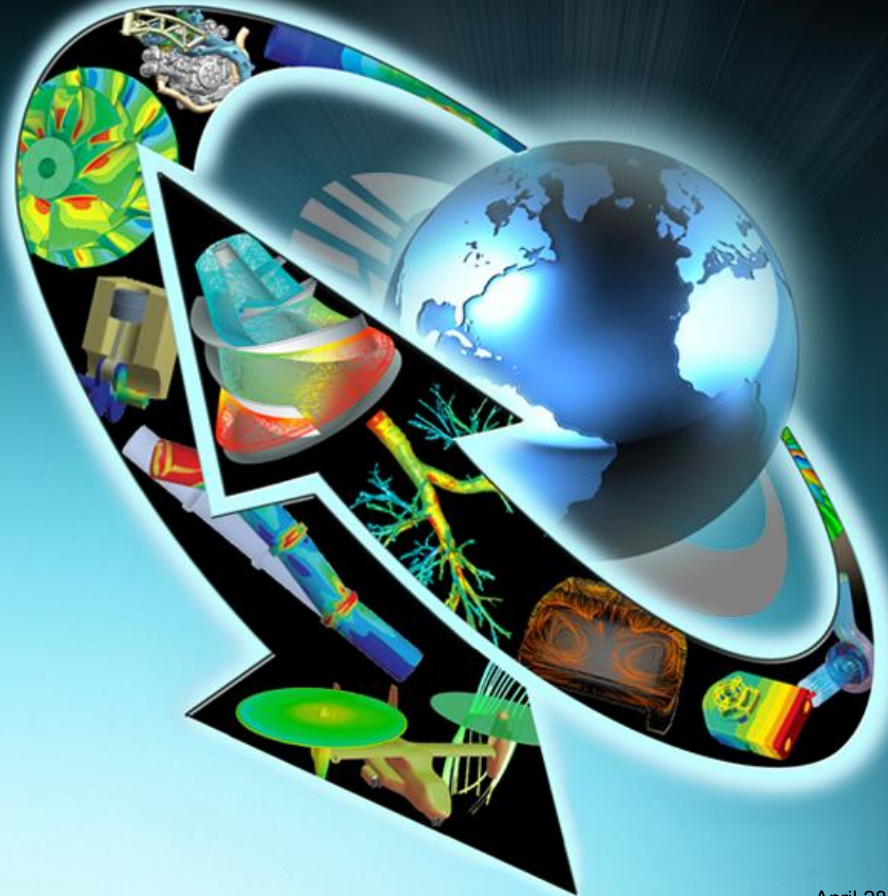




## Workshop 2

### Transonic Flow Over a NACA 0012 Airfoil.

## Introduction to CFX



# Goals

The purpose of this tutorial is to introduce the user to modelling flow in high speed external aerodynamic applications.

In this case the flow over a NACA 0012 airfoil at an angle of attack of  $1.49^\circ$  will be simulated and the lift and drag values will be compared to published results. These results were taken with a Reynolds number of  $9 \times 10^6$  and a chord length of 1m.\*

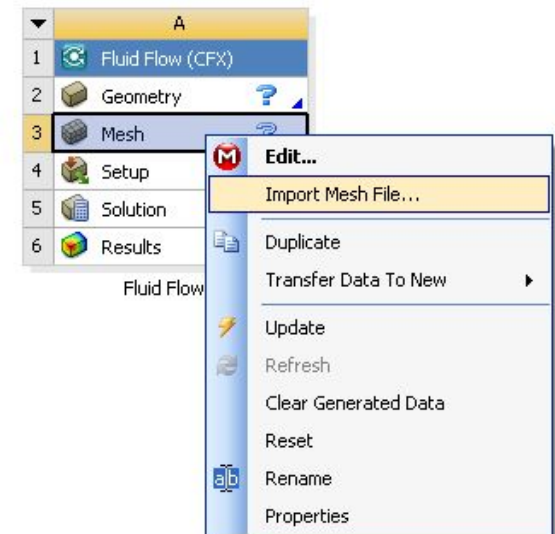
The airfoil is travelling at Mach 0.7 so the simulation will need to account for compressibility as well as turbulence effects.

To reduce the computational cost, the mesh will be made up of a 2D slice through the airfoil (one element thick).

\* NASA TM 81927 Two-Dimensional Aerodynamic Characteristics of the NACA 0012 Airfoil in the Langley 8-Foot Transonic Pressure Tunnel 1981.  
Harris, C. D.


# Start a Workbench project

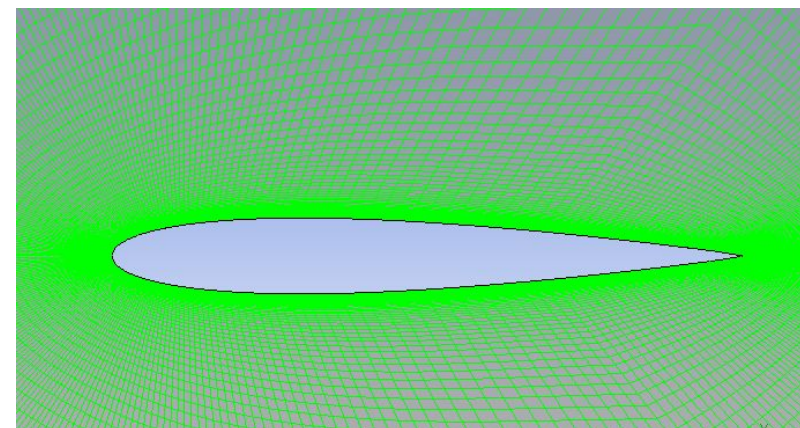
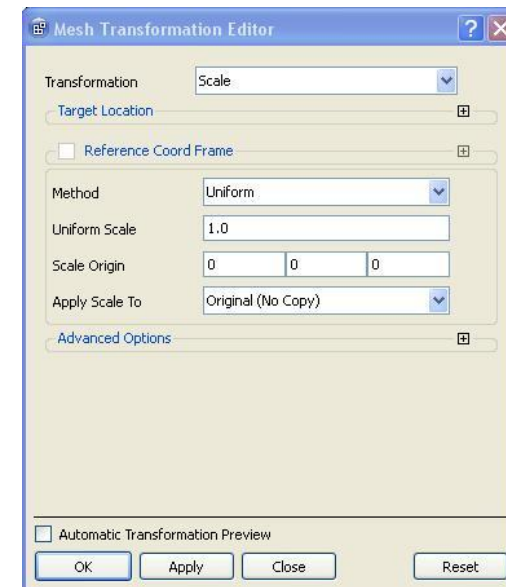
1. Launch Workbench
2. Save the new project as *naca0012* in your working directory
3. Drag a *Fluid Flow (CFX)* module from the *Analysis Systems* section of the *Toolbox* onto the *Project Schematic*
4. In the *Project Schematic* right-click on the *Mesh* cell and select *Import Mesh File*
5. Set the file filter to *FLUENT Files* and select *NACA0012.msh*
  - With the mesh file imported the *Geometry* cell will not be needed so it is removed from the *Fluid Flow* module.



