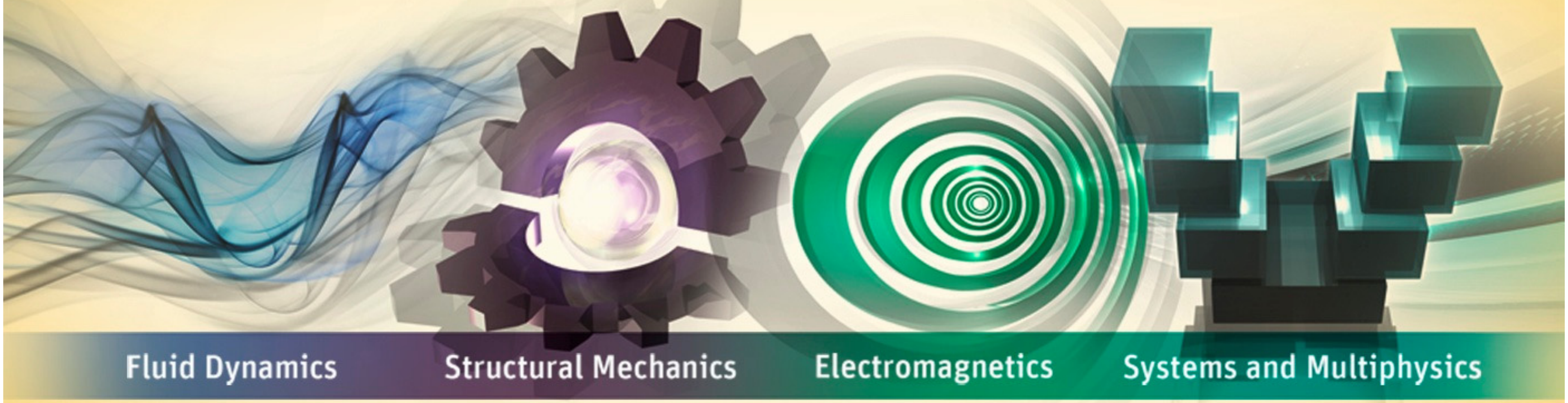
Workshop 04
Fluid flow around the NACA0012 Airfoil

14. 0 Release



Introduction to ANSYS CFX

Introduction

Workshop Description:

The flow simulated is an external aerodynamics application for the flow around a NACA0012 airfoil

Learning Aims:

This workshop introduces several new skills (relevant for many CFD applications, not just external aerodynamics):

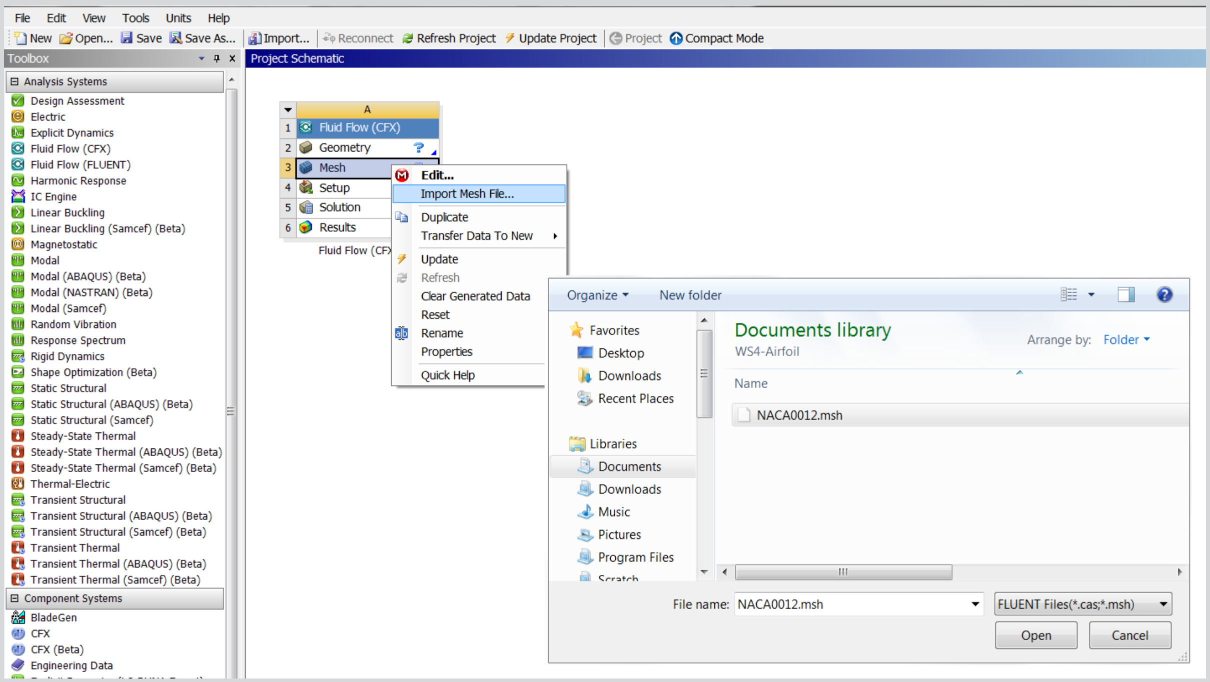
- Assessing Y+ for correct turbulence model behavior
- Modifying solver settings to improve accuracy
- Reading in and plotting experimental data alongside CFD results
- Producing a side-by-side comparison of different CFD results.

