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 $a = 10^{\circ}$ and a free stream Mach number (M_{∞}) of 0.4066 and 0.8. The chord length (C) is 1 m.

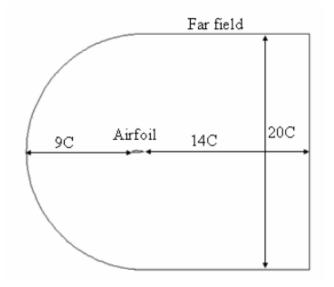


Figure 5.1: Problem Schematic

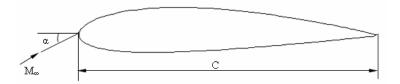


Figure 5.2: Airfoil Details

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.

 $|\mathsf{File}| \longrightarrow |\mathsf{Read}| \longrightarrow \mathsf{Case...}$

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

2. Check the grid.

 $Grid \longrightarrow Check$

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

3. Scale the grid.

 $Grid \longrightarrow Scale...$

💶 Scale Grid				
Scale Factors	Unit Conversion			
X 1	Grid Was Created In 🔳			
Y 1	Change Length Units			
Domain Extents				
Xmin (m) -9 Xmax (m) 15				
Ymin (m) -19 Ymax (m) 19				
Scale	Jnscale Close Help			

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).

 $\mathsf{Display} \longrightarrow \mathsf{Grid}...$

💶 Grid Display				
Options	Edge Type	Surfaces 📃 =		
 □ Nodes □ Edges □ Faces □ Partitions 	 All ⊂ Feature ⊂ Outline 	airfoil-bottom airfoil-top default-interior far-field		
Shrink Factor	eature Angle 20			
Surface Name	Pattern	Surface Types 🔳 🖃		
Match axis clip-surf exhaust-fan fan v				
		Outline Interior		
Display	Colors	Close Help		

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

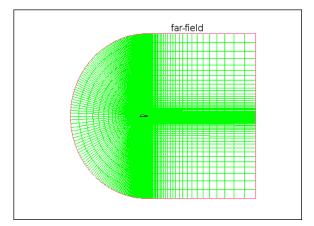


Figure 5.3: Grid Display

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.

 $\mathsf{Define} \longrightarrow \mathsf{Models} \longrightarrow \mathsf{Solver}...$

(a) Click OK to retain the default settings.

2. Enable the Spalart-Allmaras turbulence model.

Define –	\rightarrow Models —	→Viscous…
----------	------------------------	-----------

Model	Model Constants
C Inviscid C Laminar ⊙ Spalart-Allmaras (1 eqn)	Cb1 (0.1355) Cb2
C k-epsilon (2 eqn) C k-omega (2 eqn) C Reynolds Stress (5 eqn)	0.622 Cv1
Spalart-Allmaras Options	7.1
 Vorticity-Based Production Strain/Vorticity-Based Production 	on Cw2
	User-Defined Functions
	Turbulent Viscosity

- (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 wallbounded 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.

 $\mathsf{Define} \longrightarrow \mathsf{Materials...}$

Materials		X
Name	Material Type	Order Materials By
air	J	O Name O Chemical Formula
Chemical Formula	Fluent Fluid Materials	
1	air	▼ Fluent Database
	Mixture	User-Defined Database
	none	× _
Properties		
Density (kg/m3)	ideal-gas	
Cp (j/kg-k)	constant 💌 Edit	
	1006.43	
Thermal Conductivity (w/m-k)	constant 🗨 Edit	
	0.0242	
Viscosity (kg/m-s)	sutherland 🗾 Edit	
Change/Ci	eate Delete Close H	lelp

- 1. Select ideal-gas from the Density drop-down list.
- 2. Select sutherland from the Viscosity drop-down list.

Sutherland Law	×
Methods	
C Two Coefficient Method (SI Units Only) Three Coefficient Method	
Reference Viscosity, mu0 (kg/m-s) 1.716e-05	_
Reference Temperature, TO (k) 273.11	_
Effective Temperature, S (k) 110.56	
OK Cancel Help	

- (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 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 \longrightarrow Operating Conditions...

Operating Conditions	×
Pressure	Gravity
Operating Pressure (pascal) Ø	🗆 Gravity
Reference Pressure Location	
X (m) 0	
Y (m) 🔋	
OK Cancel H	lelp

- 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).

