Customer Training Material

Workshop 2

Transonic Flow Over a NACA0012 Airfoil

Introduction to ANSYS FLUENT

Release 13.0
December 2010

Goals

- **The purpose of this tutorial is to introduce the user to good techniques for accurately modelling flow in high speed external aerodynamic applications.**

- **Transonic flow will be modelled over a NACA 0012 airfoil for which experimental data has been published, so that a comparison can be made.**

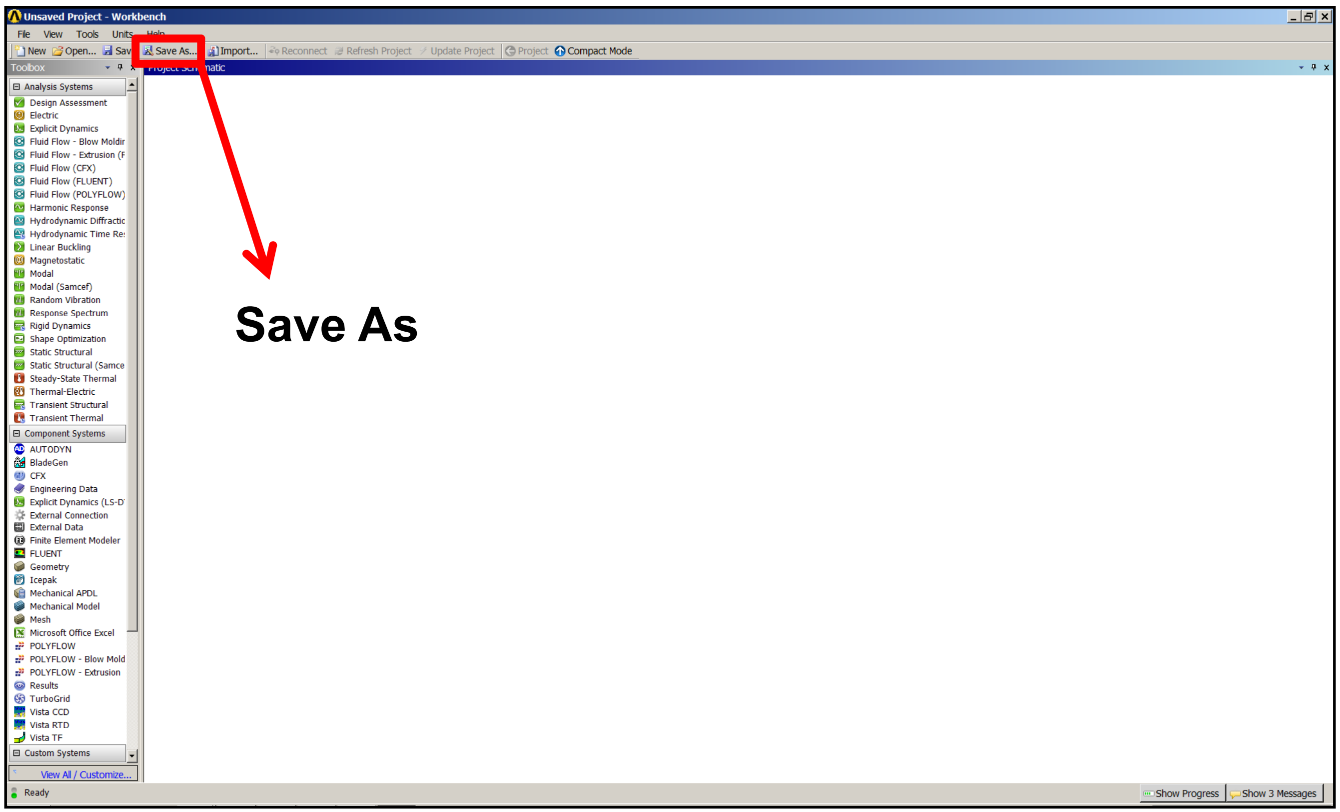
- **The flow to be considered is compressible and turbulent.**

- **The solver used is the density based implicit solver, which gives good results for high speed compressible flows.**

- **The tutorial is carried out using ANSYS FLUENT and CFD Post from within Workbench, but it could also be completed in standalone mode.**

Start a workbench project

- **Launch Workbench**
  - **START > all programs > ANSYS 13.0 > Workbench**
- **Save the new project as ‘naca0012.wbpj’ in your chosen working directory.**



Create folder structure and user files area

- **Saving the project immediately creates a folder structure which workbench will automatically manage.**

- **Included in the new folder structure is a 'user_files' area where external files can be placed. This means that these files will be archived with the project and not lost if the project is relocated.**

- **For this workshop, a mesh file has already been generated and supplied. Experimental result files and case comparison files have also been provided. Copy them all into the user_files folder:**

  working directory > naca0012_files > user_files >

  NACA0012.msh

  test-data-top.xy

  test-data-bottom.xy

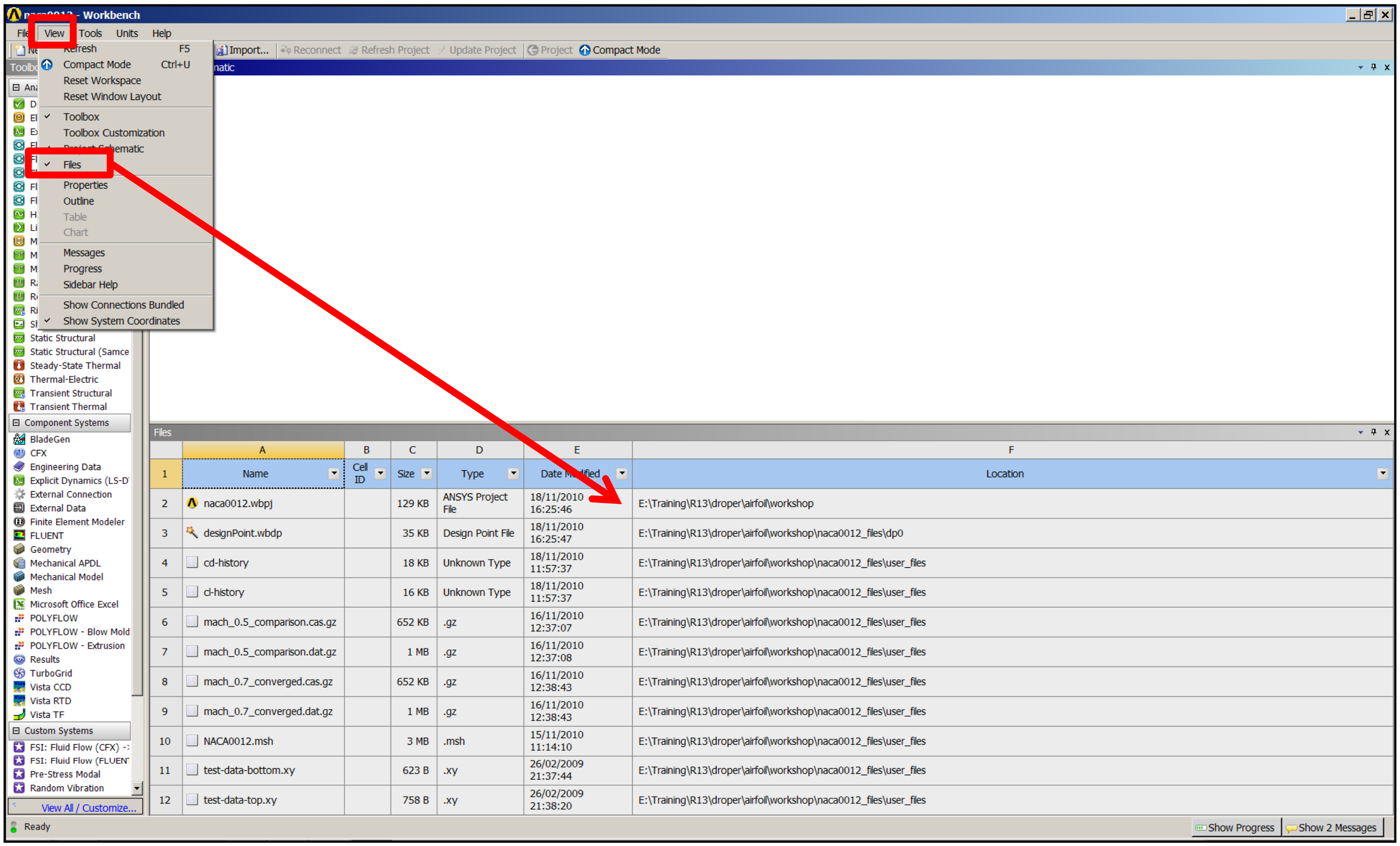
  mach_0.5_comparison.cas.gz and mach_0.5_comparison.dat.gz

  mach_0.7_converged.cas.gz and mach_0.7_converged.dat.gz

- **Shortly we will be importing the mesh file into our Fluent simulation.**

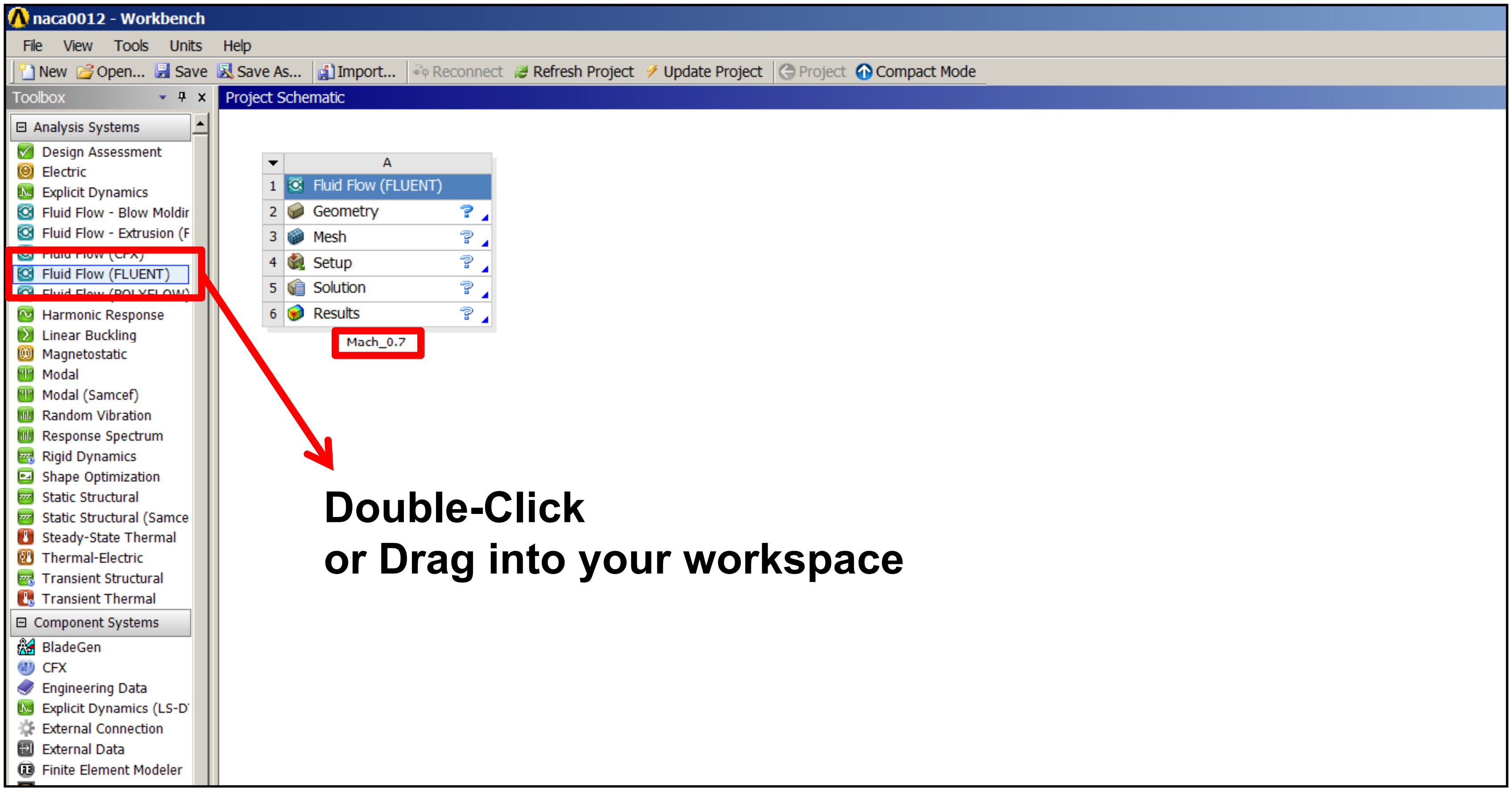
View your file locations

- **You can easily view the location of the files that are associated with your project (View > Files).**



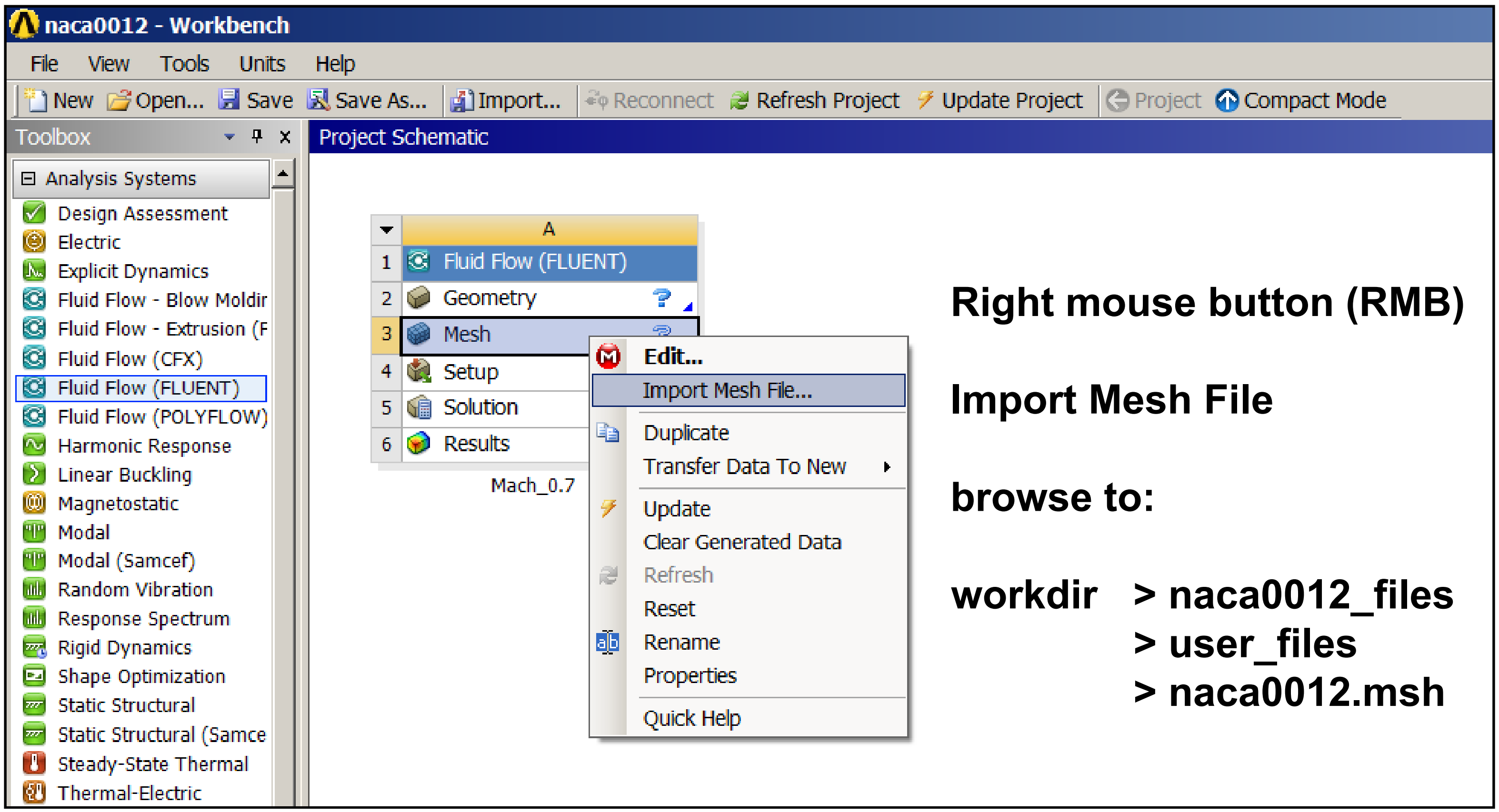
Pick a FLUENT system from the toolbox

- **Double-click or drag a Fluid Flow (FLUENT) analysis system from your toolbox into your project workspace.**
- **If you wish, rename the FLUENT analysis, e.g. Mach_0.7**



Import the supplied mesh file

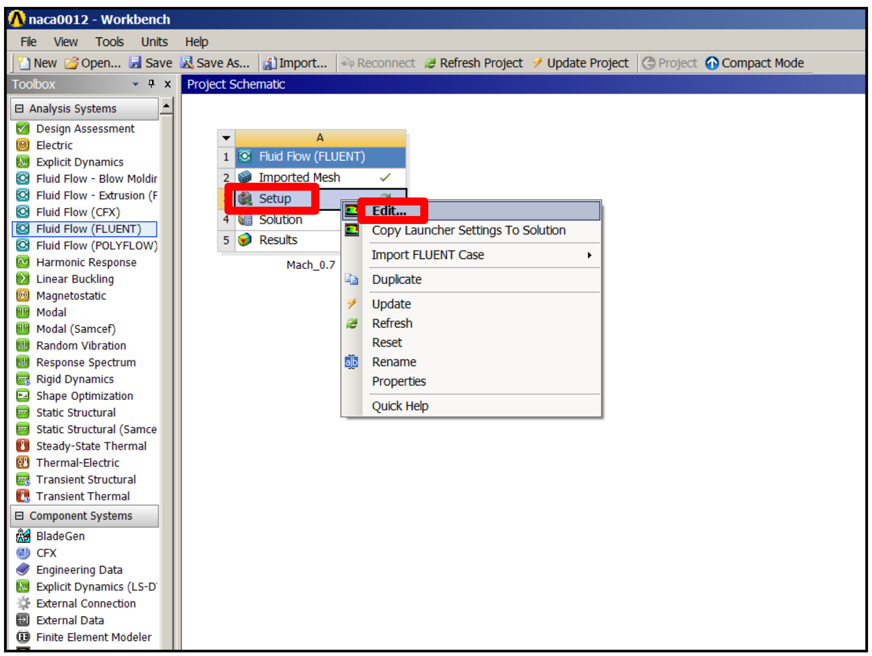
- **Import the supplied FLUENT mesh file (naca0012.msh)**
  - **Right click on Mesh (cell A3) and select 'Import Mesh File'**
  - **Browse to the mesh file which was placed into the user_files folder**



Import the supplied mesh file

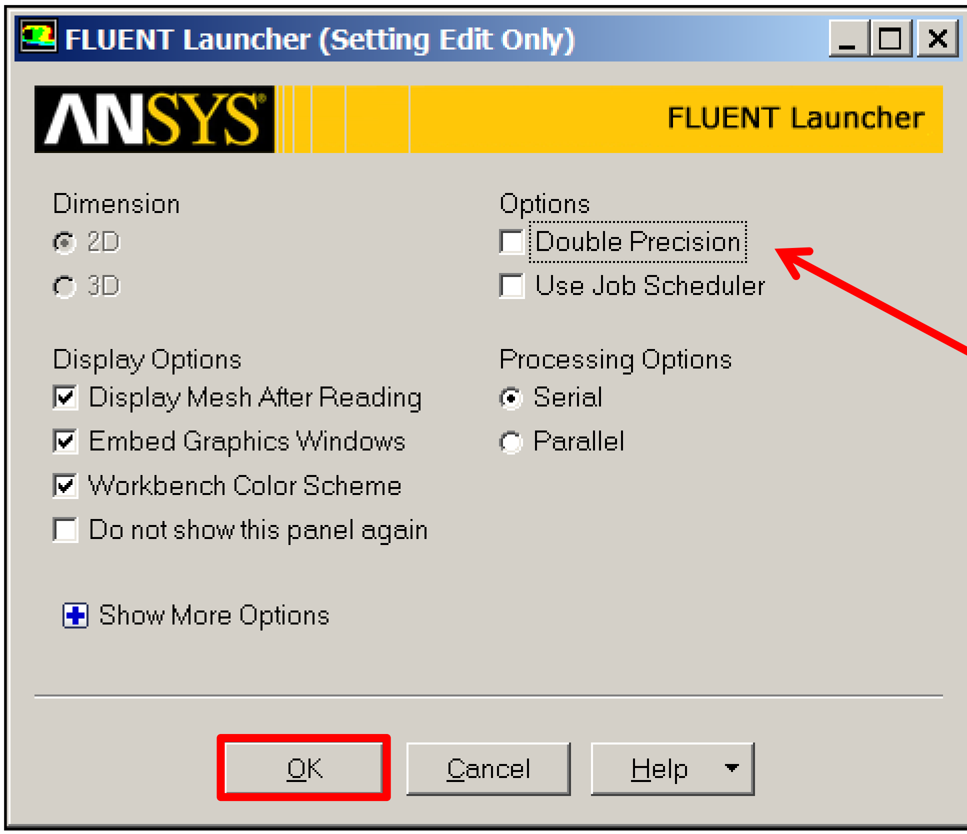
Customer Training Material

- Note that because we have imported a mesh file the geometry tab is automatically removed as it is not needed.
- Note also that the mesh cell now has a green tick to indicate that the this part of the analysis system is completed and up-to-date.
- The next cell is the setup cell, used to open FLUENT and setup the case so that it is ready to run.
- Right-click > edit (or just double-click the setup cell).