Learning Objectives:

To understand how to model an external aerodynamics problem, and skills to improve and assess solver accuracy with respect to both experimental and other CFD data.

Import the supplied mesh file

- **Start Workbench 14.0**
- Copy a CFX 'Analysis System' into the project schematic
- Import the supplied FLUENT mesh file (naca0012.msh) by:
  - Right click on Mesh (cell A3) and select 'Import Mesh File'
  - Browse to the mesh file

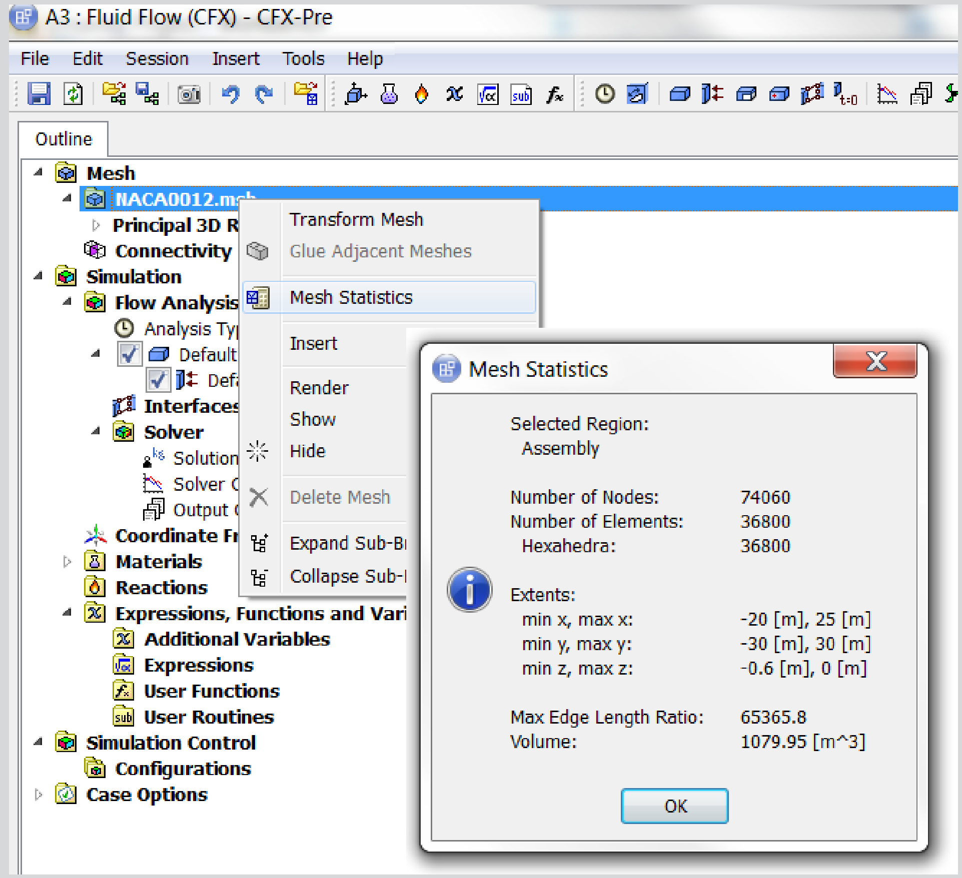


Introduction | **Setup** | Solution | Results | Summary

ANSYS

Double-click Setup to set up the case

- CFX-Pre will launch in a new window
- Check the mesh by right-clicking NACA0012.cfx → Mesh Statistics



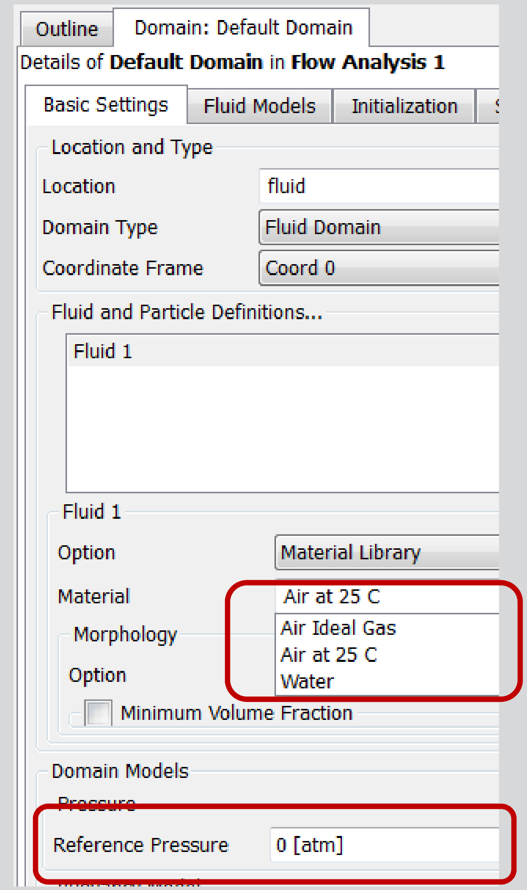
Introduction | Setup | Solution | Results | Summary

Case Setup: Choose the Material and reference pressure

Adjust the domain so that Air Ideal Gas is used along with the SST turbulence model and Total Energy model :

Default Domain → Basic Settings
→ Material → Air Ideal Gas

Default Domain → Basic Settings
→ Reference Pressure = 0 [atm]



Case Setup: Reference Pressure

Absolute pressure = operating pressure + gauge pressure

For incompressible flows it is normal to specify a large (typically atmospheric pressure) operating pressure and let the solver work with smaller 'gauge' pressures for the boundary conditions, to reduce round-off errors.