E	💶 Boundary Con	ditions 🛛 🔀		
	Zone	Туре		
	airfoil-bottom	exhaust-fan 🔼		
	airfoil-top	inlet-vent		
	default-interior	intake-fan		
	far-field	interface		
	fluid	mass-flow-inlet		
		outflow outlet-vent		
		pressure-far-field		
		pressure-inlet		
		pressure-outlet		
		symmetry		
		velocity-inlet 🛛 👱		
		ID		
		4		
		1		
	Set Copy	y Close Help		

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

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

(b) Click the Set... button.

Pressure Far-Field	X
Zone Name	
far-field	
Momentum Thermal Radiation Species UDS	DPM
Gauge Pressure (pascal) 101325	constant 🗸
Mach Number 0.4966	constant 🗸
X-Component of Flow Direction 0.984807	constant 🔹
Y-Component of Flow Direction 0.173648	constant 👻
Turbulence	
Specification Method Turbulent Viscosity F	łatio 🗸
Turbulent Viscosity Ratio 19	constant 💌
OK Cancel Help	

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

- 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 = $1.0 \times cos(10^{\circ})$ and Y component of flow direction = $1.0 \times sin(10^{\circ})$

- 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 \longrightarrow Controls \longrightarrow Solution...

2. Initialize the flow.

 $\fbox{Solve} \longrightarrow \fbox{Initialize} \longrightarrow \fbox{Initialize}...$

Solution Initialization	×
Compute From far-field	Reference Frame © Relative to Cell Zone © Absolute
Initial Values	
Gauge Pressure (pascal)	101325
X Velocity (m/s)	138.9817
Y Velocity (m/s)	24.50622
Modified Turbulent Viscosity (m2/s)	0.0001529736
Init Reset Apply	Close Help

(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.

 $\boxed{\mathsf{Solve}} \longrightarrow \boxed{\mathsf{Monitors}} \longrightarrow \operatorname{\mathsf{Residual}} \dots$

🖻 Residual Monitors 🛛 🛛 🛛					
Options	Storage		Plotting		
✓ Print✓ Plot	Iteratio	ns 1000 🔺	Wind	low 🔋 🌲	
	Normalization		Iterations	1000 韋	
	🗆 Norm	alize 🗹 Scale	Axes	Curves	
	Convergence C	riterion			
	absolute	•			
Residual	Check Monitor Conve	Absolute rgence Criteria	A		
continuity		0.001			
x-velocity		0.001			
y-velocity	V V	0.001			
energy		1e-06			
nut	v	0.001	-		
OK Plot Renorm Cancel Help					

File

- (a) Enable Plot from the Options list.
- (b) Click OK to close the Residual Monitors panel.
- 4. Save the case file (airfoil1.cas.gz).

 \rightarrow Write \longrightarrow Case...

Select File					? 🛛
Look in:	iirfoil 🔁		•	+ 🗈 💣 🖩	
My Recent Documents Desktop My Documents My Computer					
My Network Places	Case File Files of type:	airfoil1.cas.gz Case Files		•	OK Cancel
	Write Binary Fil	,			

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 \longrightarrow Iterate...

💶 Iterate 🛛 🗙
Iteration
Number of Iterations 250
Reporting Interval 1
UDF Profile Update Interval 1
Iterate Apply Close Help

- (a) Set the Number of Iterations to 250.
- (b) Click Iterate.
- (c) Close the **lterate** 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.

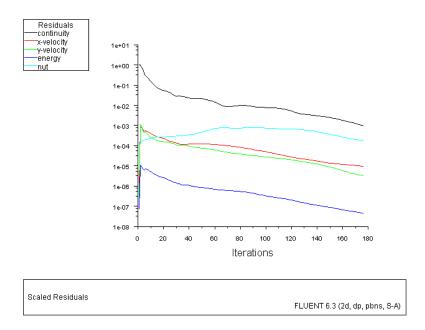


Figure 5.4: Scaled Residuals

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.

 $\mathsf{Report} \longrightarrow \mathsf{Reference Values...}$

Reference Values
Compute From
far-field 🗸
Reference Values
Area (m2) 1
Density (kg/m3) 1.176674
Depth (m) 1
Enthalpy (j/kg) 11820.15
Length (m) 1
Pressure (pascal) 101325
Temperature (k) 300
Velocity (m/s) 141.1259
Viscosity (kg/m-s) 1.7894e-05
Ratio of Specific Heats 1.4
Reference Zone
OK Cancel Help

(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 \longrightarrow Forces...

