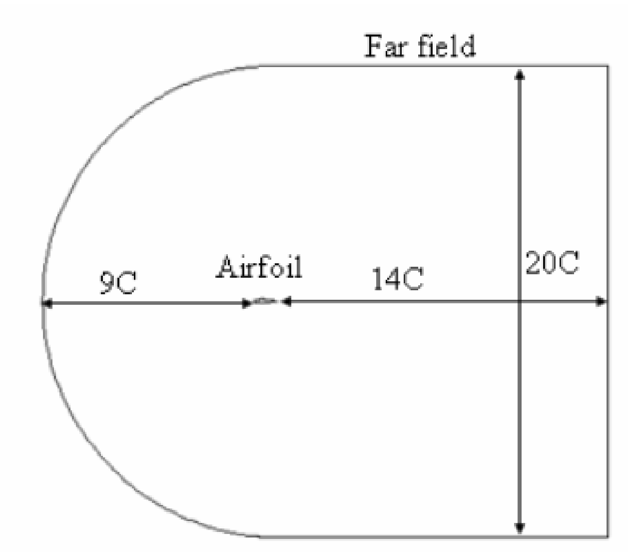


Figure 5.1: Problem Schematic

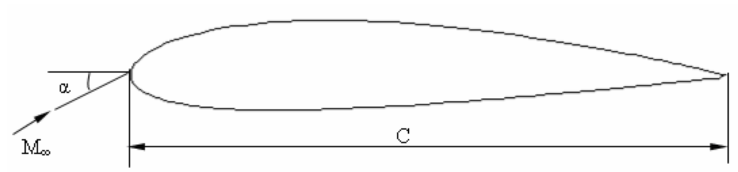


Figure 5.2: Airfoil Details

Preparation

1. Copy the mesh file, `airfoil.msh` to your working directory.
2. Start the 2D double precision solver of FLUENT.

Setup and Solution

Step 1: Grid

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

   File ⟶ Read ⟶ Case...

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

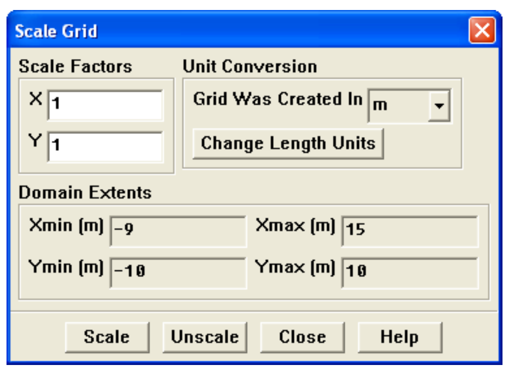
2. Check the grid.

   Grid ⟶ Check

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

3. Check the scale of the grid.

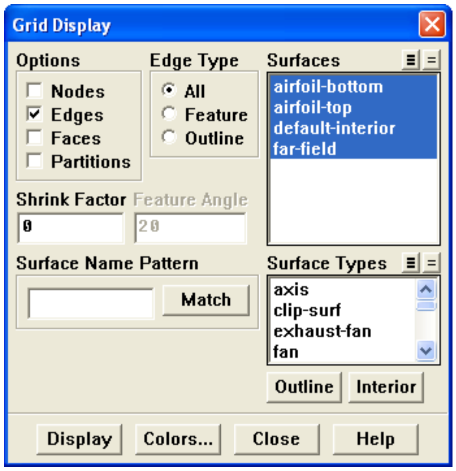
   Grid ⟶ Scale...



   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.

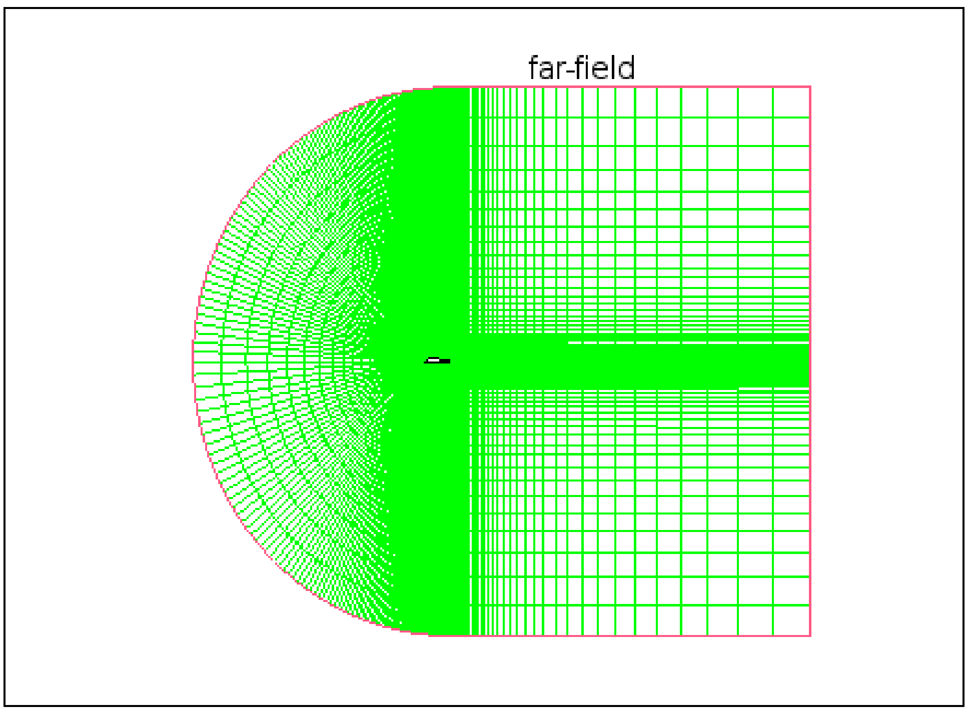


Figure 5.3: Grid Display

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

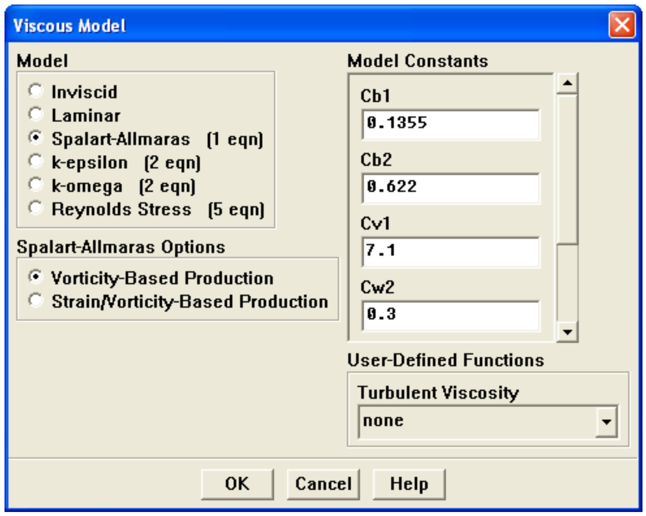
Step 2: Models

1. Retain the default solver settings.

   Define ⟶ Models ⟶ Solver...