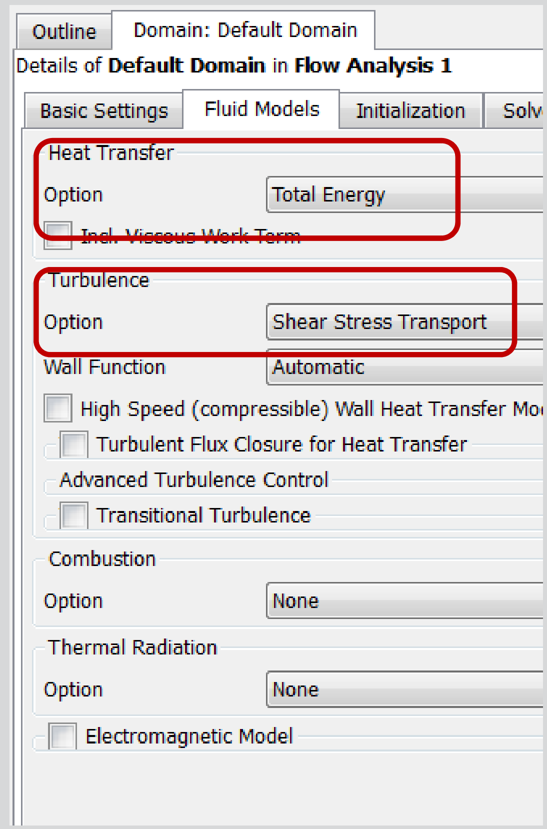
For compressible flows, the solver needs to use the absolute values in the calculation, therefore, with compressible flows, it is sometimes convenient to set to operating pressure to zero, and input/output 'absolute' pressures.

Case Setup: Choose the models

Adjust the domain so that the SST turbulence model and Total Energy model are used :

Default Domain → Fluid Models
→ Heat Transfer
→ Option = Total Energy

Default Domain → Fluid Models
→ Turbulence
→ Option = Shear Stress Transport



Case Setup: Coordinate Frame

The angle of attack is 1.55 degrees. One way of accounting for this angle is to create a new coordinate system whose z-axis is in line with the flow direction and then to use this coordinate system when applying boundary conditions.

Create a new coordinate frame:

Insert → Coordinate Frame → Name = Coord 1

Option = Axis Points

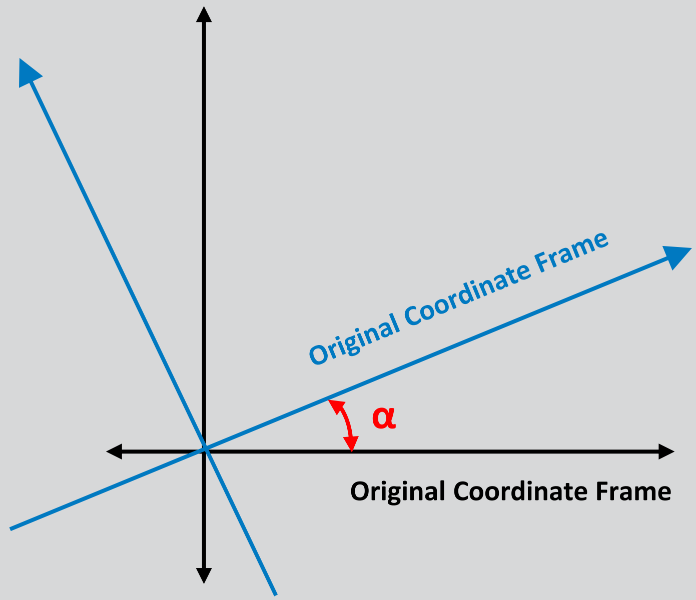
Origin = 0, 0, 0

Z axis = 0.999634, 0.027049, 0

X-Z Plane Pt = -0.02007, 0.999799, 0

Where the above values were calculated using

cos(α), sin(α), sin(α+90 [deg]), and cos(α+ 90 [deg])



Introduction | Setup | Solution | Results | Summary

Case Setup: Boundary Conditions

Create a boundary condition for the airfoil:

Insert → Boundary → Name = airfoil

Basic Settings → Boundary Type = Wall

Basic Settings → Location = airfoil_lower, airfoil_upper → OK

This will add a boundary called airfoil with the default wall settings (adiabatic, no-slip wall). To change these settings double-click on the airfoil object and change the settings under Boundary Details.