Launch FLUENT to set up the case

- **The FLUENT launcher will start.**
  - **Note that '2D' has automatically been selected based on the mesh file**
  - **Keep the default options for everything else, click OK**

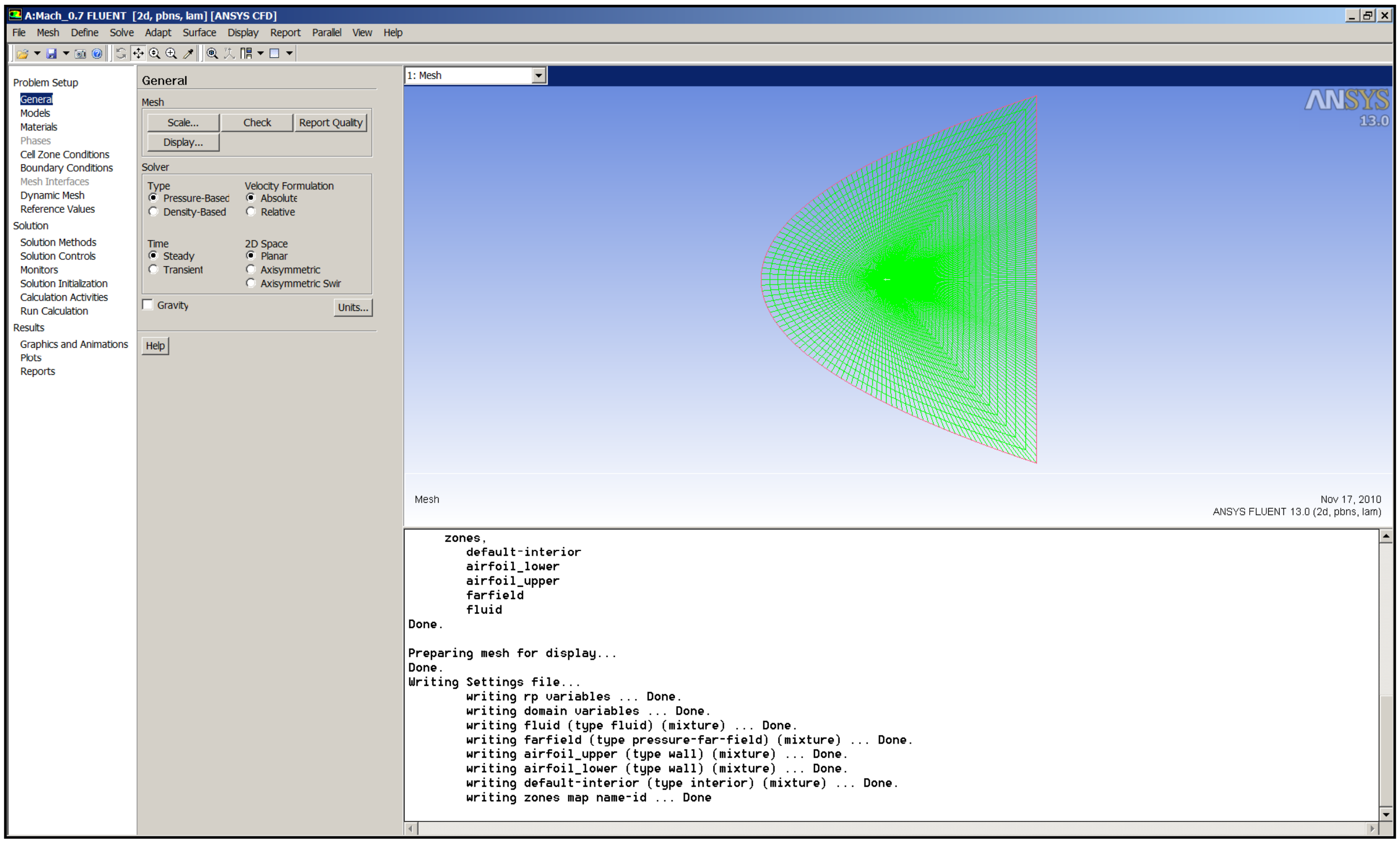


For this workshop single precision (the default) is used.

Double precision may yield more accurate drag prediction, and is recommended in real cases.

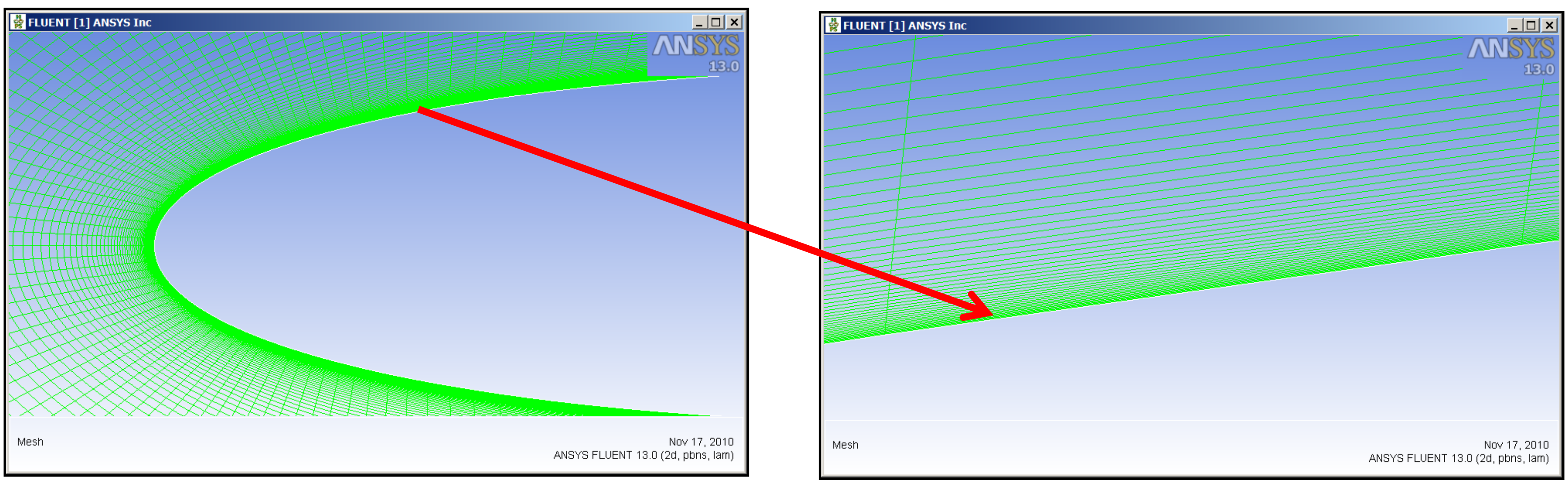
Launch FLUENT to set up the case

- **FLUENT will launch in a new window.**
- **The mesh will automatically read in and display (can be disabled).**
- **A settings file will automatically be written to the Workbench project.**



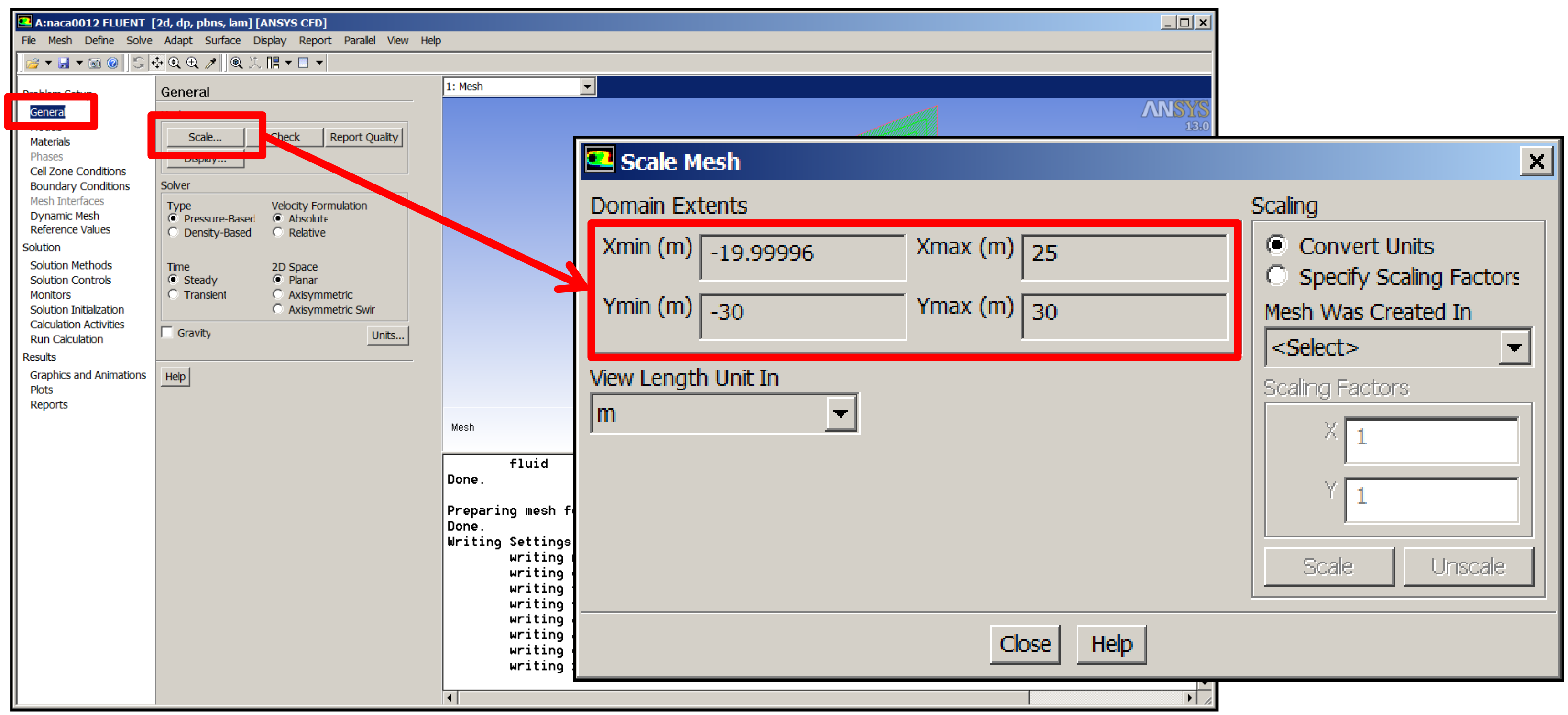
Examine the mesh

- **Zoom in and examine the mesh.**
  - **The maximum aspect ratio in this mesh is quite high, as will be examined shortly.**
  - **This is a result of placing the first grid point in the viscous sublayer close to the airfoil wall surface, for maximum accuracy in the turbulence model.**
  - **However, very long thin (stretched) cells can lead to problems, so we need to proceed with some checks.**



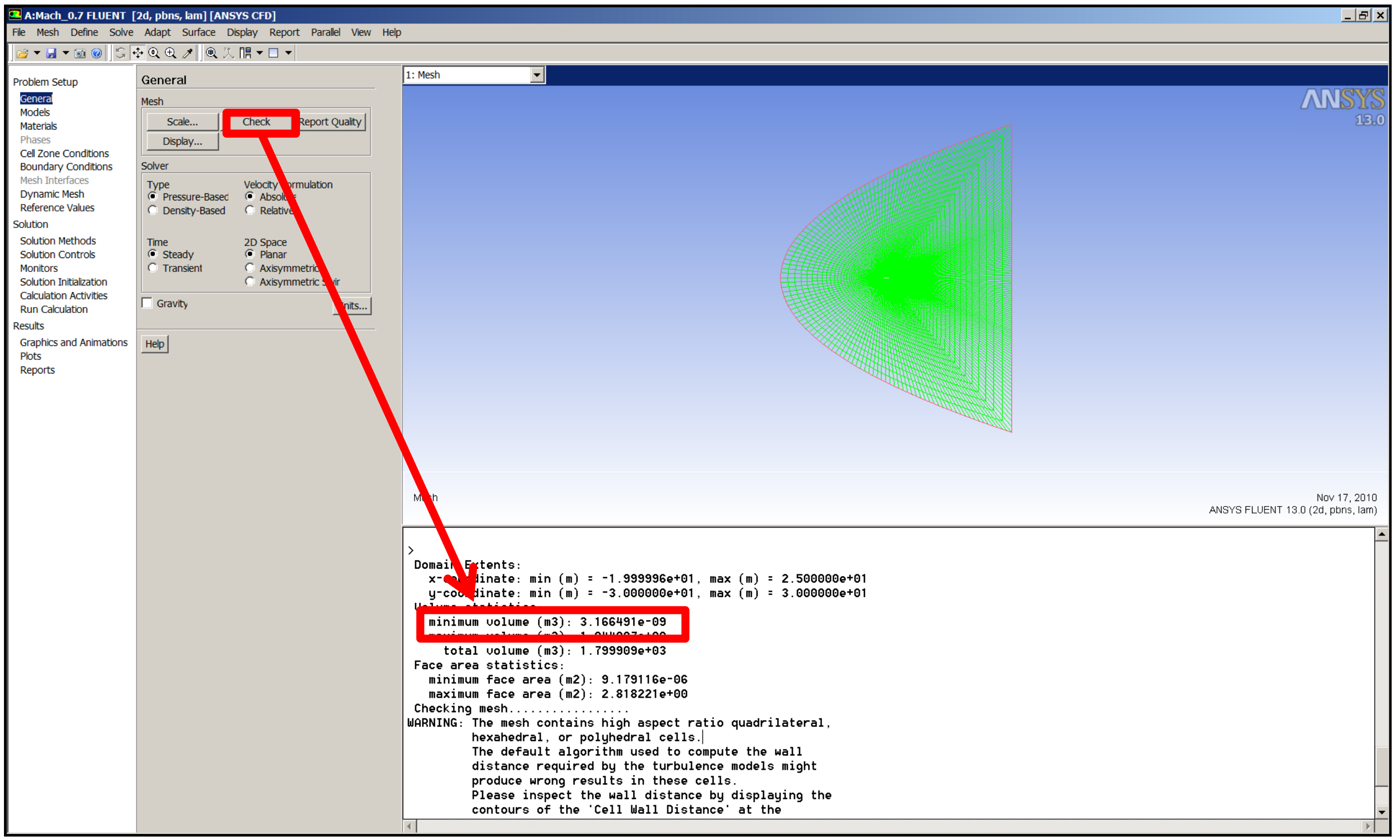
Scale the mesh

- **First of all, it is <u>always</u> important to check the scaling of the mesh, particularly when it has been imported from outside of workbench.**
- **Select General > Mesh > Scale and check the domain extents.**
- **In this case the domain extents are correct.**
- **If they are not, scale the mesh before proceeding any further!**



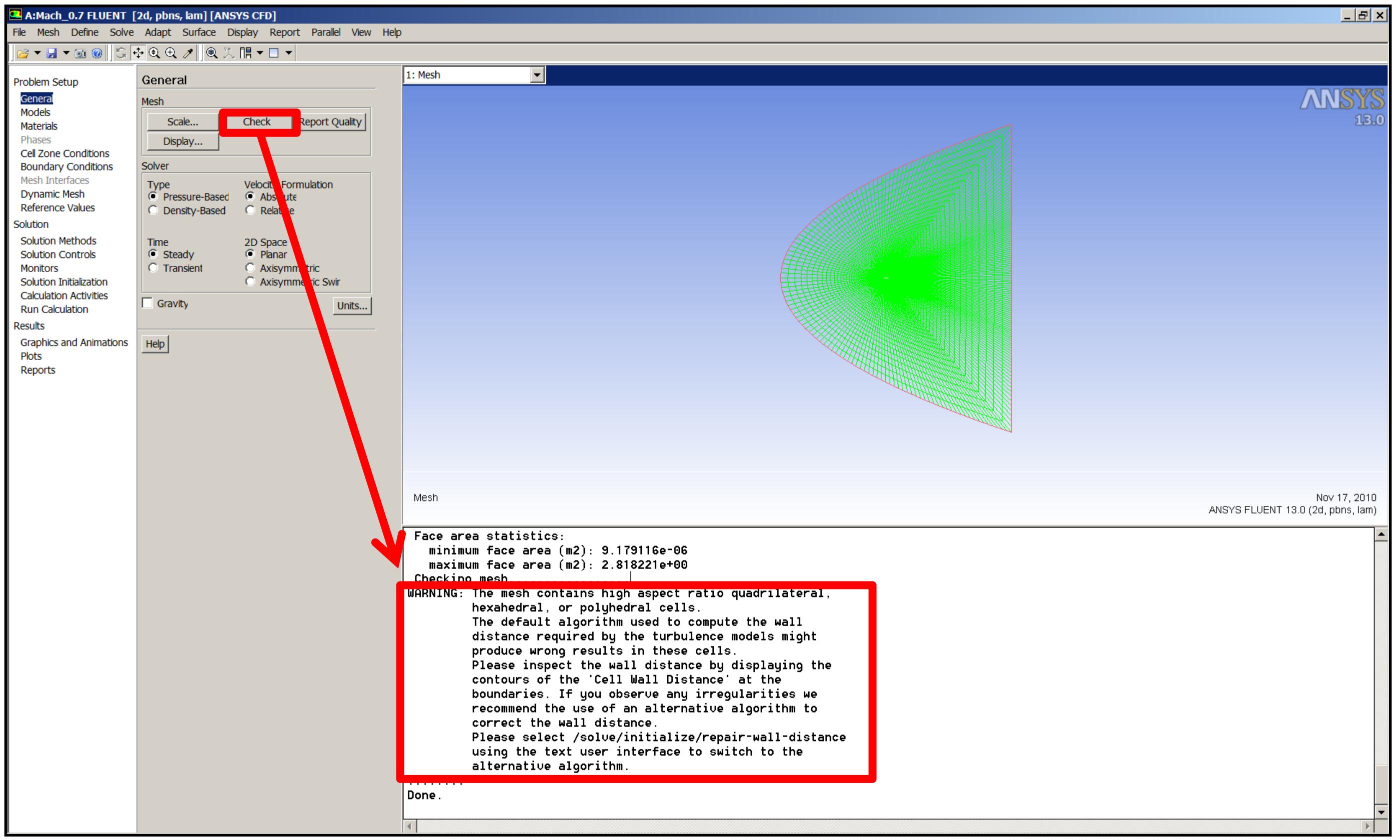
Check the mesh (general errors)

- **Check the mesh for errors: General > Mesh > Check**
  - **Review the text window output and check there are no errors**
  - **In particular make sure that the minimum cell volume is not negative**



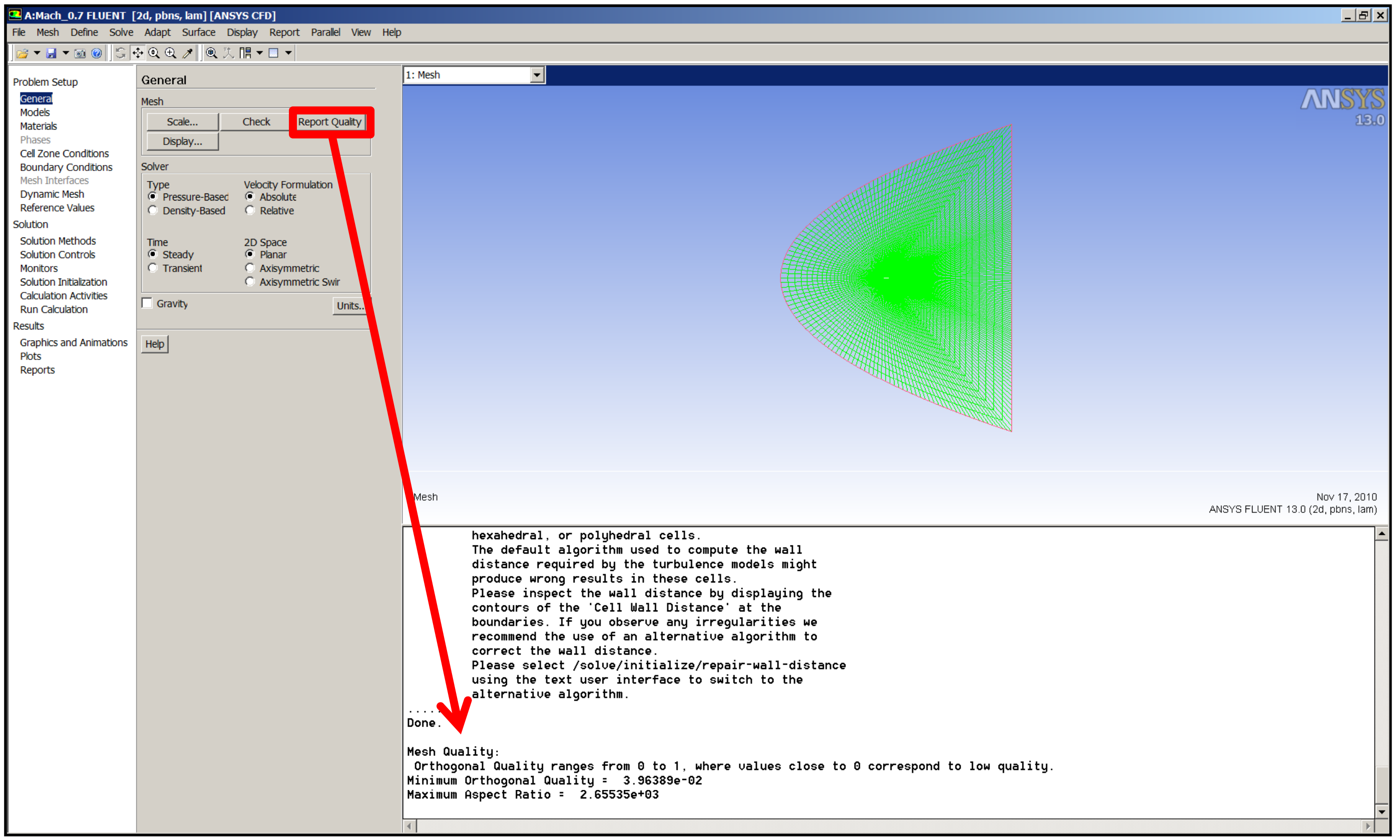
Check the mesh (aspect ratio)

- In this case there are no errors reported with the mesh.
- Note, however, that we do have a warning which is a result of the high aspect ratio cells within the boundary layer near the airfoil wall.