Force Reports		
Options Forces Moments Center of Pressure	Force Vector × 0.173648 Y 0.984807 Z 0	Wall Zones = = airfoil-bottom airfoil-top
Wall Name Pattern	Match	
Print Writ	te Close	Help

- (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 $X = 1.0 \times sin(10^\circ)$ and $Y = 1.0 \times cos(10^\circ)$.

(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].

3.7389e-06 5.0519e-08 1.9780e-04 0:00:29 79 3.6140e-06 4.9305e-08 1.9405e-04 0:00:23 78 3.5049e-06 4.8094e-08 1.9065e-04 0:00:18 77 3.4108e-06 4.6972e-08 1.8753e-04 0:00:29 76 3.3341e-06 4.5777e-08 1.8482e-04 0:00:18 74 30701 0) pressure viscous total pressure viscous total force force force coefficient coefficient coefficient n n n 1982.5663 7.0866331 1989.6529 0.16919535 0.00060478451 0.16980013 9393.6219 13.48705 9407.109 0.8016666 0.0011510062 0.8828176 	File Grid Define S	olve Adapt Surface	Display Plot Report	Parallel Help		
3.5049e-06 4.8094e-08 1.9065e-04 0:00:18 77 3.4108e-06 4.6972e-08 1.8753e-04 0:00:29 76 3.3341e-06 4.5777e-08 1.8482e-04 0:00:23 75 3.2803e-06 4.4645e-08 1.8247e-04 0:00:18 74 30701 0) pressure viscous total pressure viscous total pressure viscous force force force coefficient coefficient 1982.5663 7.0866331 1989.6529 0.16919535 0.80060478451 0.16980013 9393.6219 13.48705 9407.109 0.8016666 0.0011510062 0.8028176	3.7389e-06 5.0	519e-08 1.9780e	-04 0:00:29	79		
.4108e-06 4.6972e-08 1.8753e-04 0:00:29 76 .3341e-06 4.5777e-08 1.8482e-04 0:00:23 75 .2803e-06 4.4645e-08 1.8247e-04 0:00:18 74 0701 0) pressure viscous total pressure viscous total force force force coefficient coefficient coefficient n n n 1982.5663 7.0866331 1989.6529 0.16919535 0.00060478451 0.16980013 9393.6219 13.48705 9407.109 0.8016666 0.0011510062 0.8028176	.6140e-06 4.9	305e-08 1.9405e	-04 0:00:23	78		
3.3341e-06 4.5777e-08 1.8482e-04 0:00:23 75 3.2803e-06 4.4645e-08 1.8247e-04 0:00:18 74 30701 0) pressure viscous total pressure viscous total pressure viscous force force force coefficient coefficient n n n 1 1982.5663 7.0866331 1989.6529 0.16919535 0.00060478451 0.16980013 9393.6219 13.48705 9407.109 0.8016666 0.0011510062 0.8028176						
3.2803e-06 4.4645e-08 1.8247e-04 0:00:18 74 30701 0) pressure viscous total pressure viscous total force force force coefficient coefficient coefficient n n n 1982.5663 7.0866331 1989.6529 0.16919535 0.00060478451 0.16980013 9393.6219 13.48705 9407.109 0.8016666 0.0011510062 0.8028176						
30701 0) pressure viscous total pressure viscous total force force force coefficient coefficient coefficient n n n 1982.5663 7.0866331 1989.6529 0.16919535 0.00060478451 0.16980013 9393.6219 13.48705 9407.109 0.8016666 0.0011510062 0.8028176	3.3341e-06 4.9	777e-08 1.8482e	-04 0:00:23	75		
force force force coefficient coefficient <thcoefficient<< td=""><td>3.2803e-06 4.4</td><td>645e-08 1.8247e</td><td>-04 0:00:18</td><td>74</td><td></td><td></td></thcoefficient<<>	3.2803e-06 4.4	645e-08 1.8247e	-04 0:00:18	74		
force force force coefficient coefficient <thcoefficient<< td=""><td>30701 0)</td><td></td><td></td><td></td><td></td><td></td></thcoefficient<<>	30701 0)					
n n n 1982.5663 7.0866331 1989.6529 0.16919535 0.00060478451 0.16980013 9393.6219 13.48705 9407.109 0.8016666 0.0011510062 0.8028176						
1982.5663 7.0866331 1989.6529 0.16919535 0.00060478451 0.16980013 9393.6219 13.48705 9407.109 0.8016666 0.0011510062 0.8028176				coefficient	coefficient	coefficient
9393.6219 13.48705 9407.109 0.8016666 0.0011510062 0.8028176	n	n	n			
	1982.5663	7.0866331	1989.6529	0.16919535	0.00060478451	0.16980013
11376.188 20.573683 11396.762 0.97086195 0.0017557907 0.97261774	9393.6219	13.48705	9407.109	0.8016666	0.0011510062	0.8028176
	11376.188	20.573683	11396.762	0.97086195	0.0017557907	0.97261774

