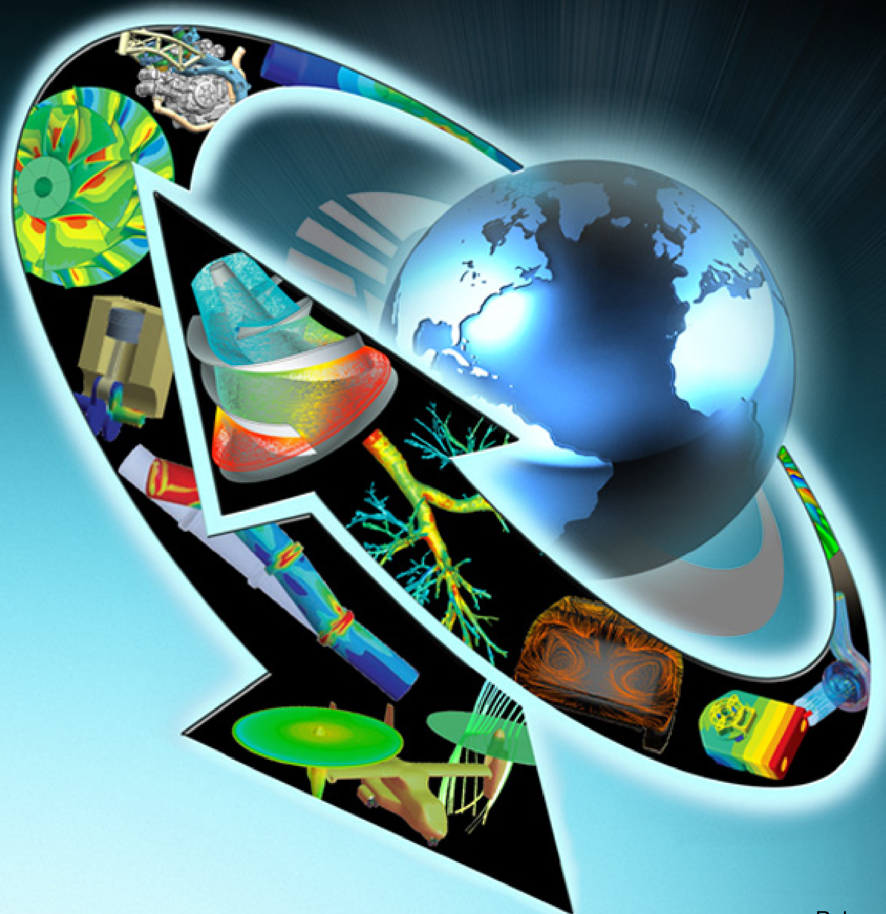
Customer Training Material

Workshop 1

Fluid Flow and Heat Transfer in a Mixing Tee



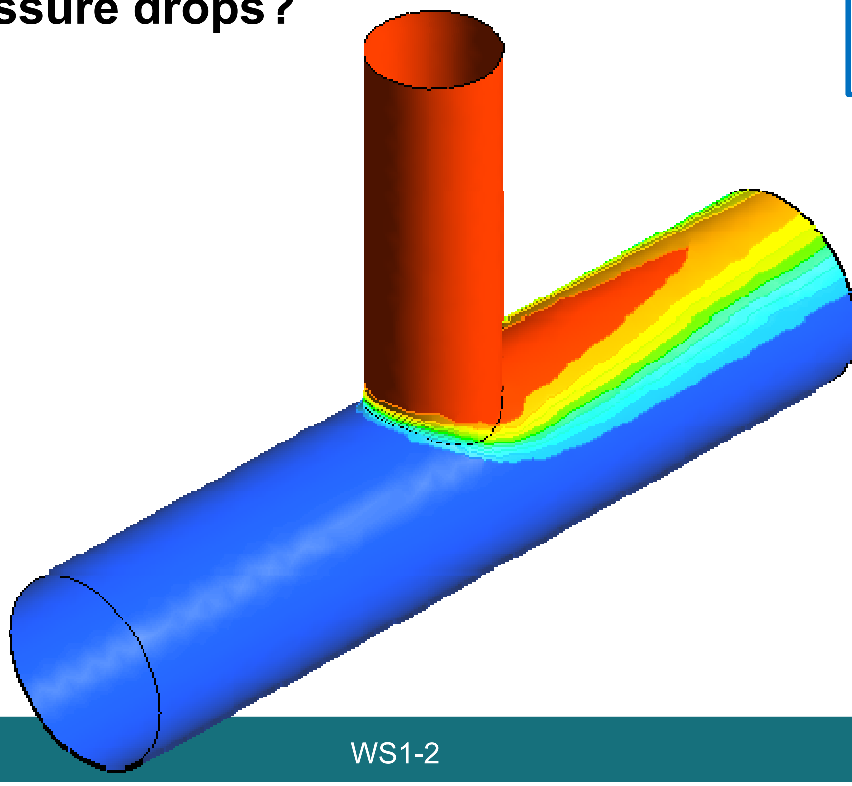
Introduction to ANSYS FLUENT

Welcome!

- **Introductory tutorial for FLUENT**
  - **Starting from existing mesh (generated in earlier tutorial)**
  - **Model set-up, solution and post-processing**

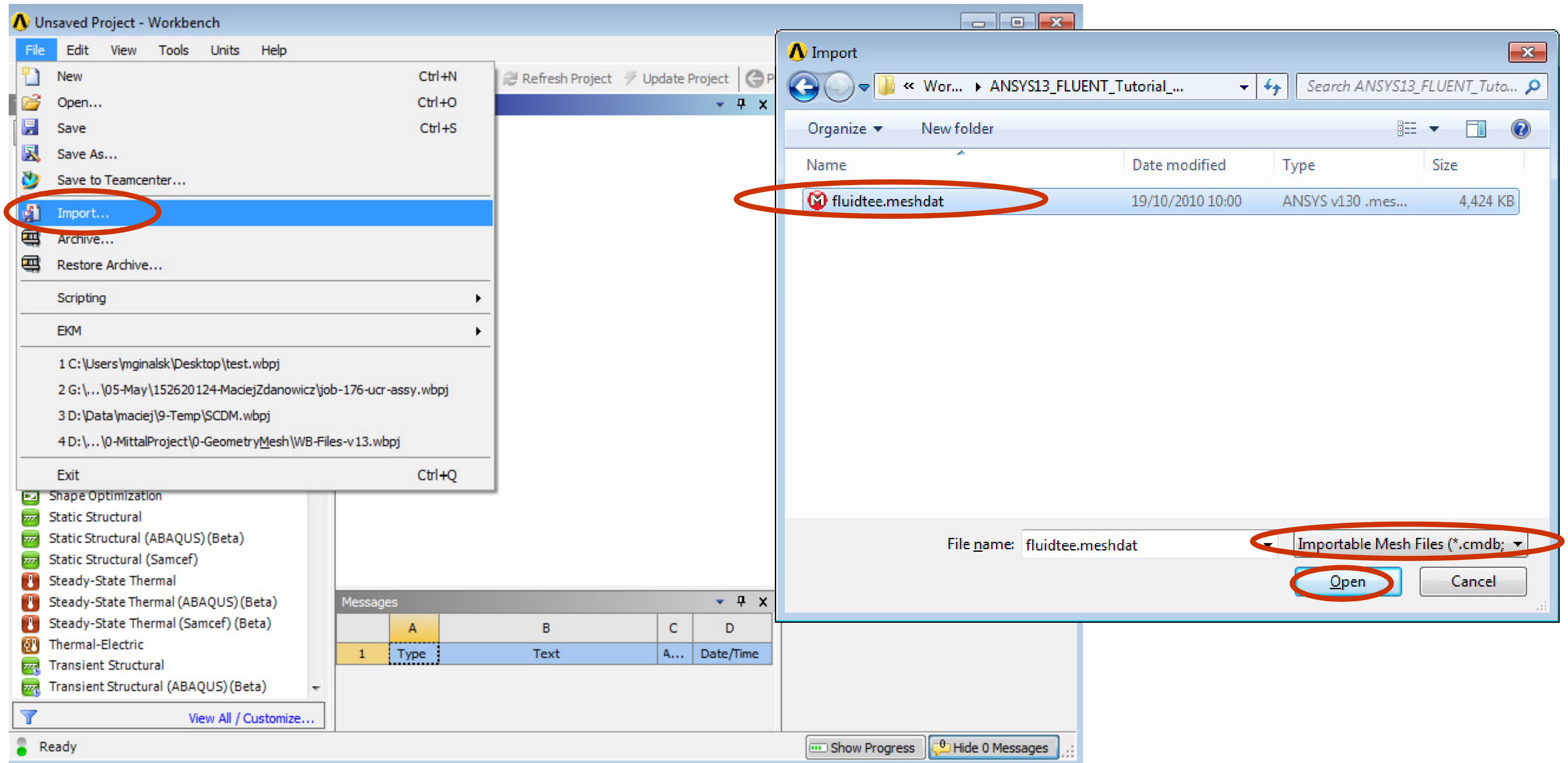
- **Mixing of cold and hot water in a T-piece**
  - **How well do the fluids mix?**
  - **What are the pressure drops?**

It's a good idea to **identify the key simulation outcomes** from the start. You can use these to monitor solution progress.



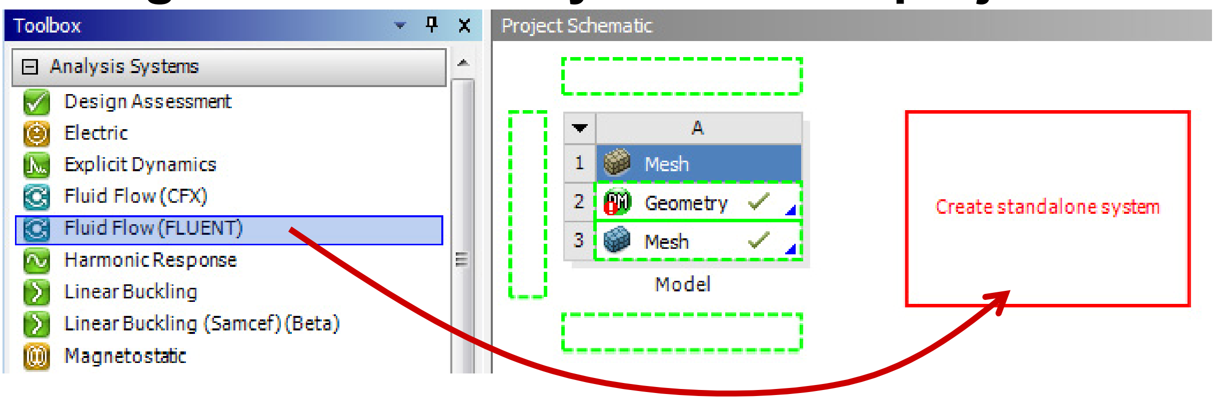
Start in Workbench

- **If starting from a ready-made mesh file (*.meshdat), start Workbench and import the file (see screenshot below)**
  - **and save the project**
- **Alternatively, start in the Workbench project that generated the mesh**



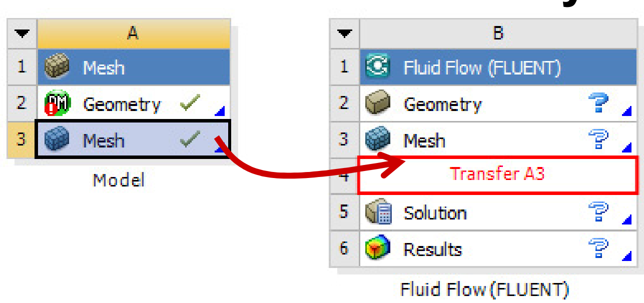
Start a FLUENT case

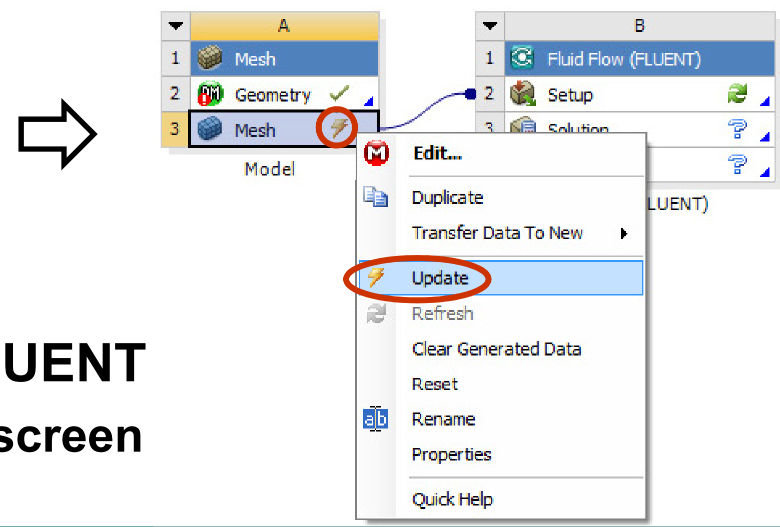
- **Drag a FLUENT analysis into the project**



Dragging the mesh into the FLUENT analysis tells Workbench which mesh format is required. The status icon changes to Update to signify that the mesh now needs to be converted and written.