2. Enable the Spalart-Allmaras turbulence model.

   Define ⟶ Models ⟶ Viscous...



   (a) Under Model, enable Spalart-Allmaras (1 eqn).

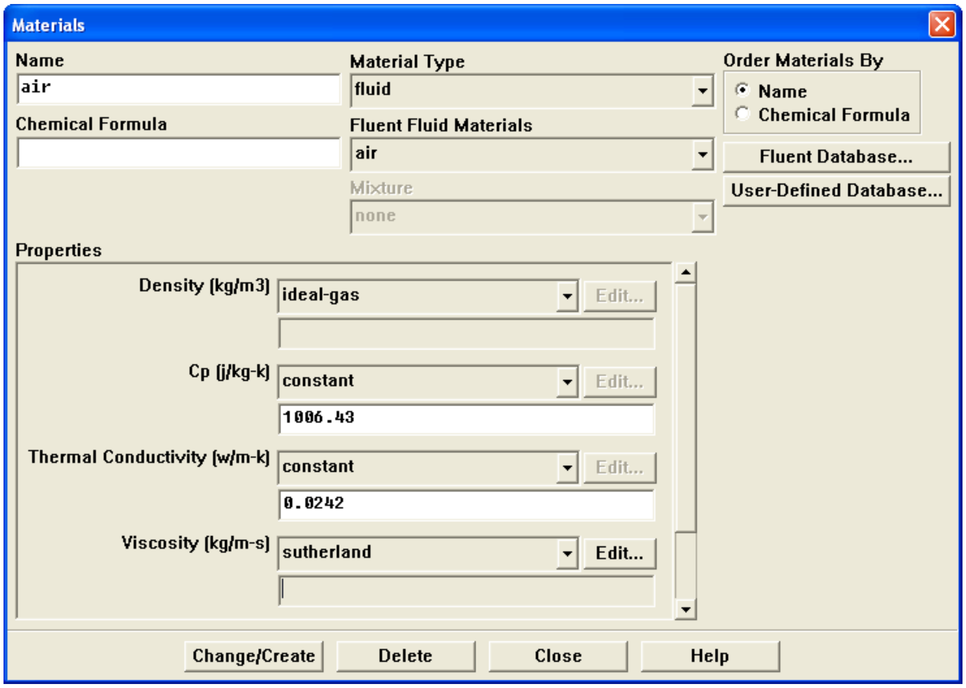
   (b) Click OK.

   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.

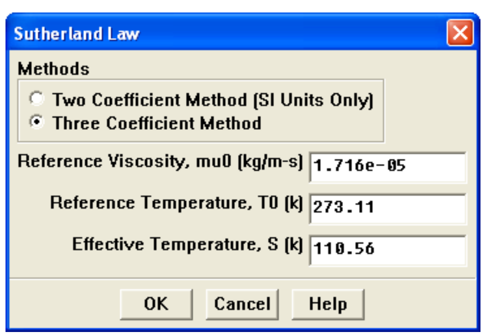
Define ⟶Materials...



1. In the Density drop-down list, select ideal-gas.

2. In the drop-down list for Viscosity, select sutherland.

   The Sutherland Law *panel opens.*



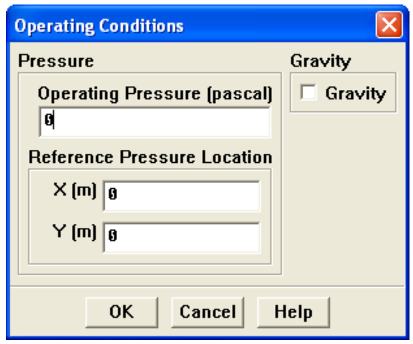
   (a) Click OK to accept the default Three Coefficient Method and other parameters.

3. Click Change/Create and close the panel.

   Density *and* Viscosity *have been made temperature-dependent. For high-speed compressible 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...



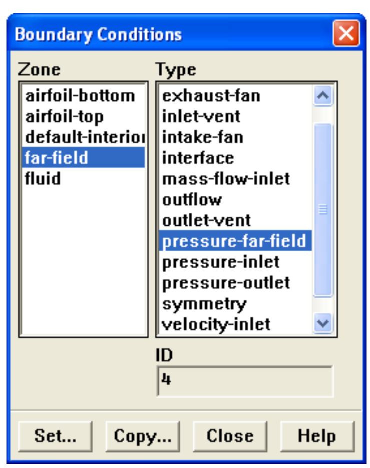
1. Set the Operating Pressure (pascal) to 0.

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).

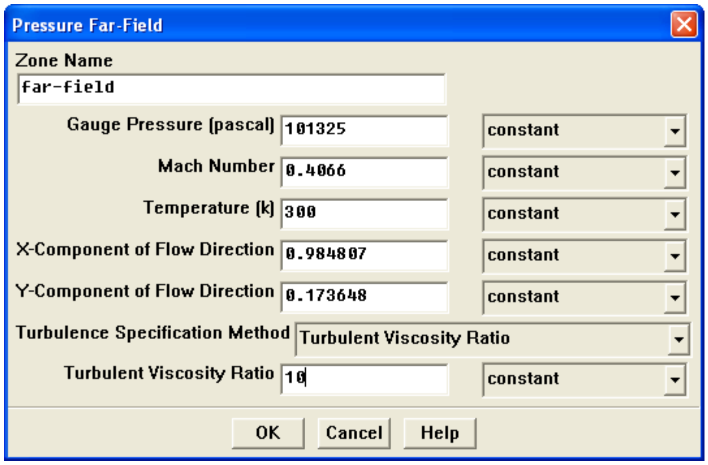


(a) Under Zone, select far-field.