Check the mesh (aspect ratio)

- **Report the mesh quality statistics:**
  - **General > Mesh > Report Quality**
  - **Note the maximum aspect ratio is high at 2655**



Check the mesh (aspect ratio)

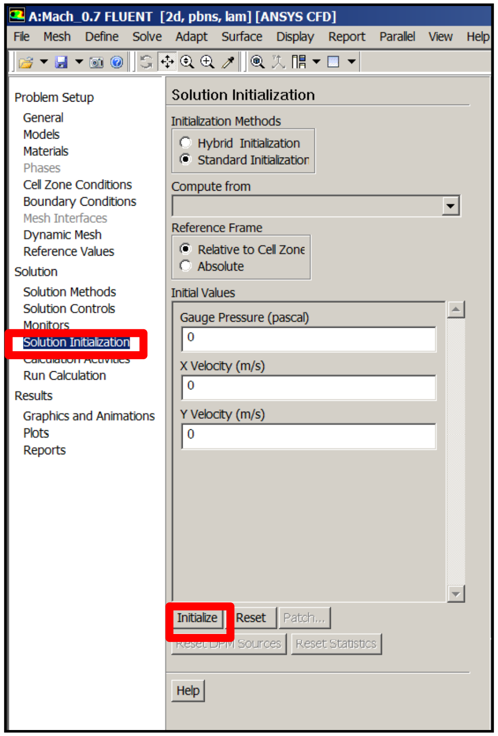
- As a guide, the cell aspect ratio should be around 5 or less in the main region of the mesh (away from the boundary layer).
- However, it is usual to have much higher aspect ratio cells than this in the boundary layer, up to around 100.
- In this case, the maximum aspect ratio in the boundary layer is much higher than this, but the mesh was designed in this way due to the need for very low Y+ values. Away from the boundary layer the maximum aspect ratio is around 5.
- For this special case the high maximum aspect ratio is justified.
- Not all cases require such a well resolved boundary layer mesh.
- Be aware that such high aspect ratio cells can give problems in the solver calculations near to the wall, hence the warning when the mesh quality metrics are reported.
- It is necessary to check that the solver is computing the wall distance correctly, as this will be used in the turbulence model when we switch it on.

Check the mesh (wall distance check)

- FLUENT suggests a method to check the wall distances that are calculated by the solver.

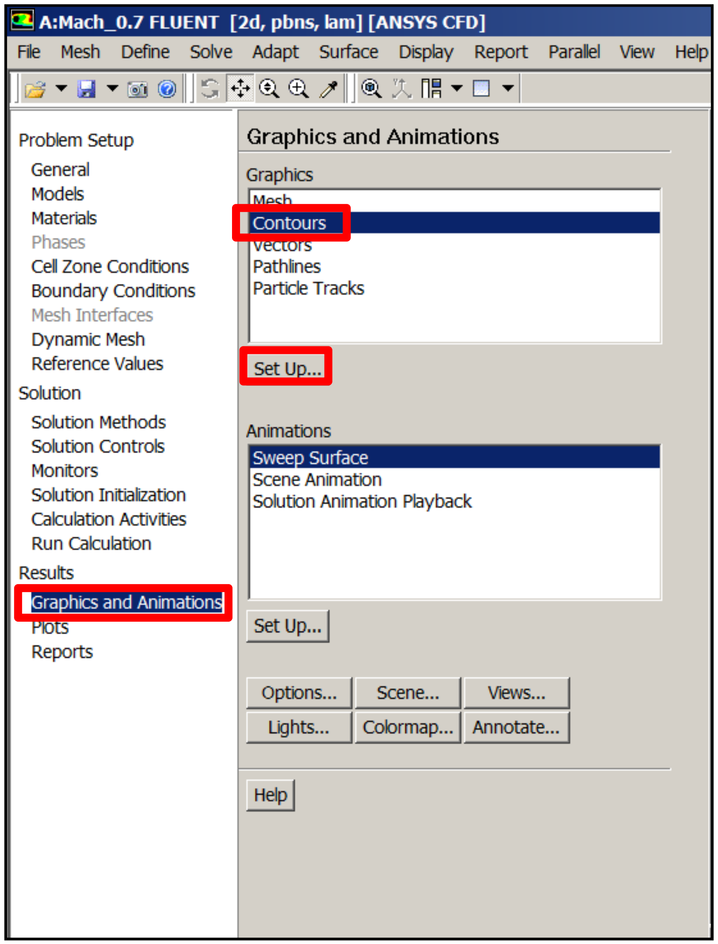
- The cells need some (any) data before we can post-process any variables, including wall distances, so initialize the flow:
  - default values are fine at this stage

- Solution Initialization > Initialize



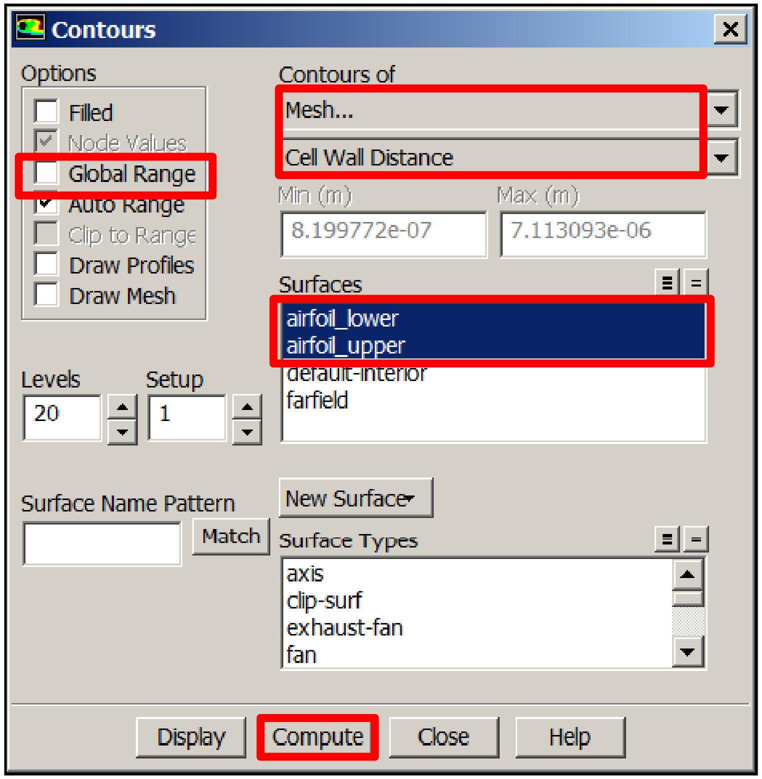
Check the mesh (wall distance check)

- **Now we can use the contour plot facility to report the wall distances.**
- **Graphics & Animations > Contours > Set Up**



Check the mesh (wall distance check)

- **Select Contours of Mesh > Cell Wall Distance**
- **Choose the airfoil wall surfaces (airfoil_lower and airfoil_upper)**
- **Deselect global range (report the local range of these surfaces only)**



Click 'Compute'

The minimum computed cell wall distance is 8.2e-07 m

The maximum computed cell wall distance is 7.1e-06 m

For the airfoil wall surfaces these are as expected from the mesh design, so we can proceed with our normal set up.

Check the mesh (some notes on Y+)

- **Y+ is the non-dimensional normal distance from the first grid point (the wall-adjacent cell center) to the wall.**

- **If the first grid point is placed within the viscous sublayer (the near-wall region where y+ <= 5), the turbulence model's enhanced wall treatment (EWT) option should be chosen. When using EWT, the intention is to integrate governing equations directly to the wall without using the Universal Law of The Wall for turbulence.**

- **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**
    - **reduces the length of cells for a given height so will reduce the aspect ratio**
    - **this would significantly increase the overall cell count**
    - **for this tutorial, the cell count has been kept low while maintaining low Y+**

Check the mesh (some notes on Y+)

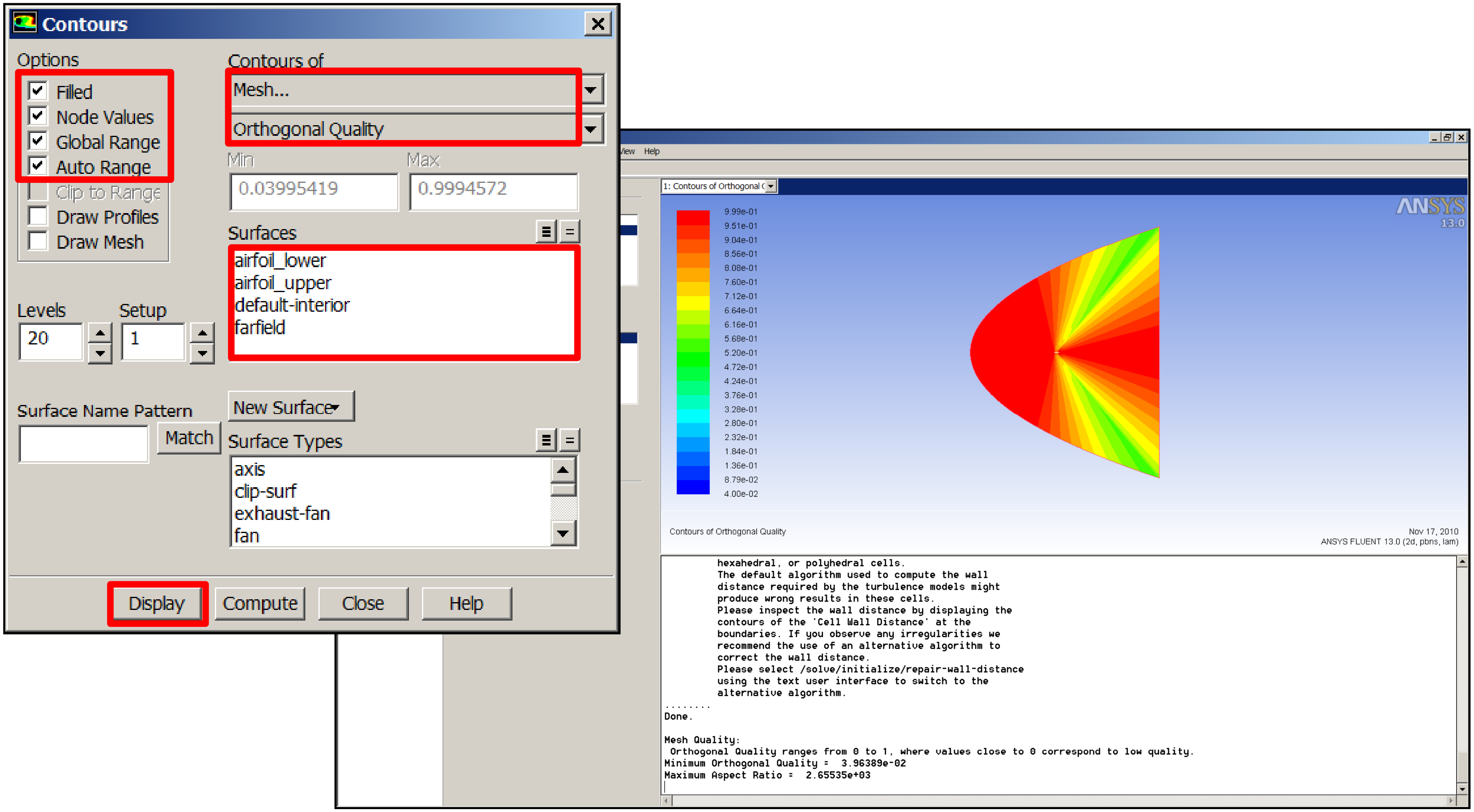
- **The aspect ratio could also be reduced, while increasing Y+ value:**
  - **by increasing the normal distance of the first grid point from the wall to give Y+ values of between 30 and 300.**
  - **This is the valid range of y+ for using the wall functions approach.**
  - **Avoid placing the first grid so that 5 < y+ < 30, which is called the buffer layer. (Due to the large variation of various turbulence source terms in the buffer layer, no existing models can handle the wall treatment well if the first grids lie in this region.)**

Check the mesh (orthogonal quality)

- In addition to maximum aspect ratio, the minimum orthogonal quality is also reported:
  – 1 is the best quality, 0 is the worst quality.
- For this mesh, the minimum value is reported as 3.96e-02 which is again the result of highly stretched cells in the boundary layer.
- This can be examined further using the contour panel again:
  – Contours of Mesh > Orthogonal Quality
    - Deselect all surfaces
    - Select global range
    - Select Filled
    - Display

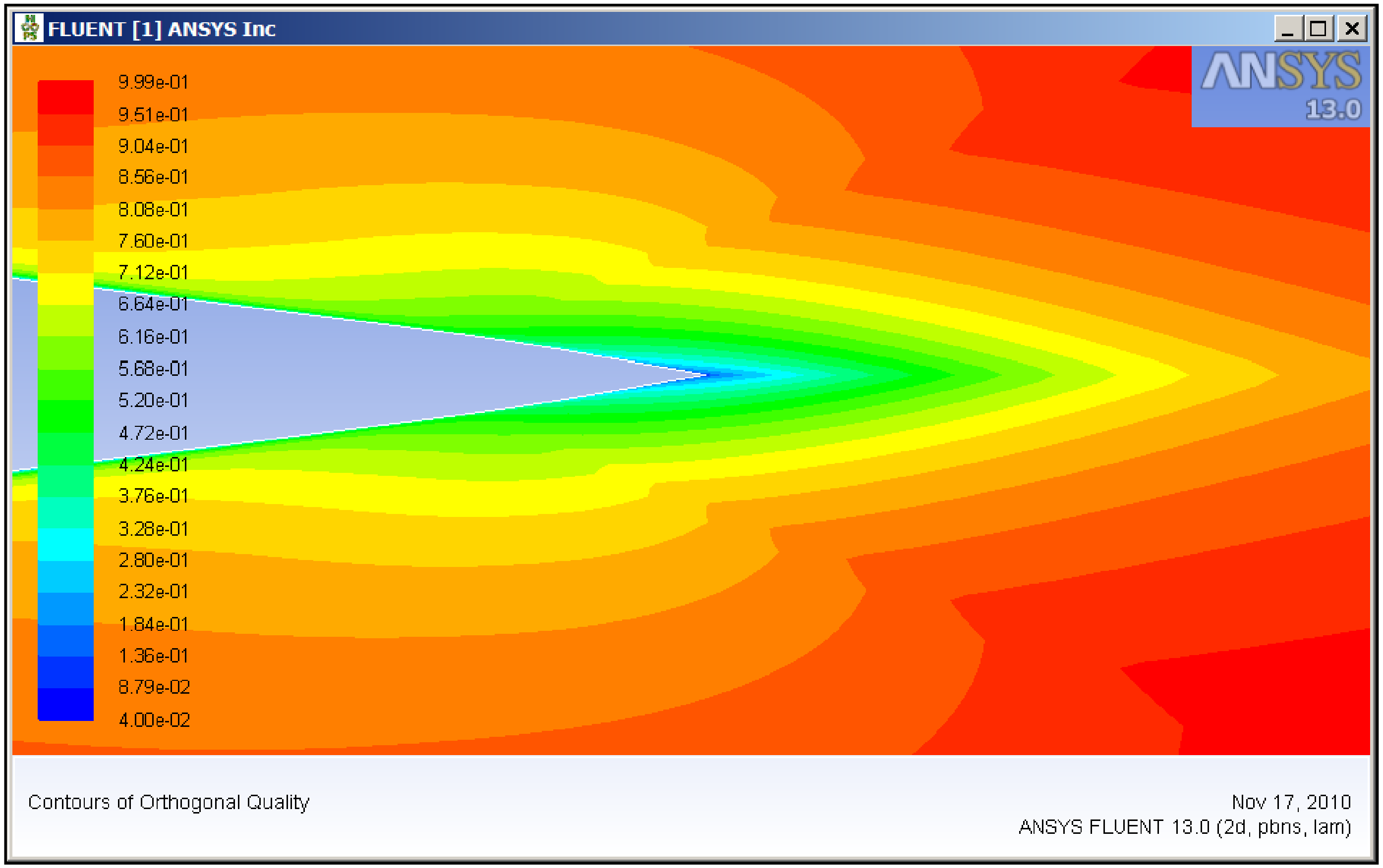
Check the mesh (orthogonal quality)

- **Most cells are good quality, many are around 1 (shown in red)**



Check the mesh (orthogonal quality)

- Zoom in to the boundary layer cells near the airfoil wall, these cells are blue (low orthogonal quality) due to the stretched cells.
- In particular this can be seen where they meet at the trailing edge.

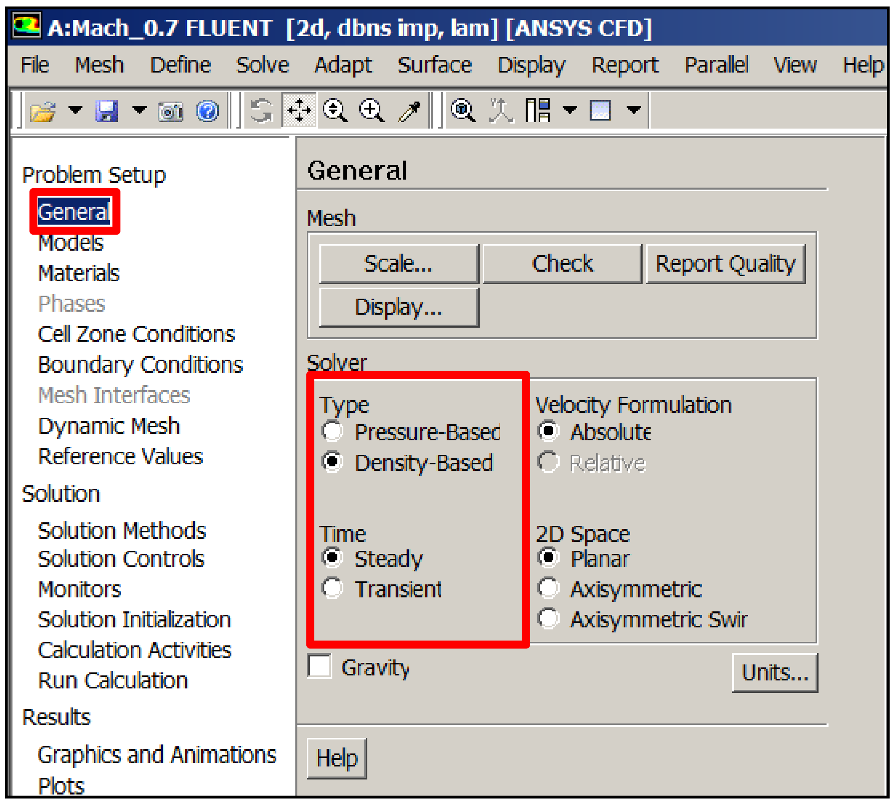


Check the mesh (proceed)

- **High aspect ratio cells and low orthogonal quality cells have been identified in the mesh.**
  - **Both are located in the stretched boundary layer cells.**
- **This is acceptable in this airfoil drag-prediction case, where low Y+ values can lead to high aspect ratio cells.**
  - **More grid points along the airfoil wall surfaces would reduce the aspect ratio, but increase the overall mesh size.**
- **The cell wall distance computed by the solver, and needed for the turbulence model, has been checked.**
- **Proceed with this mesh and set up the case.**

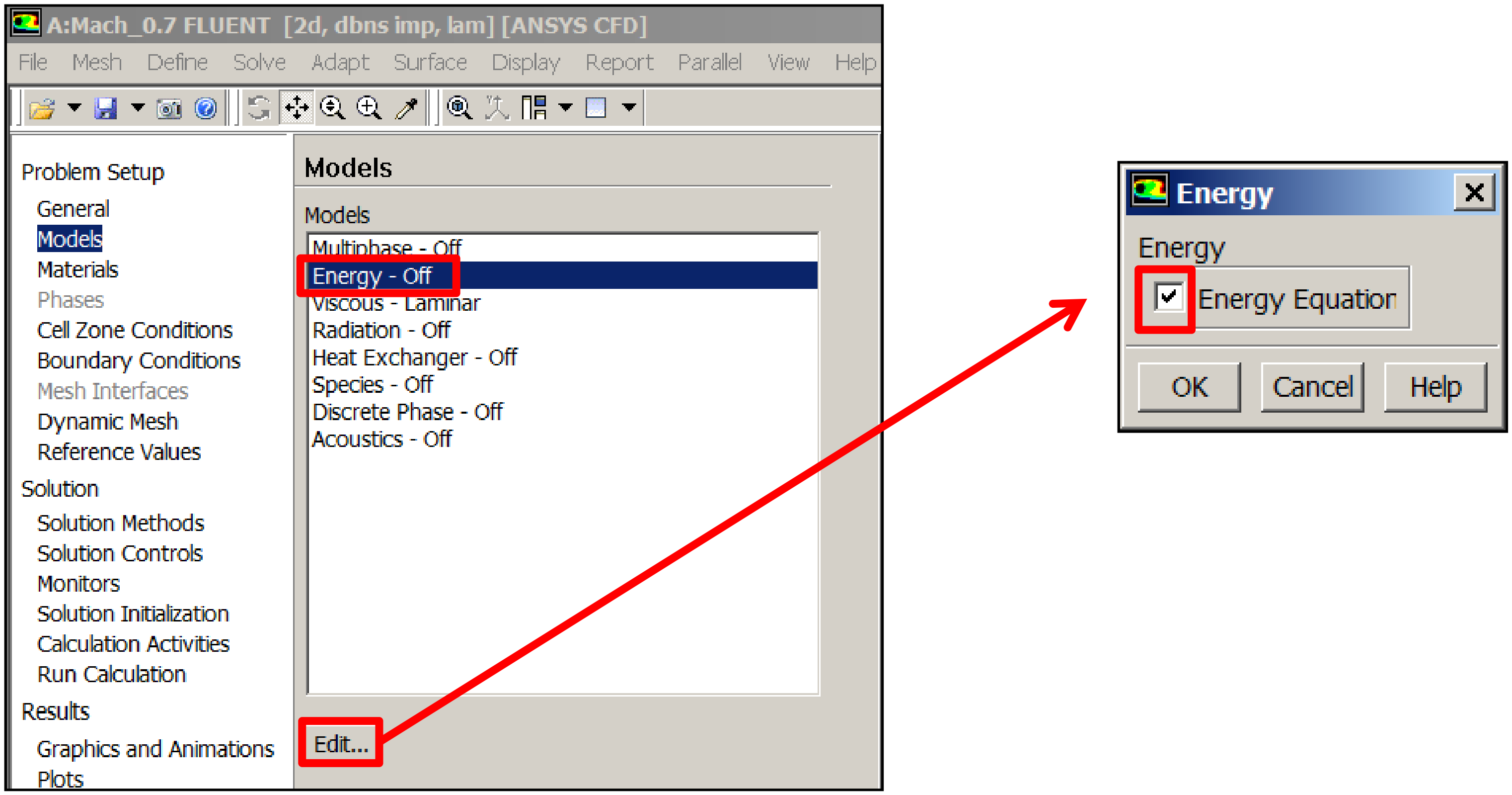
Case Setup: Choose the solver and models

- **Select the steady-state density-based solver:**
  - **General > Solver > Type > select ‘Density-Based’**
  - **General > Solver > Time > steady (default)**