8. Set the solution controls.

 $Solve \longrightarrow Controls \longrightarrow Solution...$

Solution Controls					
Equations 📃 📃	Under-Relaxation Factors				
Flow Modified Turbulent Viscosity	Pressure 9.3				
Energy	Density 1				
	Body Forces 1				
	Momentum 9.7				
Pressure-Velocity Coupling	Discretization				
SIMPLE	Density Second Order Upwind				
	Momentum Second Order Upwind 👻				
	Modified Turbulent Viscosity Second Order Upwind 🗸				
	Energy Second Order Upwind				
OK Default Cancel Help					

- (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.

Solve \longrightarrow Monitors \longrightarrow Residual...

💶 Residual Mo	nitors				
Options	Storage			Plotting	
✓ Print✓ Plot	lter	ations 10	00 🛨	Window	
	Normalizati	ion		Iterations 1	800 🛨
	□ N	lormalize	Scale	Axes	Curves
	Convergen	ce Criterio	n		
	absolute		-		
Residual	Cl Monitor Co	neck onvergenc	Absolute e Criteria	<u> </u>	
continuity		Γ	0.001		
x-velocity	~		0.001	_	
y-velocity	V		0.001		
energy	V	Γ	1e-06		
nut	~		0.001		
OK Plot Renorm Cancel Help					

- (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...

E Force Monitor	s		X
Options	Wall Zones <u>=</u> = airfoil-bottom airfoil-top	Force Vector X 0.984807 Y 0.173648 Z 0 About Z-Axis	Plot Window 1 + Axes Curves
cd-history			
Арр	ly Plot Cle	ear Close H	elp

- (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). $Plot \longrightarrow XY...$

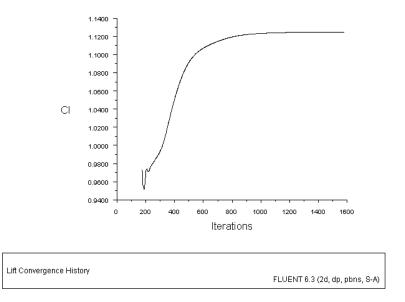


Figure 5.5: Lift Coefficient Plot

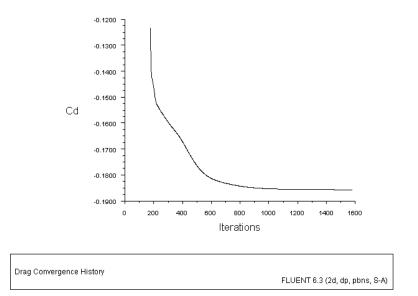


Figure 5.6: Drag Coefficient Plot

Solution XY Plot			
Options Node Values Position on X Axis Position on Y Axis Write to File Order Points	Plot Direction X 1 Y 0 Z 0	Y Axis Function Turbulence Wall Yplus X Axis Function Direction Vector	•
File Data <u>≡)</u> =	Load File Free Data	Surfaces airfoil-bottom airfoil-top default-interior far-field	
Plot /	xes Cur	ves Close Help	

- (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.

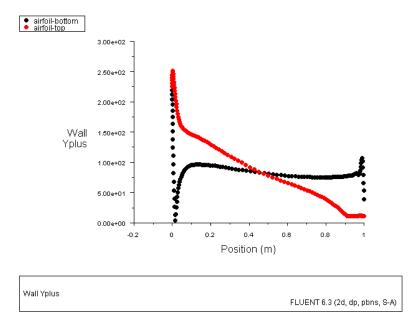


Figure 5.7: Wall Yplus Plot

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).

Write →Case &	→Write →Case & Data	
---------------	---------------------	--

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

Adapt \longrightarrow Boundary...