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

(b) Click Set....

The Pressure Far Field *panel opens.*



The flow Reynolds number is 1e+5 and the velocity can be calculated using the expression:

$$Re = \left[\frac{UH\rho}{\mu}\right]$$

(c) Set Gauge Pressure (pascal) as `101325`.

(d) Set Mach Number to `0.4066`.

(e) Set the X-component of Flow Direction and and Y-component of Flow Direction as `0.984807` and `0.173648` respectively.

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

(f) Select Turbulent Viscosity Specification Method as Turbulent Viscosity Ratio.

(g) Retain the default value of `10` for Turbulent Viscosity Ratio.

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

(h) Click OK.

2. Close the Boundary Conditions panel.

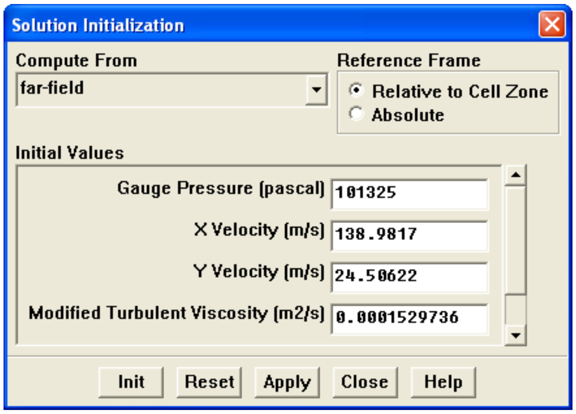
Step 6: Solution

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

   Solve ⟶ Controls ⟶ Solution...

2. Initialize the flow.

   Solve ⟶ Initialize ⟶ Initialize...



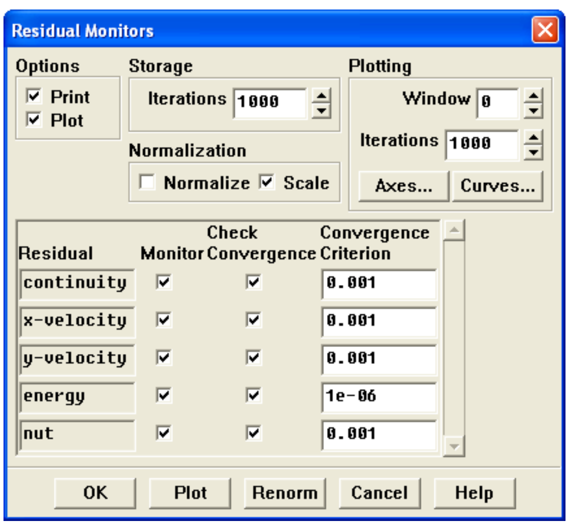
   (a) Under Compute From, select far-field.

   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 panel.

3. Enable the plotting of residuals during the calculation.

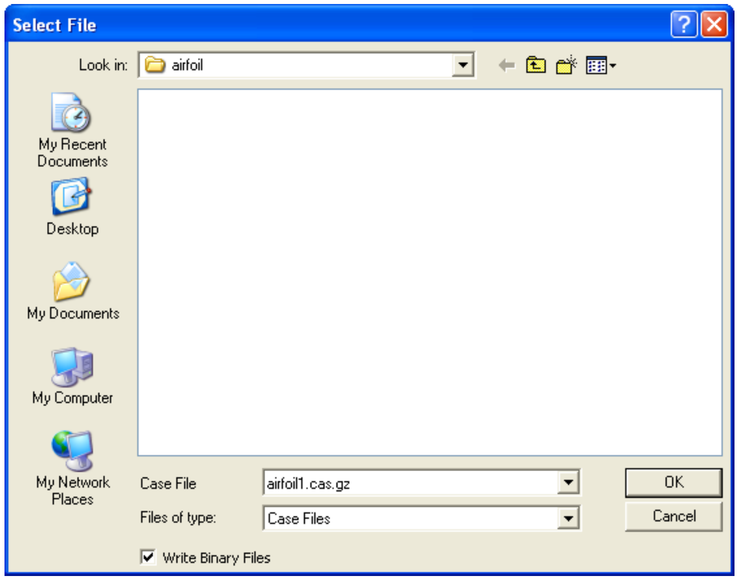
   Solve ⟶ Monitors ⟶ Residuals...



   (a) Under Options, enable Plot and click OK.

4. Save the case file (`airfoil1.cas.gz`).

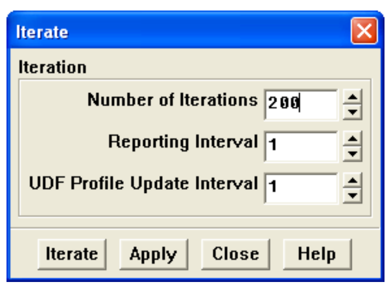
   File →Write →Case...



   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 200 iterations.

   Solve →Iterate...



   (a) Set Number of Iterations to 200.

   (b) Click Iterate.

   Iterate till the solution converges for default convergence criteria. The solution will converge in about 145 iterations. The residuals plot is shown in Figure 5.4.