Note that you could have dragged *Component System* > *CFX* onto the *Project Schematic*, as in the first workshop. The mesh would then be imported after starting CFX-Pre.

# Mesh Modification

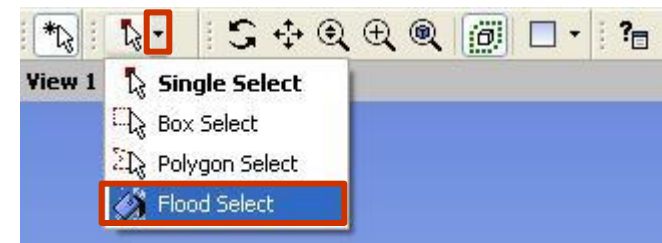
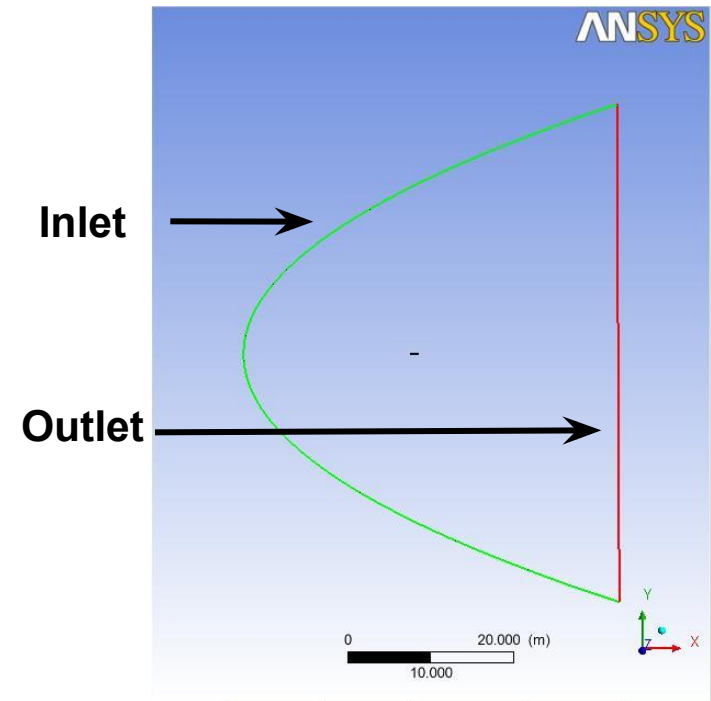
1. Open CFX-pre by right-clicking on the *Setup* cell and selecting *Edit*
  - After CFX-Pre has opened the mesh can be examined and it is clear that the scale is incorrect as the airfoil chord is 1000 m rather than 1 m, indicating the mesh was built in mm rather than m. This can be fixed using the mesh transformation options.
2. Right-click on *Mesh* in the *Outline* tree and select *Transform Mesh*
3. Change the *Transformation* to *Scale*
4. Leave the method to *Uniform* and enter a *Uniform Scale* of 0.001
5. Click *OK*
6. Select the *Fit View* icon  from the Viewer toolbar
  - Zoom in further to see the airfoil



# Mesh Modification

The mesh has been built to have a single boundary around the entire outer edge. This needs to be split into inlet and outlet regions. While it is better to create the correct mesh regions when generating the mesh, CFX-Pre can be used to modify the mesh regions.

1. Right-click *Mesh* in the *Outline* tree and select *Insert > Primitive Region*
2. Click on the *Start Picking* button
3. From the drop down selection menu select *Flood Select* (see image to the right)
4. In the viewer select any element from the front curved boundary
  - The flood fill will select all cells where the change in angle is less than  $30^\circ$
5. Click in the *Move Faces To* field and type *Inlet*
6. Click *OK*

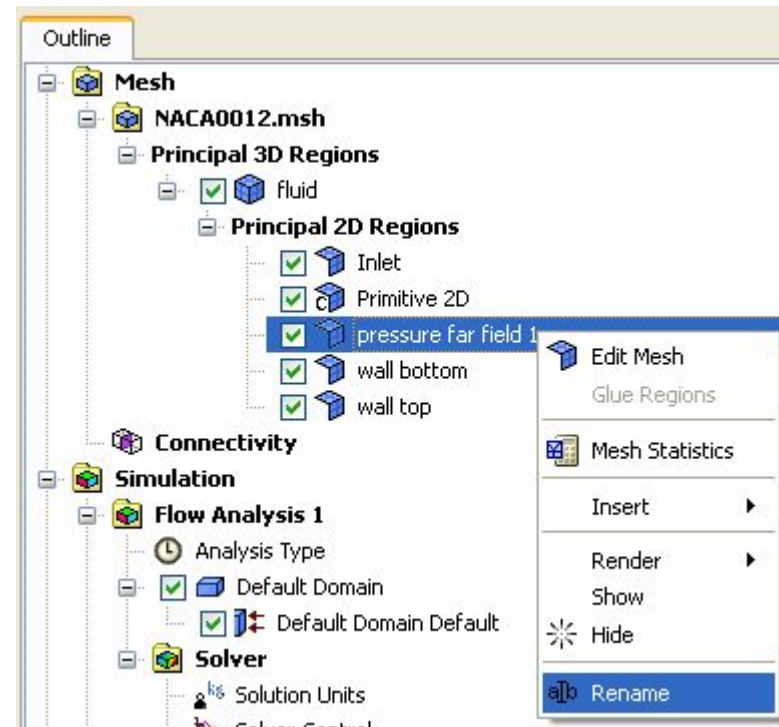




# Mesh Modification

The remaining section will now be renamed “Outlet”.