- **Drag the existing mesh into the FLUENT analysis Setup cell**
  - **then Update the mesh (Right-click on ) to convert and write the mesh for the FLUENT analysis.**





- **Double-click on Setup to launch FLUENT**
  - **click OK on the FLUENT Launcher screen**

FLUENT interface

The main commands are reached from the **navigation pane**

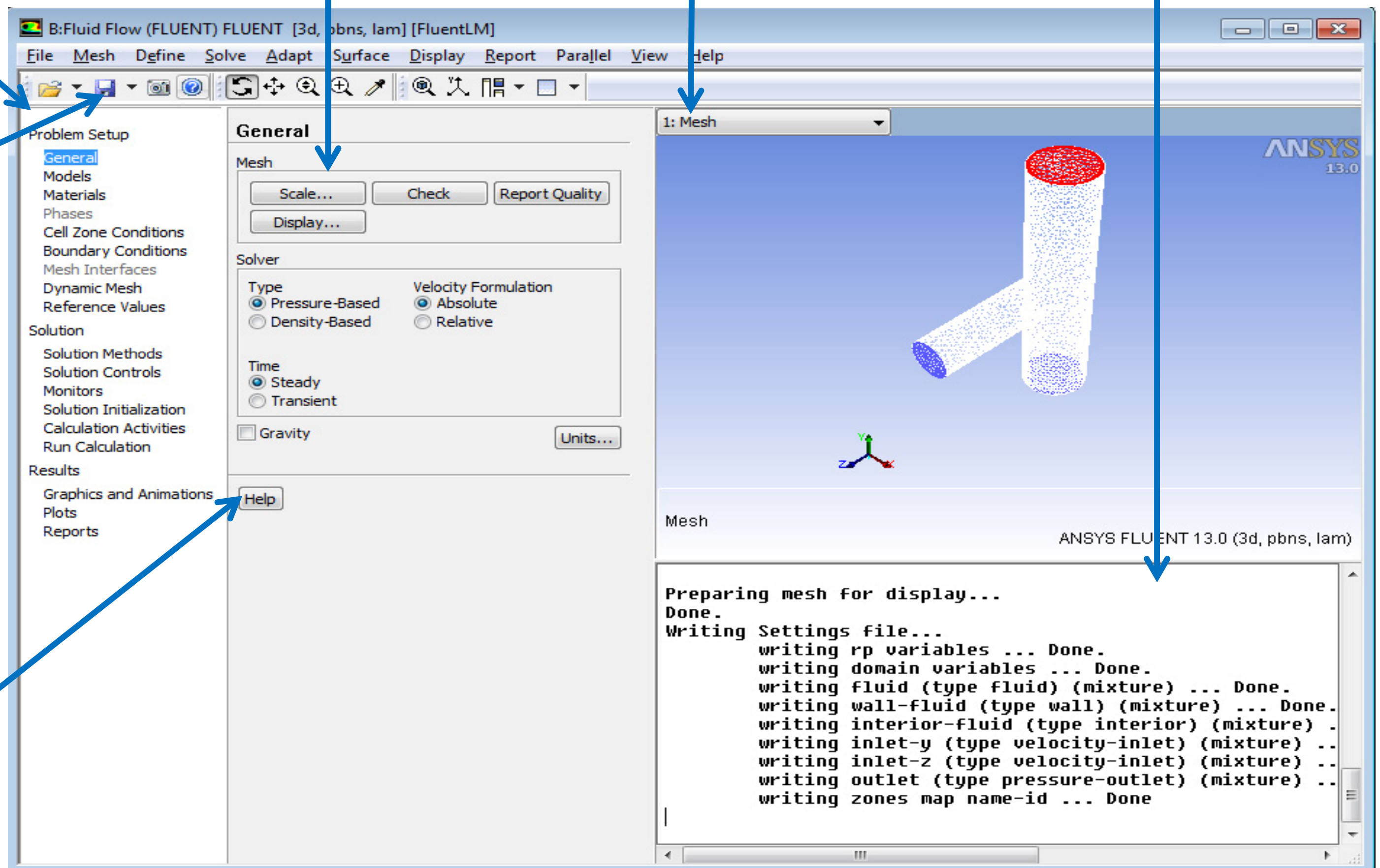
Each item in the navigation pane brings up a new **task page**. A typical workflow will tackle these in order

One or more **graphics windows** will be available (shown here with reduced size)

The **console window** displays text, and can accept TUI (Text User Interface) commands

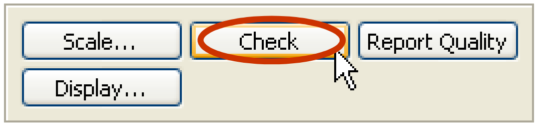
Some useful commands have **toolbar buttons**

The **Help** button brings up context-sensitive help pages



Mesh scale and check

- **Press 'Check'**
  - **- review the text output**




```
Domain Extents:
  x-coordinate: min (m) = -7.619977e-02, max (m) = 7.619977e-02
  y-coordinate: min (m) = -3.556000e-01, max (m) = 3.556000e-01
  z-coordinate: min (m) = -7.620000e-02, max (m) = 3.746500e-01
Volume statistics:
  minimum volume (m3): 3.882075e-08
  maximum volume (m3): 1.872221e-06
    total volume (m3): 1.534207e-02
Face area statistics:
  minimum face area (m2): 7.836273e-06
  maximum face area (m2): 3.103300e-04
Checking mesh........................
Done.
```

The **mesh check** ensures that each cell is in a correct format and connected to other cells as expected. It is recommended to check every mesh immediately after reading it. Failure of any check indicates a badly formed or corrupted mesh which will need repairs prior to simulation.

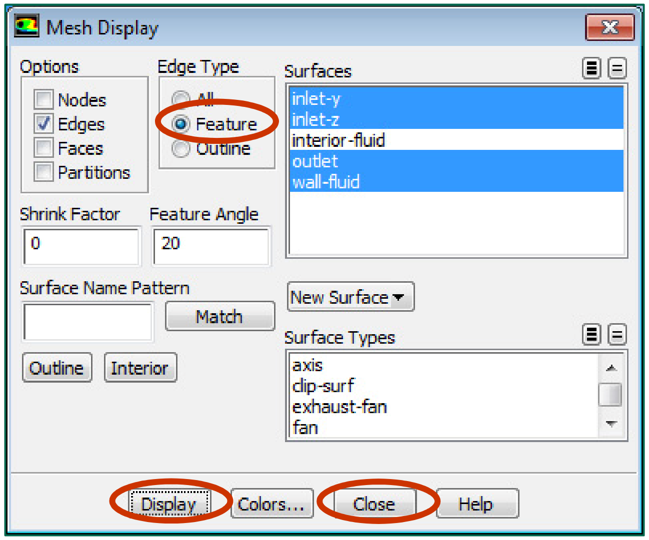
- **Press 'Report quality'**
  - **- review the text output**

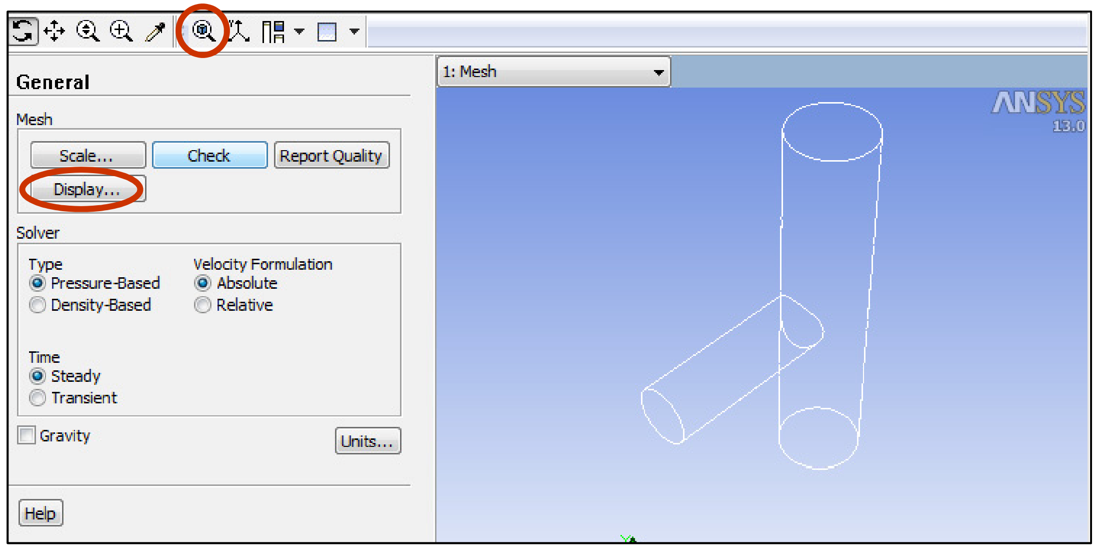
```
        writing inlet-y (type velocity-inlet) (mixture) ... Done.
        writing inlet-z (type velocity-inlet) (mixture) ... Done.
        writing outlet (type pressure-outlet) (mixture) ... Done.
        writing zones map name-id ... Done

Mesh Quality:
 Orthogonal Quality ranges from 0 to 1, where values close to 0 correspond to low quality.
Minimum Orthogonal Quality =  3.70920e-01
Maximum Aspect Ratio =   3.48184e+01
```

Mesh quality is very important for getting a converged, accurate solution. The worst cells will have an orthogonal quality closer to 0, with the best cells closer to 1. The minimum orthogonal quality for all types of cells should be more than 0.01, with an average value that is significantly higher (0.2). The maximum aspect ratio is 34.8, which is high, but acceptable in inflation layers. If the mesh quality is unacceptable it is best to remesh before proceeding. There are other possible remedies in FLUENT, such as conversion to polyhedral cells.

Display geometry

- **Press ‘Display’**
  - **set ‘Edge Type’ to ‘Feature’, press ‘Display’ and then ‘Close’**

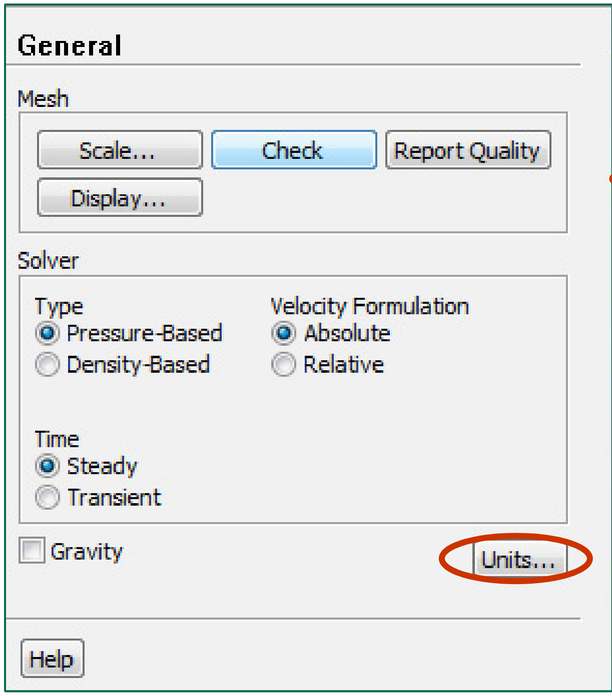


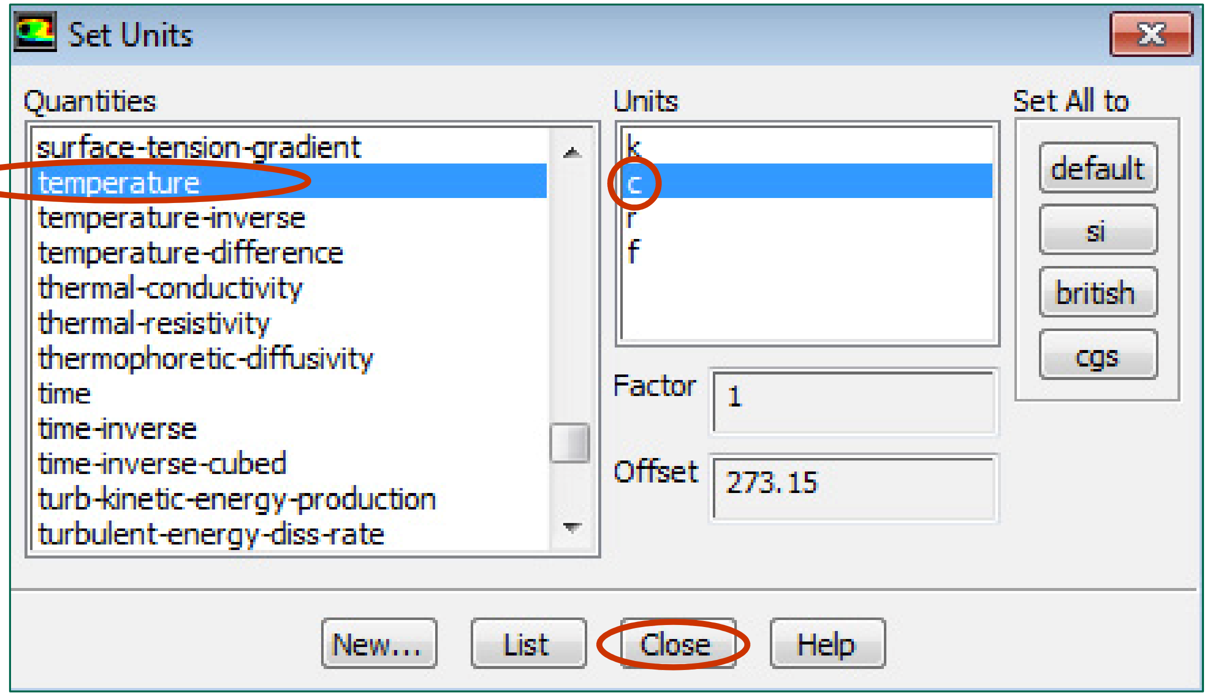


- **Adjust the view if you like**
  - **in rotation mode:**
    - **drag left-mouse-button rotates**
    - **drag middle-mouse-button zooms (to zoom in, drag down and right)
(to zoom out, drag up and left)**
    - **click middle-mouse-button centre origin on click**

Change units of temperature

- **Click ‘Units’**
  - **select ‘Temperature’ to be ‘c’ (Celsius)**
  - **click ‘Close’**



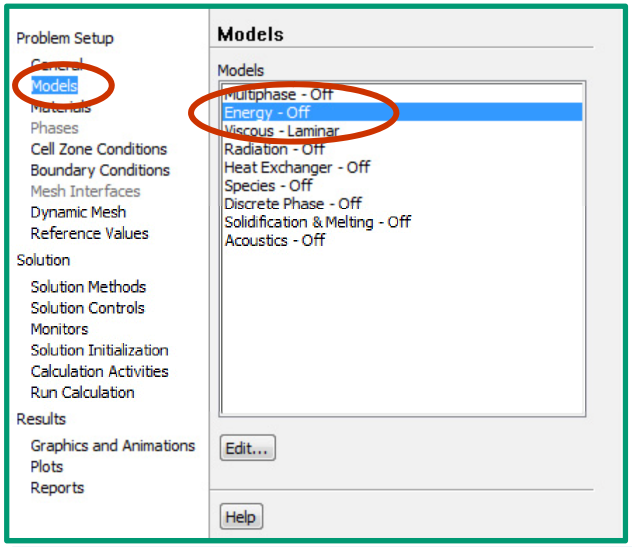


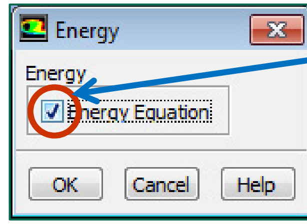
FLUENT stores values in **SI units**. Most postprocessing can be converted to other units.