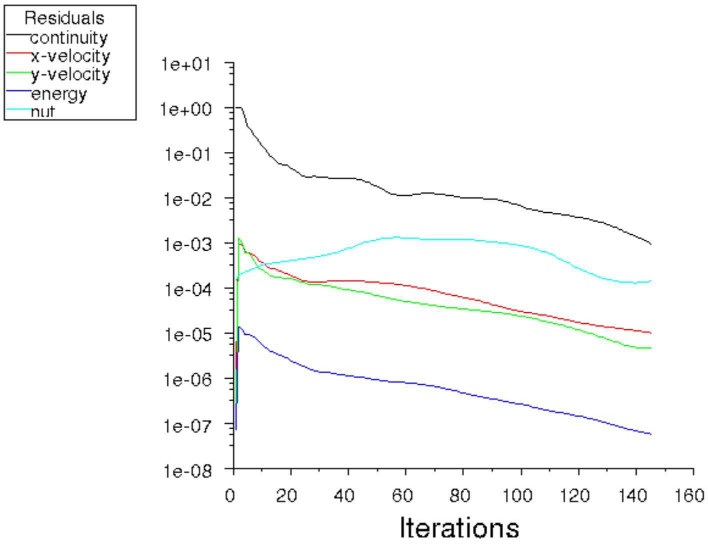
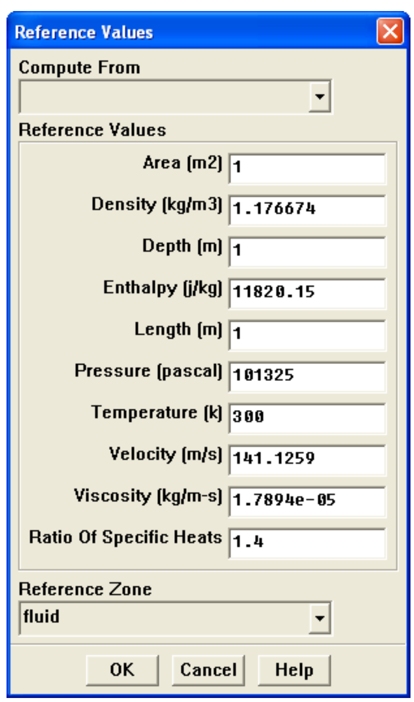


Figure 5.4: Scaled Residuals

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...



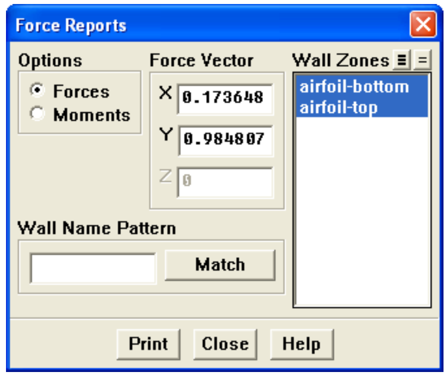
(a) In the Compute From drop-down list, select far-field.

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

(b) Click OK.

7. Check the lift coefficient.

Report ⟶Forces...



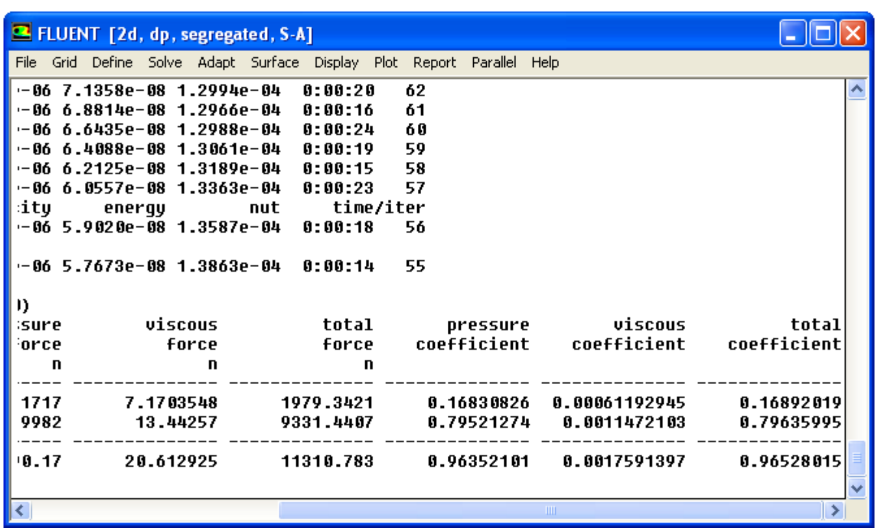
(a) Under Wall Zones, select airfoil-bottom and airfoil-top.

(b) Under Force Vector, set X and Y as 0.173648 and 0.984807 respectively.

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* $Y = 1.0 \times cos(10°)$.

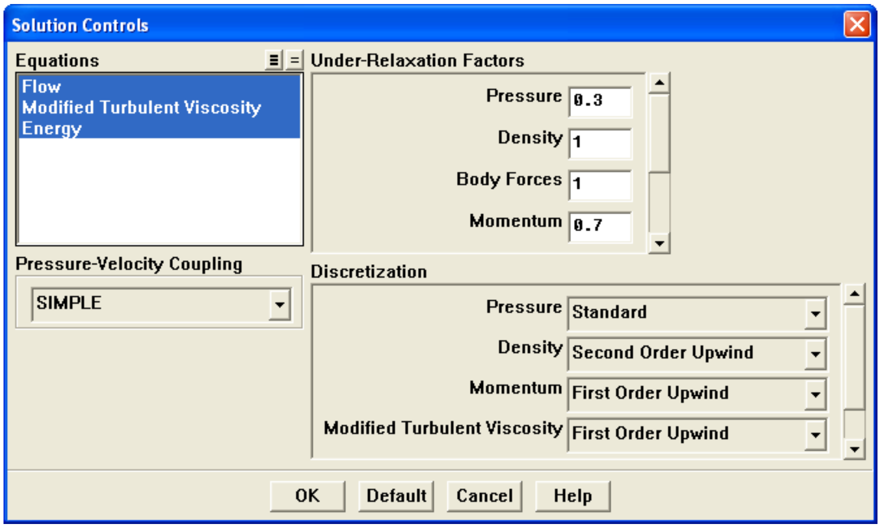
(c) Click Print.

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 0.9652. The expected lift coefficient for NACA-2415 for current operating conditions is about 1.25 [1].*



8. Set the solution controls.

   Solve ⟶ Controls ⟶ Solution...

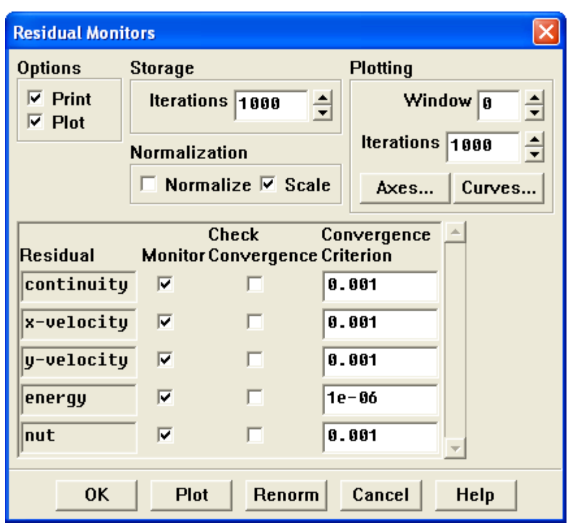


   (a) Select Second Order Upwind for Density

   (b) Click OK.