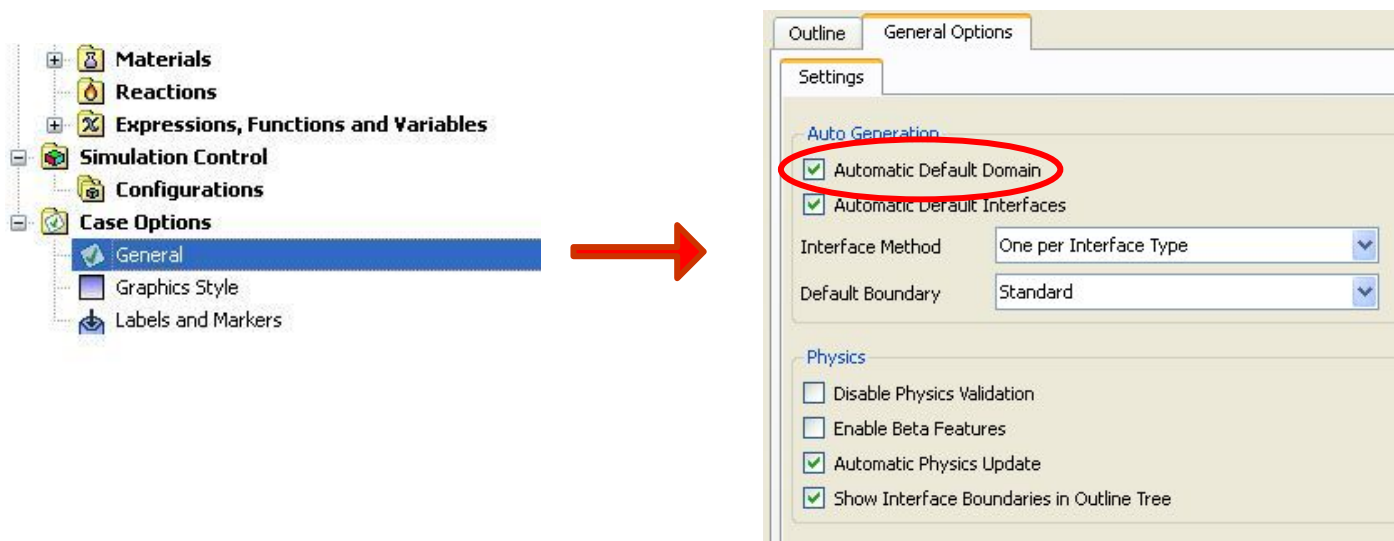
1. Expand the *Mesh* section of the tree so the list of *Principal 2D Regions* is visible. Note that this list now contains the location *Inlet*
2. Click on the region *pressure far field 1* to confirm it is the region representing the outlet
  - It will be highlighted in the Viewer
3. Right-click on *pressure far field 1* and select *Rename*. Change the name to *Outlet*



# Domain Setup

Usually the option to automatically generate domains is active, this can be checked by editing *Case Options* > *General* in the Outline tree.

1. Check that *Automatic Default Domain* is active the click OK.



2. Right-click on *Default Domain* in the Outline tree and rename it to *Fluid*
3. Double-click on *Fluid* to edit the domain settings

# Domain Setup

This case involves high speed aerodynamics so it is important to include compressibility.

It is important to set the correct operating pressure so that the intended Reynolds number is achieved. The simulation will take place at 288 [K] in air; this allows the speed of sound to be calculated. This can then be converted into a free-stream velocity using the Mach number. Using the definition for Reynolds number the fluid density can be obtained, which can then be used to determine the operating pressure for the simulation, assuming an ideal gas.

$$c = \sqrt{\gamma RT} = \sqrt{1.4 \times 287 \times 288} = 340.17 [m/s]$$

$$u = M \times c = 0.7 \times 340.17 = 238.12 [m/s]$$

$$Re = \frac{\rho u l}{\mu} \Rightarrow \rho = \frac{Re \mu}{u l} = \frac{9e^6 \times 1.82e^{-5}}{238.12 \times 1} = 0.688 [kg/m^3]$$

$$P = \rho R T = 0.688 \times 287 \times 288 = 56867 [Pa]$$

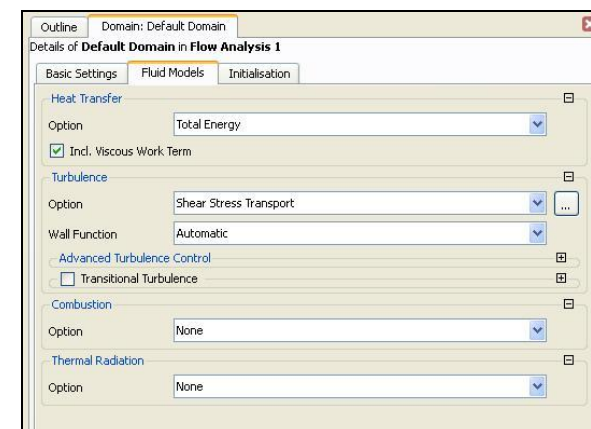
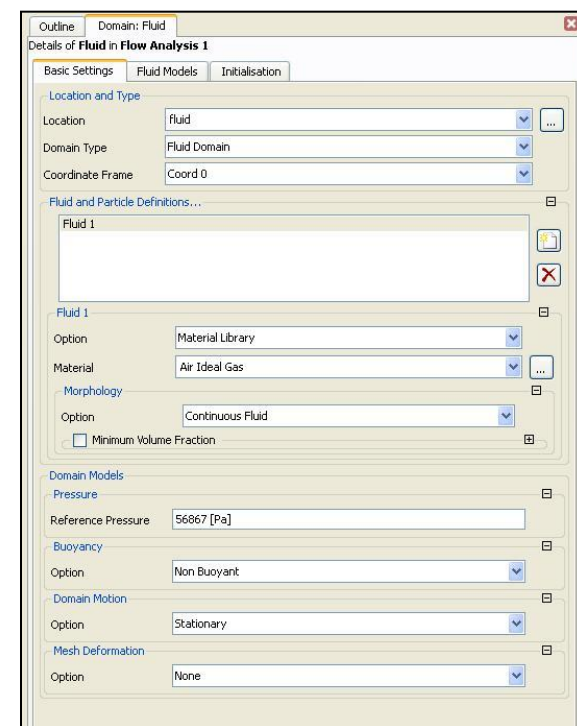
$c$  = Speed of sound  
 $R$  = Gas Constant  
 $\gamma$  = Ratio of specific heats  
 $T$  = Temperature  
 $u$  = Free-stream velocity  
 $M$  = Mach number  
 $Re$  = Reynolds number  
 $\mu$  = Dynamic viscosity  
 $\rho$  = Density