Introduction | Setup | Solution | Results | Summary

Case Setup: Boundary Conditions

Create a boundary condition for the inlet:

Insert → Boundary → Name = inlet→ OK
Basic Settings → Boundary Type = Inlet
Basic Settings → Location = inlet
Basic Settings → Coordinate Frame = Coord 1
Boundary details → Mass and Momentum → Option = Cart. Vel. Components
Boundary details → Flow Direction → U = 0 [m/s]
Boundary details → Flow Direction → V = 0 [m/s]
Boundary details → Flow Direction → W = 0.7 * 340.29 [m/s]
Boundary details → Turbulence → Option = Intensity and Eddy Viscosity Ratio
Boundary details → Turbulence → Fractional Intensity = 0.01
Boundary details → Turbulence → Eddy Viscosity Ratio = 1
Boundary details → Heat Transfer → Option = Static Temperature
Boundary details → Heat Transfer → Static Temperature = 283.34 [K]

This will create an inlet boundary condition with air flowing at a speed flow with Ma = 0.7 at an angle of attack (α) of 1.55 deg.

Case Setup: Boundary Conditions

Create a boundary condition for the outlet:

Insert → Boundary → Name = outlet→ OK
Basic Settings → Boundary Type = Outlet
Basic Settings → Location = outlet
Boundary details → Mass and Momentum → Option = Average Static Pressure
Boundary details → Mass and Momentum → Relative Pressure = 73048 [Pa]

Case Setup: Boundary Conditions

It is important to place the farfield (inlet and outlet) boundaries far enough from the object of interest.

For example, in lifting airfoil calculations, it is not uncommon for the far-field boundary to be a circle with a radius of 20 chord lengths.

This workshop will compare CFD with wind-tunnel test data therefore we need to calculate the static conditions at the far-field boundary.

We can calculate this from the total pressure, which was atmospheric at 101325 Pa with a Mach number of 0.7 in the test.

The wind tunnel operating conditions for validation test data give the total temperature as $T_0 = 311$ K

$$\frac{p_o}{p} = \left[1 + \left(\frac{\gamma - 1}{2}\right) M^2\right]^{\frac{\gamma}{\gamma - 1}}$$

where

p_o = total pressure = 101325 Pa

p = static pressure

$\gamma = 1.4$ for air

M = Mach No. = 0.7

$$\therefore \frac{p_o}{p} = 1.3871$$

$$p = 73048 \text{ Pa}$$

$$\frac{T_0}{T} = 1 + \left(\frac{\gamma - 1}{2}\right) M^2$$

where

T_0 = total temperature = 311 K

T = static temperature

$\gamma = 1.4$ for air

M = Mach number = 0.7

$$\therefore \frac{T_0}{T} = 1.098 \text{ and so } T = 283.24 \text{ K}$$

Introduction | **Setup** | Solution | Results | Summary

Case Setup: Boundary Conditions

Create a boundary condition for the symmetries:

Insert → Boundary → Name = symmetry → OK
Basic Settings → Boundary Type = Symmetry
Basic Settings → Location = sym1,sym2

Introduction | Setup | Solution | Results | Summary

Case Setup: Solution Monitors

Set up residual monitors so that convergence can be monitored

Insert → Solver → Output Control → Monitor → Monitor Objects

Monitor Points and Expressions → Add New Item → Name = Lift Coef → OK

→ Option = Expressions → Expression Value =

force_x_Coord 1()@airfoil * 2 / (massFlowAve(Density)@inlet * (massFlowAve(Velocity)@inlet)^2*0.6 [m]* 1[m])

Monitor Points and Expressions → Add New Item → Name = Drag Coef → OK

→ Option = Expressions → Expression Value =

force_z_Coord 1()@airfoil * 2 / (massFlowAve(Density)@inlet * (massFlowAve(Velocity)@inlet)^2*0.6 [m]* 1[m])

Lift and drag coefficients are defined (perpendicular and parallel respectively) relative to the free-stream flow direction, not the airfoil.

The expressions must match the names for the airfoil and inlet boundary conditions. To ensure that the correct boundary names and functions are being used, try using the right mouse button in the Expression Value field instead of typing the expression manually.