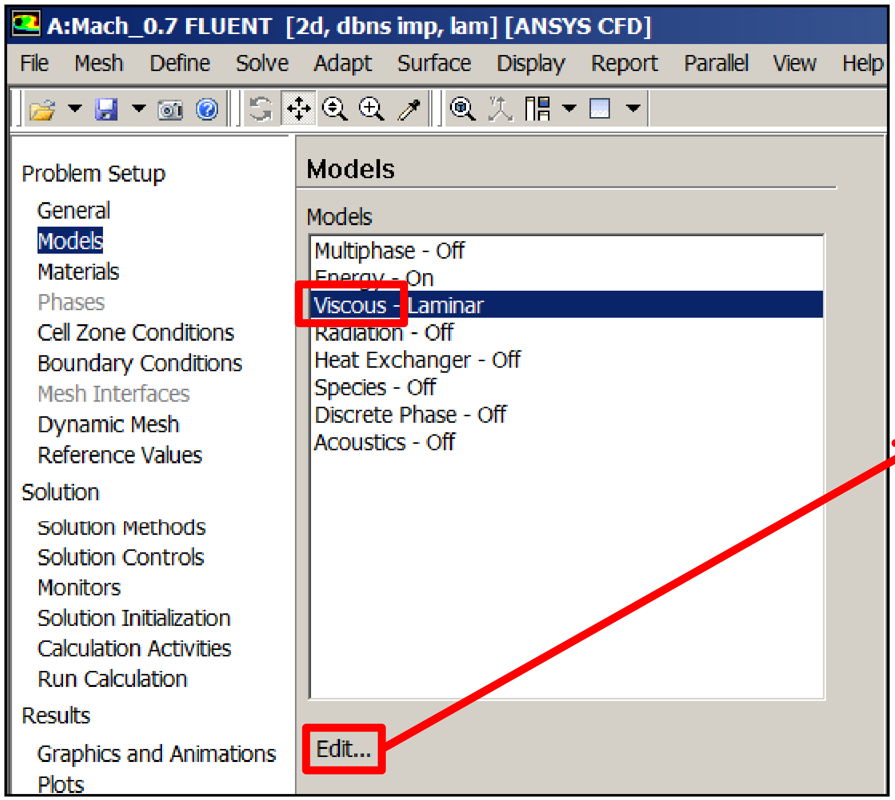
Case Setup: Choose the solver and models

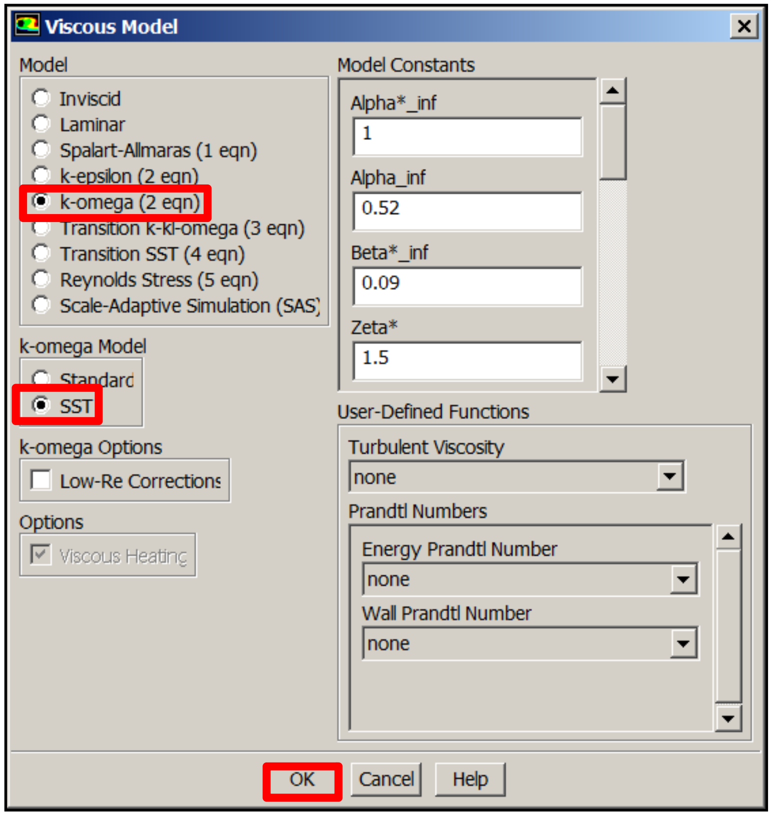
- **Turn on the energy equation.**
  - **Models > Energy > Edit > Toggle 'On'**
  - **This is needed because the flow is compressible and temperature will be variable.**



Case Setup: Choose the solver and models

- **Select the turbulence model to be used:**
  - **Models > Viscous > Edit**
    - **Choose k-omega (2-eqn)**
    - **Select SST, then OK**



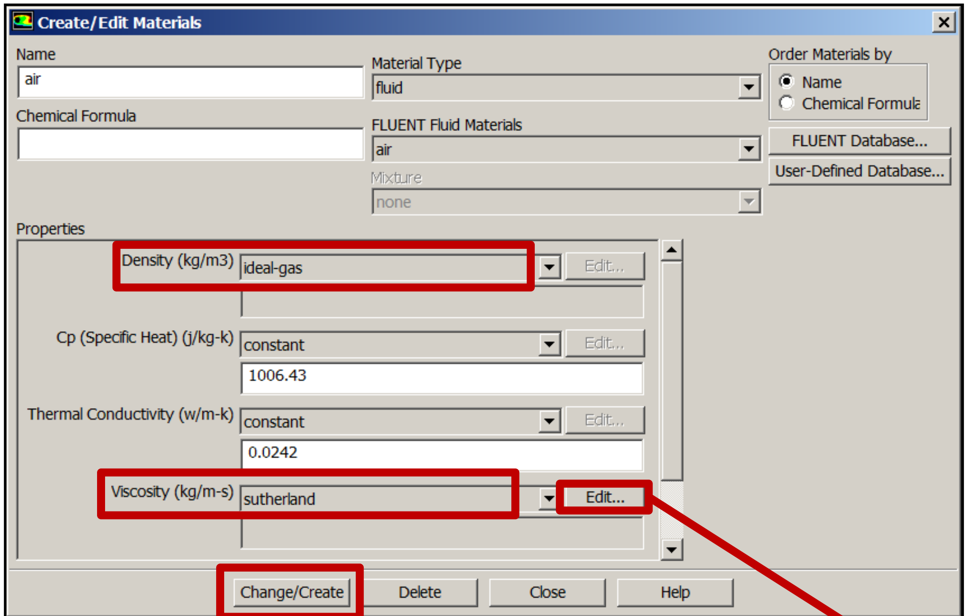


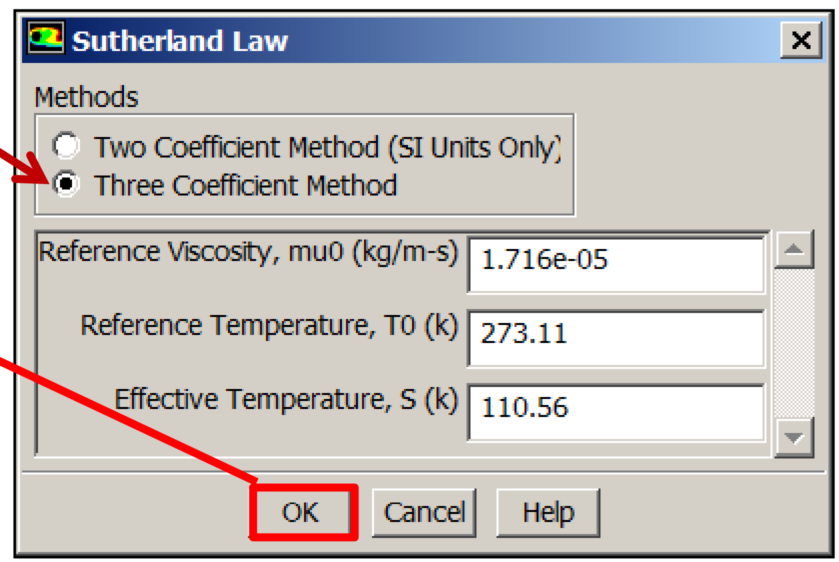
Case Setup: Define Materials

- Constant density air is the default fluid.
  - This must be modified for compressible flow.

- Materials > Fluid > Air > 'Create/Edit'
  - For Density, select 'Ideal Gas'
  - For Viscosity, select 'Sutherland'
    - Accept the default settings for the Three Coefficient Method
    - The Sutherland law for viscosity is well suited for high-speed compressible flow.
  - For simplicity, we will leave Cp and Thermal Conductivity as constant.
    - Ideally, in high speed compressible flow modeling, these should be temperature dependent as well.

Case Setup: Define Materials



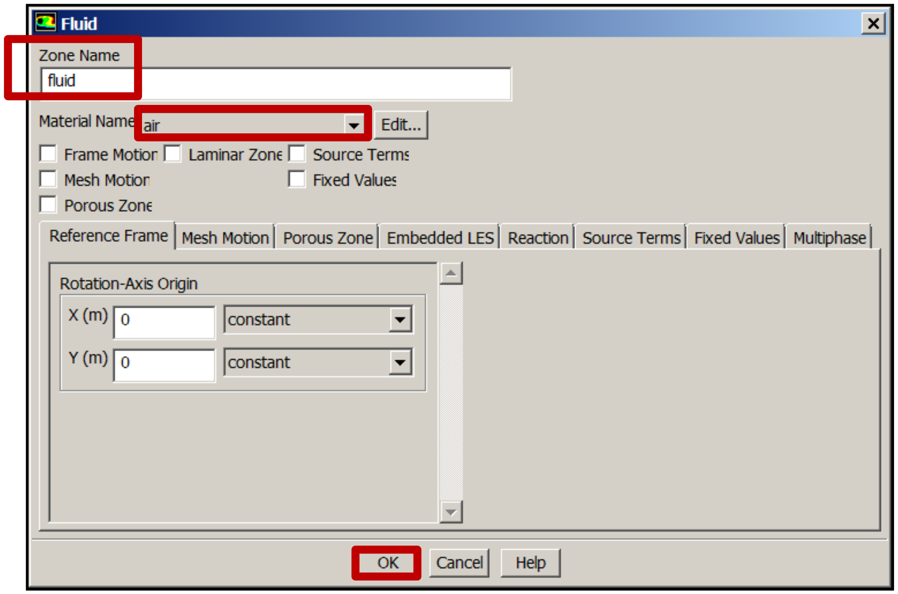


Click 'Change/Create'

Close the materials panel.

Case Setup: Assign Materials

- Check that 'air' is assigned to the cell zone 'fluid'
- Cell Zone Conditions > Zone > 'fluid' > edit
  - In this case there is only one cell zone (with the default name of fluid)
  - In this case there is also only one material (air, which has been modified)
  - There can be multiple zones with user-specified names and multiple materials if required
- There is nothing else to assign to the fluid cell zone for this case
  - Other tabs are enabled when the corresponding options are selected



Case Setup: Operating Conditions

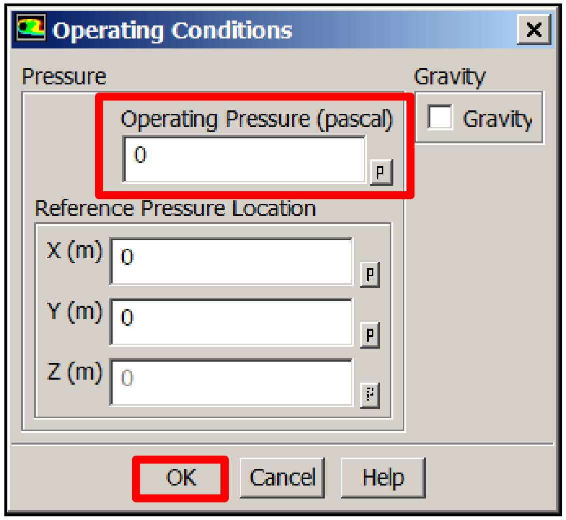
- **Cell Zone Conditions > Operating Conditions**
  - **Note: can also be accessed from the boundary conditions panel**
- **Absolute pressure = operating pressure + gauge pressure**
  - **For incompressible flows (where pressure changes are relatively small) it is normal to specify a large operating pressure and let the solver work with smaller 'gauge' pressures (FLUENT input/output values), to reduce round-off errors.**
  - **For *compressible* flows, the solver needs to use the absolute values in the calculation anyway, so numerically this makes no difference.**

Therefore, with compressible flows, it is sometimes convenient to set to operating pressure to zero, and input/output 'absolute' pressures.

This is simply a user preference. Other times it may be convenient to set a bulk flow operating pressure.

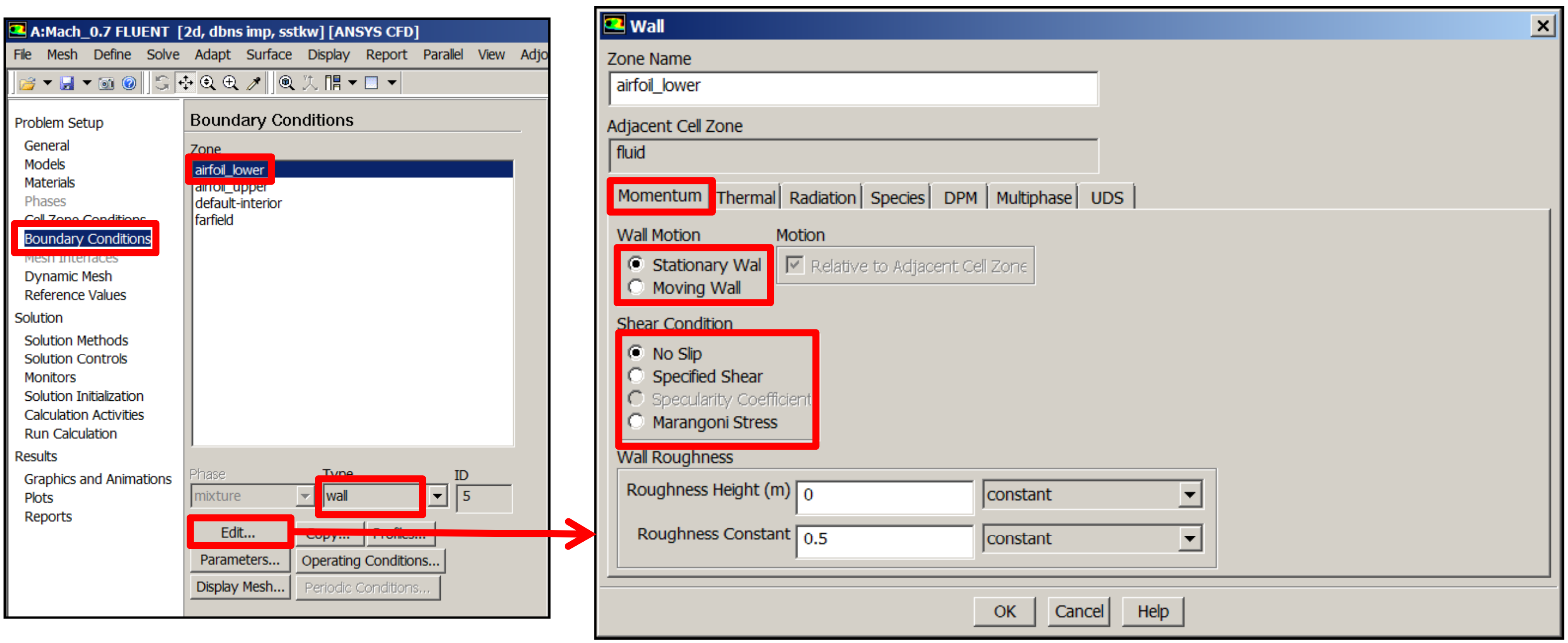
The main thing to remember is that whatever operating pressure is set, this will be *added* to any gauge input pressures.

Set Operating Pressure to 0 Pa



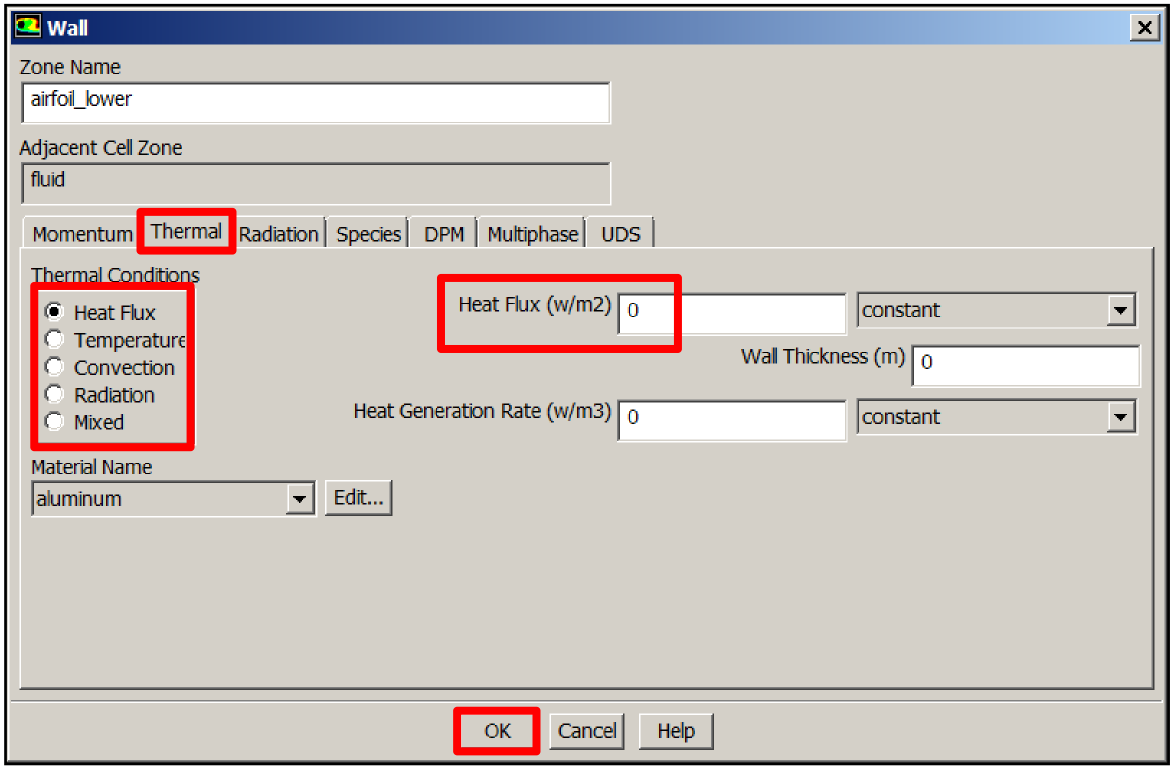
Case Setup: Boundary Conditions

- **Boundary Conditions > Zone > select ‘airfoil_lower’**
- **Check that the boundary zone type is set to ‘wall’**
  - **If not, use the drop down list to change!**
- **Edit > Momentum Tab**
  - **leave the default settings which correspond to a stationary wall (relative to the fluid zone) with a no-slip condition.**



Case Setup: Boundary Conditions

- **Thermal Tab**
  - **Remember that the energy equation is switched on so thermal boundary conditions must be applied.**
  - **Leave the default setting (zero heat flux) which corresponds to an adiabatic wall**



Case Setup: Boundary Conditions

- **Boundary Conditions > Zone > ‘airfoil_upper’**
  - **Check that the boundary zone type is set to ‘wall’**
    - **If not, use the drop down list to change!**
  - **Edit > Momentum Tab and Thermal Tab**
    - **Repeat as for airfoil_lower (no-slip, adiabatic)**
- **Boundary Conditions > Zone > ‘farfield’**
  - **Check that the boundary zone type is set to ‘pressure-far-field’**
    - **If not, use the drop down list to change!**
  - **Edit > Momentum Tab and Thermal Tab**
    - **See next slides**

Case Setup: Boundary Conditions

- **Set the far field boundary conditions:**
  - **The pressure-far-field boundary is applicable only when using the ideal-gas law.**
  - **It is important to place the far-field boundary 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.**
  - **We need to input the static pressure at a far-field boundary.**
  - **We can calculate this from the total pressure, which was atmospheric at 101325 Pa in the test.**
  - **We have already set the operating pressure to zero, so we are now working in absolute pressure values. Hence the 'gauge' static pressure input will be equal to the 'absolute' static pressure value, which we will calculate to be 73048 Pa, for a Mach number of 0.7**
  - **On the 'Momentum' tab**
    - **set the gauge static pressure to 73048 Pa**
    - **set the Mach Number to 0.7**

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

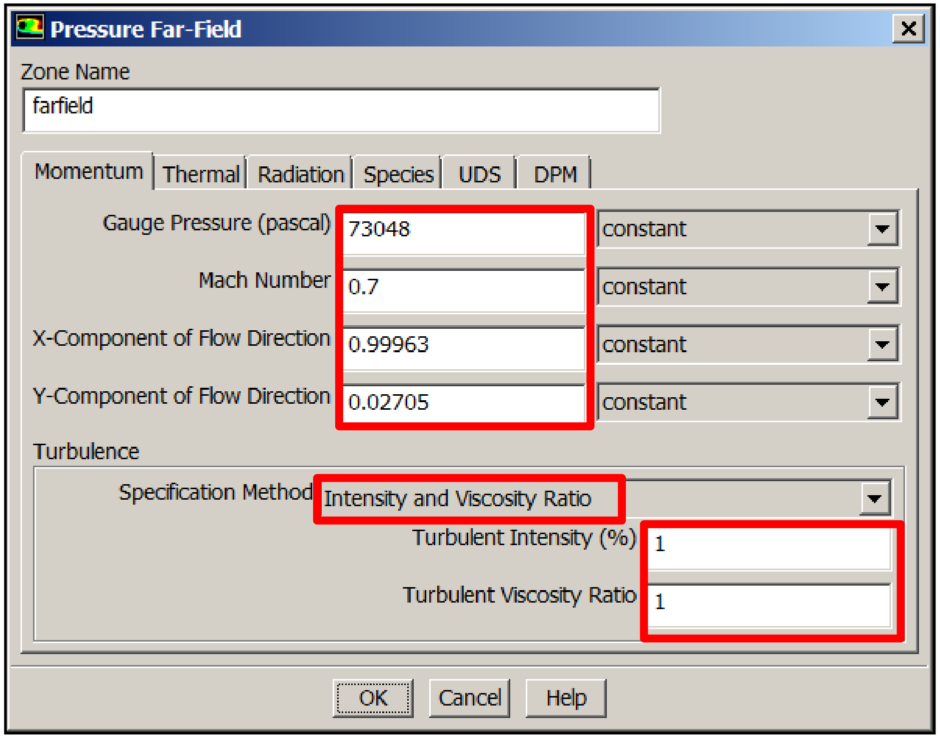
where

$$p_o = \text{total pressure} = 101325\ Pa$$
$$p = \text{static pressure}$$
$$\gamma = 1.4 \text{ for air}$$
$$M = \text{Mach No.} = 0.7$$
$$\therefore \frac{p_o}{p} = 1.3871$$
$$\boxed{p = 73048\ \text{Pa}}$$