# Domain Setup

In the *Fluid* domain *Basic Settings* tab:

1. Set the Material to *Air Ideal Gas*
2. Set the *Reference Pressure* to 56867 [Pa]
  - Make sure you change set the units
3. Move to the *Fluid Models* tab
4. Set the *Heat Transfer Option* to *Total Energy*
  - This is required for compressible simulations
5. Enable *Incl. Viscous Work Term*
  - This includes viscous heating effects
6. Set the *Turbulence Option* to *Shear Stress Transport*
7. Click OK



# Boundary Conditions

An outlet relative pressure of 0 [Pa] will now be applied. This pressure is relative to the operating pressure of 56867 [Pa].

$$\text{Absolute Pressure} = \text{Reference Pressure} + \text{Relative Pressure}$$

1. Right-click on the domain *Fluid* in the Outline tree and select *Insert > Boundary*, naming the boundary *Outlet*
2. Change the *Boundary Type* to *Outlet* and check that the location is set to *Outlet*
3. Move to the *Boundary Details* tab and set the *Mass and Momentum* option to *Average Static Pressure* with a value of 0 [Pa]
4. Click *OK*

The sides of the domain will use symmetry conditions since this is a 2D simulation.

1. Insert a *Symmetry* boundary called *Sym Left*, at the location *sym left*
2. Insert a *Symmetry* boundary called *Sym Right*, at the location *sym right*

# Boundary Conditions

The mesh has been constructed so that the airfoil is at 0° angle of attack. To apply the required angle of 1.49° the flow direction at the inlet must be adjusted. The values will be created using expressions.

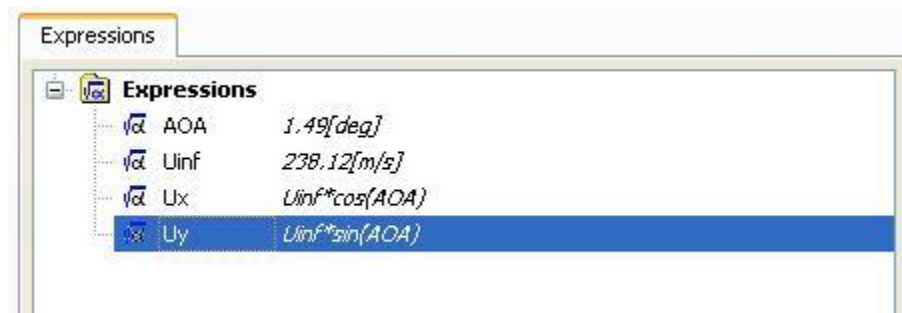
1. Right-click on *Expression* in the tree and select *Insert > Expression*. Call it *Uinf*.
2. Set the *Definition* to  $238.12 [m\ s^{-1}]$  then click *Apply*
  - All expressions must have the appropriate dimensions
3. In the expression editor add the following expressions by right-clicking on *Expressions* and selecting *Insert > Expression*

$$AOA = 1.49[deg]$$


$$U_x = U_{inf} \cos(AOA)$$

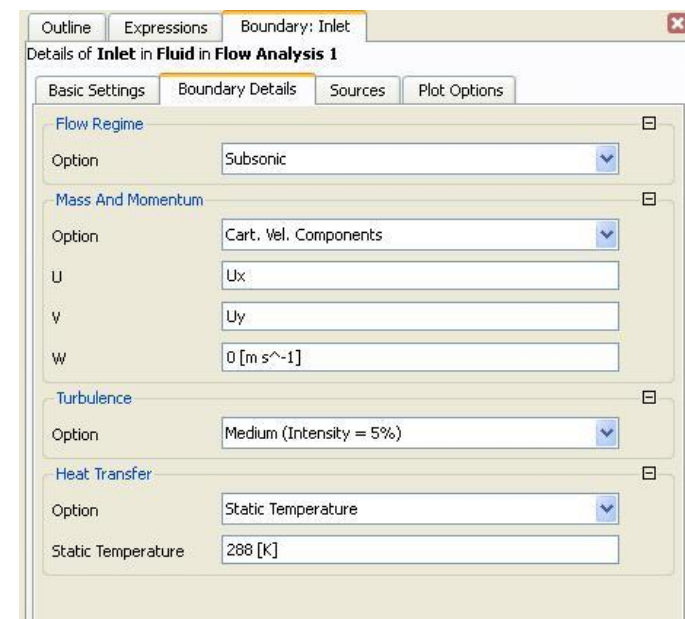
$$U_y = U_{inf} \sin(AOA)$$

4. Return to the main Outline tree



# Boundary Conditions

1. Right-click on *Fluid* and insert a boundary called *Inlet*
2. The *Boundary Type* should be set to *Inlet* by default and a *Location* of *Inlet* should also be selected by default
3. Move to the *Boundary Details* tab
4. Change the *Mass and Momentum* option to *Cart. Vel. Components*
5. Enter the U, V and W values as  $U_x$ ,  $U_y$  and  $0 [m s^{-1}]$ 
  - Use the Expression icon  to allow the  $U_x$  and  $U_y$  expressions to be entered
6. Set *Static Temperature* = 288 [K]
7. Click OK