Activate models

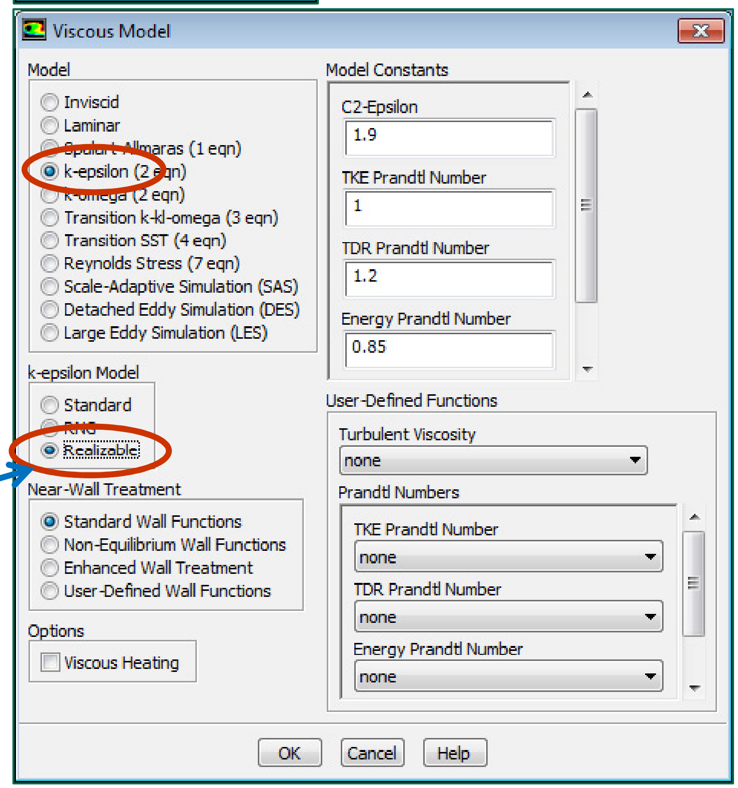
ANSYS
Customer Training Material

- **Double-click (or click and press 'Edit...') the following models:**
  - **Energy Equation: On**
  - **Viscous model: 'k-epsilon', 'Realizable'**





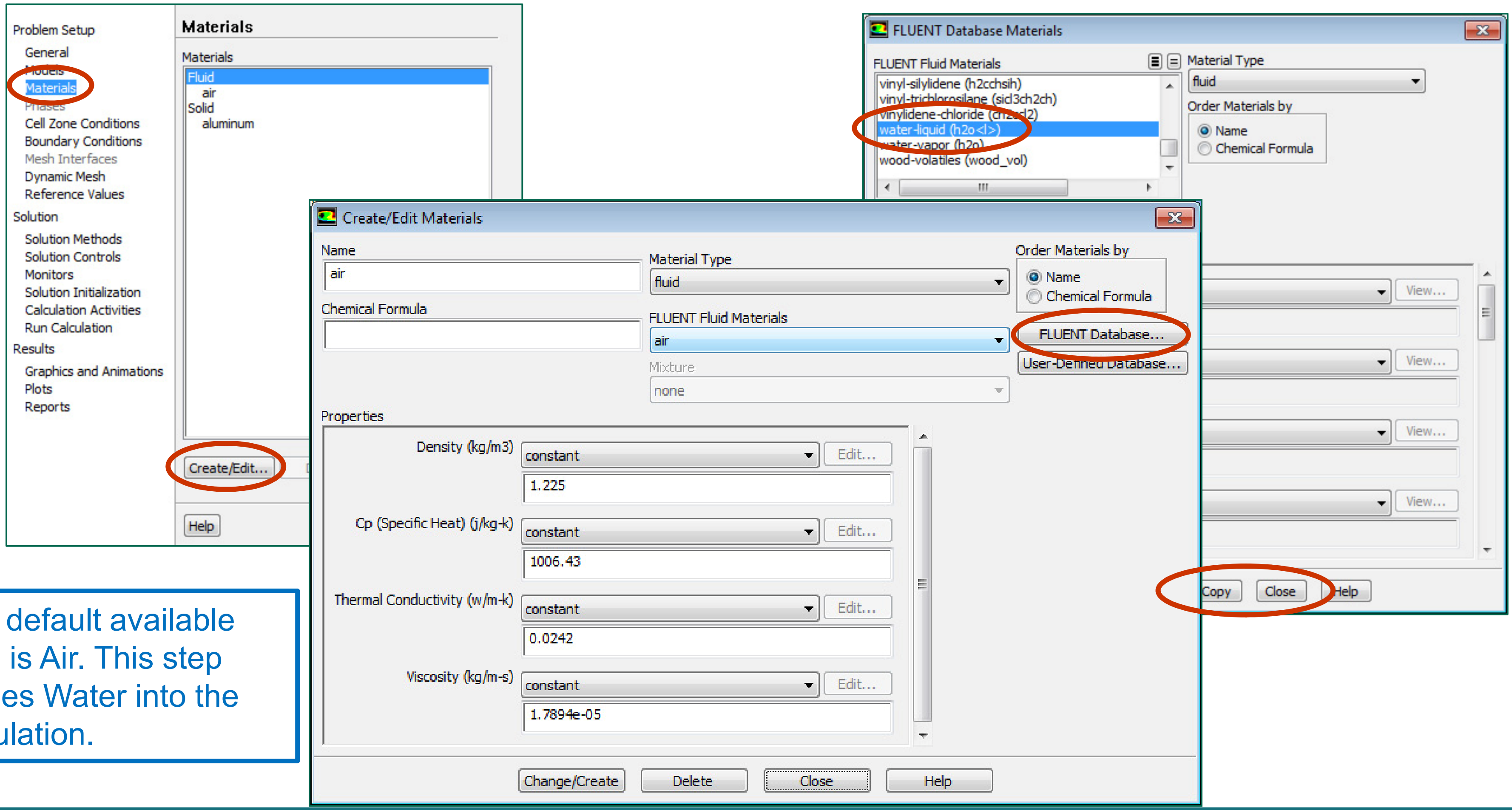
Activating the **Energy equation** allows temperature dependent problems to be solved.



Turbulence modelling is a complex area. The choice of model depends on the application. Here, the Realizable k-epsilon model is used. This is an improvement on the well-established Standard k-epsilon model. Accept the remaining default settings.

Define a new material

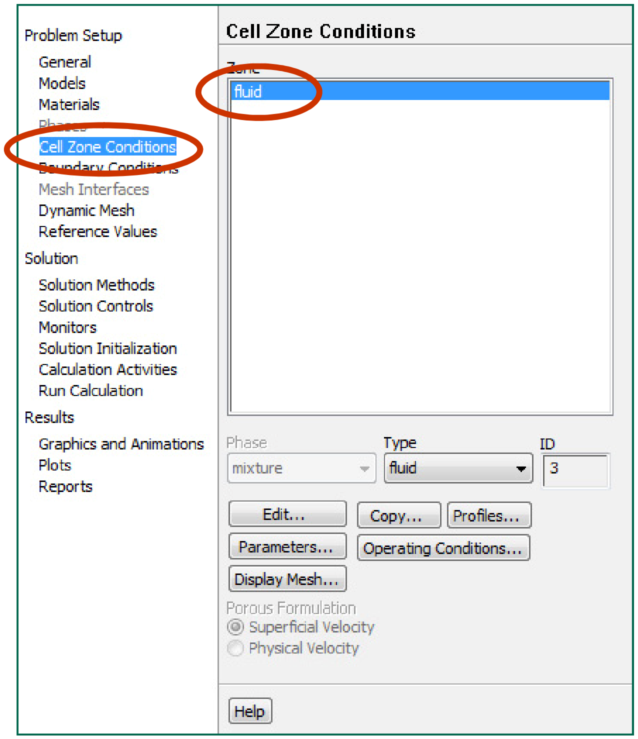
- **In Materials, click ‘Create/Edit...’**
  - **click ‘FLUENT Database...’**
  - **select ‘water-liquid’, press ‘Copy’, then close both windows**

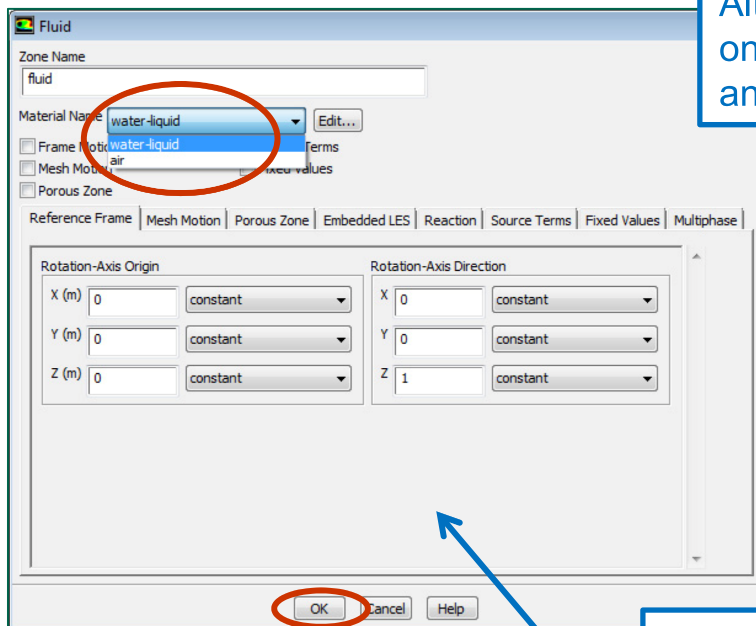


The default available fluid is Air. This step copies Water into the simulation.

Cell Zone Conditions

- In ‘Cell Zone Conditions’, double-click the zone called ‘fluid’
  - Change Material Name to ‘water-liquid’
  - accept all other settings



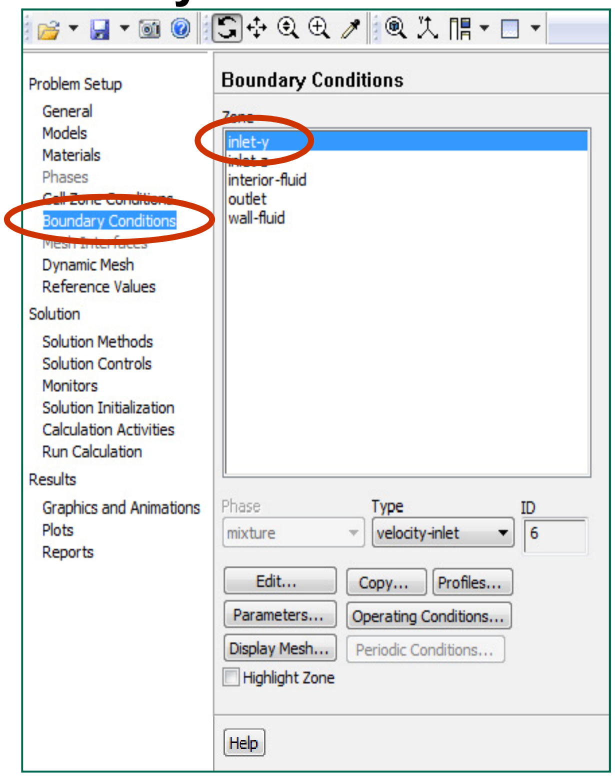


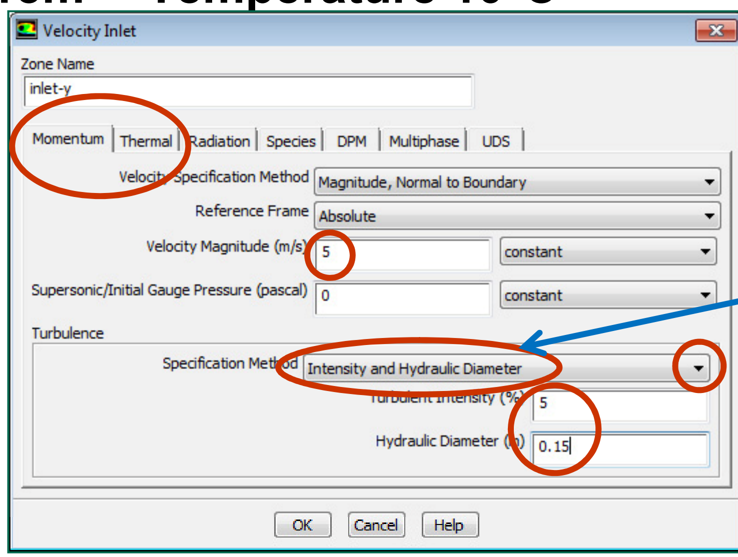
Alternatively, click once on ‘fluid’ to highlight it, and then click ‘Edit...’.

Throughout the problem setup there are many options that are left to default settings as they are not relevant to this particular type of analysis.