   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.

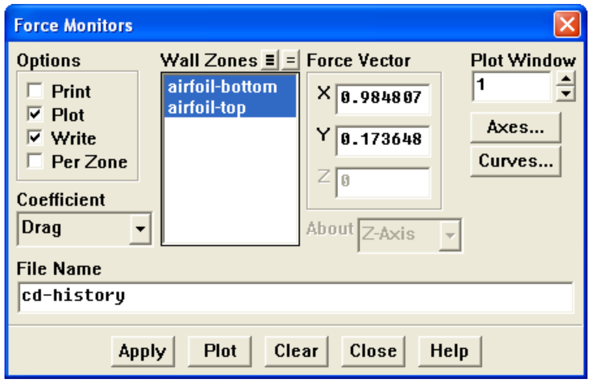


   (a) Deselect Check Convergence for all the equations.

   (b) Click OK.

10. Set the monitors for lift and drag coefficients.

    Solve ⟶ Monitors ⟶Force...



    (a) Under Wall Zones, select airfoil-bottom and airfoil-top.

    (b) Set Plot Window to 1.

    (c) Under Force Vector, enter 0.984807 and 0.173648 for X and Y respectively.

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

    (d) Under Coefficient, select Drag from the drop-down list.

    (e) Under Options, select Plot to enable plotting of the drag coefficient.

    (f) Under Options, select Write to save the monitor history to a file.

    (g) Under File Name, enter `cd-history`.

    (h) Click Apply.

    (i) Similarly, set the monitor for Lift.

    Note: *For* Lift, *under* Force Vector, *use the values 0.173648 and 0.984897 for* X *and* Y *respectively and set* Plot Window *to 2. The filename is* `cl-history`.

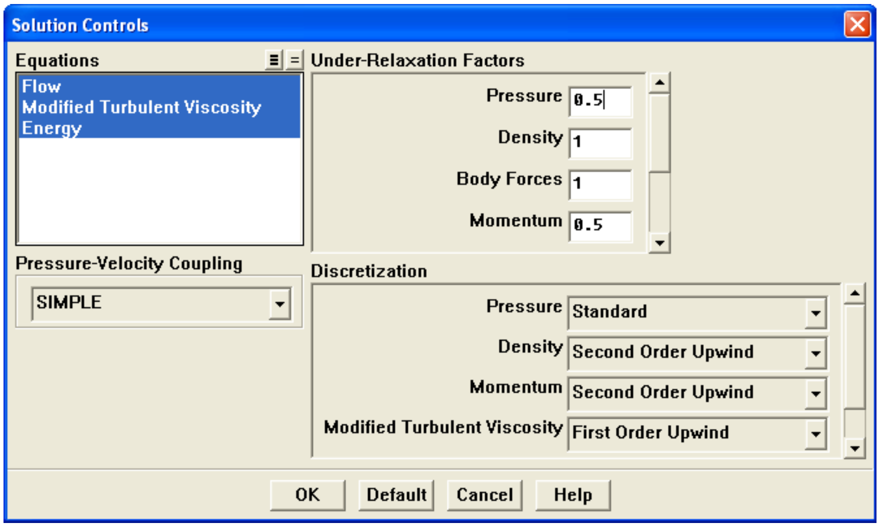
11. Run the solution till the drag and lift monitors reach constant values.

    This is observed at about 700 iterations.

    Second order schemes can then be enabled for Momentum equation.

12. Set Solution controls.

    Solve ⟶ Controls ⟶ Solution...



    (a) Select Second Order Upwind for Momentum.

    (b) Under Under-Relaxation Factors, set `0.5` for both Pressure and Momentum .

    (c) Click OK.

13. Iterate the solution till the drag and lift coefficients have reached a constant value as shown in Figures 5.5 and 5.6.