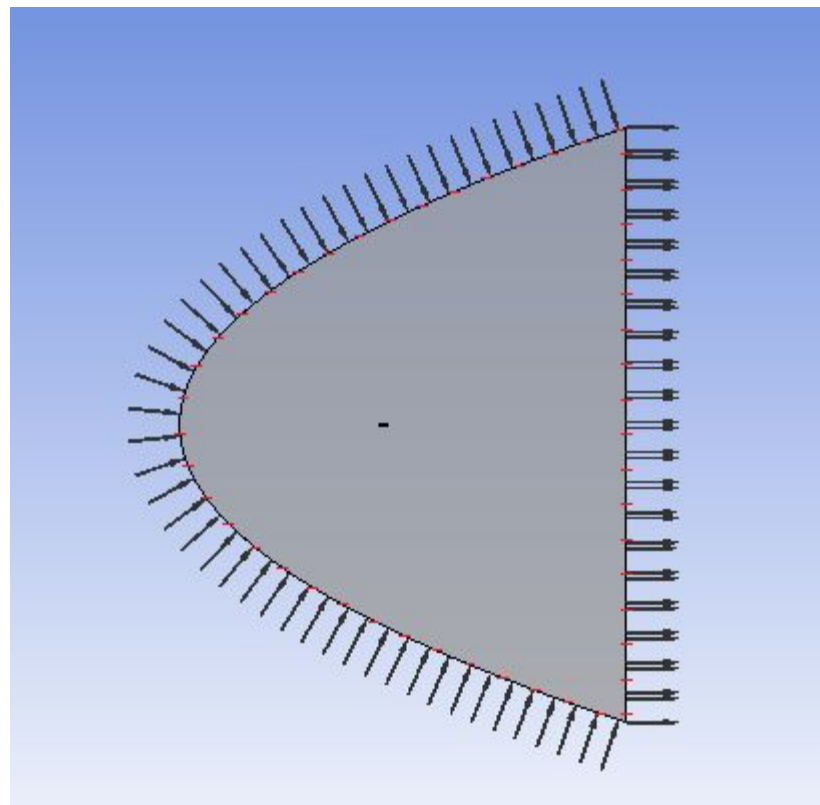


# Boundary Conditions

The Viewer indicates the locations of the inlet and outlet boundaries. Note that the arrows do not represent the applied flow direction.

The final boundary condition is the wall around the airfoil. This should already exist as *Fluid Default*.

1. Edit *Fluid Default* to check that only the *wall bottom* and *wall top* regions remain in the default boundary
2. Click *Close*
3. Rename *Fluid Default* to *Airfoil*





# Monitors

For this simulation the lift and drag are the quantities of interest, so monitor points will be added to track their values and ensure they reach a steady value. The lift and drag coefficients will be created using expressions. Remember that the free-stream flow is offset from the x-direction so the forces will have to be adjusted to account for the angle of attack.

1. Enter the following expressions, **or** select *File > Import > CCL* and load the file *Airfoil.ccl*. If loading the CCL file, use the *Append* option as shown

$F_y = \text{force}_y()@Airfoil$

$F_x = \text{force}_x()@Airfoil$

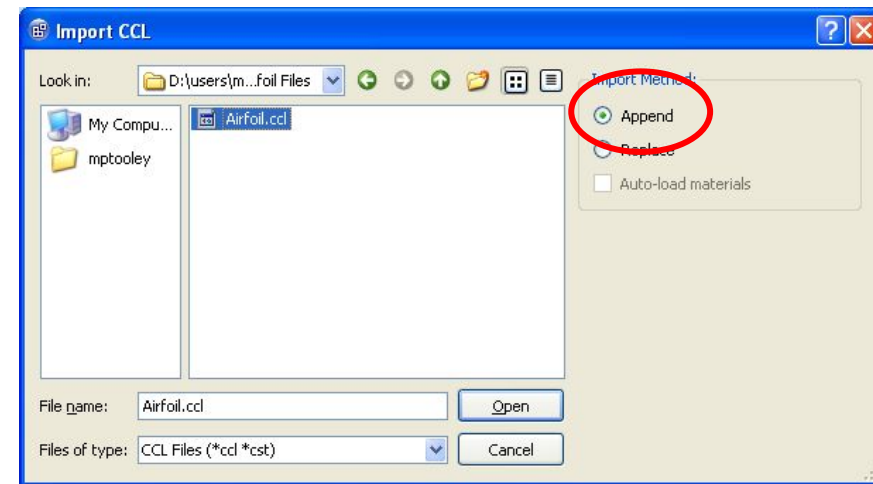
$Lift = \cos(AOA) * F_y - \sin(AOA) * F_x$

$Drag = \cos(AOA) * F_x + \sin(AOA) * F_y$


$Denom = 0.5 * \text{massFlowAve}(\text{Density})@Inlet * U_{inf}^2 * 1[m] * 0.1[m]$

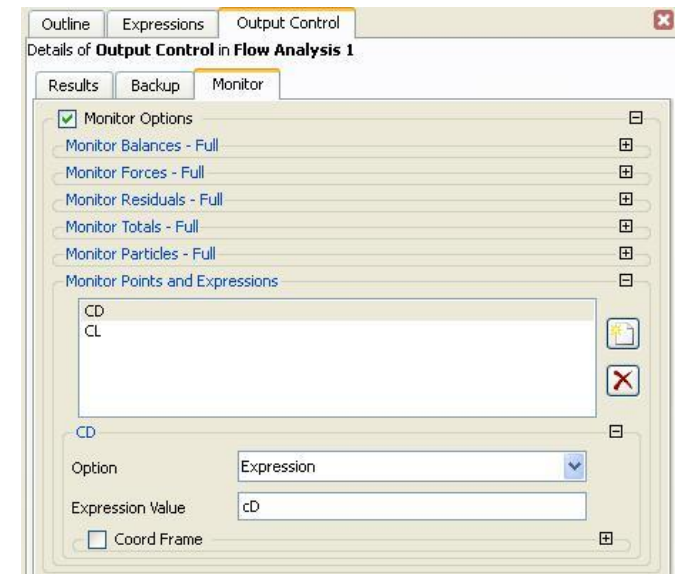
$c_L = Lift / Denom$

$c_D = Drag / Denom$



# Monitors

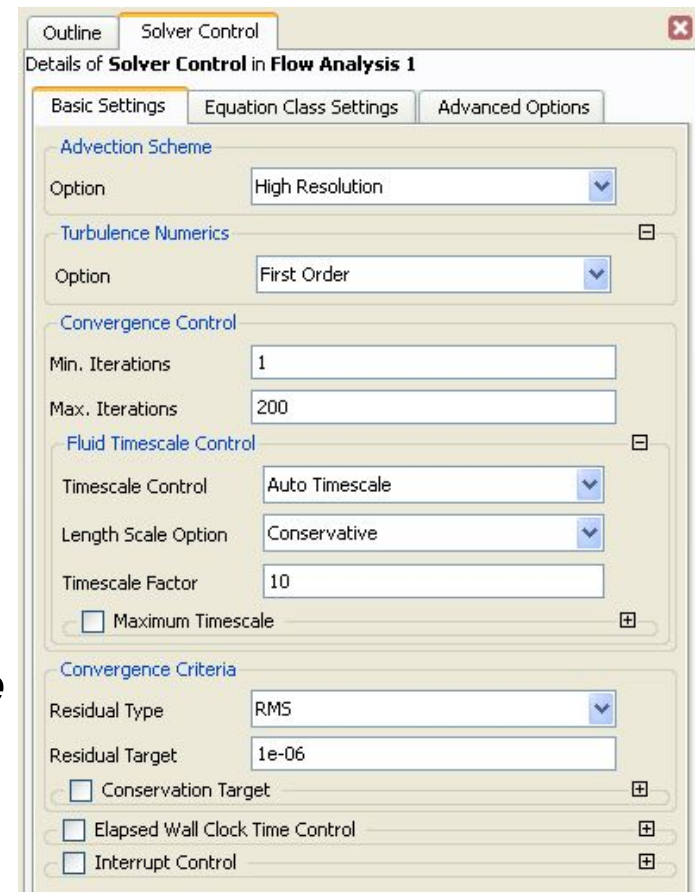
1. Edit *Output Control* from the *Outline* tree and go to the *Monitor* tab
2. Check the *Monitor Options* box
3. Click on the *Add New Item* icon  and name it *CL*
4. Set the *Option* to *Expression* and enter *cL*
  - This is the monitor point for the Coefficient of Lift. Note that all names and expressions are case sensitive, so the monitor point is named “CL” and it refers to the expression named “cL”.
5. Add a new item called *CD* and set it to the expression *cD*
  - This is the Coefficient of Drag
6. Click *OK*