Solution Control and Solve

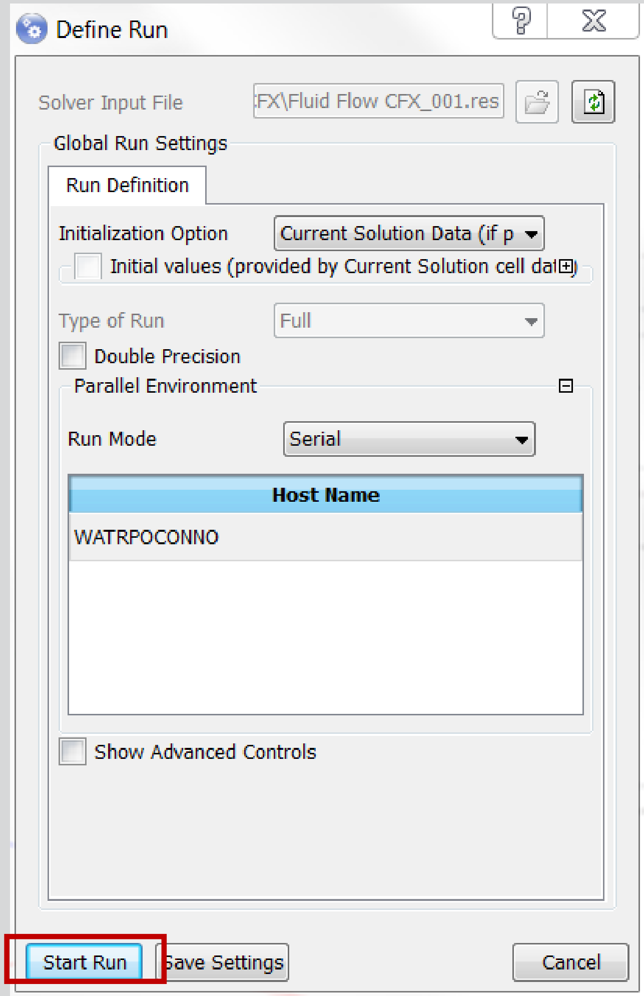
Insert → Solver → Solution Control

Basic Settings → Min. Iterations = 100

Basic Settings → Max. Iterations = 200 → OK

Return to Workbench and double-click Solution.

In the Define Run window, click Start Run.



Introduction | Setup | **Solution** | Results | Summary

Run Calculation

Review the convergence plots. The solution will complete when either the default residual targets (1e-4) have been satisfied or when the default maximum number of iterations (200) has been reached.

Click User Points to review the lift and drag coefficient convergence.

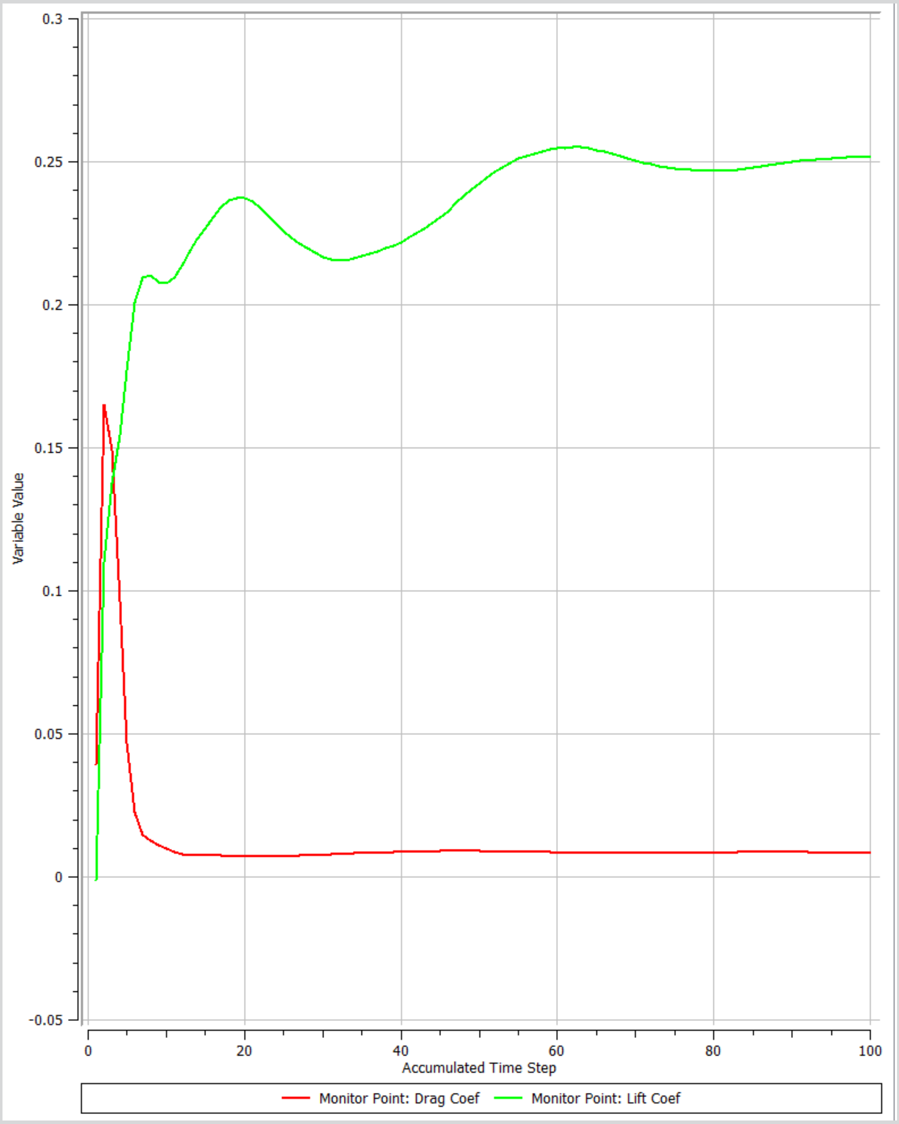
From Reference [1],

Cl = 0.241 and Cd = 0.0079

The CFD solution calculates

Cl = 0.252 and Cd = 0.0085

Further iterations and mesh refinement would improve the solution.