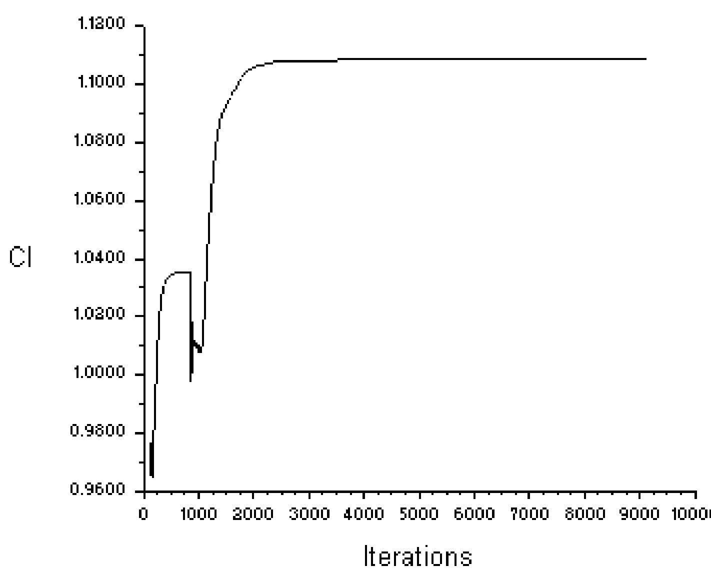


Figure 5.5: Lift Coefficient Plot

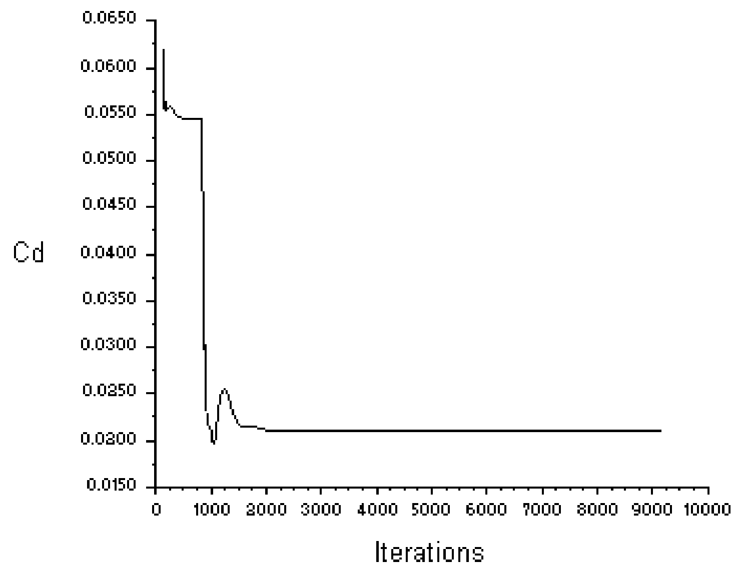
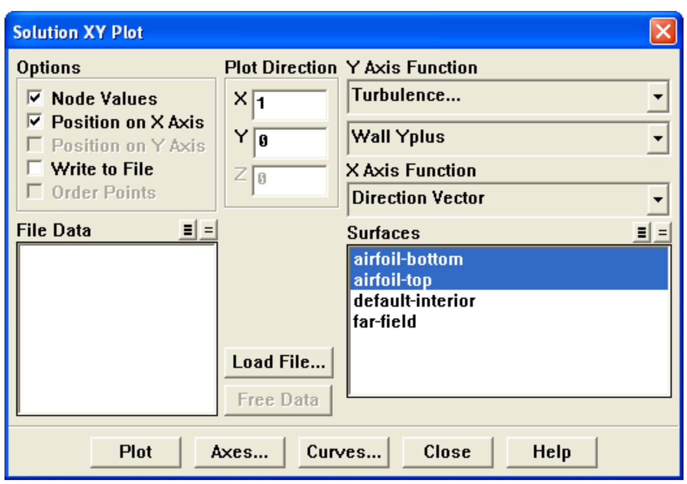


Figure 5.6: Drag Coefficient Plot

The lift coefficient reported in **`cl-history`** *is 1.1083, which is still slightly deviating from the expected value.*

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

    Plot ⟶XY...



(a) Under Y Axis Function, select Turbulence... and Wall Yplus.

(b) In the Surfaces list, select airfoil-bottom and airfoil-top.

(c) Deselect Node Values.

Wall Yplus *is available only for cell values.*

(d) Click Plot.

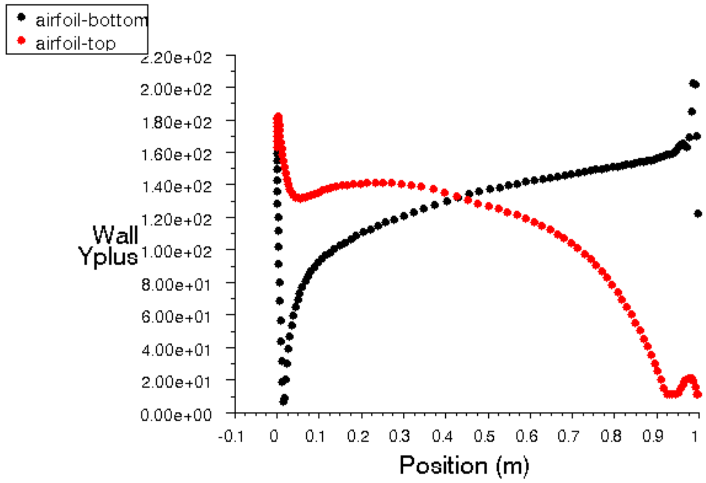


Figure 5.7: Wall Yplus Plot

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.

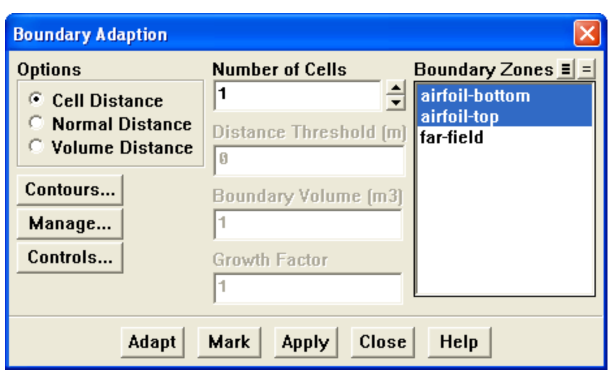
In this case, the Y^+ values are slightly high and maximum is about 180. The lift coefficient can be better predicted if the Y^+ value is kept in the range of 30-100. In order to meet this condition, refine the cells adjacent to wall using Adapt.

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

    File ⟶ Write ⟶ Case & Data...

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

    Adapt ⟶ Boundary...



(a) Under Boundary Zones, select airfoil-bottom and airfoil-top.

(b) Click Mark.

FLUENT *will print the number of cells marked for refinement in console window*

(c) Click Adapt.

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

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.*

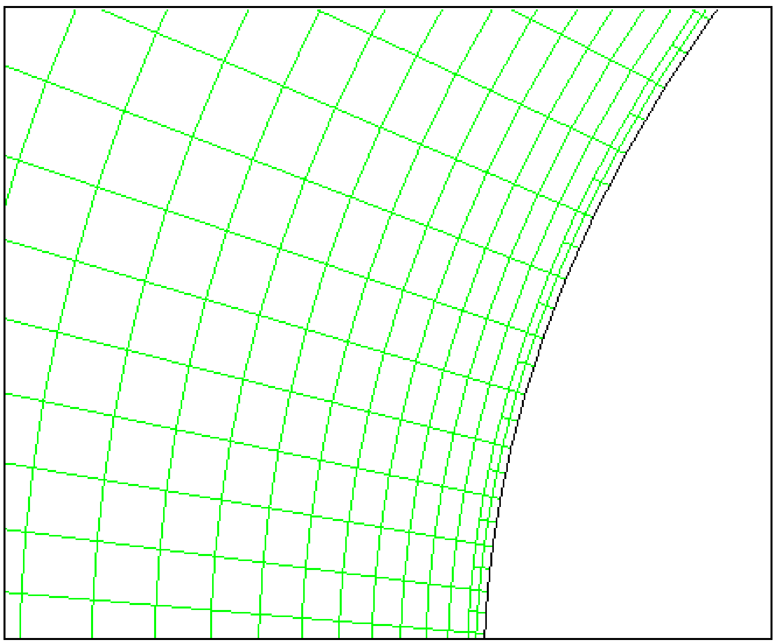


Figure 5.8: Adapted Grid

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

    The coefficients reach a constant value at about 10000 iterations.