Case Setup: Boundary Conditions

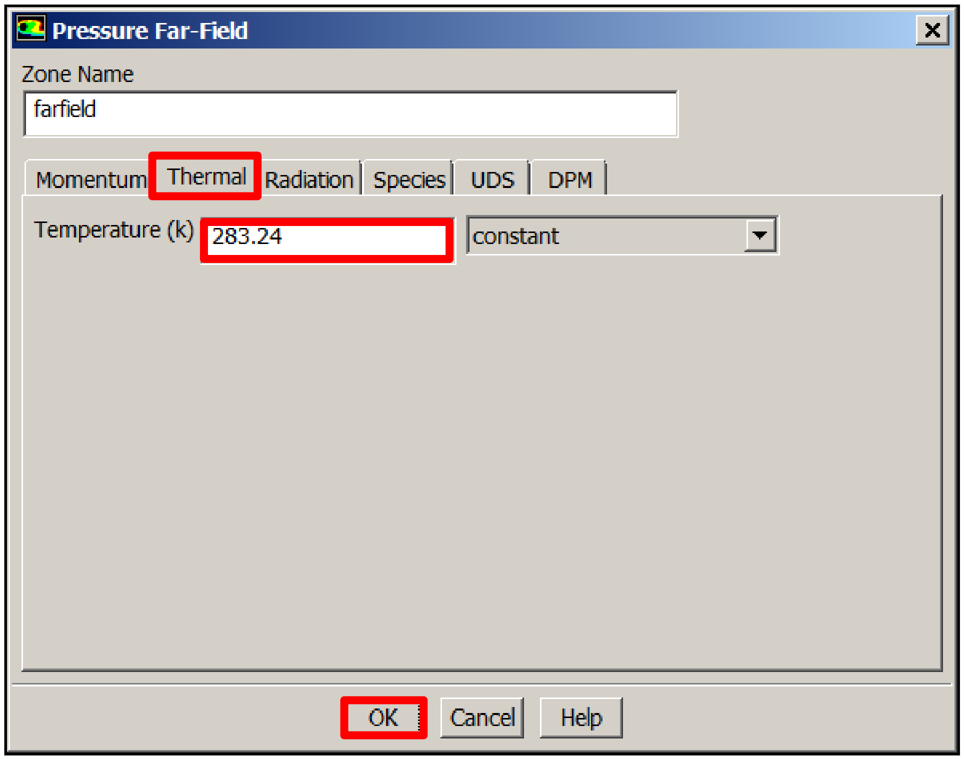
- The angle of attack (α) in this case is 1.55 deg. The x-component of the flow is cos α and the y-component is sin α.
  - Set the flow direction components as shown
  - It is common practice to adjust the numerical α from the experimental α in order to match the lift obtained in the wind tunnel, and then to determine the drag associated with this lift. This adjustment of α is carried out to counter the effects of the wind tunnel enclosure.

- Set reasonable boundary conditions for the far field turbulence:
  - Select 'Intensity and Viscosity Ratio'
  - Set the intensity to 1%
  - Set the viscosity ratio to 1



Case Setup: Boundary Conditions

- **Select the thermal tab.**
  - **The wind tunnel operating conditions for validation test data give the total temperature as T_0 = 311 K**
  - **We can therefore calculate the static temperature to be 283.24 K**



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

where

T_0 = total temperature = 311K

T = static temperature

$\gamma = 1.4$ for air

M = Mach number = 0.7

$$\therefore \quad \frac{T_0}{T} = 1.098 \text{ and so } \boxed{T = 283.24\,\mathrm{K}}$$

Case Setup: Reference Values

- **Set the reference values**
  - **These are not used to compute the flow solution, but they are used to report coefficients, such as Cp.**
- **Use the free-stream as a reference condition.**

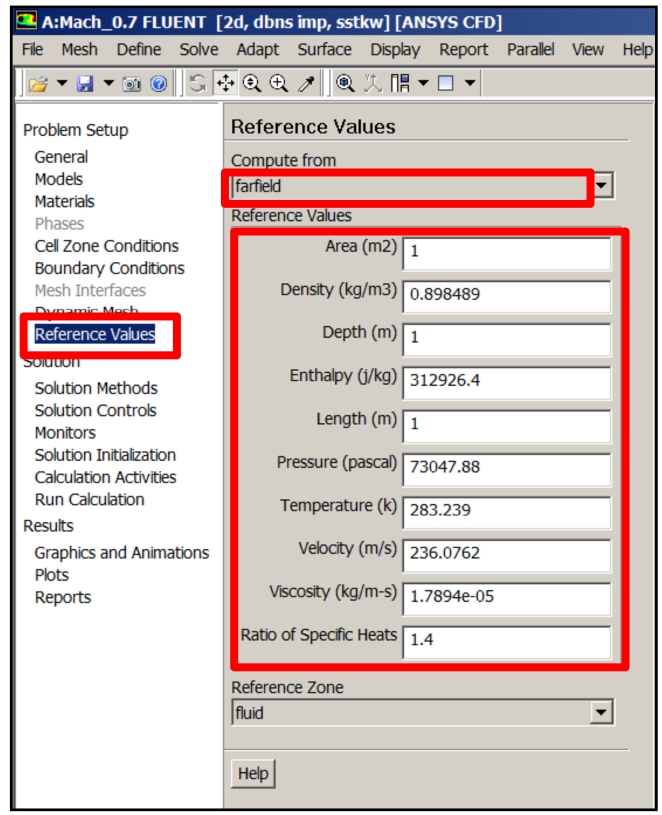
Select 'compute from farfield' in the drop down list.

Reference values for velocity, density, temperature, etc. will update from the free-stream values.

Set the following to represent a chord length of 1m with unit depth:

reference length	**= 1m**
reference depth	**= 1m**
reference area	**= $1m^2$**

These are *not* updated from the farfield conditions, but the defaults happen to be 1 already.



Case Setup: Solution Methods

- **The CFD model itself is now defined, but the solver settings need to be modified. These dictate how fast, stable and accurate (for the given mesh and model choices) the solution will be.**

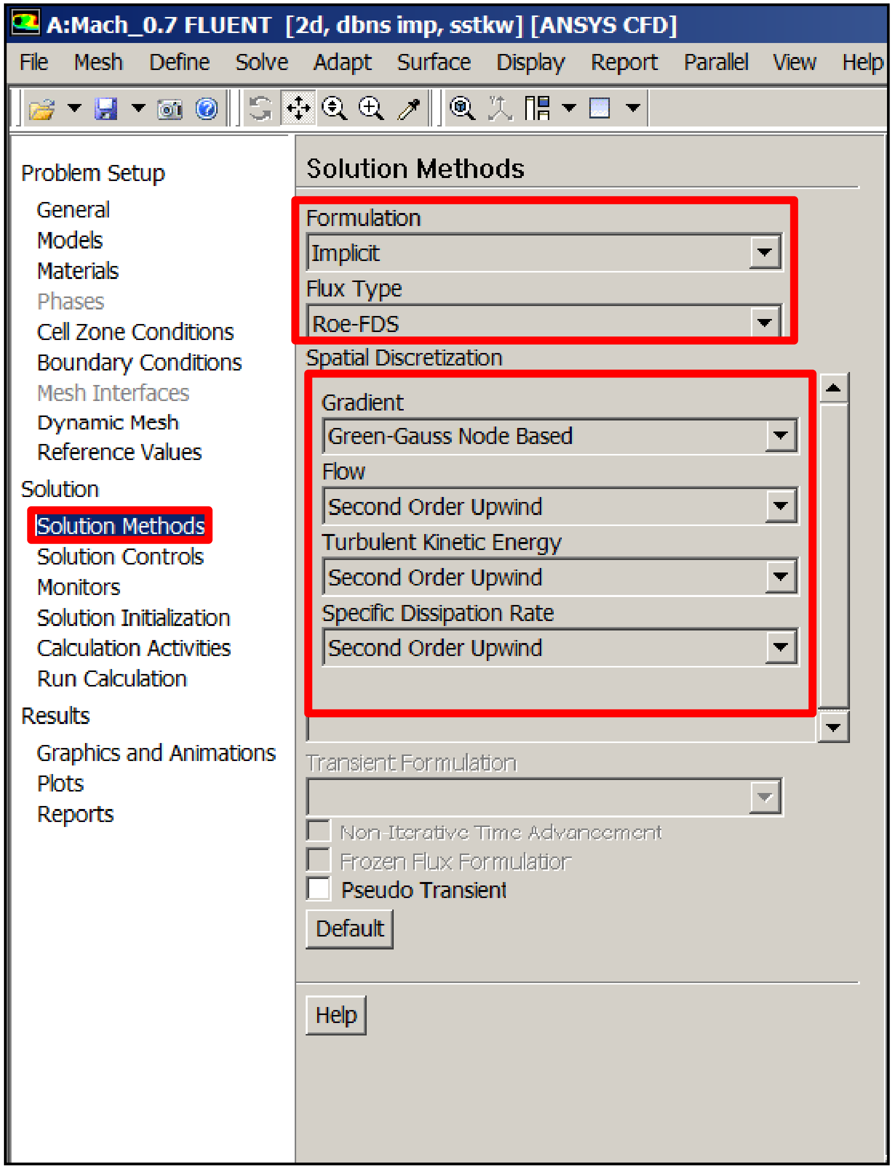
- **Solution Methods >**

  – **Keep the default settings for implicit formulation and Roe-FDS flux type.**
    - **The explicit formulation is only normally used for cases where the characteristic time scale is of the same order as the acoustic time scale, for example the propagation of high Mach number shock waves.**
    - **The implicit formulation is more stable and can be driven much harder to reach a converged solution in less time.**

  – **Change the gradient method to Green-Gauss Node Based.**
    - **This is slightly more computationally expensive than the other methods but is more accurate.**

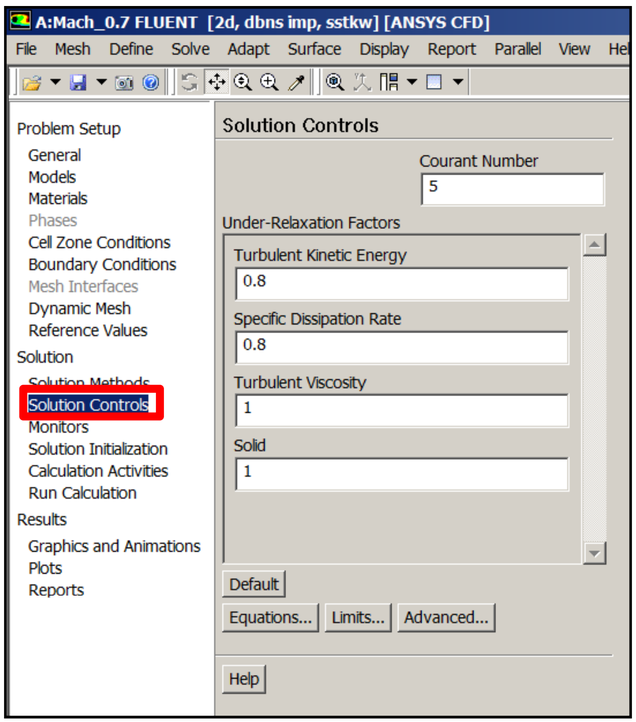
  – **Select Second Order Upwind for flow and turbulence discretization.**
    - **To accurately predict drag, the default 1st order schemes are not sufficient.**

Case Setup: Solution Methods



Case Setup: Solution Controls

- **The Courant number (CFL) determines the internal time step and affects the solution speed and stability.**
  - **The default CFL for the density-based implicit formulation is 5.0.**
  - **It is often possible to increase the CFL to 10, 20, 100, or even higher, depending on the stability of the solution. It may be that a lower CFL is required during startup (when changes in the solution are highly nonlinear), but it can be increased as the solution progresses.**

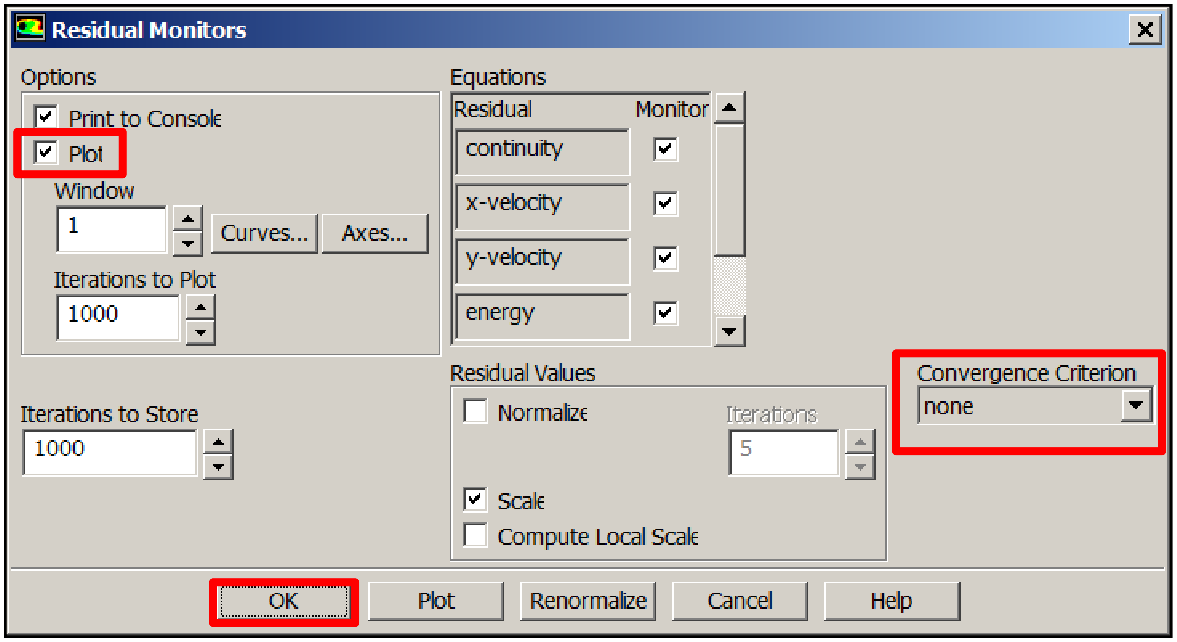


As we will be using automatic 'solution steering', the choice of CFL at this stage is not important for this case.

Keep the default under-relaxation factors (URFs) for the uncoupled parameters.

Case Setup: Solution Monitors

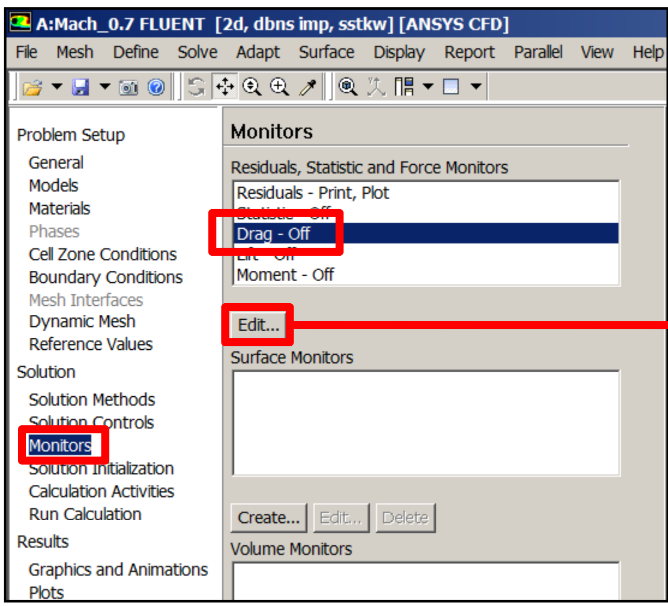
- Set up residual monitors so that convergence can be monitored
  - Monitors > Residuals > Edit
  - Make sure 'Plot' is on
  - Turn off convergence targets by setting the criterion to 'none'.
    - This means that the calculation will not stop at pre-defined convergence criteria, but residual convergence can still be plotted.



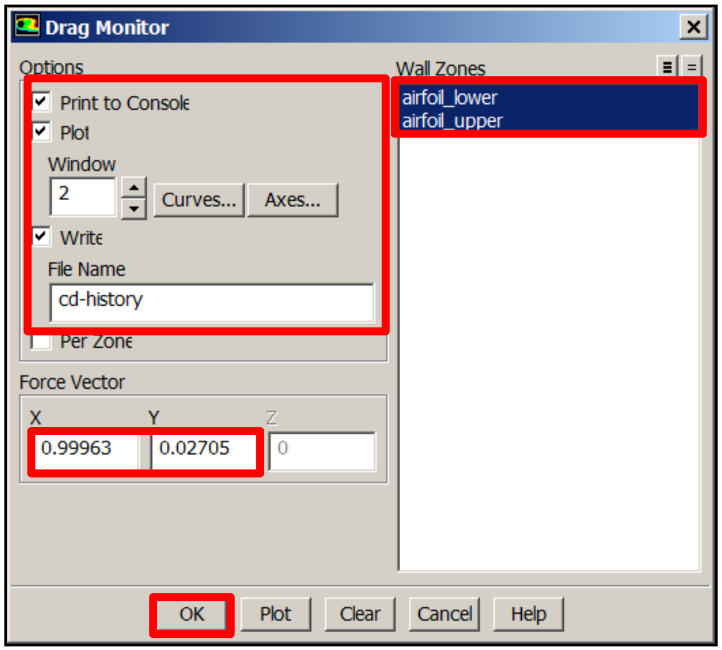
Case Setup: Solution Monitors

- **Set up a monitor for the drag coefficient on the airfoil.**

- **Select both wall zones and toggle on 'Print', 'Plot' and 'Write'.**
  - **Remember that α is 1.55° so we need to use the force vector as shown.**
    - **Lift and drag are defined (perpendicular and parallel respectively) relative to the free-stream flow direction, not the airfoil.**

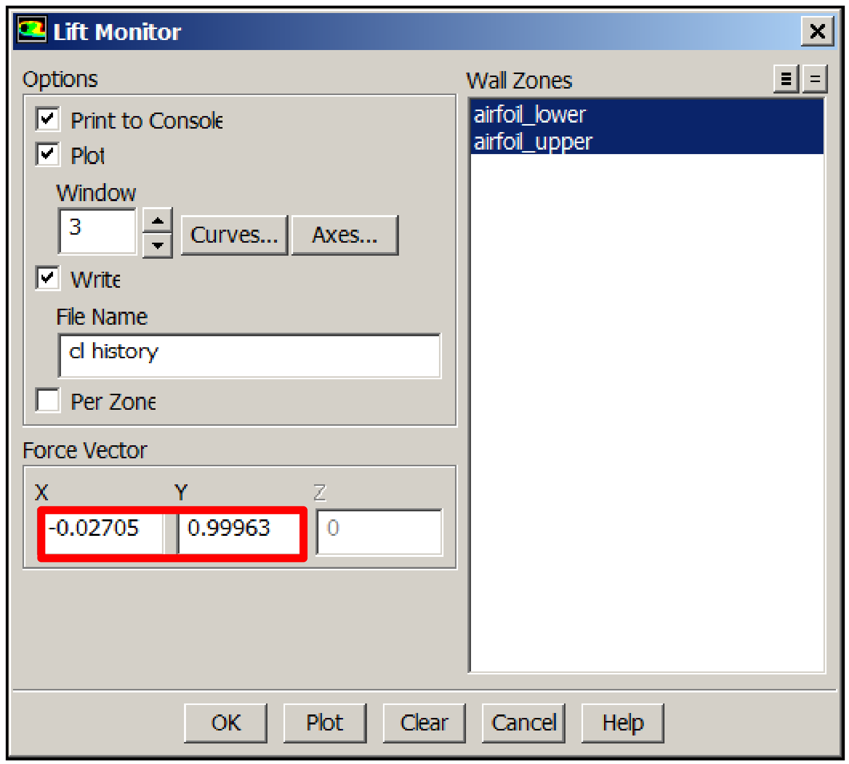


You can specify which window FLUENT uses to display plots. For now, accept the defaults.