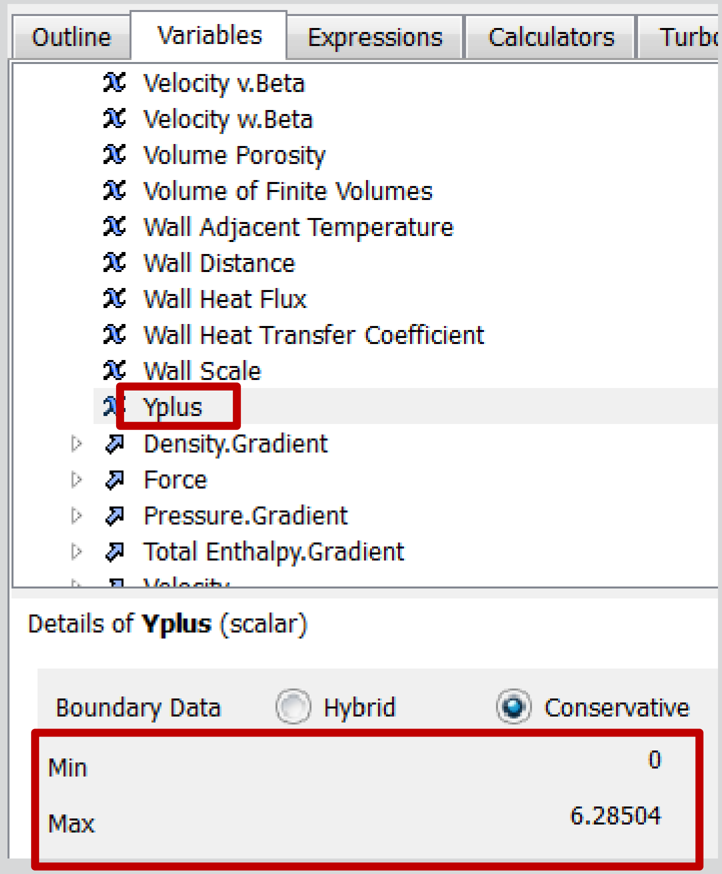


Introduction | Setup | Solution | Results | Summary

Check the mesh (Y+)

Variables → Yplus

The maximum Y+ is 6.28504.



Introduction | Setup | Solution | **Results** | Summary

Check the mesh (some notes on Y+)

y^+ is the non-dimensional normal distance from the first grid point to the wall and is covered tomorrow in Lecture 6

When using SST, the intention is to integrate governing equations directly to the wall without using the Universal Law of The Wall for turbulence. For such cases, the first grid point should be placed within the viscous sublayer (near-wall region, $y^+ \leq 2$).

The aspect ratio could be reduced, while keeping the same y^+ value:

- By keeping the same first cell distance and increasing the number of nodes along the wall surface. This reduces the length of cells for a given height so will reduce the aspect ratio whilst significantly increasing the overall cell count

The aspect ratio could be reduced, while increasing y^+ value:

- Increasing the normal distance of the first grid point from the wall to give y^+ larger values wall functions will begin to be.

Introduction | Setup | Solution | **Results** | Summary

Post Processing

Plot the y^+ values along the airfoil surfaces

Insert → Location → Polyline → Name = Airfoil curve

Geometry → Method = Boundary Intersection

→ Boundary List = airfoil → Intersect with = sym1 → Apply

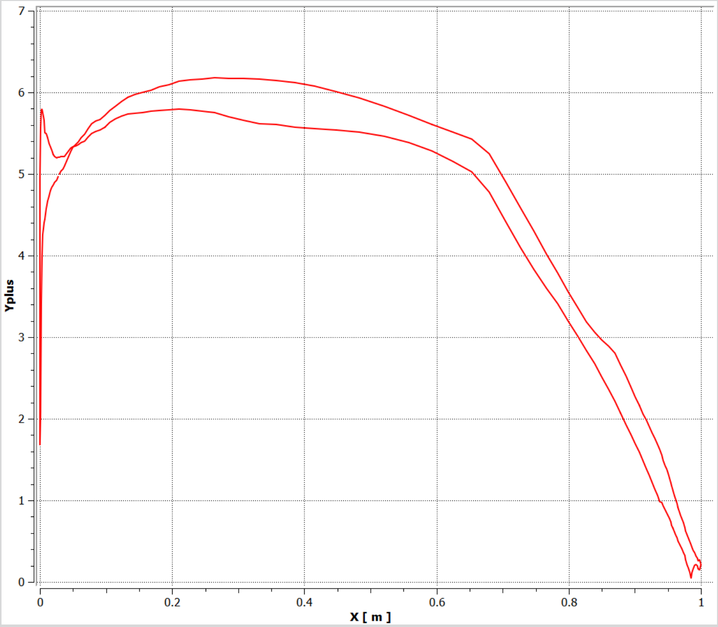
Insert → Chart → Name = Yplus on airfoil

Data Series → Location = Airfoil Curve → X Axis → Variable = X → Y Axis

→ Variable = Yplus → Apply

We can see that $y^+ \approx 2.5$ for much of the surface

In order to obtain a good drag prediction, and for the turbulence model to work effectively, the mesh is well resolved near to the wall, such that the first grid point is located in the viscous sub-layer, with y^+ of 5 or less.