    The final lift coefficient obtained was about 1.1225.

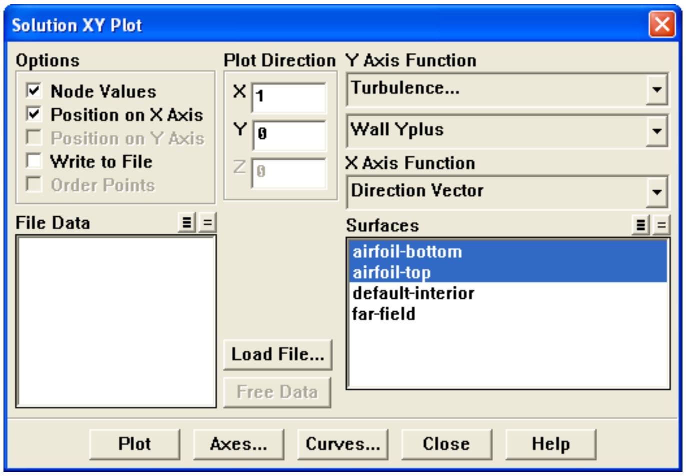
18. Save the case and data files (`airfoil3.cas.gz` and `airfoil3.dat.gz`).

    File ⟶ Write ⟶Case & Data...

Step 7: Postprocessing

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

   Plot →XY Plot...



   (a) Under Y Axis Function, select Turbulence... and Wall Yplus.

   (b) In the Surfaces list, select airfoil-bottom and airfoil-top.

   (c) Deselect Node Values.

   (d) Click Plot.

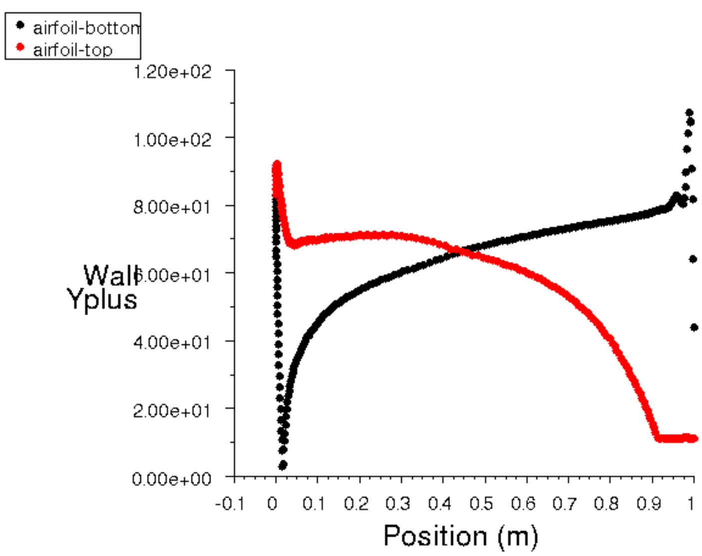
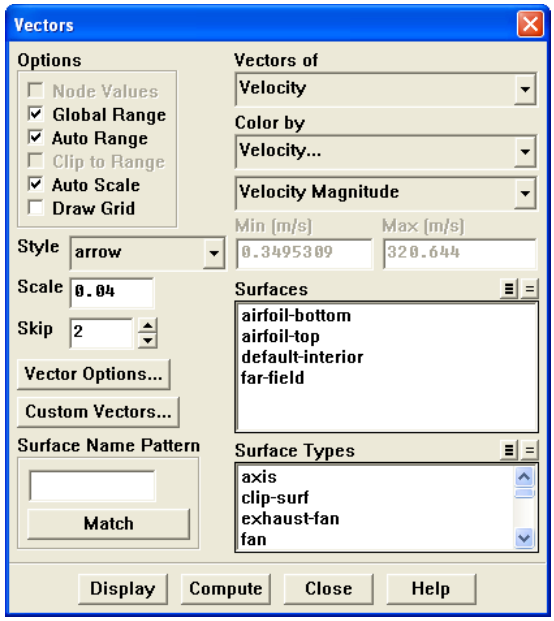


Figure 5.9: Wall Yplus Plot After Adaption

After adaption, the Y^+ values have reduced and most of the cells are below 80. This shows that the grid resolution is well within the recommended range.

2. Display velocity vectors (Figure 5.10).

   Display ⟶Vectors...



   (a) Click Vector Options....

   The Vector Options *panel opens.*

   i. Select Fixed Length.

   Set the Fixed Length *option to display all the vectors with the same length. Increase* Skip *to reduce the number of vectors, so that the display does not appear cluttered due to large number of vectors.*

   ii. Click Apply and close the panel.

   (b) Set the Scale to `0.04` and increase the value of Skip to 2.

   (c) Click Display.

   The flow hits the airfoil at an angle which corresponds to the angle specified at the far-field boundary. To view the velocity vectors around airfoil, you need to zoom the view.

3. Display pressure distribution (Figure 5.11).

   Display ⟶Contours...

   (a) Under Contours of, retain the default options, Pressure... and Static Pressure.

   (b) Under Options, enable Filled.

   (c) Click Display.

   To display the value of the contour in the console window, right-click on a point.