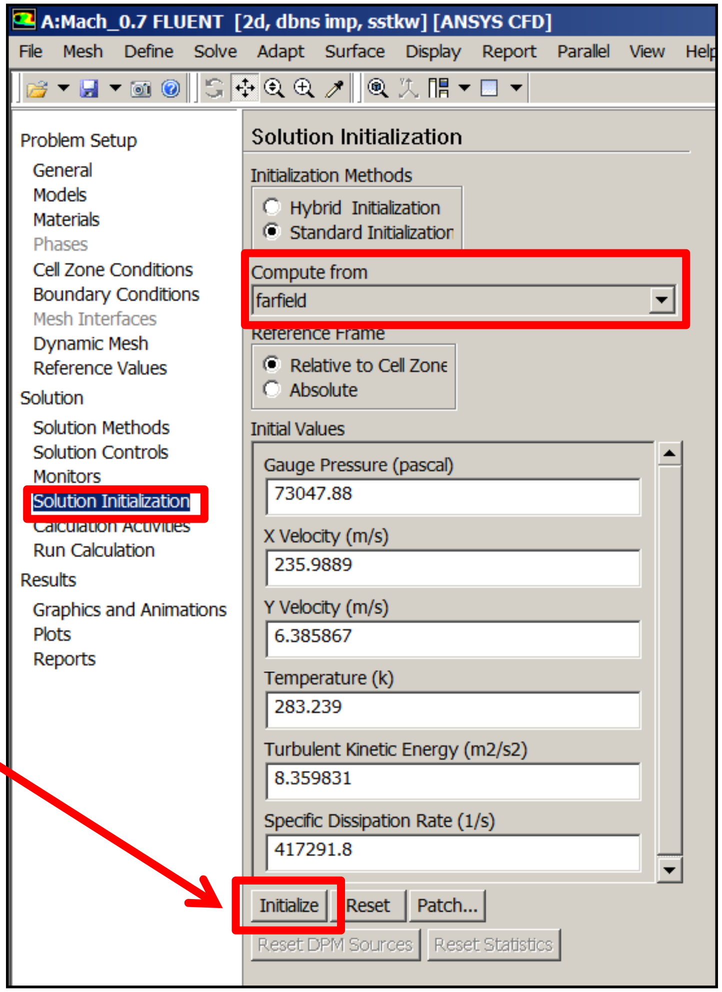
Case Setup: Solution Monitors

- **Set up a monitor for the lift coefficient on the airfoil in the same way.**
  - **Note the force vector is different to the drag monitor**



Case Setup: Solution Initialization

- **Initialize the flow field based on the farfield boundary:**
  - **Solution Initialization > Compute from > farfield**



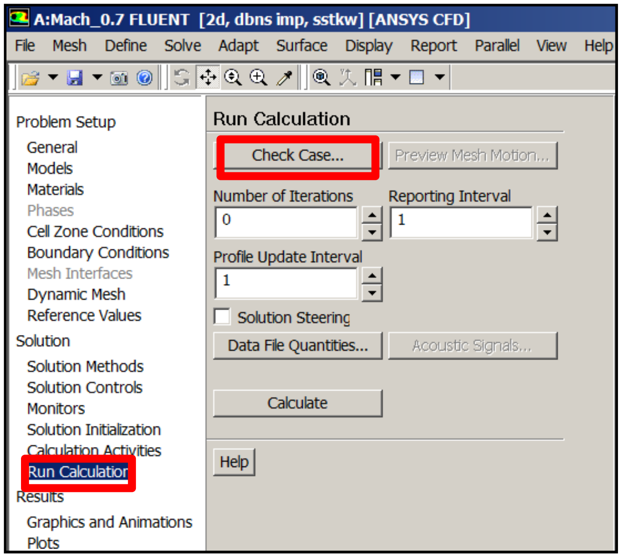
- **'Initialize'**

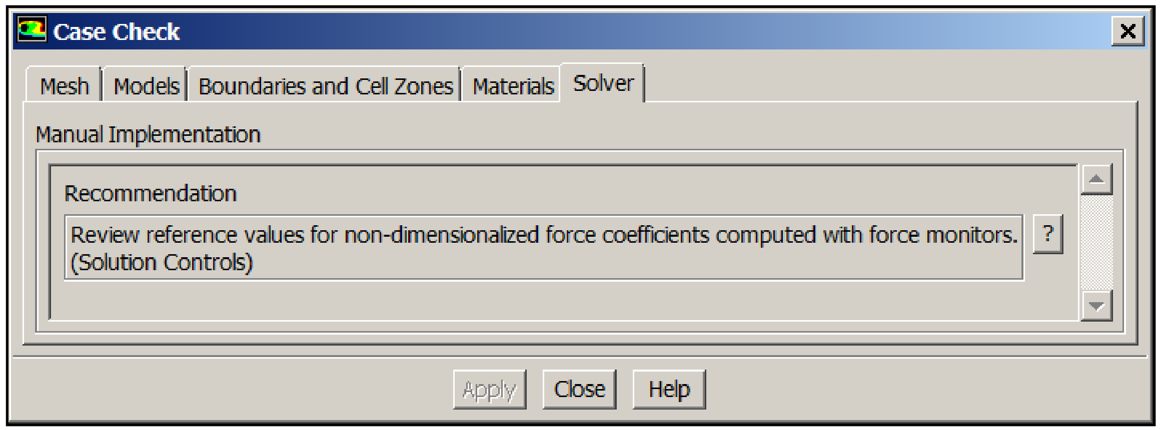
There will be existing data from the earlier initialization used for checking the cell wall distances, etc.

Click OK when prompted to discard the old data and overwrite with the new flow data.

Case Setup: Case Check

- **Run Calculation > Check Case**
  - **Check the case file and make sure there are no reported issues.**





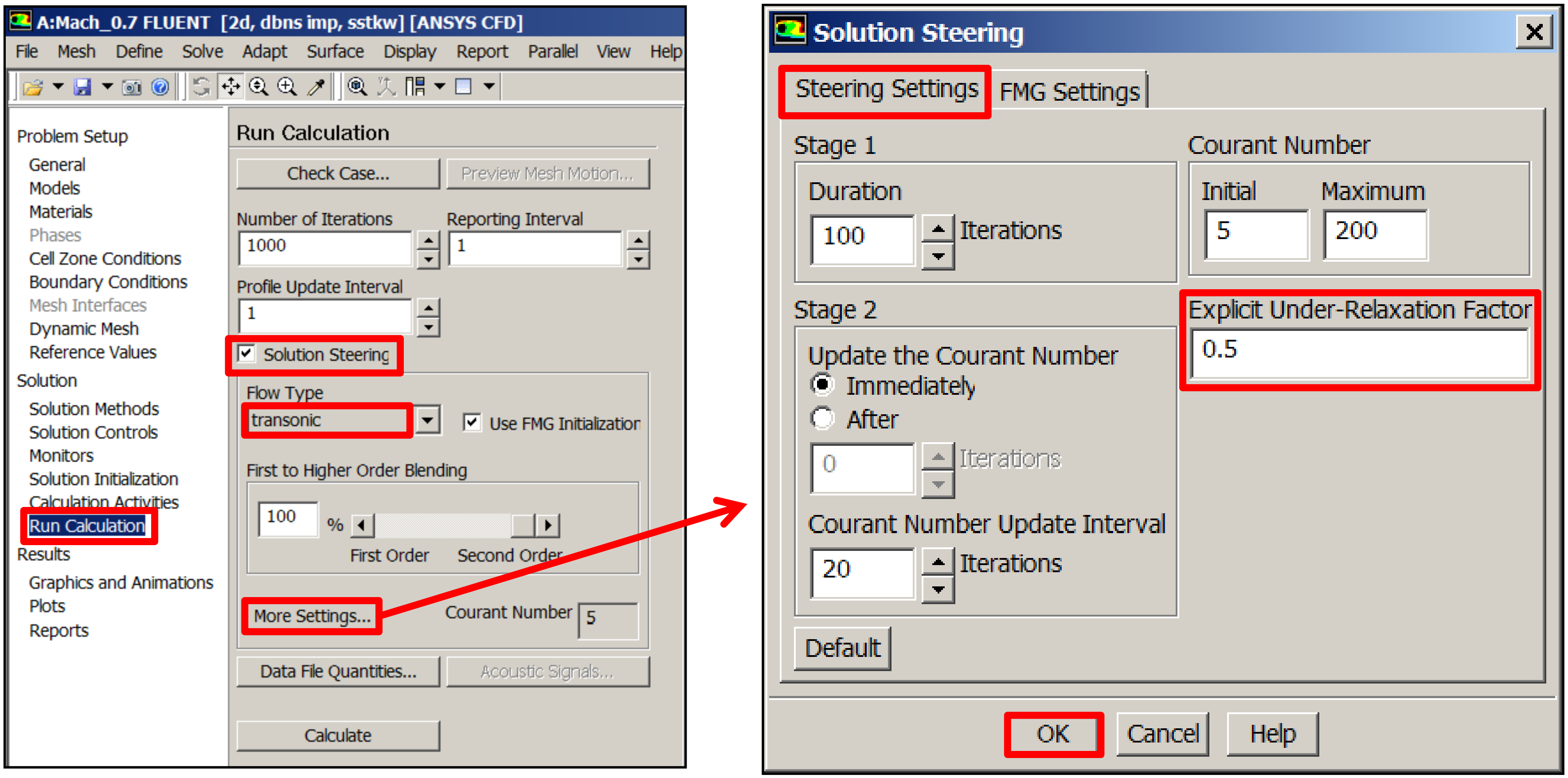
  - **Any potential problems or reminders associated with the case setup will be raised in the case check panel.**
  - **In this case, in the solver tab, there is a recommendation to check the reference values for the force monitors.**
    - **Since we have already set these we can ignore this reminder.**

Case Setup: Solution Steering

- **Solution Steering**
  - uses 'Full-Multi-Grid (FMG) Initialization' which will compute a quick, simplified solution based on a number of coarse sub-grids. This quick solution can help to get a stable starting point and is a better 'initial guess' for the main calculation
  - employs robust first order discretization in the early-stages of the main computation, then blends to the more accurate second order schemes as the solution stabilizes
  - gradually ramps up the CFL in line with stability

- **Run Calculation >**
  - Toggle on Solution Steering
  - Change the flow type to 'transonic'
  - More settings > reduce the Explicit Under-Relaxation Factor to 0.5
    - this is recommended for second order discretization and should make the solution more stable

Case Setup: Solution Steering

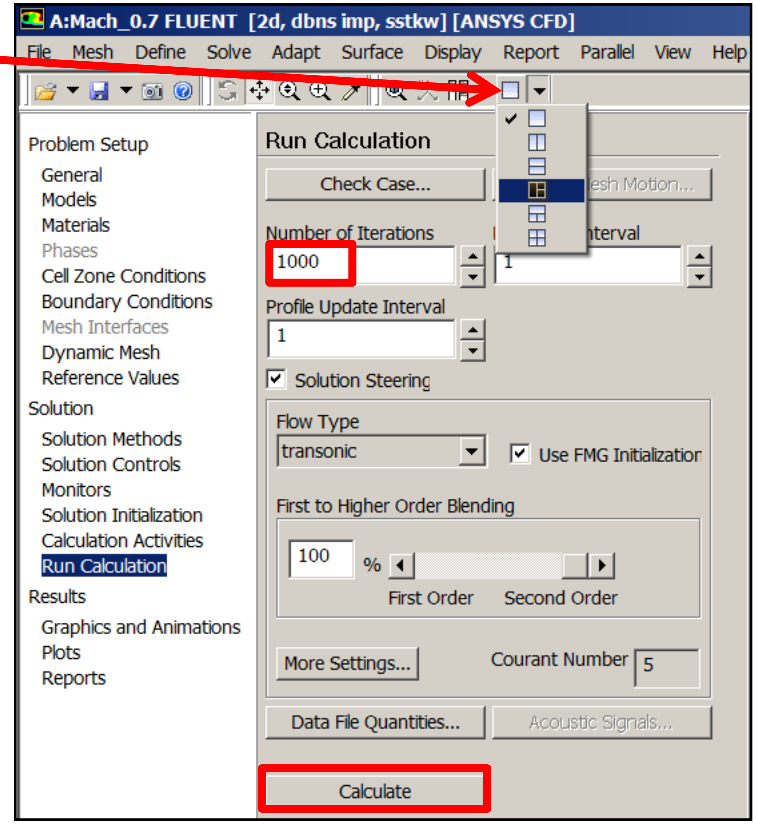


Run Calculation

- **Set three graphics windows**
  - **For the residual, lift and drag monitors**

- **File > Save Project!**

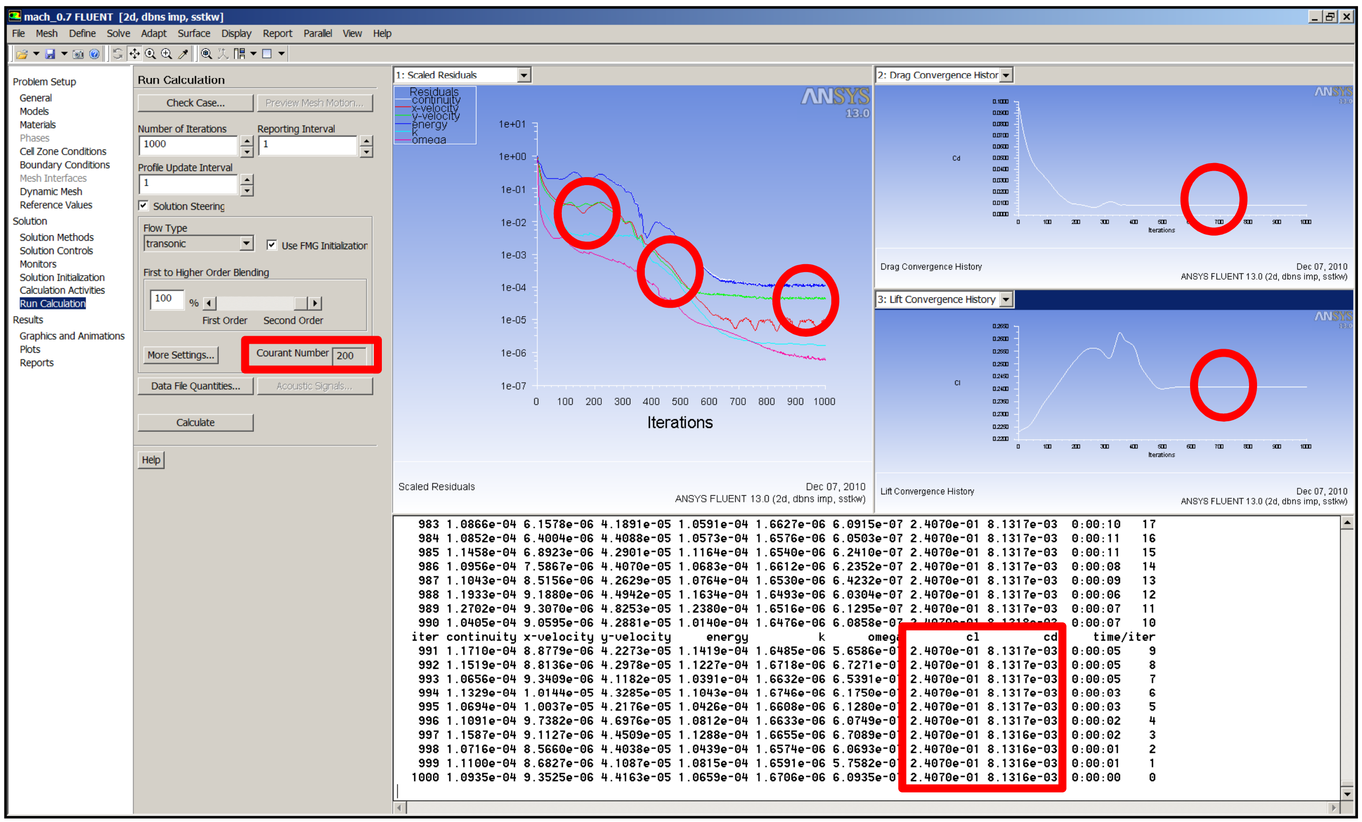
- **Run Calculation >**
  - **Set the number of iterations to 1000**
  - **Press 'Calculate'.**
  - **Note: It is good practice to run and then check the FMG first (by setting the main iteration number to zero and then pressing calculate) before starting the main calculation iterations. The FMG calculation can diverge just as the main calculation can do. Check for non-physical velocities, temperatures, etc. For this workshop, the FMG has already been checked.**
  - **The calculation should take about 15 minutes. Time for a coffee? If you can't wait, the converged case and data files can be read from your user_files folder.**



Run Calculation

ANSYS
Customer Training Material

- **After 1000 iterations the calculation has converged.**
  - **Note that the CFL has been steadily ramped up during the calculation by the solution steering algorithm. This can be seen in the CFL value displayed and the effect on the residuals is also evident.**
  - **The residuals have converged to low values and the drag and lift monitors are no longer changing.**



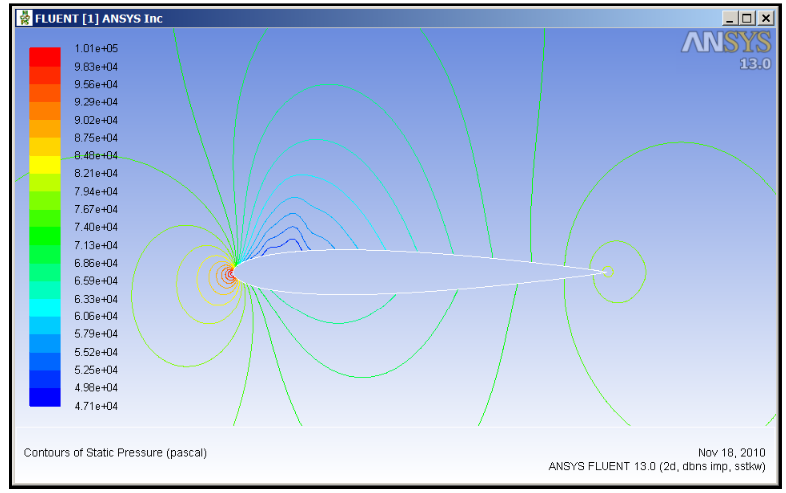
Post Processing [FLUENT]

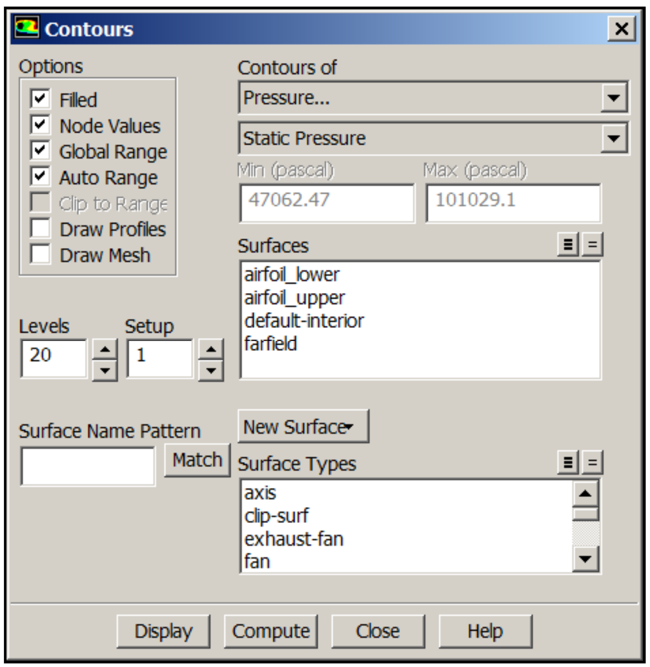
- **Compare the predicted Cl and Cd against the experimental values.**
- **From Reference 1, Cl = 0.241 and Cd = 0.0079**
- **The CFD solution predicts Cl = 0.241 and Cd = 0.0081**
- **Good agreement can be seen.**

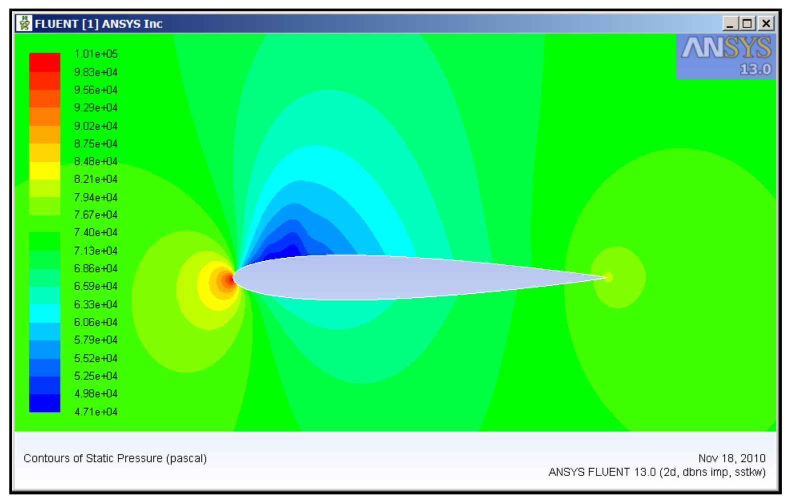
985	1.0885e-04	9.1351e-06	4.1838e-05	1.0611e-04	1.6634e-06	6.3964e-07	2.4070e-01	8.1316e-03	0:00:09	15
986	1.0923e-04	8.9679e-06	4.5414e-05	1.0643e-04	1.6565e-06	5.8666e-07	2.4070e-01	8.1317e-03	0:00:09	14
987	1.0816e-04	8.7660e-06	4.2697e-05	1.0546e-04	1.6602e-06	5.8345e-07	2.4070e-01	8.1317e-03	0:00:10	13
988	1.1473e-04	8.9421e-06	4.4616e-05	1.1181e-04	1.6739e-06	6.2707e-07	2.4070e-01	8.1318e-03	0:00:07	12
989	1.1331e-04	9.7683e-06	4.5691e-05	1.1044e-04	1.6798e-06	6.0941e-07	2.4070e-01	8.1318e-03	0:00:07	11
990	1.1234e-04	1.0138e-05	4.5041e-05	1.0949e-04	1.6628e-06	6.0598e-07	2.4070e-01	8.1318e-03	0:00:05	10
iter	continuity	x-velocity	y-velocity	energy	k	omega	cl	cd	time/iter	
991	1.1395e-04	9.9580e-06	4.2357e-05	1.1108e-04	1.6654e-06	6.3059e-07	2.4070e-01	8.1317e-03	0:00:06	9
992	1.0847e-04	9.4838e-06	4.1618e-05	1.0579e-04	1.6610e-06	6.3300e-07	2.4070e-01	8.1317e-03	0:00:06	8
993	1.0494e-04	8.7773e-06	4.0221e-05	1.0229e-04	1.6541e-06	6.0910e-07	2.4070e-01	8.1317e-03	0:00:04	7
994	1.0903e-04	8.5851e-06	4.6681e-05	1.0623e-04	1.6648e-06	6.3260e-07	2.4070e-01	8.1317e-03	0:00:04	6
995	1.0685e-04	8.9753e-06	4.1830e-05	1.0416e-04	1.6700e-06	6.3715e-07	2.4070e-01	8.1317e-03	0:00:04	5
996	1.0714e-04	9.5702e-06	4.3891e-05	1.0441e-04	1.6649e-06	6.2310e-07	2.4070e-01	8.1317e-03	0:00:02	4
997	1.1706e-04	1.0298e-05	4.4222e-05	1.1416e-04	1.6732e-06	6.3410e-07	2.4070e-01	8.1318e-03	0:00:02	3
998	1.1552e-04	1.0030e-05	4.0622e-05	1.1261e-04	1.6671e-06	6.2916e-07	2.4070e-01	8.1317e-03	0:00:01	2
999	1.1918e-04	9.6996e-06	4.7243e-05	1.1618e-04	1.6717e-06	6.3061e-07	2.4070e-01	8.1318e-03	0:00:01	1
1000	1.1389e-04	9.5101e-06	4.5165e-05	1.1099e-04	1.6633e-06	6.3366e-07	2.4070e-01	8.1318e-03	0:00:00	0

Post Processing [FLUENT]

- **Examine the contours of static pressure**
  - **Graphics and Animations > Contours**
  - **Turn off 'Filled' to just display the contour lines.**
  - **Turn on 'Filled', display again.**
    - **Note the high static pressure at the nose, and low pressure on the upper (suction) surface**