Post Processing

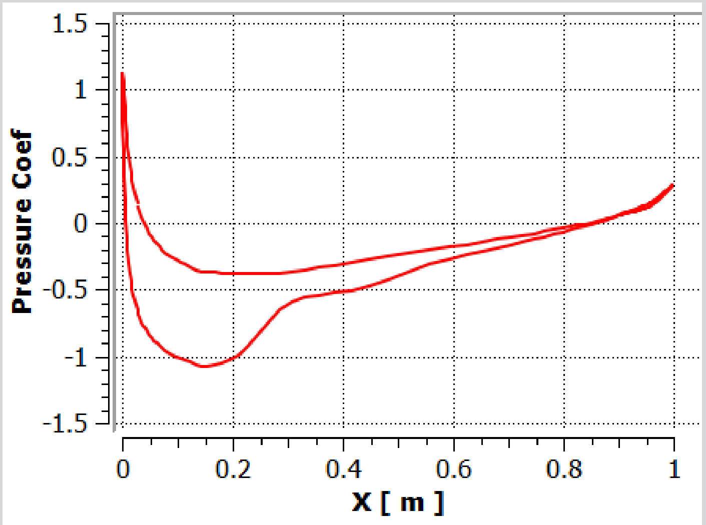
Plot the pressure coefficient (Cp) along the upper and lower airfoil surfaces

Insert → Variable → Name = Pressure Coef → OK

→ Method = Expression → Expression =

(p-73048 Pa])/(0.5*massFlowAve(density)@inlet*
(massFlowAve(Velocity)@inlet)^2)

Follow same charting instructions used for the y+ chart but set the Y Axis variable to Pressure Coef.

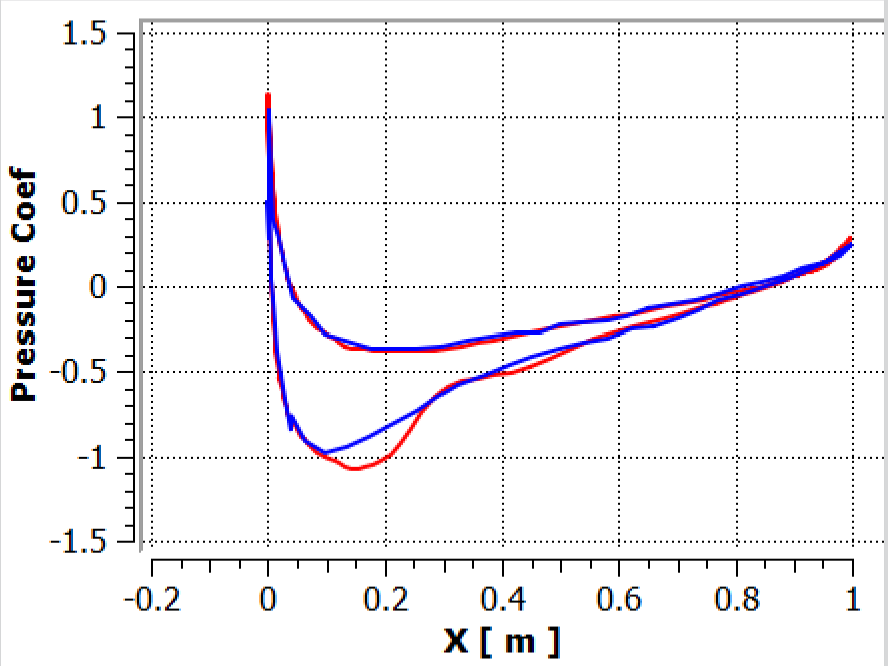


Introduction | Setup | Solution | **Results** | Summary

Post Processing

Compare the CFX result with test data by editing the details of the graph created in the previous section to include another data series.

Data Series → New → Name = Experimental → Data Source → File → Browse → select ExperimentalData.csv → Apply



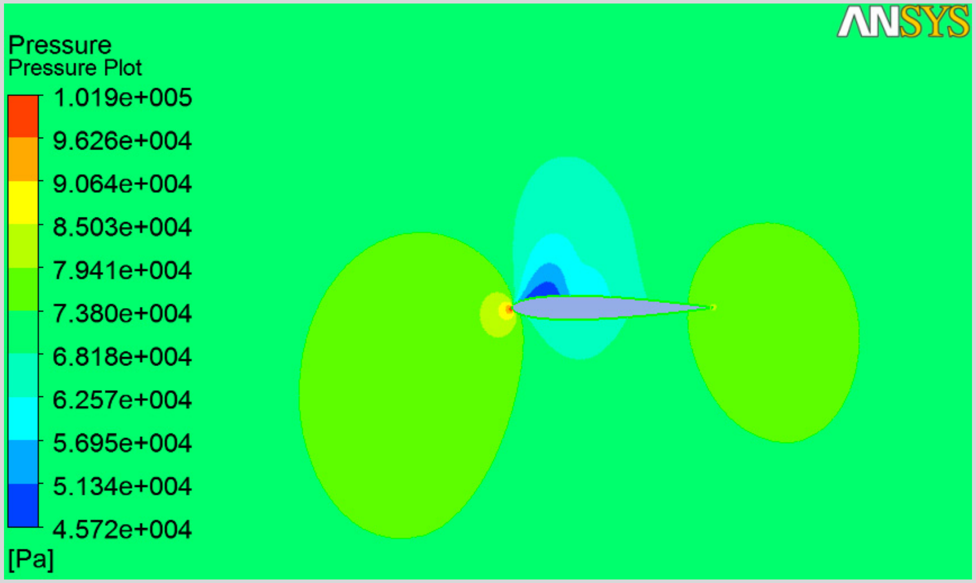
Post Processing

Examine the contours of static pressure

Insert → Contour → Name = Contour Plot

Geometry → Locations = symmetry → Variable = Pressure → Apply

Note the high static pressure at the nose, and low pressure on the upper (suction) surface. The latter is expected as the airfoil wing is generating lift.