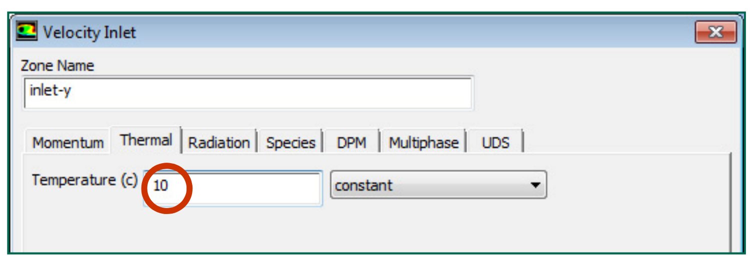
Boundary Conditions

- **In 'Boundary Conditions', double-click the zone called 'inlet-y' and set the following under the Momentum tab. Set Temperature under the Thermal tab.**
  - **Velocity Magnitude 5m/s   Turbulent Intensity 5%**
    Hydraulic Diameter 0.15m   Temperature 10°C





Inlet flows bring turbulence with them. The quantities depend on the upstream conditions so they are user inputs. For flow in pipes, turbulent intensity is typically 5 to 10%. The length-scale of the turbulence can be deduced from the pipe diameter.



Boundary Conditions

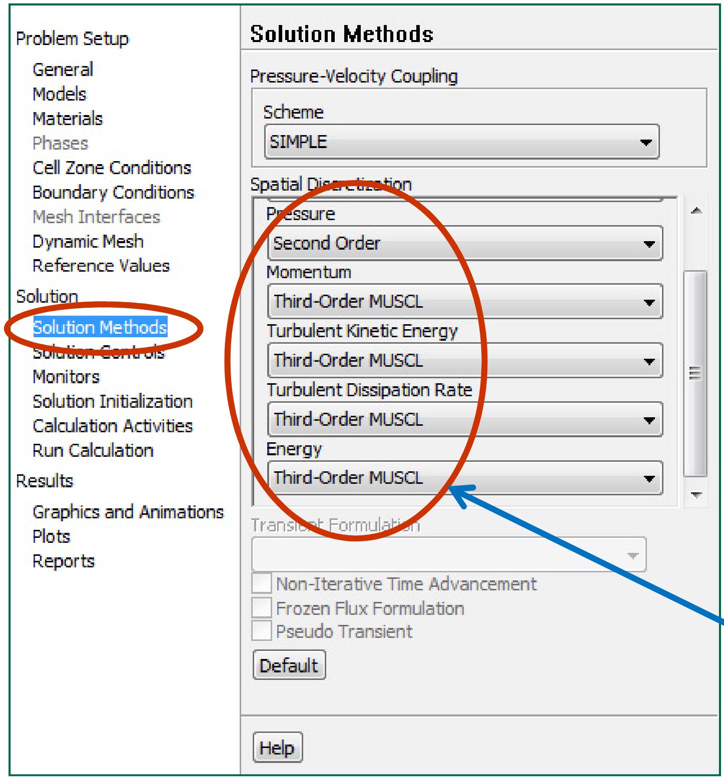
- **Double-click the zone called 'inlet-z' and apply the following settings**
  - **Velocity Magnitude 3m/s Turbulent Intensity 5%**
    Hydraulic Diameter 0.10m Temperature 90°C

- **Double-click the zone called 'outlet'**
  - **for this problem the outlet gauge pressure is 0**
  - **turbulence must still be specified and is referred to as 'Backflow' when specified on an outlet.**
  - **for backflow: Turbulent Intensity 5%**
    Hydraulic Diameter 0.15m Temperature 30°C

The simulation may predict that flow enters the model through parts of the outlet. This **backflow** will bring turbulence and energy back into the model. However, the model cannot predict how much (because the flow is coming from outside of the model). It is therefore necessary to specify backflow conditions. Ideally, the geometry should be selected such that flow enters the model only at well-defined inlets. The backflow settings then do not affect the final solution (although they may be used in intermediate iterations).

Second-order discretization

- In 'Solution methods'
  - accept default option for Gradient method
  - select 'Second Order' for pressure
  - and 'Third Order MUSCL' for all other quantities



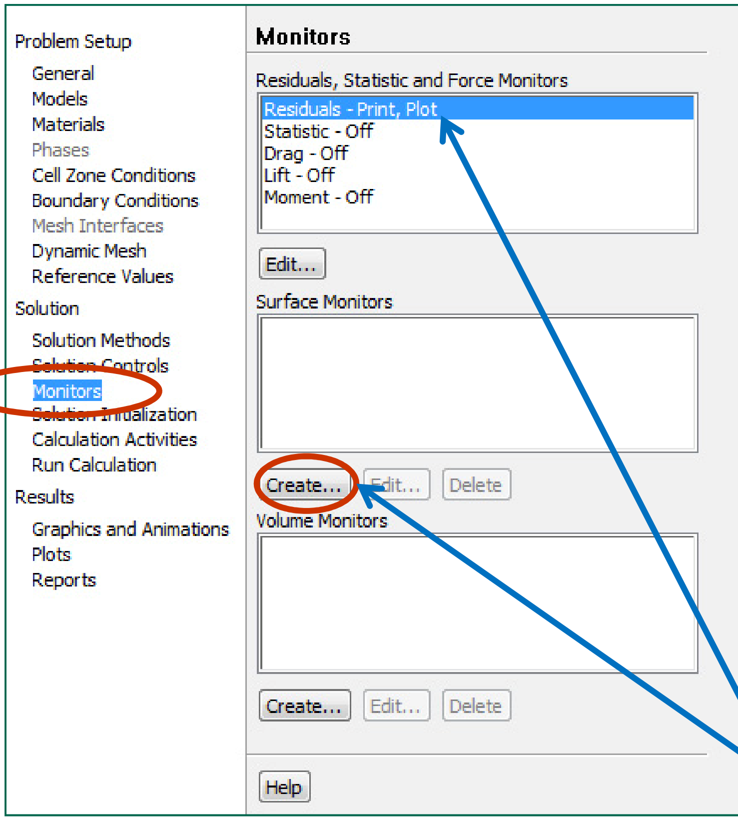
Discretization schemes define how the solver calculates gradients and interpolates variables to non-stored locations. The default schemes are First Order – generally more stable but less accurate than other schemes. Often, users run First Order discretization initially and switch to higher-order schemes for the final solution. This case is simple enough to use higher-order schemes from the start.

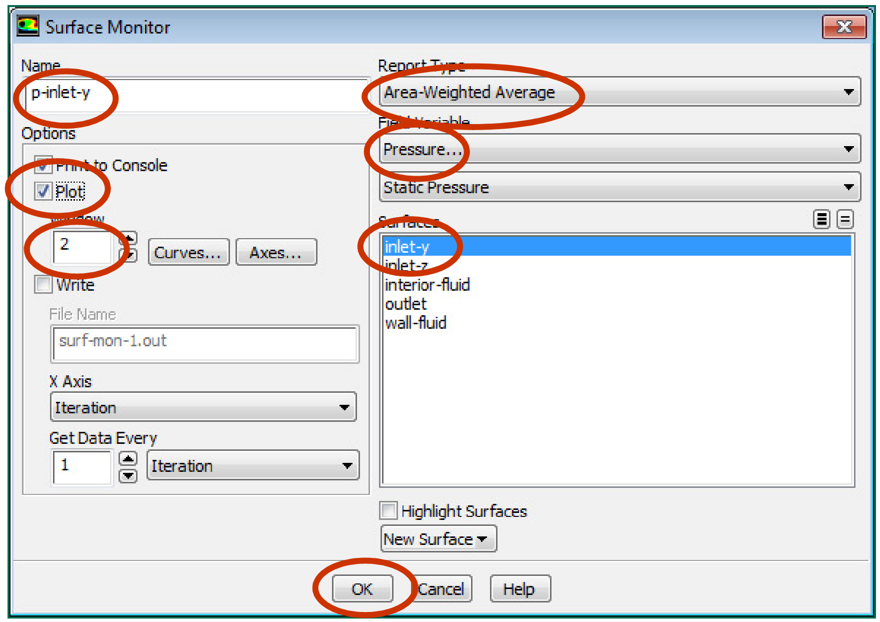
Monitors

ANSYS
Customer Training Material

- In 'Monitors', press 'Create...' for a Surface Monitor
  - Name 'p-inlet-y' Plot in window 2
    Area-Weighted Average Pressure inlet-y

Accept 'Static pressure' in the sub-category menu.





By default, FLUENT reports values of the **residuals**, which are indications of the errors in the current solution. These should decrease during the calculation. There are guidelines on the reductions that indicate a solution is 'converged'.
It is also recommended to observe other important solution quantities. In the current case, we will use pressure drops and temperature as **monitors**.

Monitors

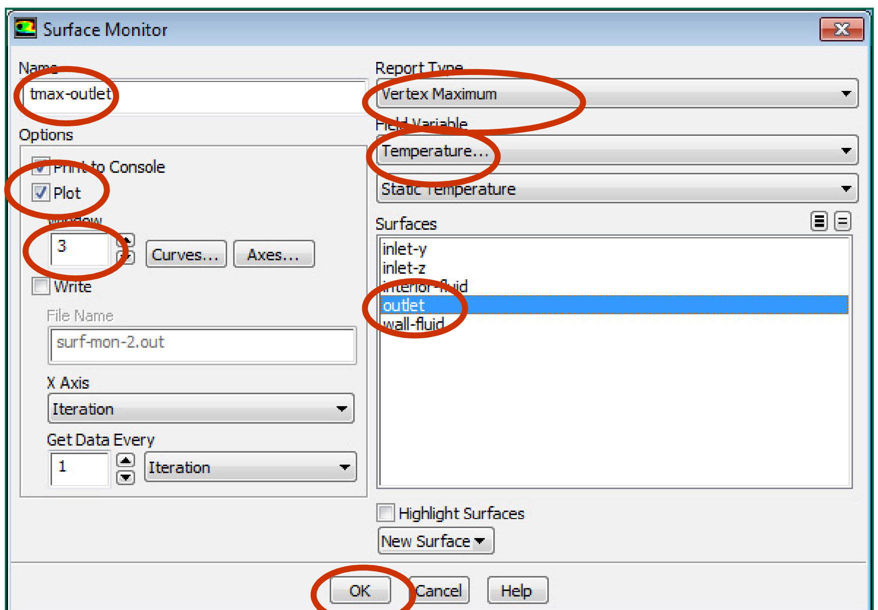
Customer Training Material

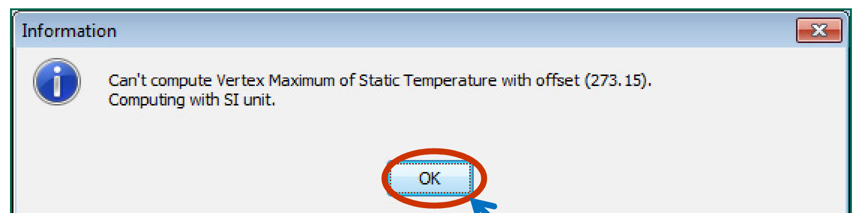
- **In Monitors, press ‘Create…’ for a Surface Monitor**
  - **Name ‘p-inlet-z’ Plot in window 2**
    Area-Weighted Average Pressure inlet-z

Not the default, 3 (which puts the new monitor in a new window).

- **In Monitors, press ‘Create…’ for a Surface Monitor**
  - **Name ‘tmax-outlet’ Plot in window 3**
    Vertex maximum Temperature outlet

Accept ‘Static pressure’ and ‘Static temperature’ in the sub-menu.

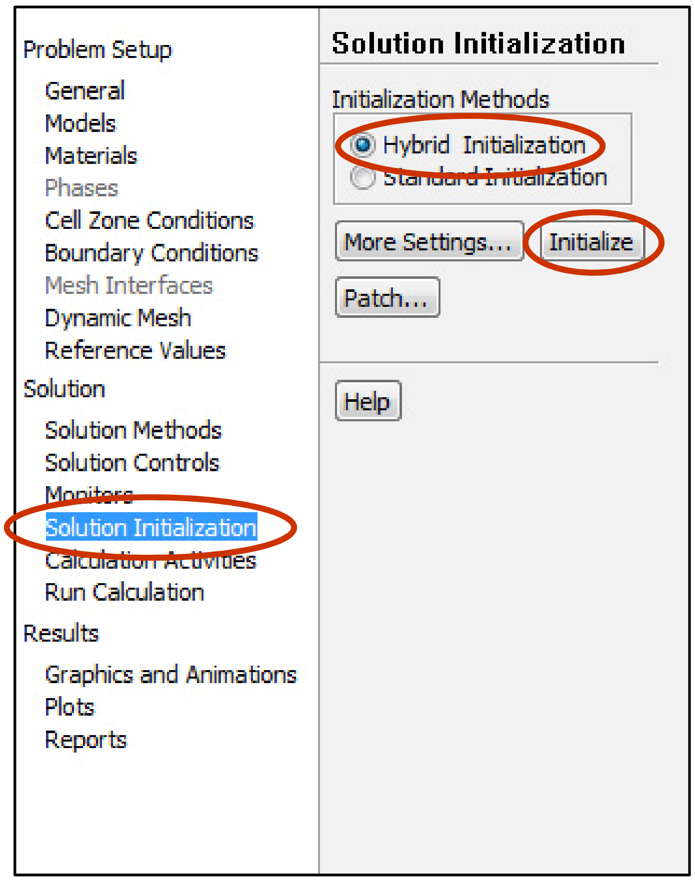




Here is an instance where FLUENT does not convert units. Click OK.