# Solver Control

Open the *Solver Control* section from the Outline tree


1. Increase the *Max. Iterations* to 200
2. Change the *Timescale Factor* from 1 to 10
  - A larger timescale can accelerate convergence, but too large a timescale will cause the solver to fail
3. Set the *Residual Target* to  $1e-6$ 
  - This is a tighter convergence criteria and is discussed further below
4. Click *OK*
5. This case is now ready to run so click on *File > Save Project* then close the CFX-Pre window to return to the main Workbench window



# Running the Simulation

1. In Workbench right-click on *Solution* and select *Update*
2. After the solver has started right-click on *Solution* again and select *Display Monitors*
  - This will open the Solver Manager and allow the residuals and monitors to be viewed

In the Solver Manager the *User Points* tab displays the monitor points setup in the *Output Control* section of CFX Pre. This will include the values of *CL* and *CD*. These should converge to a steady value before the convergence criteria is met. Otherwise the run should be extended. Many cases will be converged when an RMS residual level of  $1e-4$  is reached. For this case this is inadequate since the lift and drag had not reached steady values when the residuals were at  $1e-4$ , hence a tighter convergence criteria was used.

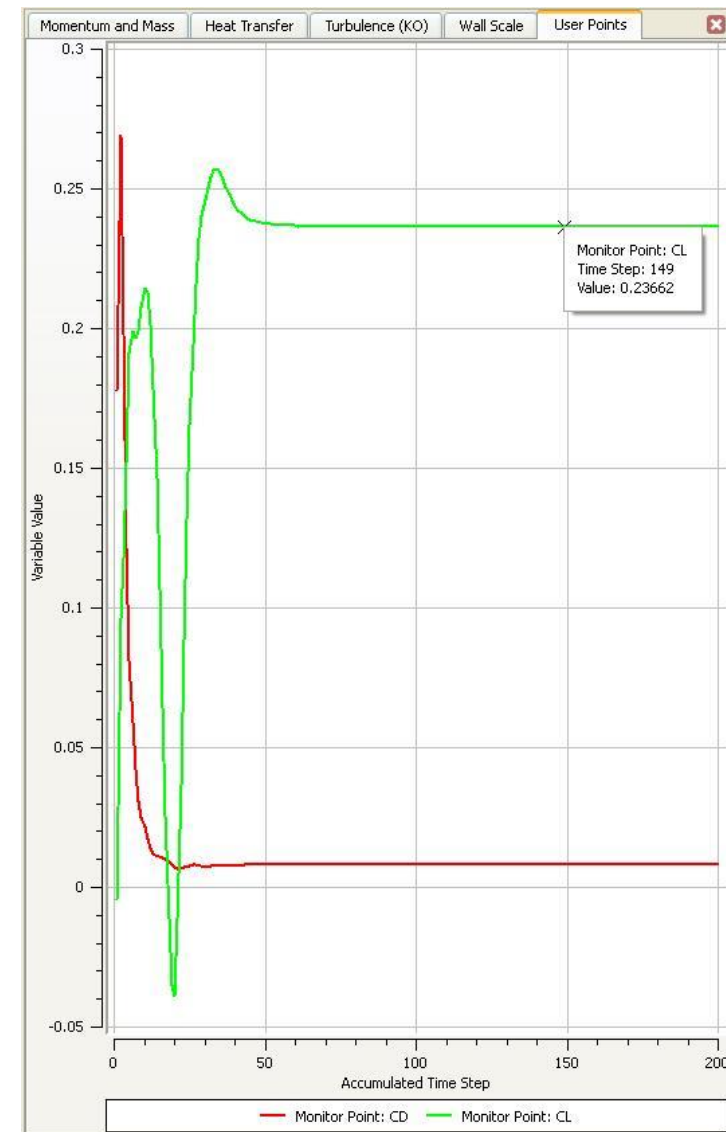
3. Check through all of the residuals and monitor values. The values of *CD* and *CL* become steady after about 50 iterations. You can click the *Stop* button  from the toolbar to stop the run at this point.

# Monitor Values

Before exiting the Solver Manager the converged values of CL and CD can be viewed by clicking on the monitor lines.

The values extracted should be CL=0.236 and CD=0.0082. These values compare well to published values\* of CL=0.241 and CD=0.0079.

Now close the solver and return to the Workbench window.



\* AIAA-87-0416 Numerical Simulation of Viscous Transonic Airfoil Flows 1987. Thomas J Coakley, NASA AMES Research Centre.