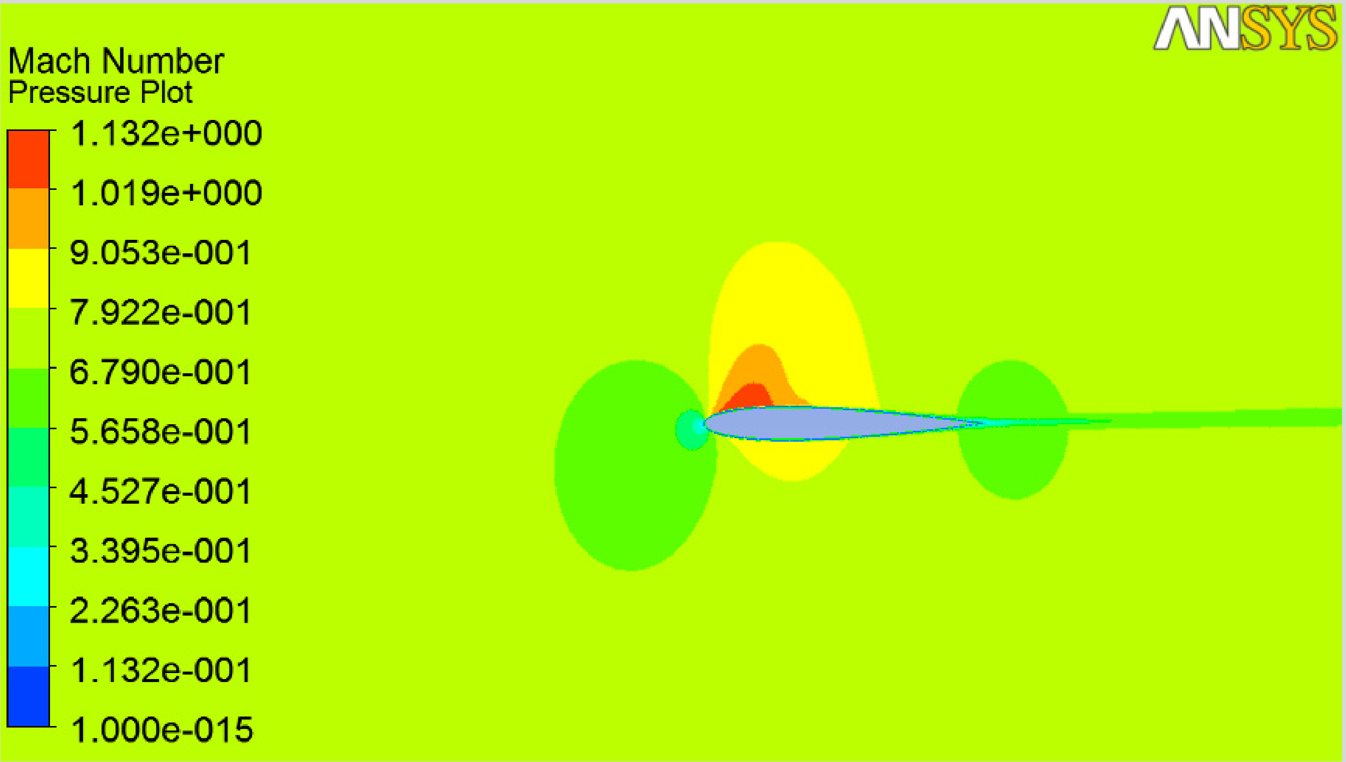


Introduction | Setup | Solution | **Results** | Summary

Post Processing

Examine the contour of Mach Number

Notice that the flow is locally supersonic (Mach Number > 1) as the flow accelerates over the upper surface of the wing



Introduction | Setup | Solution | **Results** | Summary

Wrap-up

This workshop has shown the basic steps that are applied during CFD simulations:

- Defining material properties.
- Setting boundary conditions and solver settings
- Running a simulation whilst monitoring quantities of interest
- Postprocessing the results

One of the important things to remember in your own work is, before even starting the ANSYS software, is to think WHY you are performing the simulation:

- What information are you looking for?
- What do you know about the flow conditions?

In this case we were interested in the lift (and drag) generated by a standard airfoil and how well the solver predicted these when compared to high quality experimental data

Knowing your aims from the start will help you make sensible decisions of how much of the part to simulate, the level of mesh refinement needed, and which numerical schemes should be selected

References

T.J. Coakley, “Numerical Simulation of Viscous Transonic Airfoil Flows,” NASA Ames Research Center, AIAA-87-0416, 1987

C.D. Harris, “Two-Dimensional Aerodynamic Characteristics of the NACA 0012 Airfoil in the Langley 8-foot Transonic Pressure Tunnel,” NASA Ames Research Center, NASA TM 81927, 1981