Initialization

- **In 'Solution Initialization'**
  - **select 'Hybrid Initialisation' under Initialization Methods**
  - **click 'Initialize'**

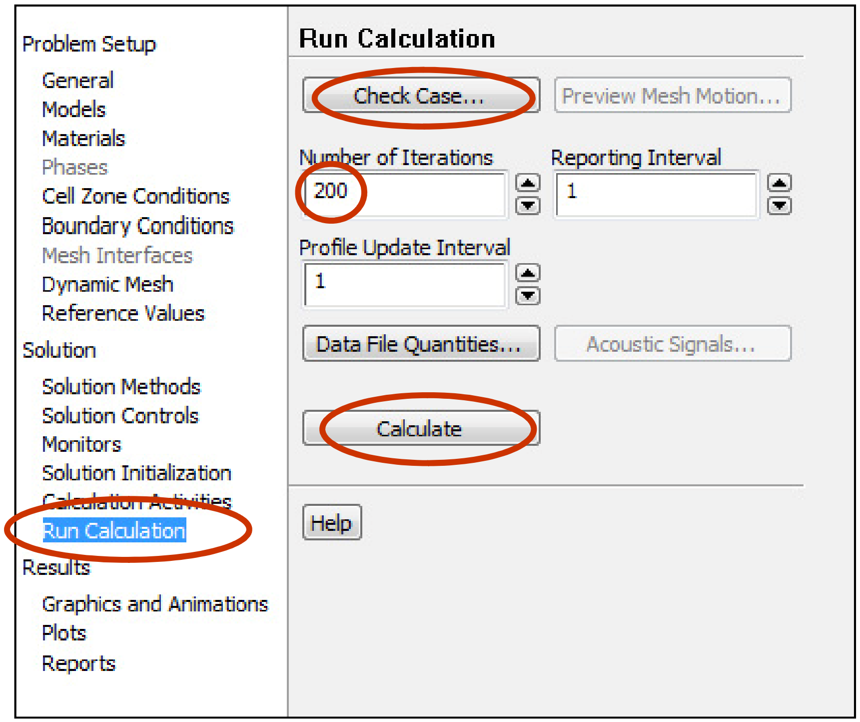


Initialization creates the initial solution that the solver will iteratively improve. Generally, the same converged solution is reached whatever the initialization, though convergence is easier if they are similar. Basic initialization imposes the same values in all cells. You can improve on this in various ways – for example, by patching different values into different zones. Several features, including patching and post-processing, are not available until after initialization.

Hybrid Initialization Method – is an efficient method of initializing the solution based purely on the setup of the simulation with no extra information required. This method produces a velocity field that conforms to complex domain geometries and a pressure field which smoothly connects high and low pressure values.

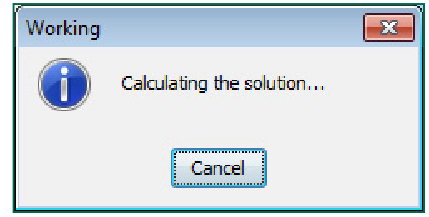
Calculate

- In ‘Run Calculation’
  - click ‘Check Case...’
    - see ‘No recommendations to make at this time’
  - set ‘Number of Iterations’ to 200
  - click ‘Calculate’

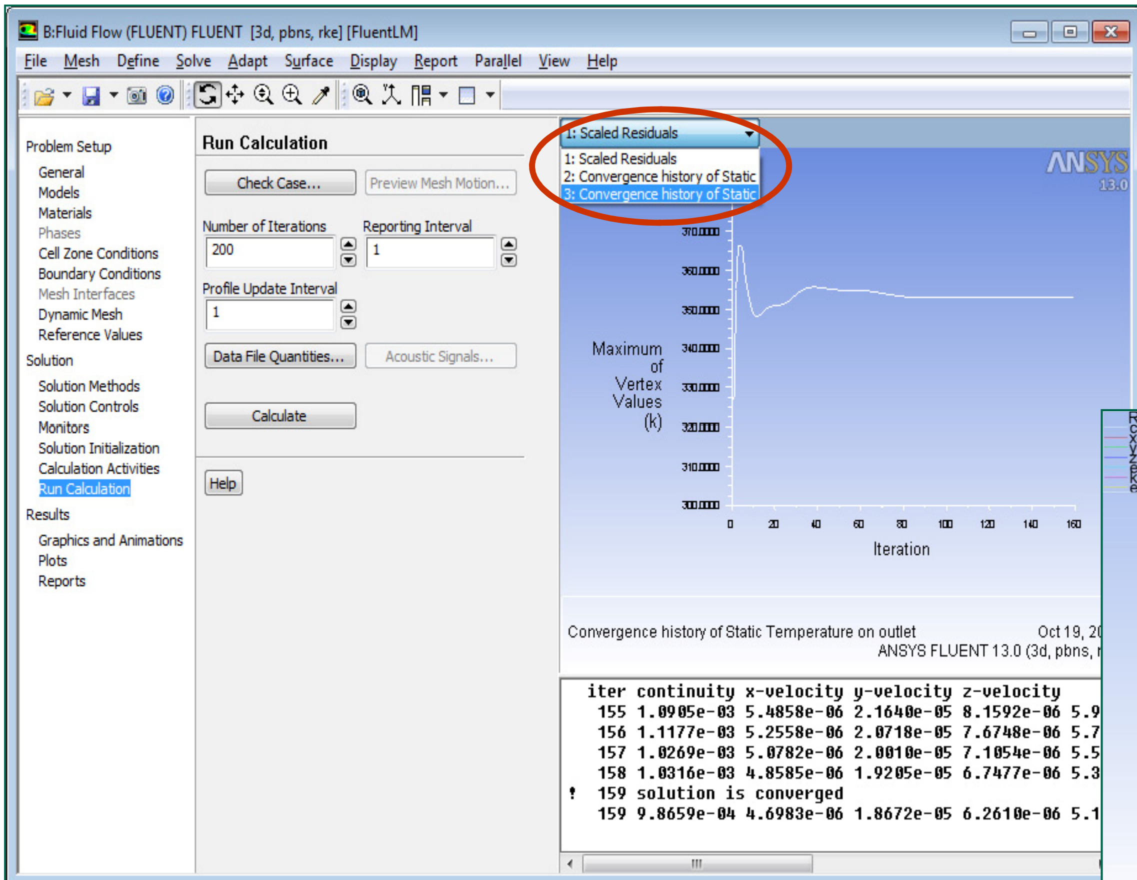


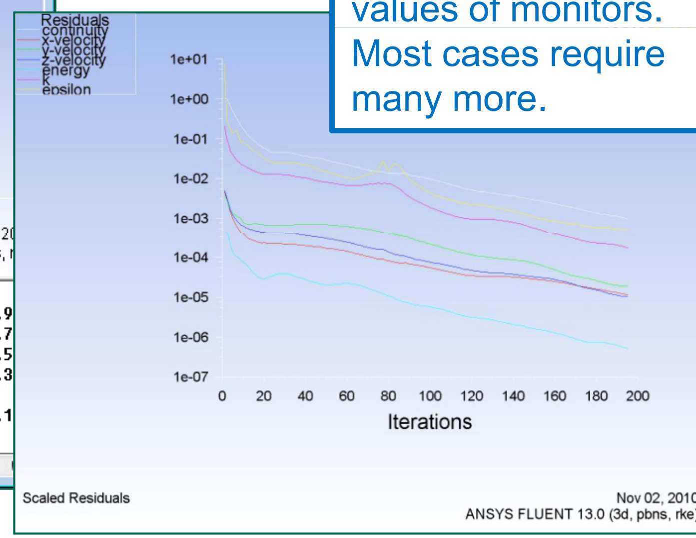
Calculating

- **While calculating, review residuals and monitors**
  - **change graphic windows using the drop-down box**



An alternative way to stop calculation is to press **CTRL-C**.





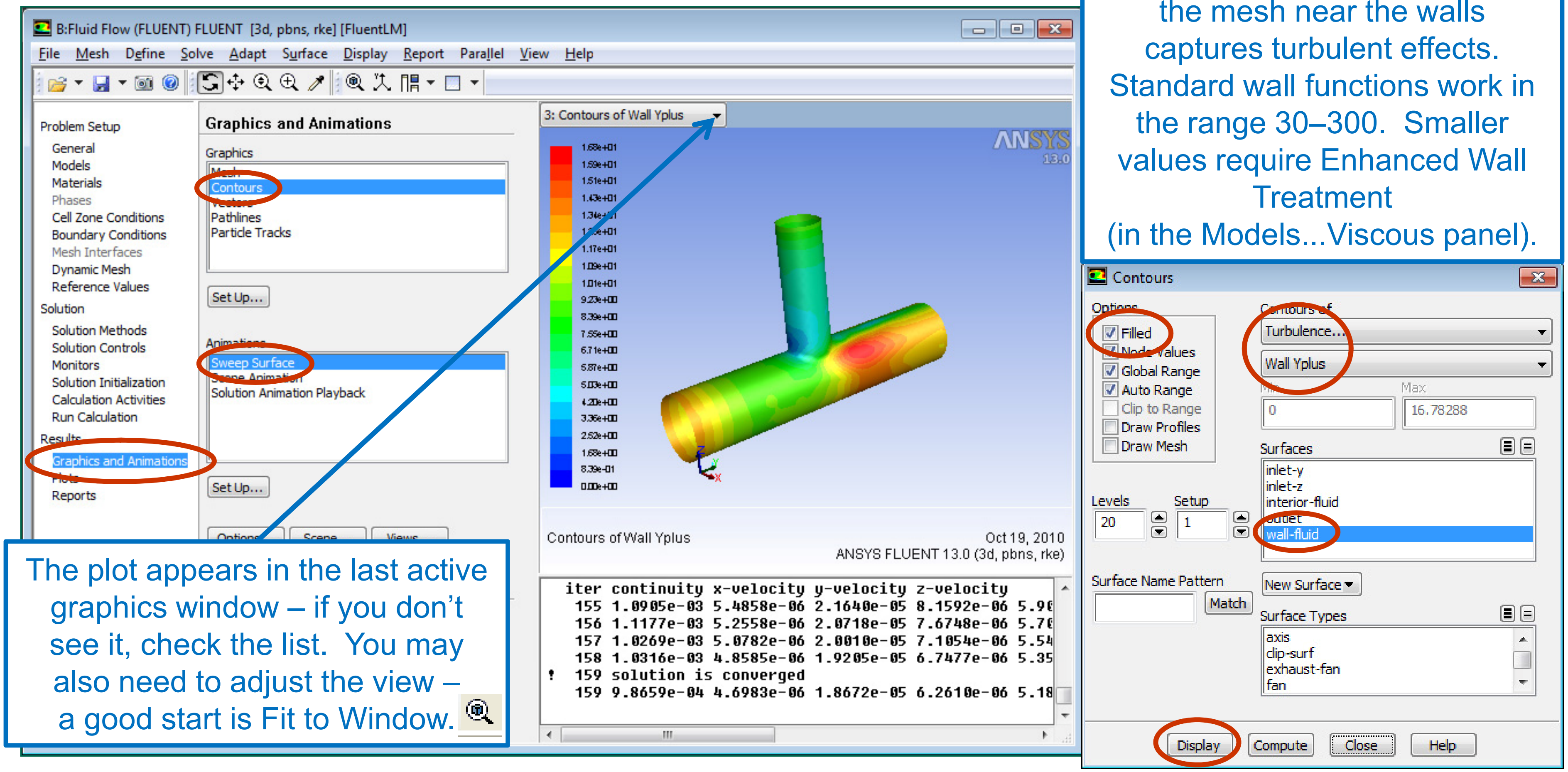
In this simple case, 200 iterations (or fewer) are enough to reach low residuals and stable values of monitors. Most cases require many more.

Preliminary post-processing

ANSYS
Customer Training Material

- In 'Graphics and Animations', select Contours, press 'Set Up...'
  - select 'Filled' contours of 'Turbulence...Wall Yplus' on 'wall-fluid'
  - press 'Display'

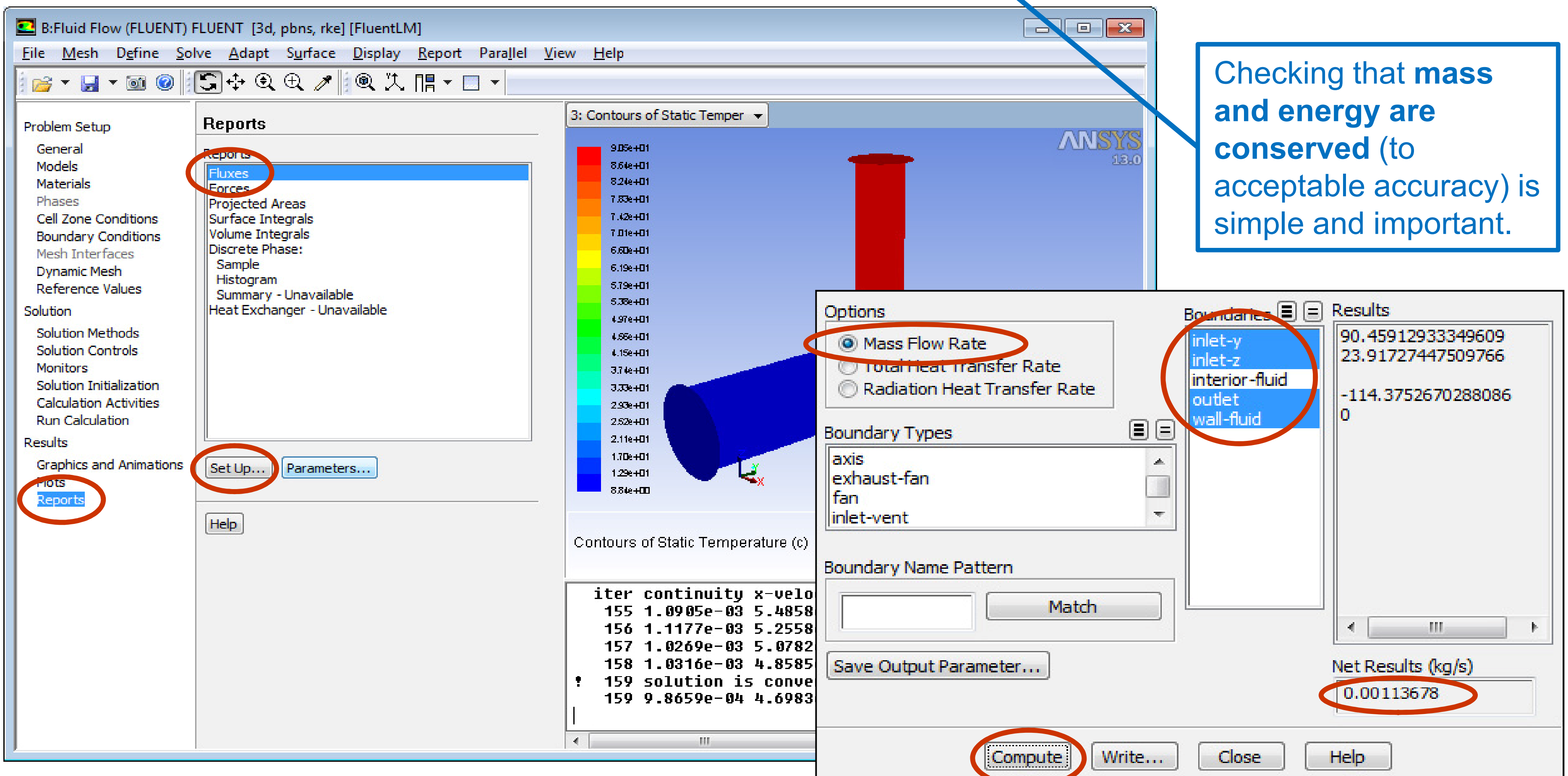
Yplus is a measure of whether the mesh near the walls captures turbulent effects. Standard wall functions work in the range 30–300. Smaller values require Enhanced Wall Treatment (in the Models...Viscous panel).