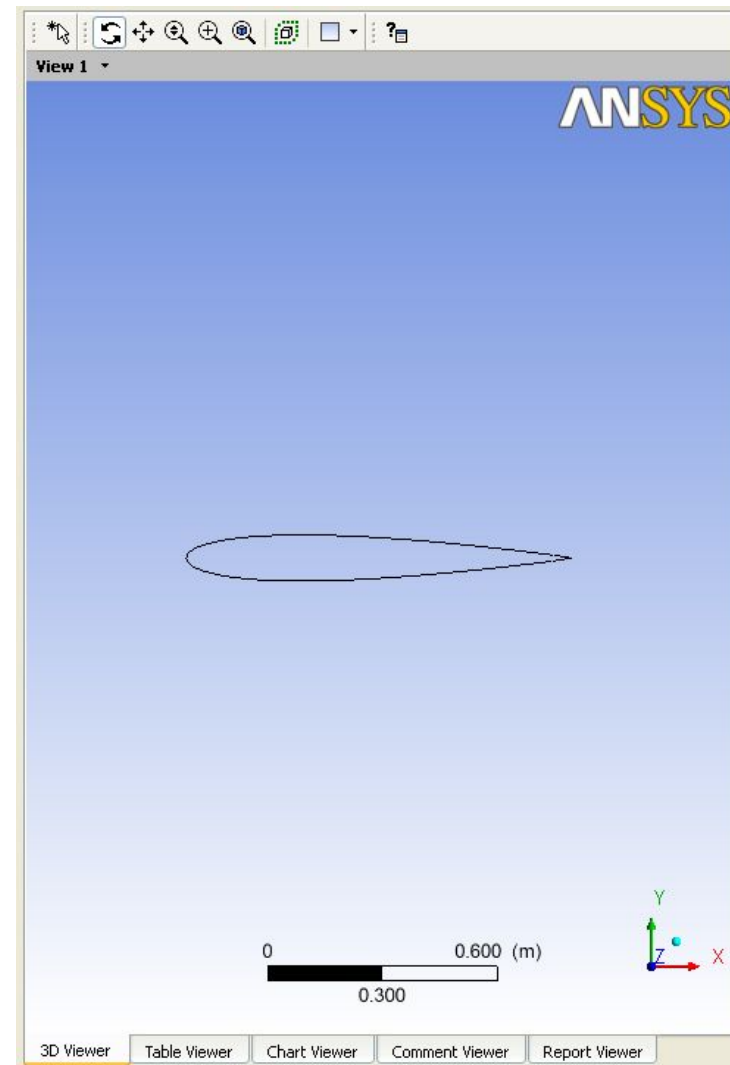


# Post-processing

1. Right-click in the *Results* cell and select *Edit* to open CFD-Post. The results should automatically be loaded

This case required a large domain to allow the boundary conditions to be imposed without a large artificial restriction on the flow. However during post-processing the main interest will be in the flow close to the airfoil.

2. Click on the Z-axis in the bottom right corner of the Viewer to orientate the view
3. Use the box zoom (right mouse button) so the Viewer displays the region around the airfoil

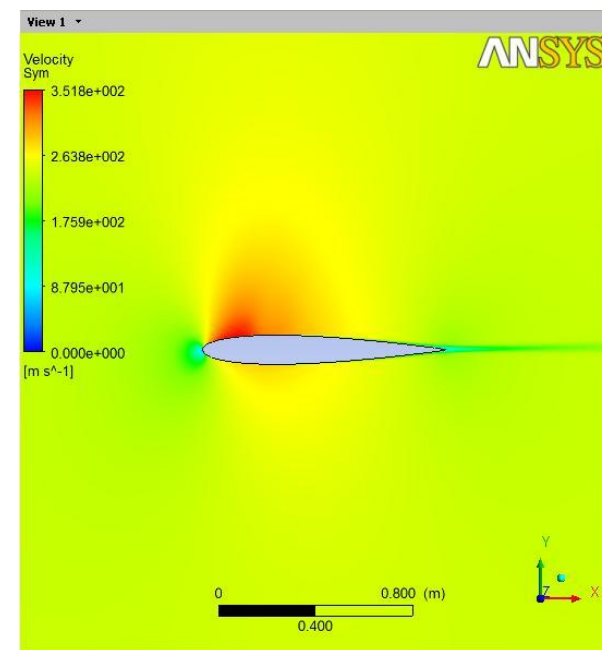


# Post-processing

When looking at the flow around an airfoil, plots of several variables can be of interest such as velocity, pressure and Mach number.


1. In the tree turn on the visibility of *Sym Left* by clicking in the check box
2. Double-click on *Sym Left* to bring up the details section
3. Under the *Colour* tab change the mode to *Variable* and select *Velocity* using the *Global Range*, then click *Apply*

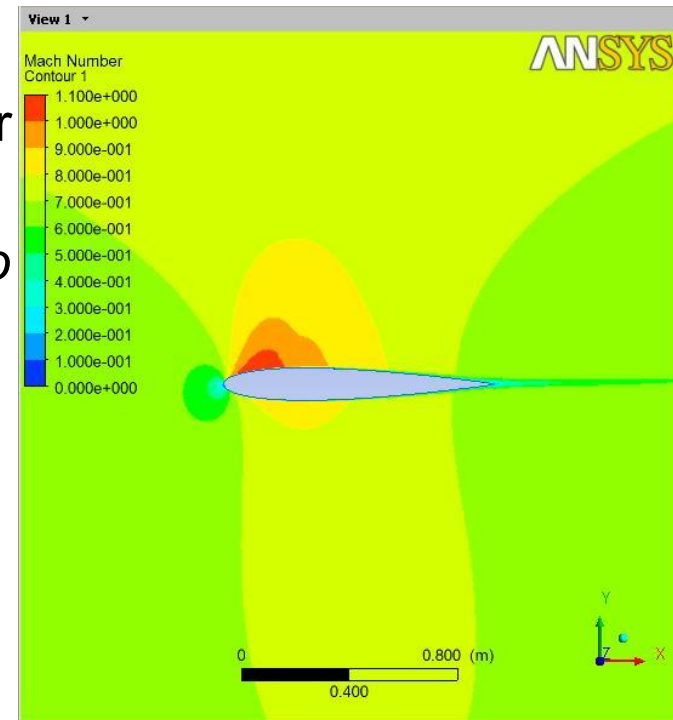
Notice that the maximum velocity is around 350 [m/s]. This is higher than the sonic speed of 340 [m/s] calculated earlier for free-stream conditions.



# Post-processing

To plot the Mach number a contour plot will be used so the supersonic region can clearly be identified.