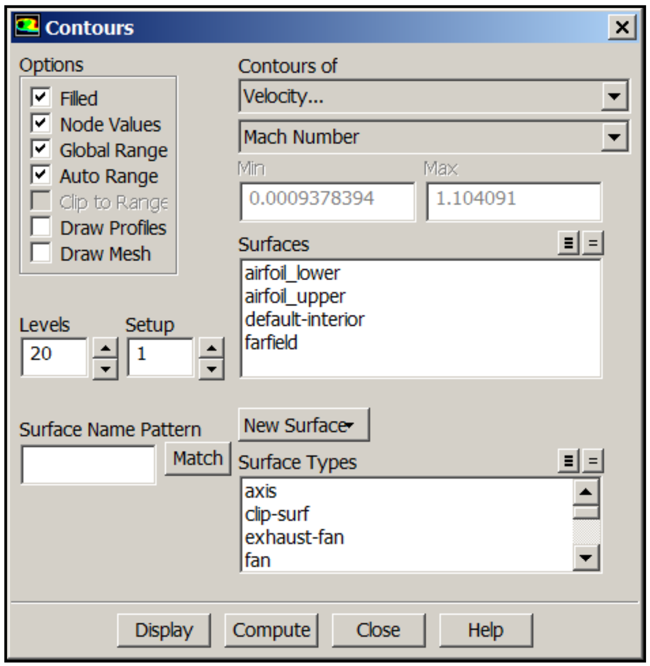


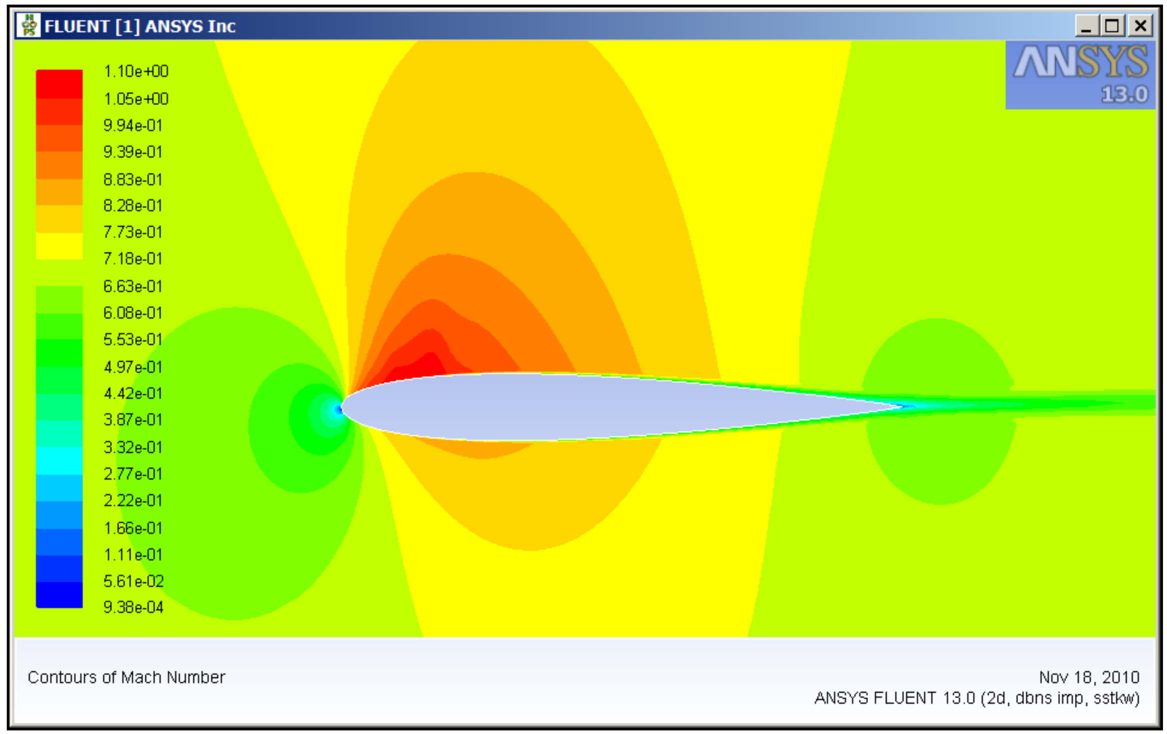




Post Processing [FLUENT]

- **Examine the contours of Mach Number**
  - **notice that the flow is now locally supersonic (Mach Number > 1)**

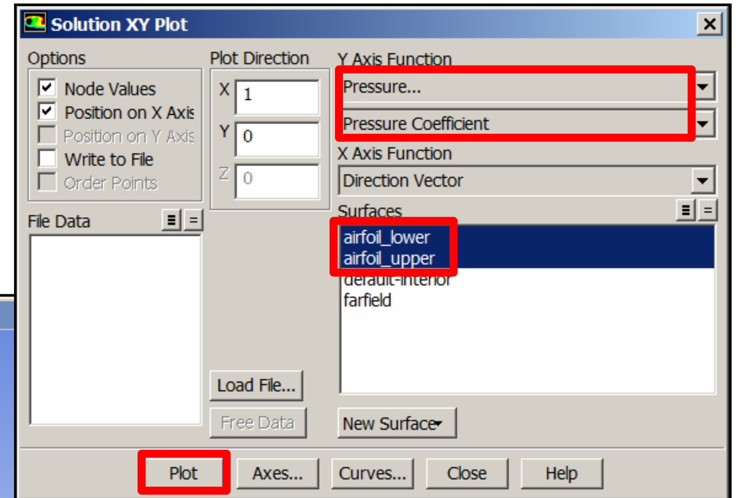


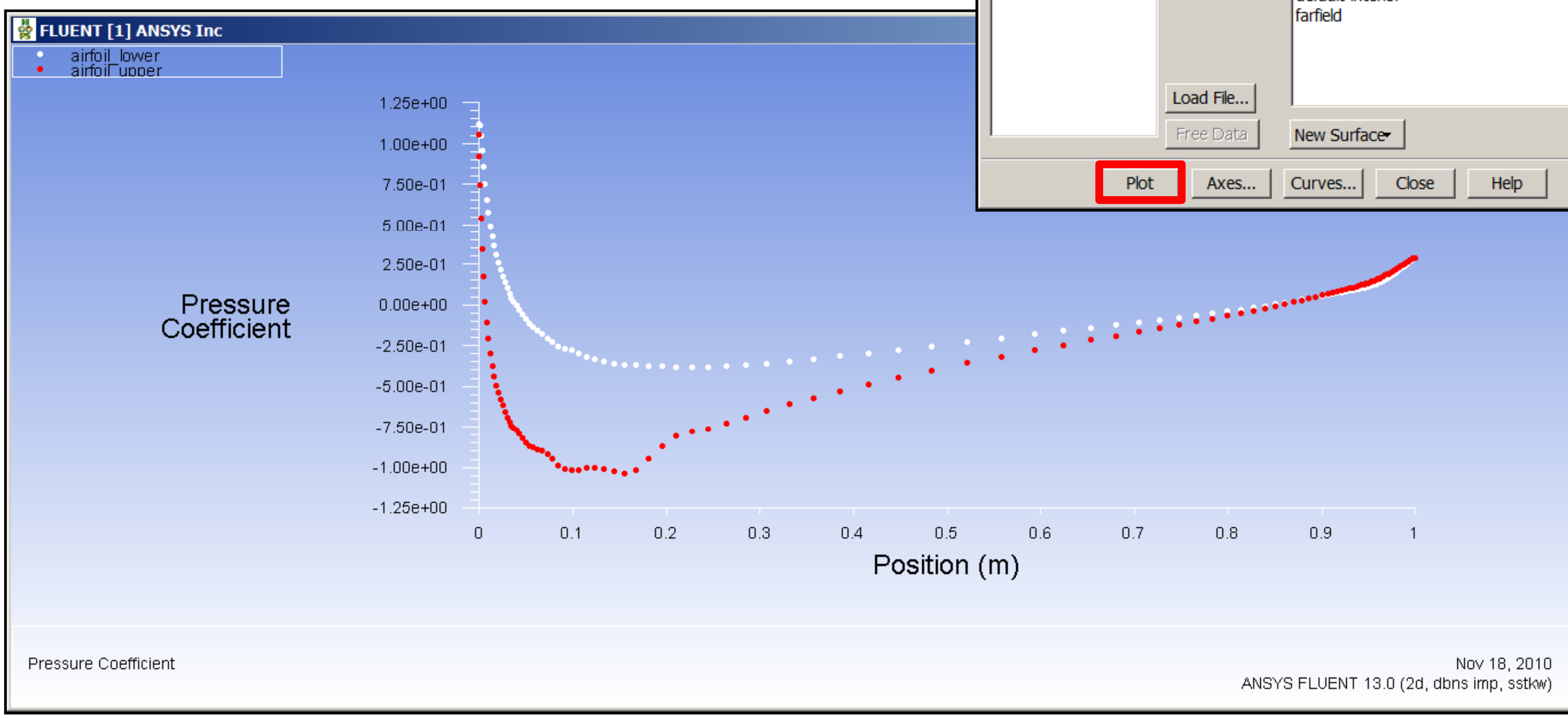


Post Processing [FLUENT]

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

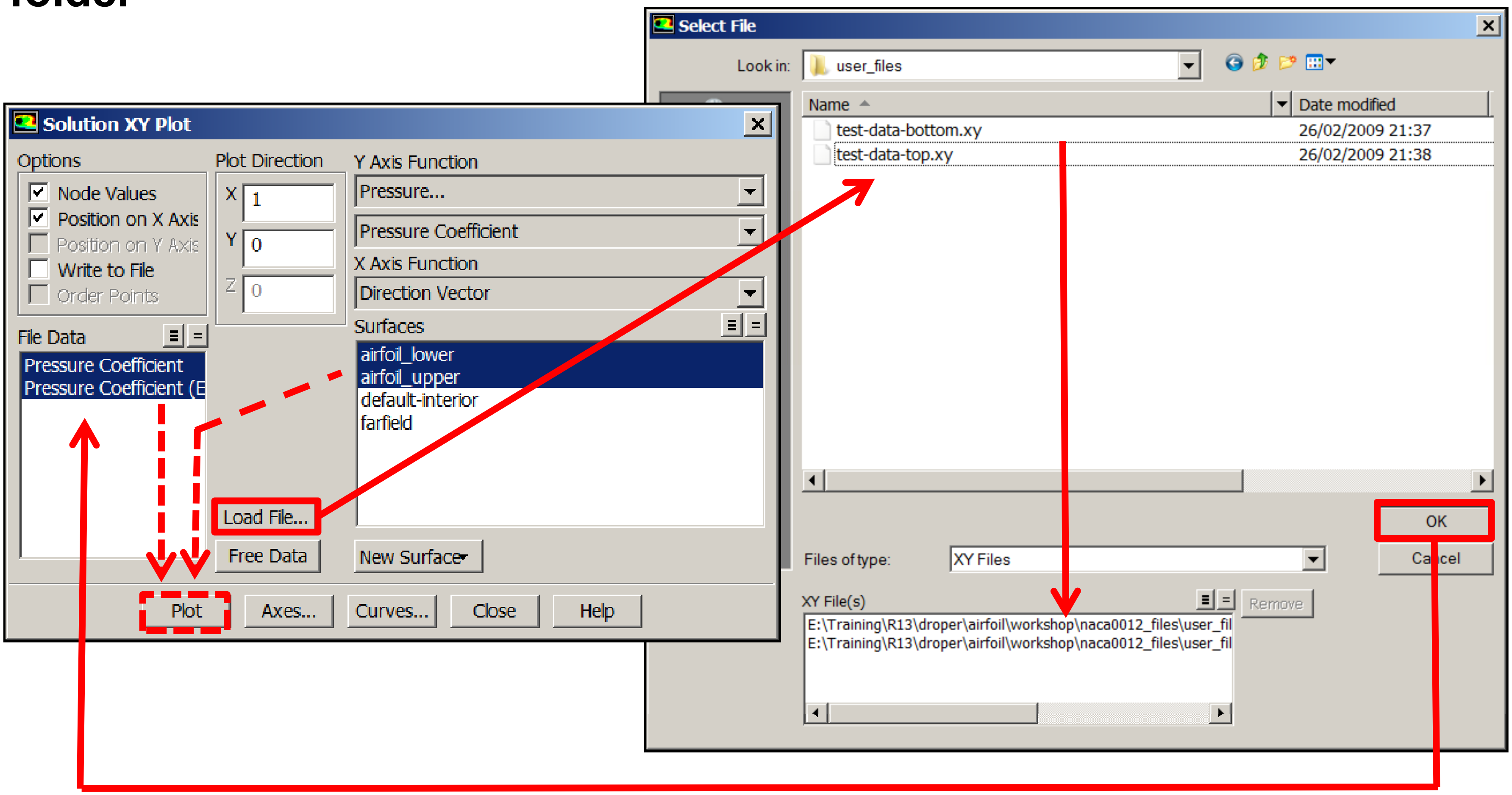
- **Results > Plots > XY Plot > Set Up**





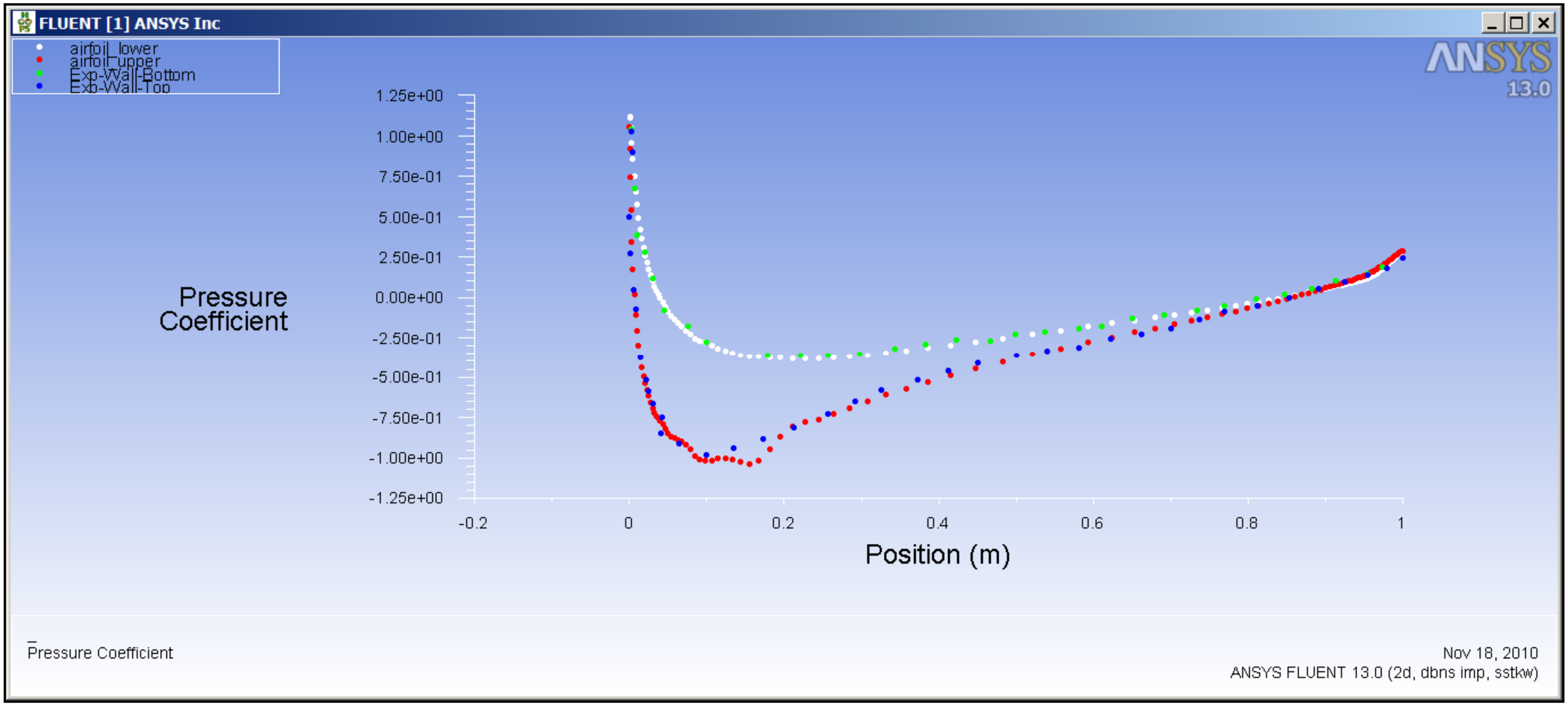
Post Processing [FLUENT]

- **Now load the test Cp data for comparison**
- **Load File > browse to the .xy files placed earlier in the user_files folder**



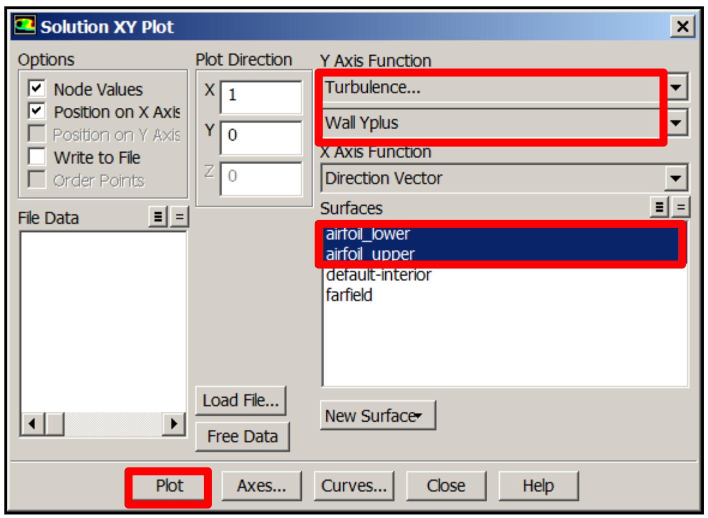
Post Processing [FLUENT]

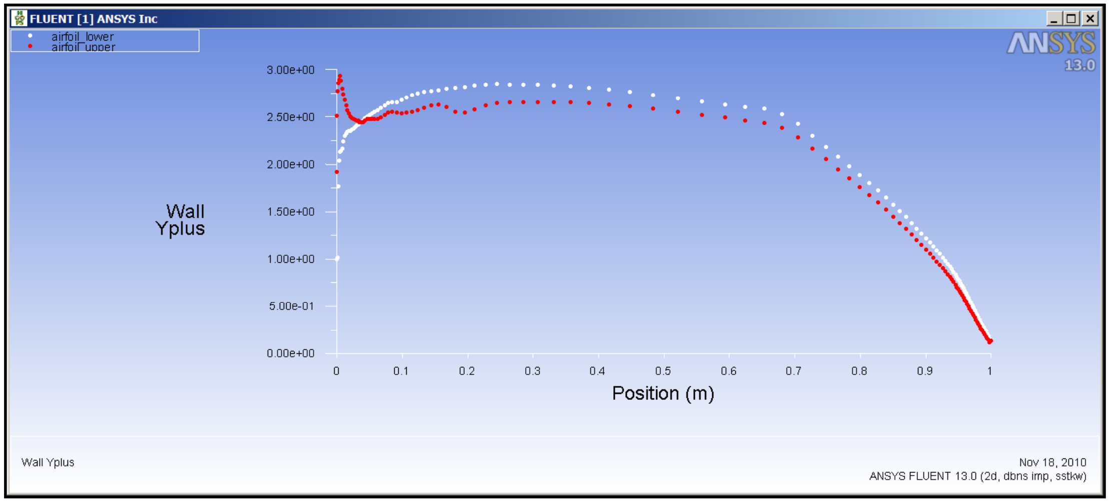
- **Once loaded, plot the CFD and experimental Cp results together.**
- **A good agreement can be seen.**



Post Processing [FLUENT]

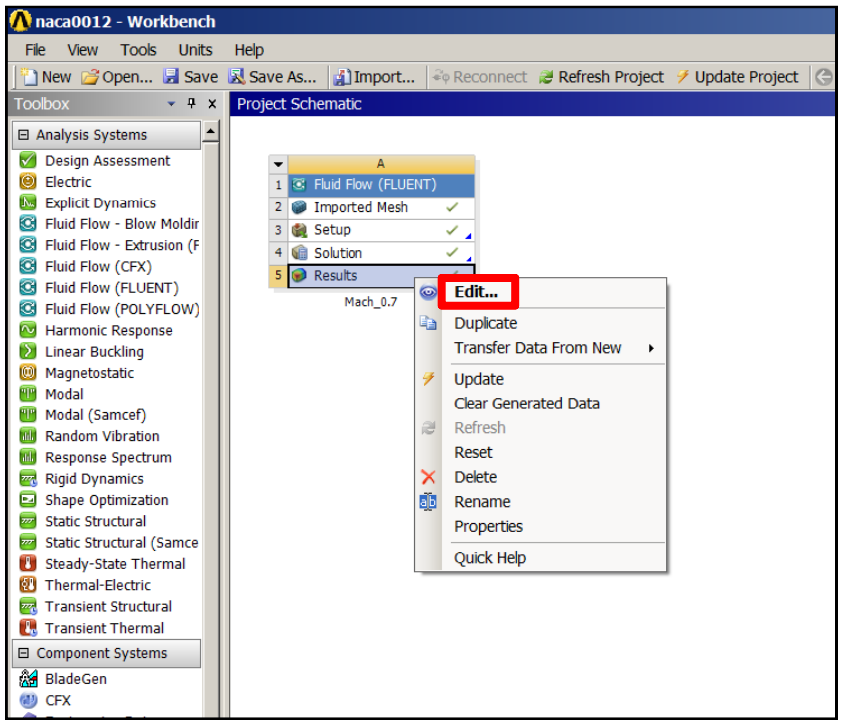
- In order to obtain a good drag prediction, and for the turbulence model to work effectively, the mesh is well resolved near to the wall, such that the first grid point is located in the viscous sub-layer, with Y+ of 5 or less (ideally around 1).
- XY Plot > Turbulence > Wall Y Plus > along both of the airfoil walls.
- We can see that Y+ is around 2.5 for much of the surface, max Y+ < 3





Post Processing [CFD POST]

- **File > Save Project (save the project from the FLUENT file menu)**
- **File > Close FLUENT**
- **Additional post-processing will now be performed in CFD Post.**
- **Return to the Workbench Project window.**
- **Click on 'Refresh Project' and notice the Results cell update.**
- **Right click the Results cell and select Edit. This will launch CFD Post.**