The plot appears in the last active graphics window – if you don't see it, check the list. You may also need to adjust the view – a good start is Fit to Window.

Check mass and heat balance

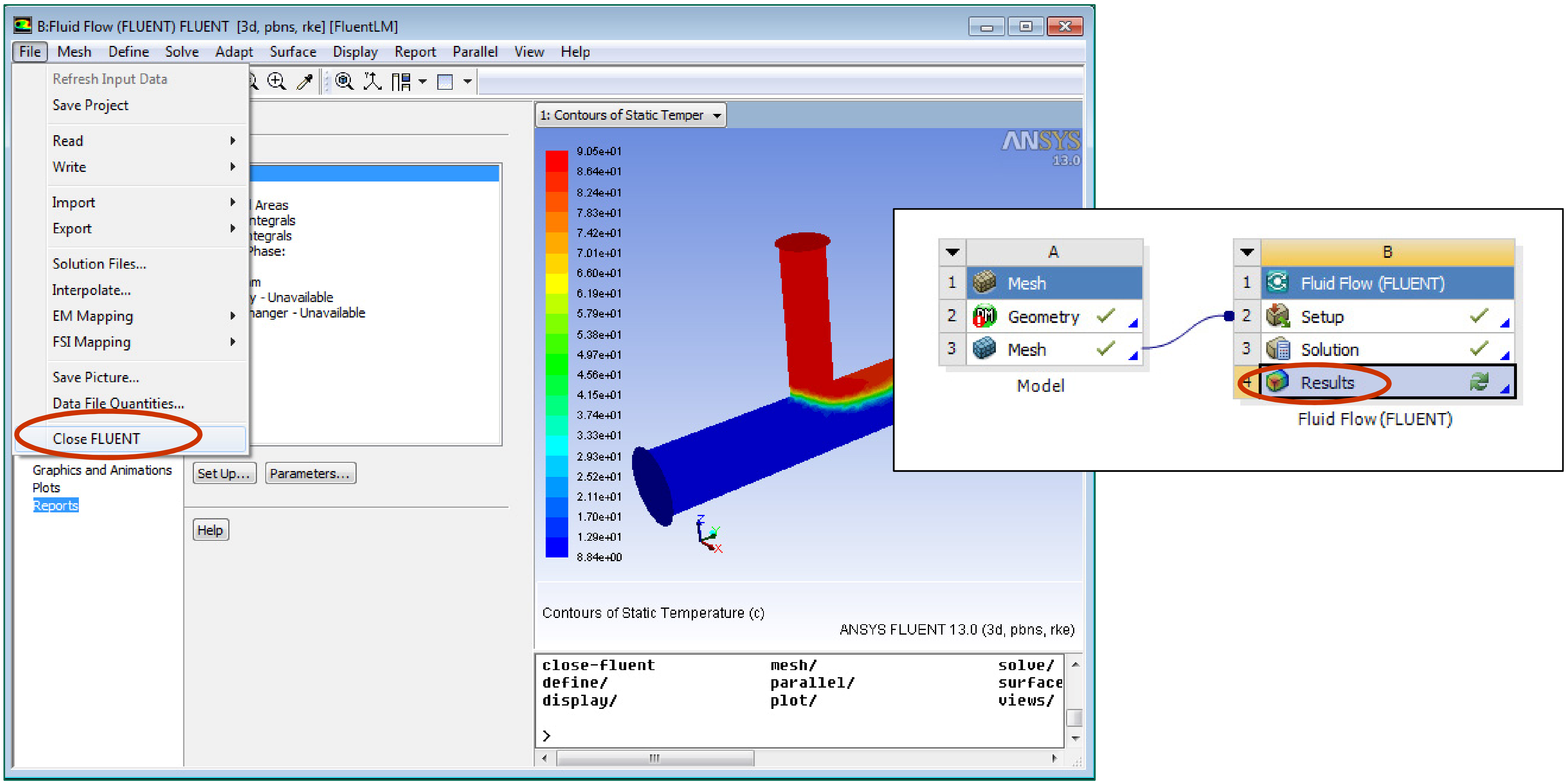
- **In 'Reports', select 'Fluxes' and press 'Set Up...'**
  - **compute 'Mass Flow Rate' and 'Total Heat Transfer Rate' for inlets and outlets – check that 'Net Results' are small**



Checking that **mass and energy are conserved** (to acceptable accuracy) is simple and important.

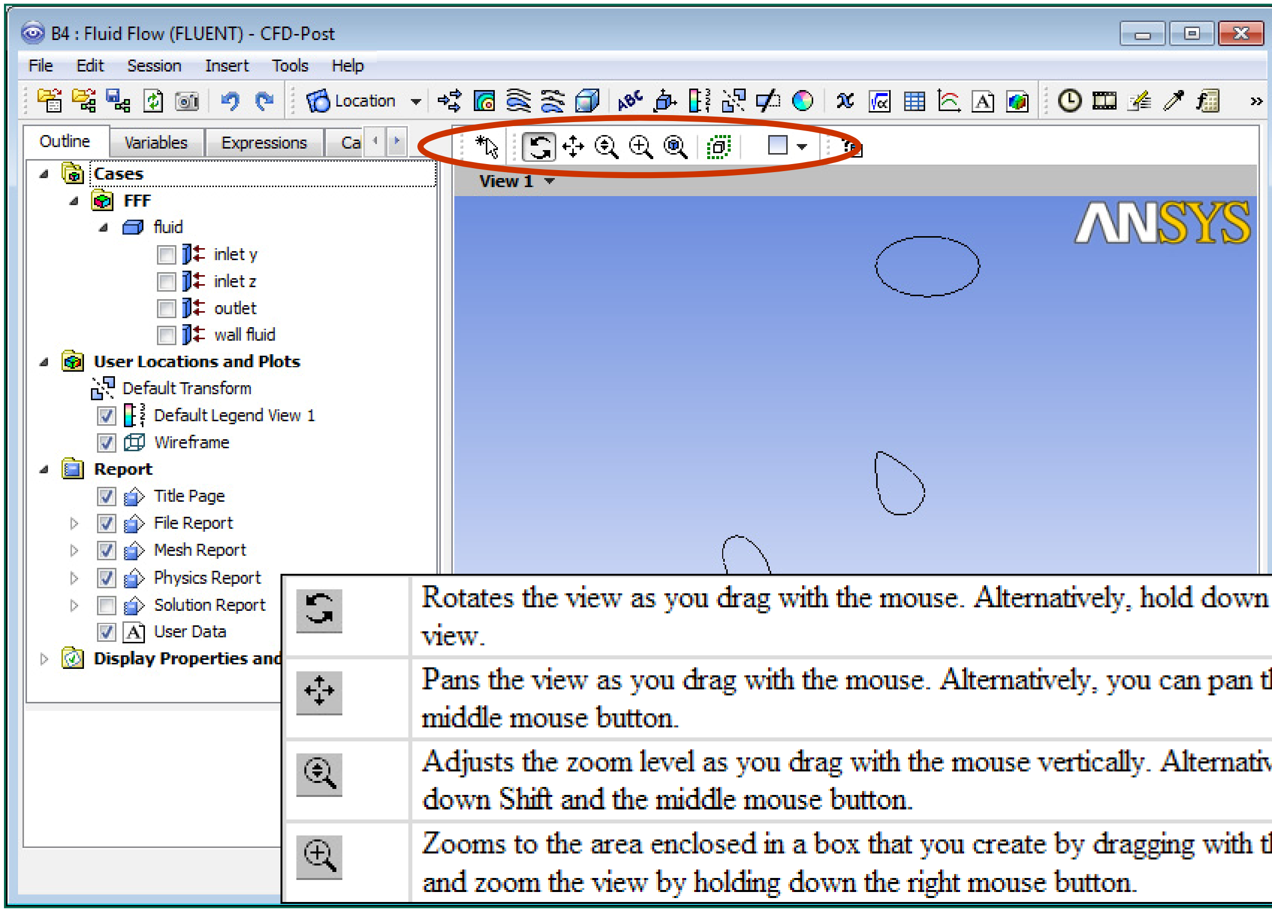
Exit FLUENT

- **Exit FLUENT – simulation case and data files are written on exit**
- **In Workbench, double-click 'Results' to launch CFD-Post**
  - **in the FLUENT session, we have completed Setup *and* Solution**



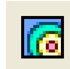
CFD-Post

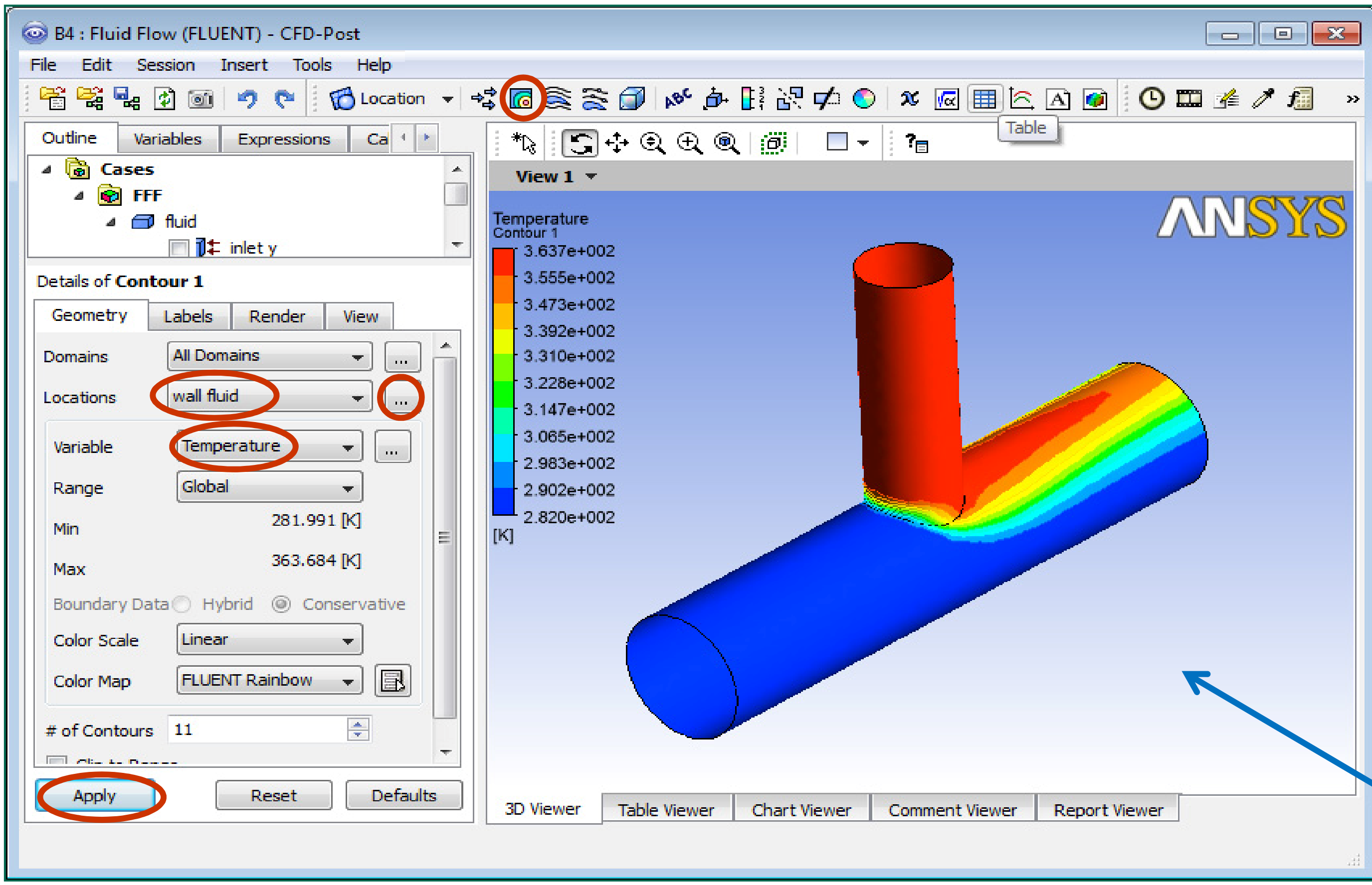
- **The results are loaded**
- **CFD-Post initially displays the outline (wireframe) of the model**
  - **viewer toolbar buttons allow you to manipulate the view**



	Rotates the view as you drag with the mouse. Alternatively, hold down the middle mouse button to rotate the view.
	Pans the view as you drag with the mouse. Alternatively, you can pan the view by holding down **Ctrl** and the middle mouse button.
	Adjusts the zoom level as you drag with the mouse vertically. Alternatively, you can zoom the view by holding down Shift and the middle mouse button.
	Zooms to the area enclosed in a box that you create by dragging with the mouse. Alternatively, you can drag and zoom the view by holding down the right mouse button.