1. Select *Insert > Contour* or click on the contour icon 
2. Accept the default name then set *Locations to Sym Left* and the *Variable to Mach Number*
3. Change the *Range to User Specified* and enter *0 to 1.1* as the range
4. Set *# of Contours to 12*, then click *Apply*
5. Turn off the Visibility of *Sym Left* so that the previous velocity plot is hidden
6. Try plotting other variables such as Pressure or Density, use the Local or Global Range when limits are not known



# Post-processing

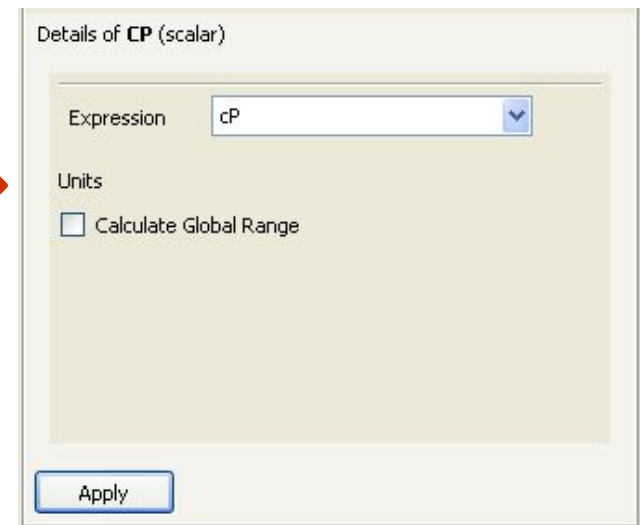
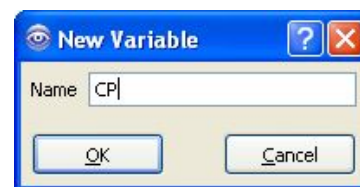
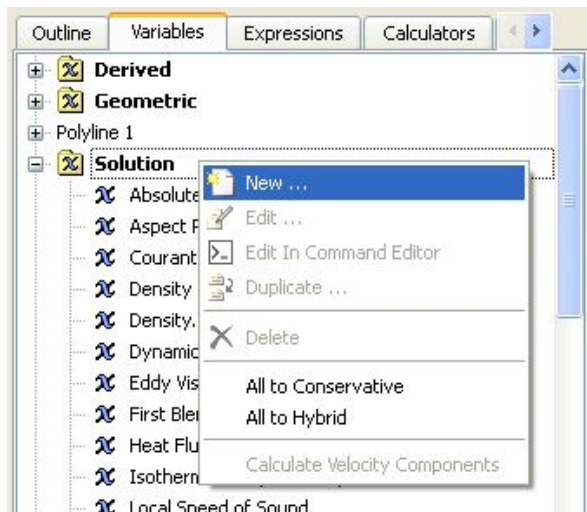
To plot the pressure coefficient distribution around the airfoil a polyline is needed to represent the airfoil profile and a variable needs to be created to give CP.

1. Create a Polyline using *Location > Polyline* from the toolbar
2. Change the *Method* to *Boundary Intersection*
3. Set *Boundary List* to *Airfoil*
4. Set *Intersect With* to *Sym Left* and then click *Apply*
5. Turn off visibility of the previous created Contour plot to see the Polyline

A line will be created around one end of the airfoil. For full 3D cases XY planes can be create at various span locations and used to extract Polylines.

# Post-processing


5. Move to the *Expressions* tab and right-click to create a new expression named  $cP$  with the definition:  
 $Pressure/(0.5*massFlowAve(Density)@Inlet*Uinf^2)$
6. Move to the *Variables* tab and right-click to create a new variable named  $CP$ .
7. Set the *Method* to *Expression* and select  $cP$ . Click *OK*.





# Post-processing

A chart showing the pressure distribution around the airfoil will now be created.

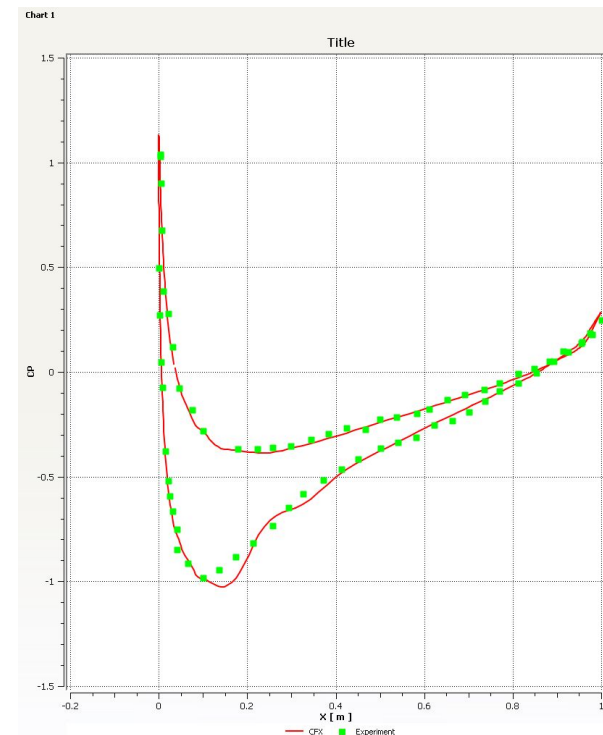
1. Insert a chart using *Insert > Chart* or selecting 
2. In the *General* tab leave the type as *XY*
3. Move to the *Data Series* tab and enter a new series. Set the location to *Polyline 1*
4. Move to the *X Axis* tab and change the variable to *X*
5. Move to the *Y Axis* tab and change the variable to *CP*
6. Click *Apply* and the chart is generated

These values can be compared with experimental results.\*

\* AIAA-87-0416 Numerical Simulation of Viscous Transonic Airfoil Flows 1987. Thomas J Coakley, NASA AMES Research Centre.

# Post-processing

7. Return to the *Data Series* tab and change the name to *CFX*
8. Insert a new series and give it the name *Experiment*
9. Change the *Data Source* to *File* and select the file *CP.csv*
10. On the *Line Display* tab, set *Line Style* to *None* and *Symbols* to *Rectangle*. Also ensure that *Symbol Colour* is a different colour from the currently plotted CFX line
11. Click *Apply* and both data series are drawn



# Summary

The workshop has covered:

- Loading an existing mesh
- Scaling the mesh
- Generating New Regions from existing 2D Primitives
- Setting up and running a high speed compressible flow simulation over an airfoil
- Extracting lift and drag forces and comparing with experimental data
- Examining the flow patterns around the airfoil
- Comparing the pressure distribution to experimental values

# Scope for further work.

This simulation is a good match to experimental work but further steps could be taken if required, including:

- Refining the mesh, particularly in the wake region.
- Applying a transition model to account for the small region of laminar flow around the nose of the airfoil.
- Adding additional airfoil features such as a finite thickness trailing edge that will be used on all “real airfoils”.
- Simulating the whole wing to account for spanwise variations.

Adding more features to a simulation will usually increase the computational cost, so one of the most important step in any simulation is to decide which features need to be included and which can be left out.