Boundary Adaption		×
Options Cell Distance Normal Distance Volume Distance Contours Manage Controls	Number of Cells 1 Distance Threshold (m) 0 Boundary Volume (m3) 1 Growth Factor 1	Boundary Zones 🗐 = airfoil-bottom airfoil-top far-field
Adapt	Mark Apply Close	Help

- (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).

 $\mathsf{File} \longrightarrow \mathsf{Write} \longrightarrow \mathsf{Case} \And \mathsf{Data...}$

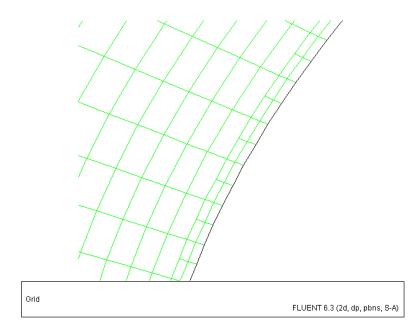


Figure 5.8: Adapted Grid

Step 7: Postprocessing

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

 $\mathsf{Plot} \longrightarrow \mathsf{XY} \mathsf{Plot}...$

Solution XY Plot		×
Options Node Values Position on X Axis	× 1	Y Axis Function Turbulence
Position on Y Axis Write to File Order Points	Y 0 Z 0	Wall Yplus • X Axis Function Direction Vector
File Data <u>≡</u> <u>=</u>		Surfaces E = airfoil-bottom airfoil-top default-interior far-field
Plot A	Load File Free Data	ves Close Help

- (a) Select Turbulence... and Wall Yplus in the Y Axis Function lists.
- (b) Select airfoil-bottom and airfoil-top in the Surfaces list.
- $(c)\ Deselect\ \mbox{Node}\ \mbox{Values}.$
- (d) Click Plot and close the Solution XY Plot panel.

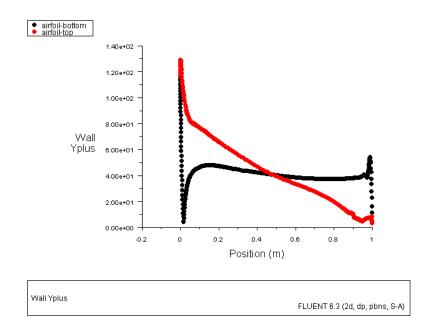


Figure 5.9: Wall Yplus Plot After Adaption

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).

 $\mathsf{Display} \longrightarrow \mathsf{Vectors...}$

Vectors			×
Options	Vectors of Velocity		•
 ✓ Global Range ✓ Auto Range ✓ Clip to Range 	Color by Velocity		•
✓ Auto Scale □ Draw Grid	Velocity Magnit	ude	•
Style arrow 🗸	Min (m/s) 0.5244651	Max (m/s) 320.8784	
Scale 0.04	Surfaces		II.
Skip 2	airfoil-bottom airfoil-top default-interior far-field		
Custom Vectors			
Surface Name Pattern Match	Surface Types axis clip-surf exhaust-fan fan		
Display Com	pute Close	Help	

(a) Click Vector Options....

 ${\rm i.}~{\rm Select}$ 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.

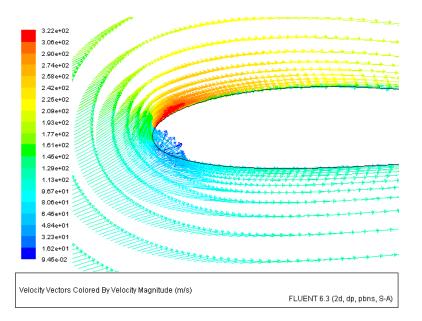


Figure 5.10: Velocity Vectors

3. Display pressure distribution (Figure 5.11).

 $\mathsf{Display} \longrightarrow \mathsf{Contours...}$

- (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.

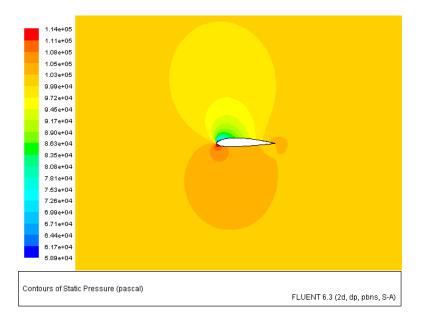


Figure 5.11: Contours of Static Pressure

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.