Temperature contour plot

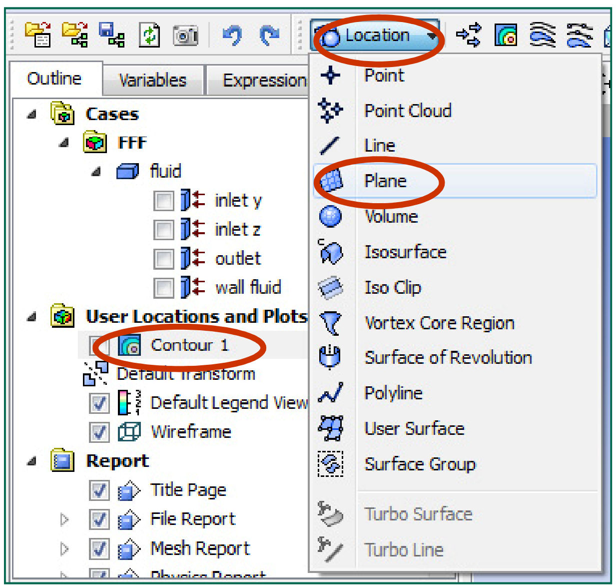
- **Press the contour button** 
  - **accept the default name 'Contour 1'**
  - **set 'Locations' to be 'wall fluid', and 'Variable' to be 'Temperature'**
  - **press 'Apply'**

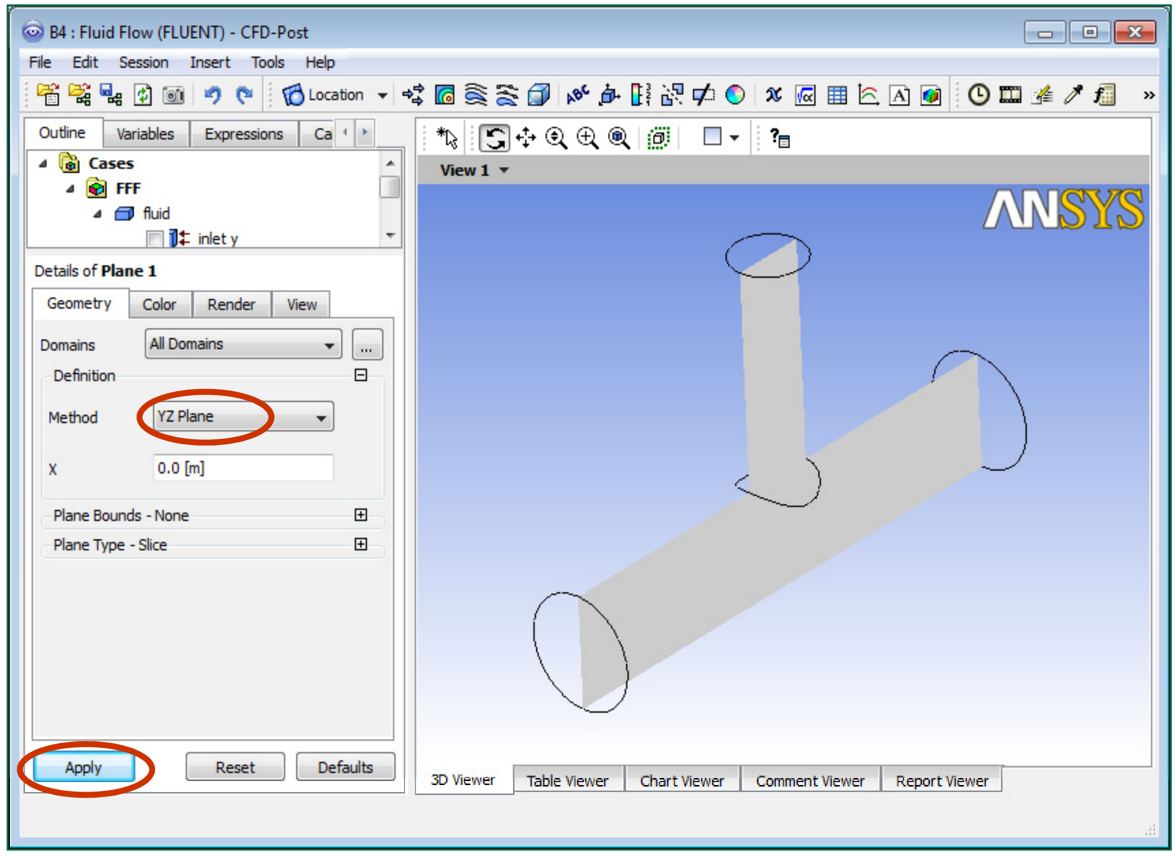


Try changing the view by rotate, zoom and pan tools.

Create a plane

- **Hide the contour plot by unchecking it in the tree view**
- **In the ‘Location’ menu, select ‘Plane’**
  - **accept the default name ‘Plane 1’**
  - **set ‘Method’ to be ‘YZ Plane’, accept ‘X’ as 0.0 and press ‘Apply’**

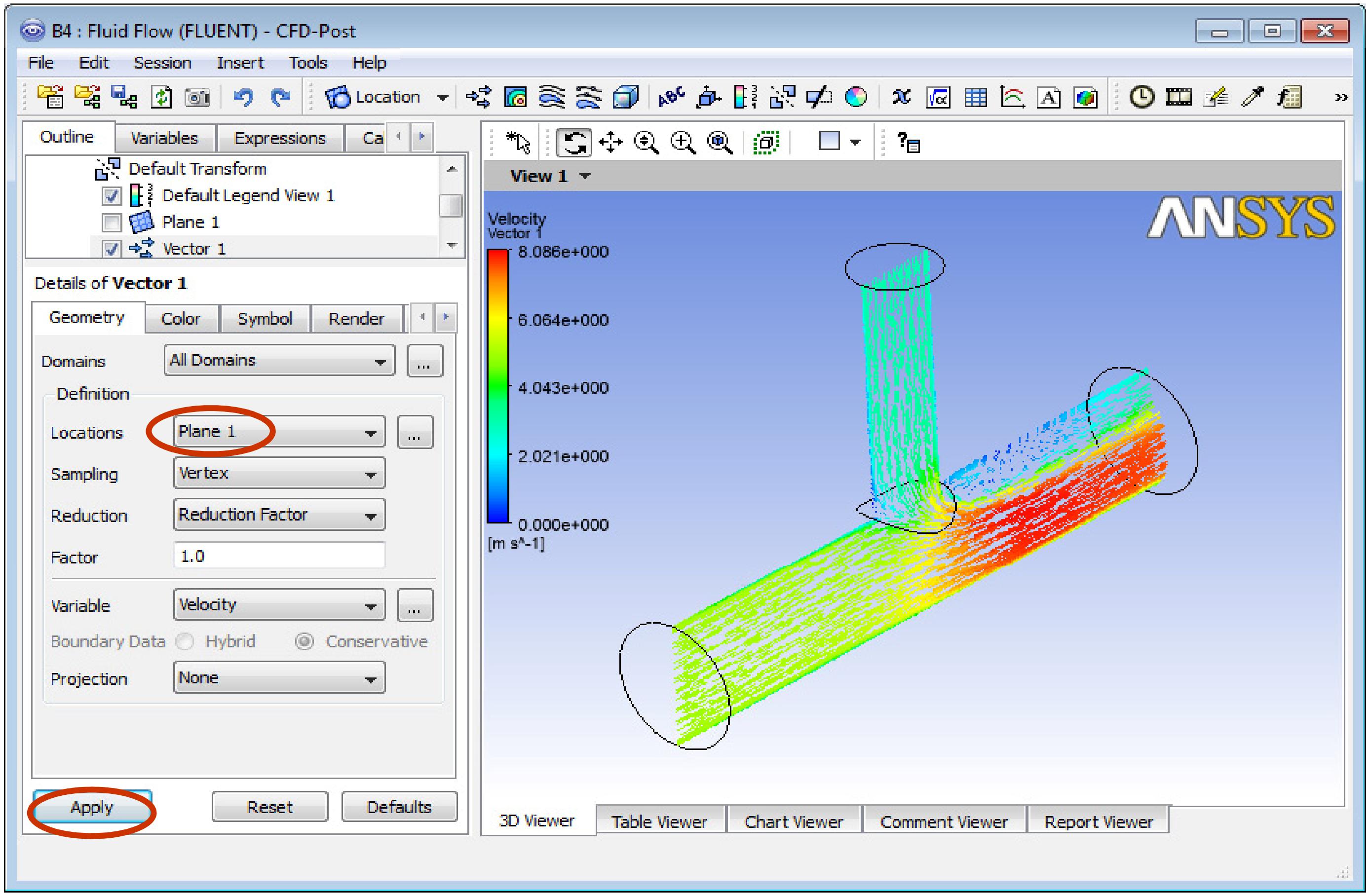




Velocity vector plot

- **Hide the plane by unchecking it in the tree view**
- **Press the Vector button (accept default name)**
  - **set 'Locations' to be 'Plane 1' and press 'Apply'**

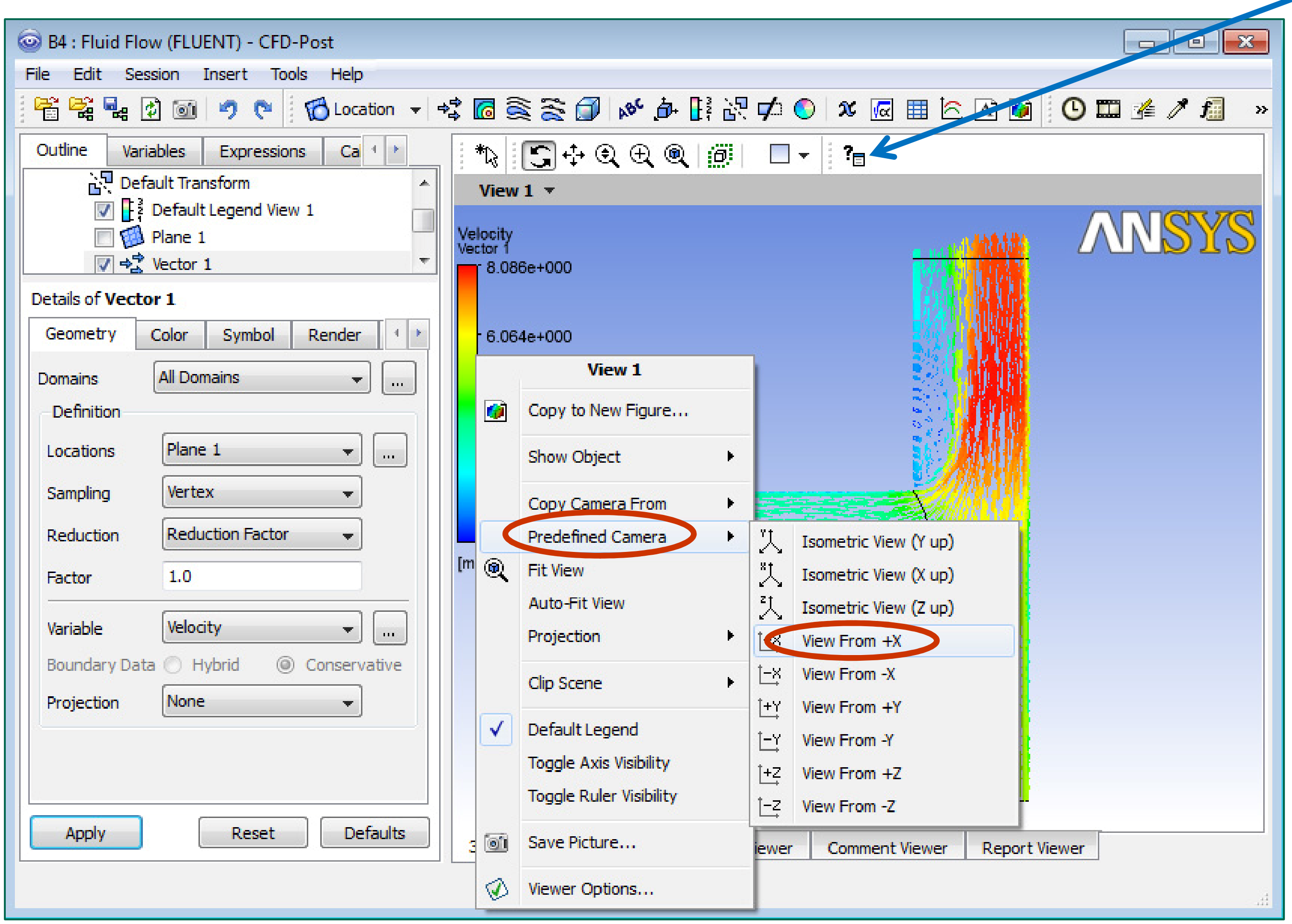
The plane is used only as a location for the vector plot.

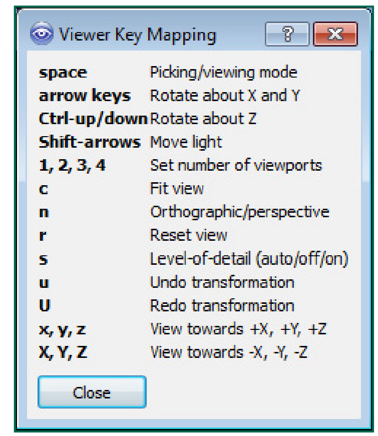


Predefined Camera views and shortcuts

- **Since the vector plot is on the YZ-plane, select a normal view**
  - **click with the right mouse-button in the view window**
  - **select ‘Predefined Camera’ then ‘View From +X’**

Alternatively, press ‘x’. Keyboard shortcuts are listed by pressing here.