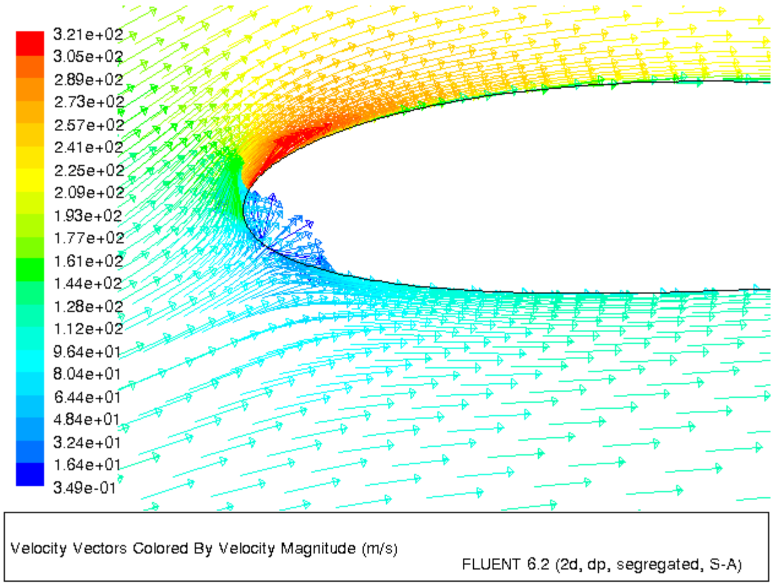


Figure 5.10: Velocity Vectors

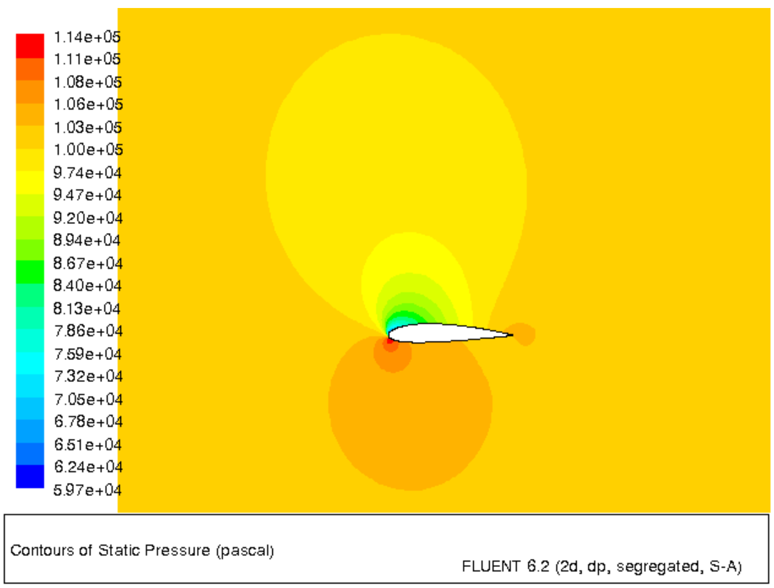


Figure 5.11: Contours of Static Pressure

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.

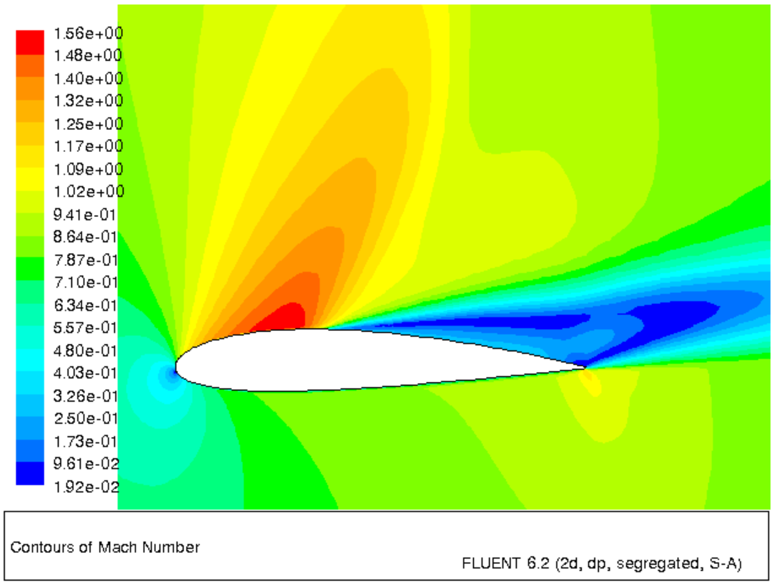


Figure 5.12: Contours of Mach Number

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.

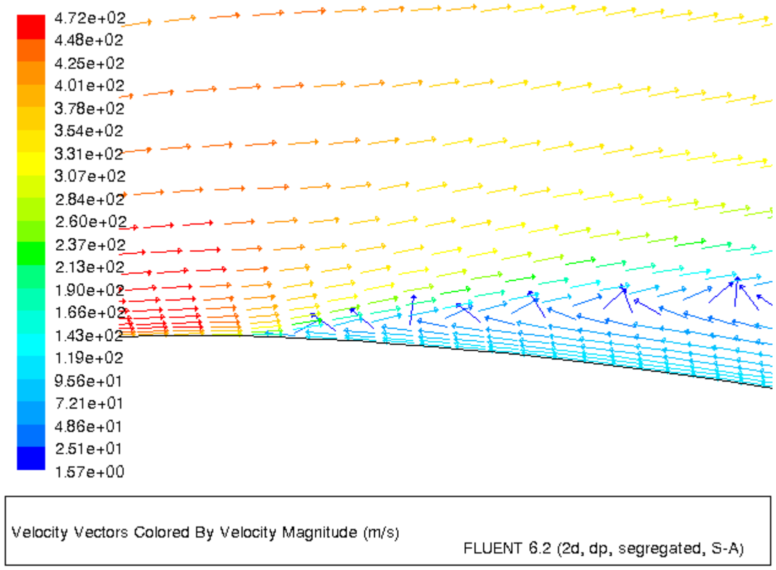


Figure 5.13: Velocity Vectors

Summary

The flow field at airfoil is highly dependent on the free stream condition. In order to predict correct lift and drag coefficients higher order discretization schemes should be used. On the mesh part it is important to resolve the near wall cell, such that Y^+ is about 1. Due to the mesh count constraint of 20,000 cells, it is not possible to resolve the mesh further. With the current mesh, results are deviating about 10% from the expected values, but flow features are captured correctly.

References

[1] J.D. Anderson, Jr., *Introduction to Flight*, McGraw Hill Higher Education.

Exercises/Discussions

1. What will be the lift and drag coefficients for different angles of attack. Run the simulations with a range of angle of attack to trace the complete curve and predict the stall angle for NACA-2415. Verify its match with the values reported in literature.

2. Simulate the flow with different turbulence models and compare their performance for drag and lift predictions.

3. Run the simulation with different Mach numbers at free stream.

4. Coupled solver can be used for higher Mach number flows and the results can then be compared with segregated solver.

5. Check for unsteadiness flow features when angle of attack is increased.

Links for Further Reading

- http://www.nasg.com/index-e.html
- http://www.aae.uiuc.edu/m-selig/
- http://airtrafficcontrol.no-ip.org:8080/airfoil.htm
- http://www.vzlu.cz/htmfiles/HSaerodynamics.htm