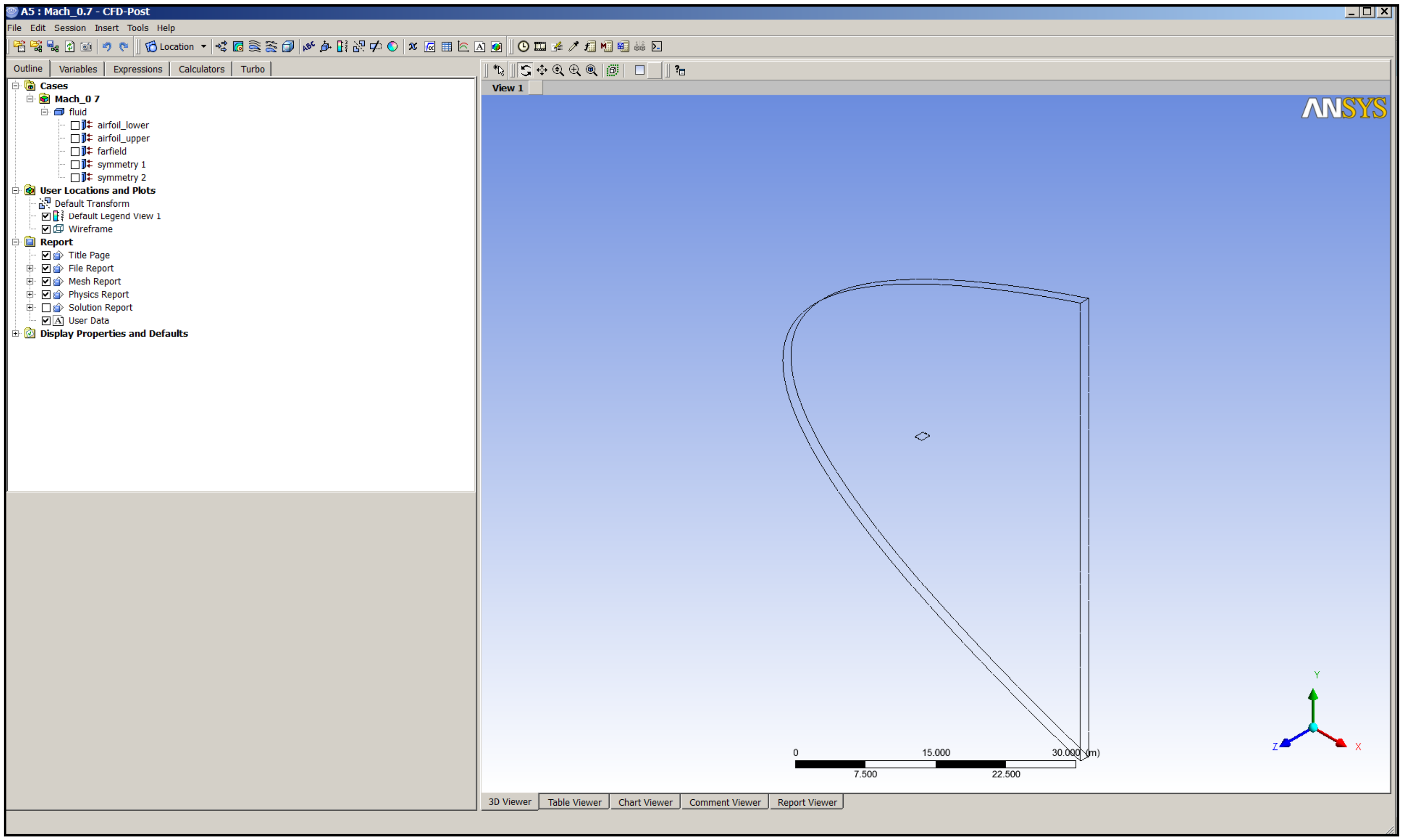


Post Processing [CFD POST]

ANSYS

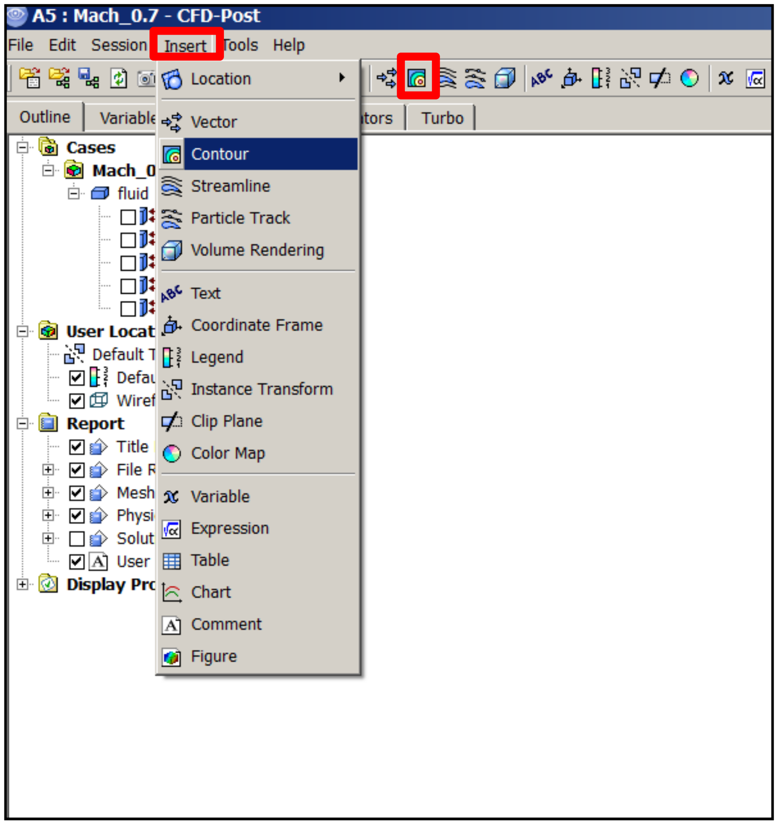
Customer Training Material

- **CFD Post works in 3D, so a unit thickness will be automatically be added to the 2D airfoil, with symmetry side boundaries.**

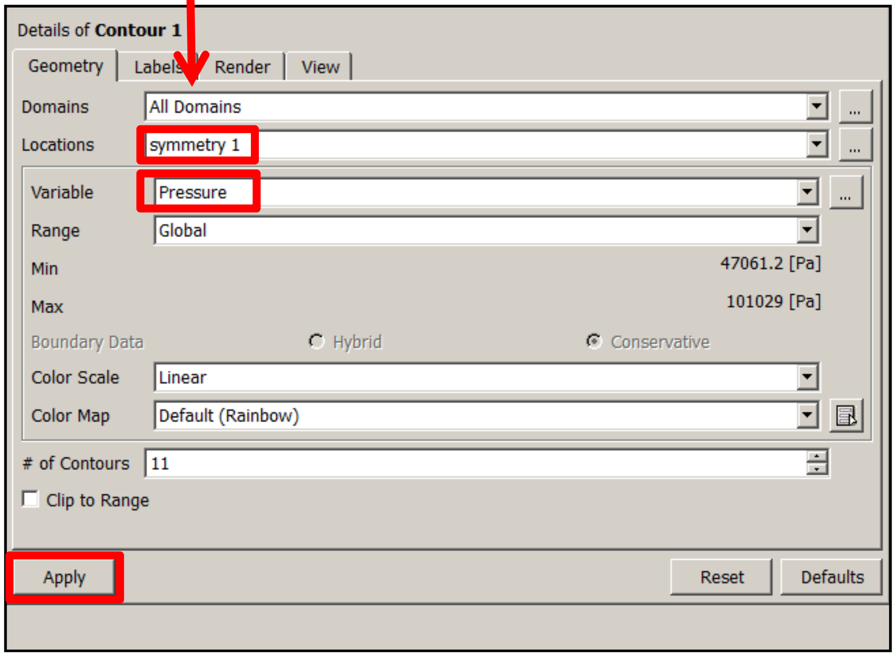


Post Processing [CFD POST]

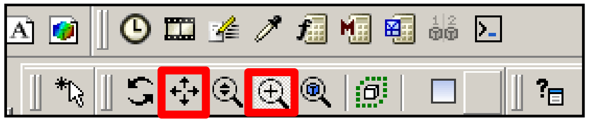
- **Insert a new Contour and accept the default name ‘Contour 1’**
  - **Insert > Contour (or use the icon)**
- **In the details panel, choose an existing location: ‘symmetry 1’**
- **Choose the variable to be ‘Pressure’ then click Apply**



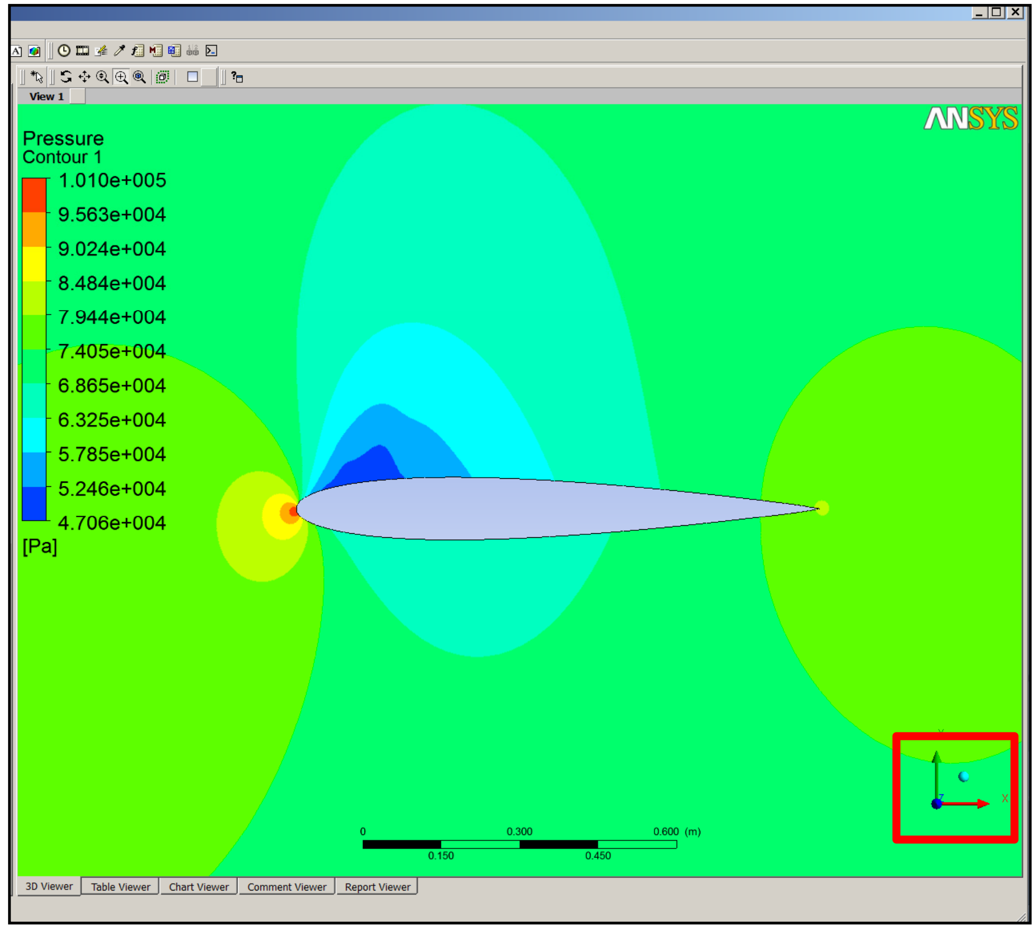
Choose all domains, so that the contour will apply to both cases when a comparison is carried out shortly.



Post Processing [CFD POST]



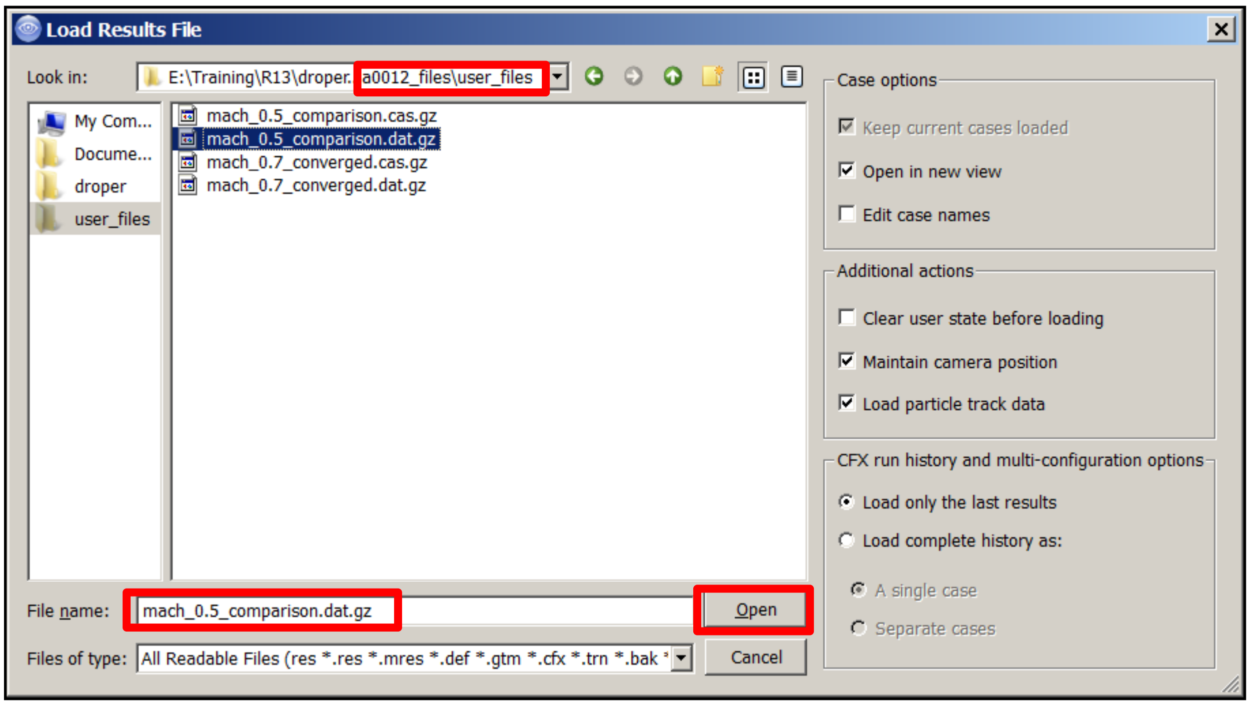
- **Click on the Z-axis in the coordinate system triad to orient in the XY plane.**
- **To zoom in:**
  left mouse button
  (or mouse wheel)
- **To pan:**
  <CTRL> + middle mouse
- **Or use the toolbar icons to switch between modes**



Post Processing [CFD POST]

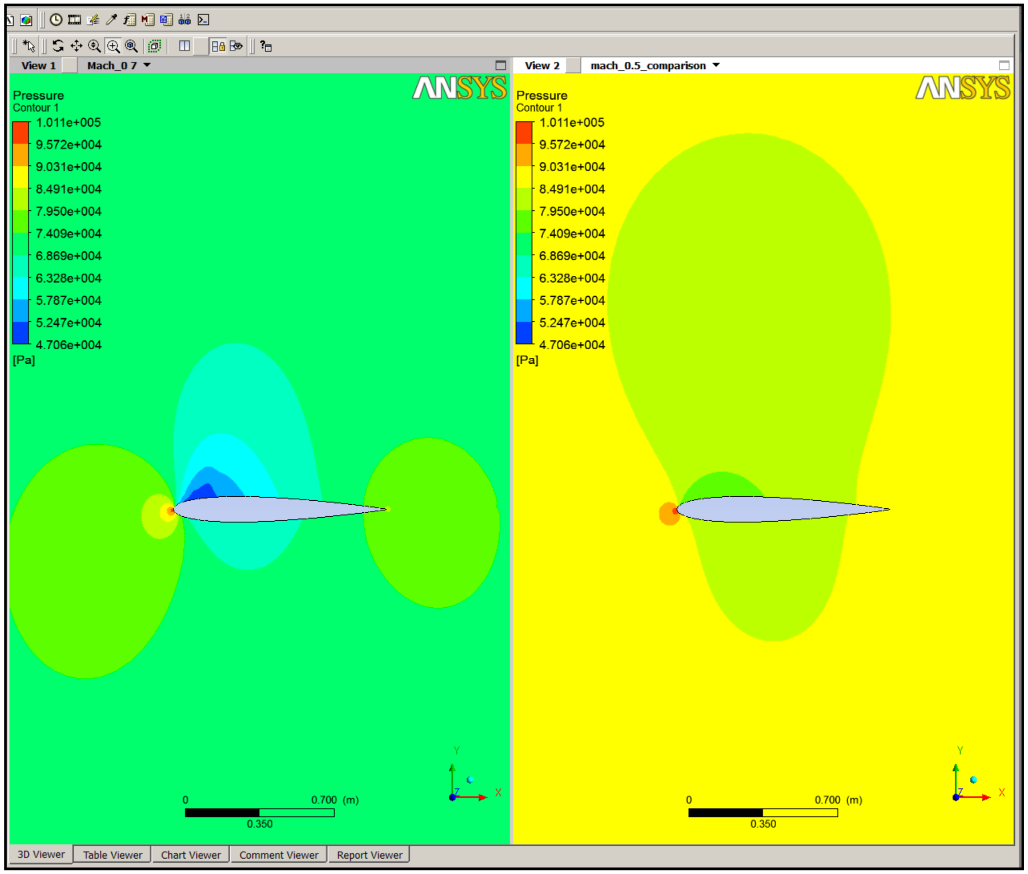
Customer Training Material

- A useful feature in CFD Post is the ability to load multiple sets of CFD and/or test data, and to then compare any two of them together to generate a difference plot.
- File > Load Results > browse to the user_files folder
  - Load ‘mach_0.5_comparison.dat.gz’
  - Click OK if an information dialog box appears



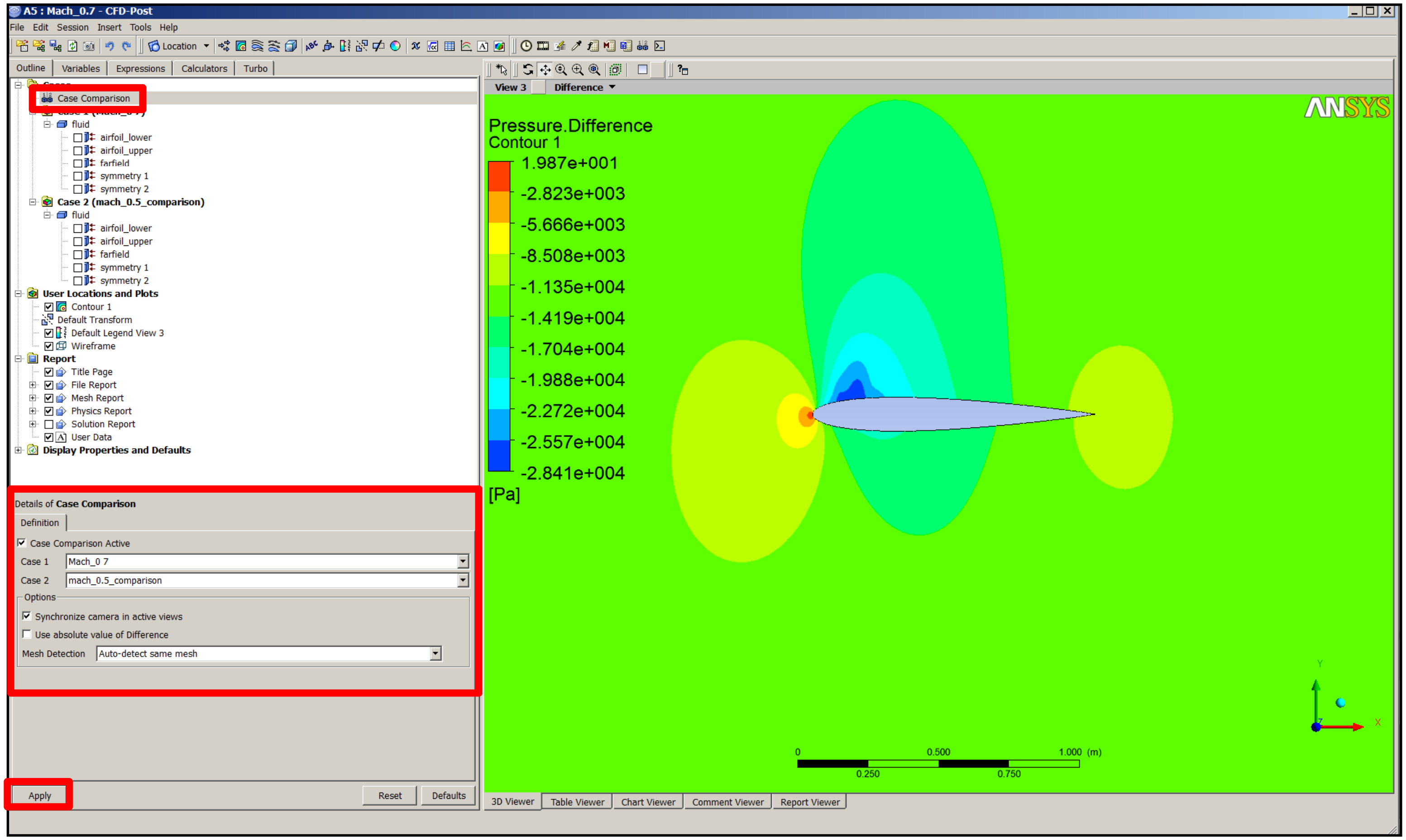
Post Processing [CFD POST]

- **Make sure that two windows are open and that each case is displayed in a different window.**
- **Lock the views so they are synchronised.**



Post Processing [CFD POST]

- **Double-click 'Case Comparison' to open the details panel.**
- **Activate, and select the cases to compare (there are only two here).**
- **Apply. The pressure difference between the two cases is plotted.**



Summary

- In this workshop ANSYS FLUENT has been used within a workbench project to compute the transonic, compressible flow over a naca0012 airfoil.

- The implicit density based solver with solution steering was employed.

- Computed results have been compared to published experimental data and good agreement was achieved.

- A case comparison has been carried out within CFD Post to compare the pressure fields at Mach 0.5 and Mach 0.7

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.