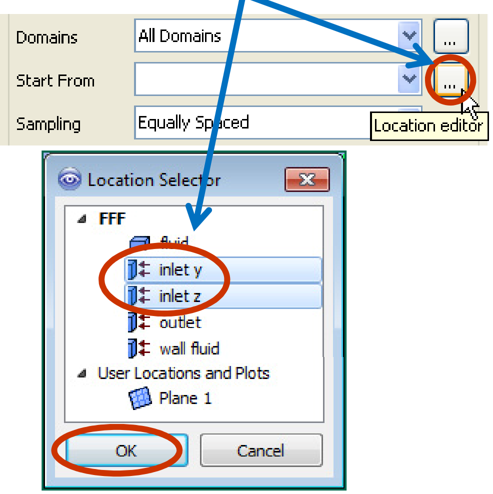


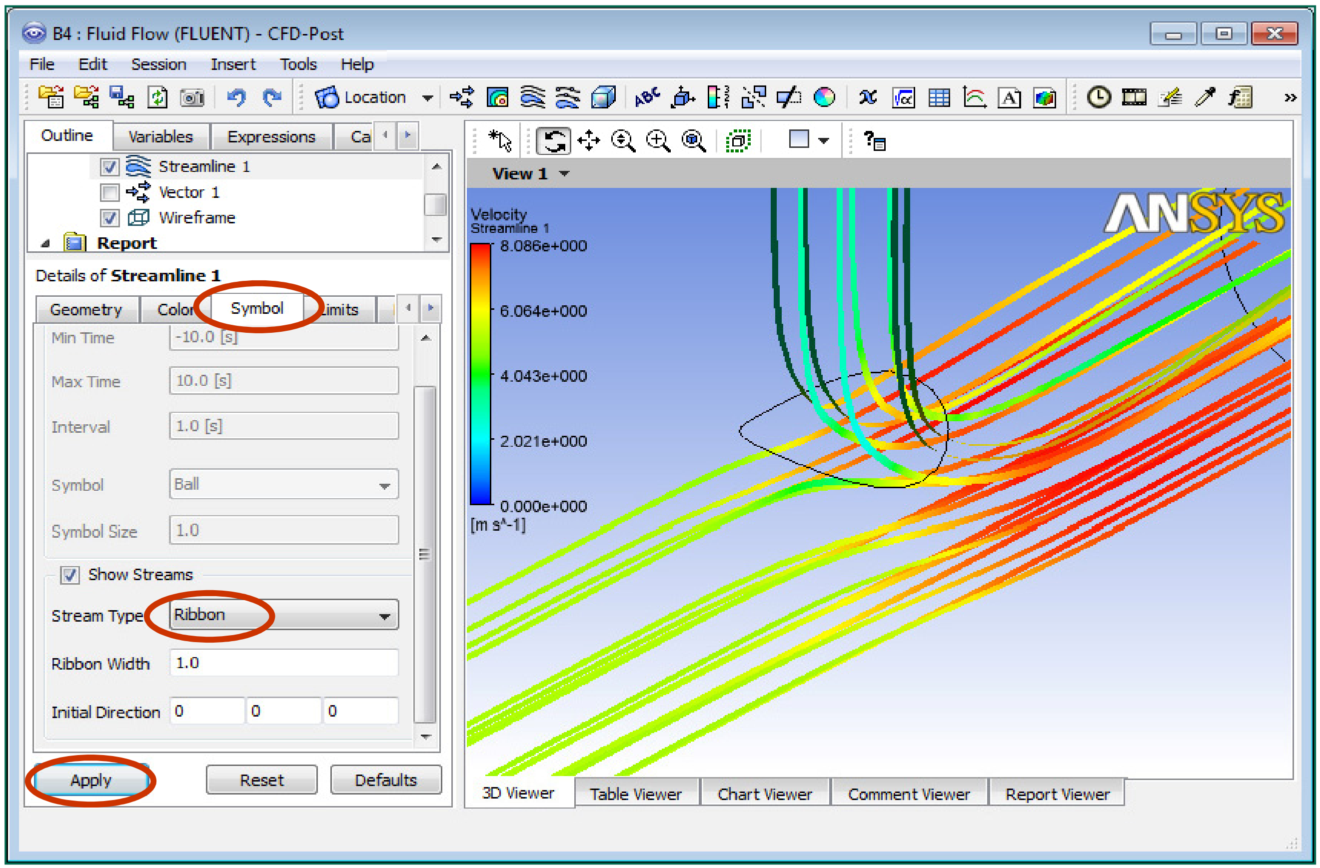
Streamline plot

- **Hide the vector plot by unchecking it in the tree view**
- **Press the Streamline button (accept default name)**
  - **set 'Start from' to be 'inlet y' *and* 'inlet z'**
  - **in the 'Symbol' tab, set 'Stream Type' to be 'Ribbon**
  - **Press 'Apply'**

Ribbons give a 3-D representation of the flow direction.
In the current plot, the colour depends on the flow velocity.

To select multiple locations, press the 'Location editor' button, and press CTRL while clicking.

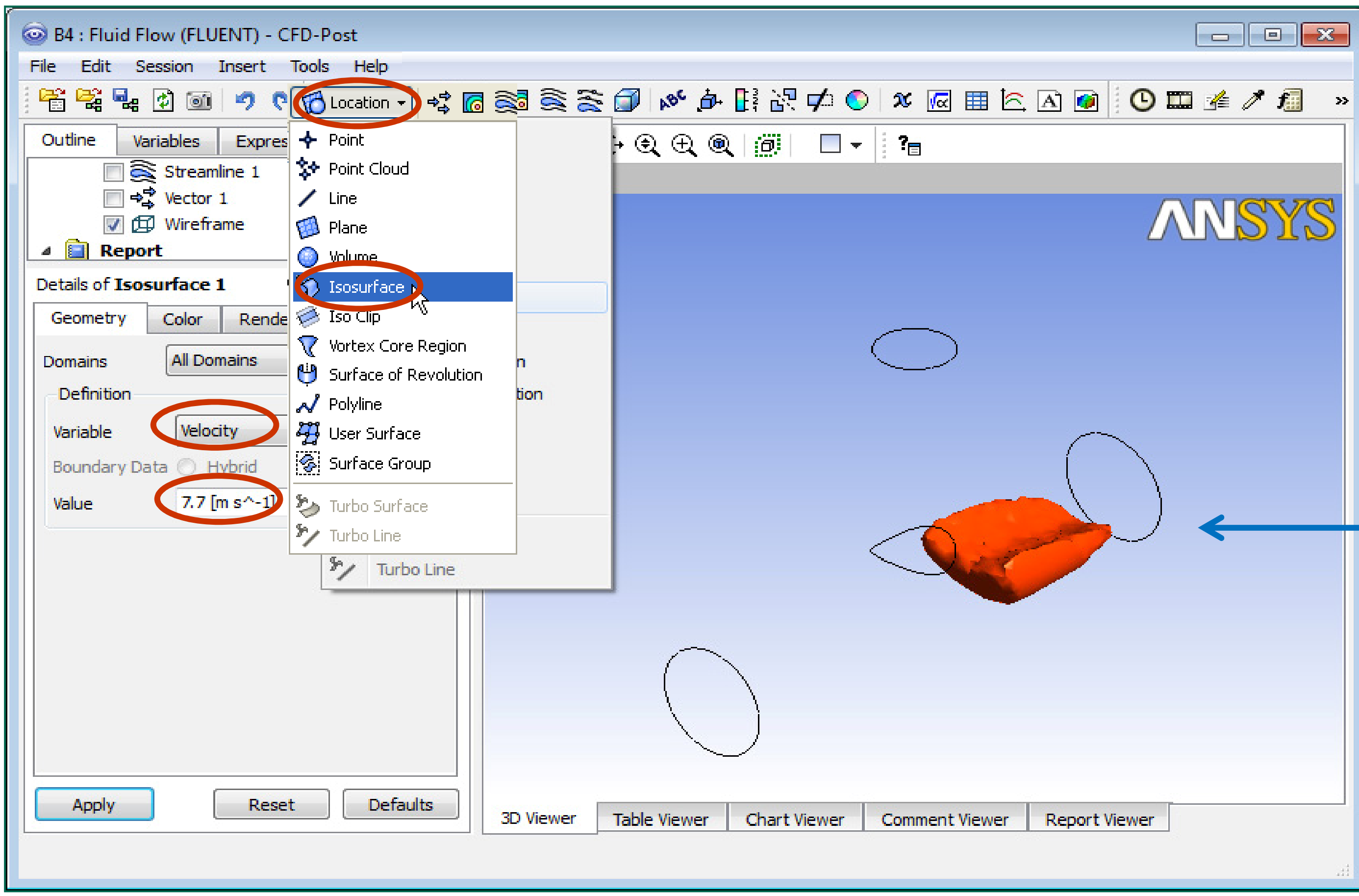




Velocity isosurface

- **Hide the streamline plot by unchecking it in the tree view**
- **In the ‘Location’ menu, select ‘Isosurface’ accepting the default name**
  - **in the ‘Geometry tab’, set ‘Variable’ to ‘Velocity’ and ‘Value’ to ‘7.7 [m s^-1]’ – Click ‘Apply’**

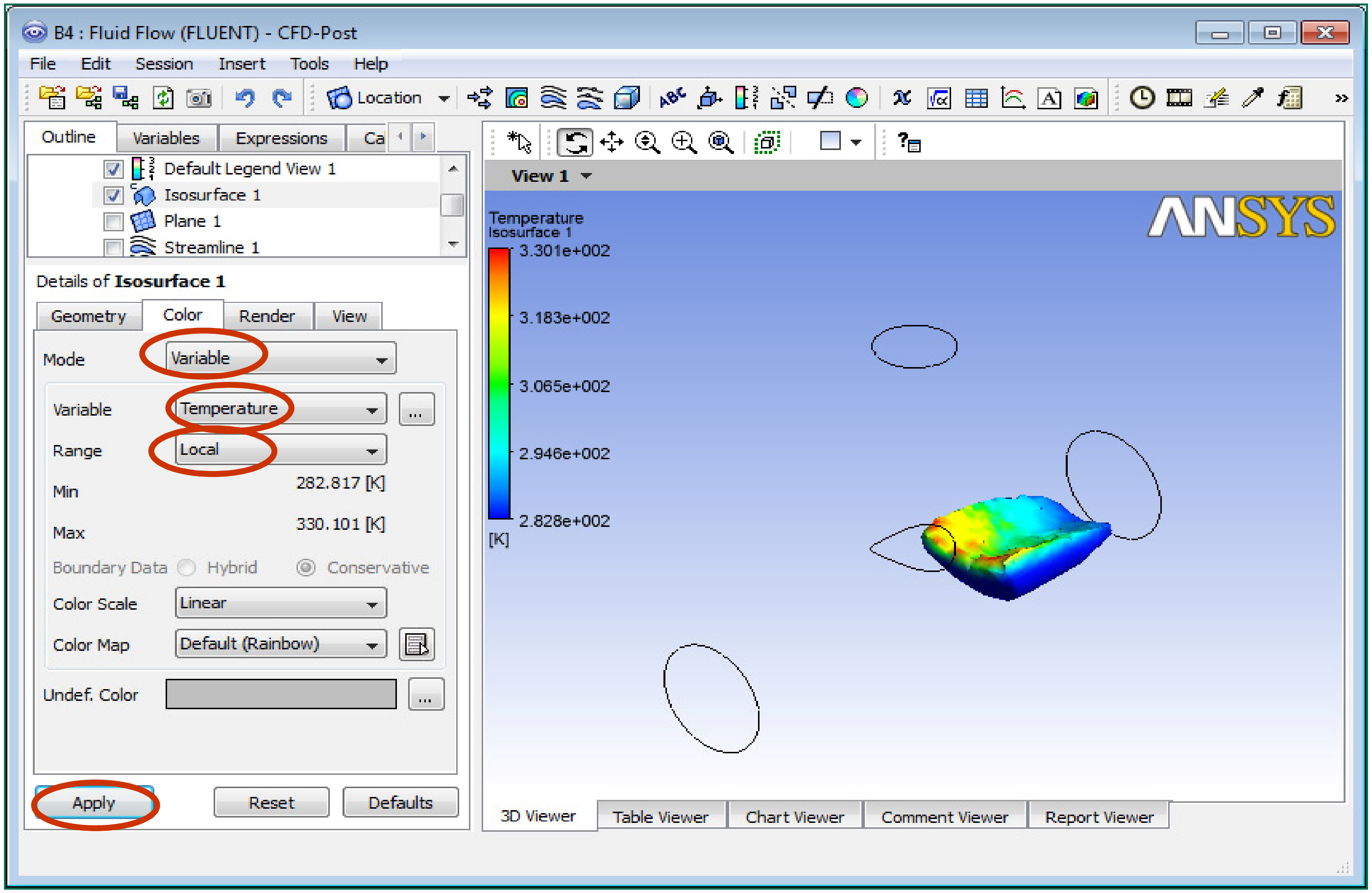
This is just one example – you can try other values.



The velocity magnitude is greater than 7.7m/s inside the isosurface , and less than that outside it.

Velocity isosurface

- **By default, the isosurface is colored by velocity magnitude**
- **In the ‘Colour’ tab**
  - **select ‘Mode’ to be ‘Variable’, ‘Variable’ to be ‘Temperature’, ‘Range’ to be ‘Local’, and press ‘Apply’**



This is the end of the tutorial. To be able to revisit this problem, quit CFD-Post and **save the project** in Workbench.

Further work

- There are many ways the simulation in this tutorial could be extended

- Better inlet profiles
  – current boundary conditions (velocity inlets) assume uniform profiles
  – specify profiles (of velocity, turbulence, etc), or
  – extend the geometry so that inlets and outlets are further from junction

- Mesh independence
  – check that results do not depend on mesh
  – re-run simulations with finer mesh(es)
    - generated in Meshing application, or
    - from adaptive meshing in FLUENT

Actually, the current mesh is probably **not** fine enough – one indication of this is that low-order discretization gives different answers.

Note that, by default, there is no gravity in the model – this is a setting in the General task page.

- Temperature-dependent physical properties
  – density
    - differences could lead to buoyant forces (with gravity turned on)
    - quite small effects in this case
  – viscosity, etc