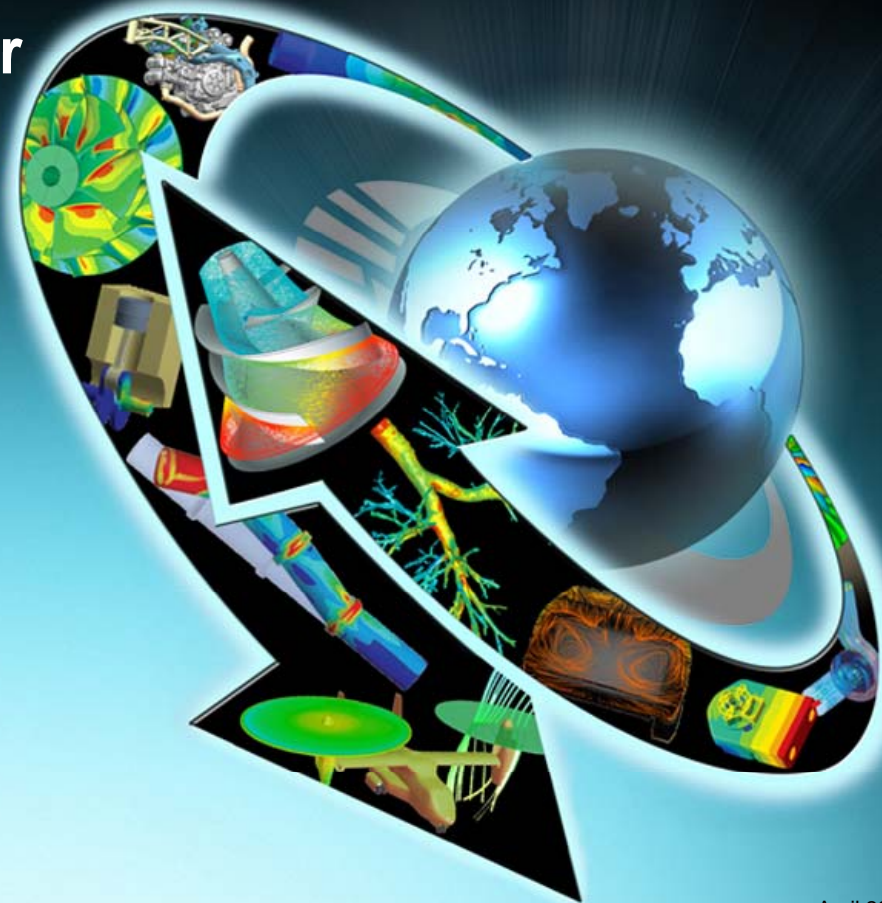




Workshop 1

Fluid Flow and Heat Transfer in a Mixing Tee

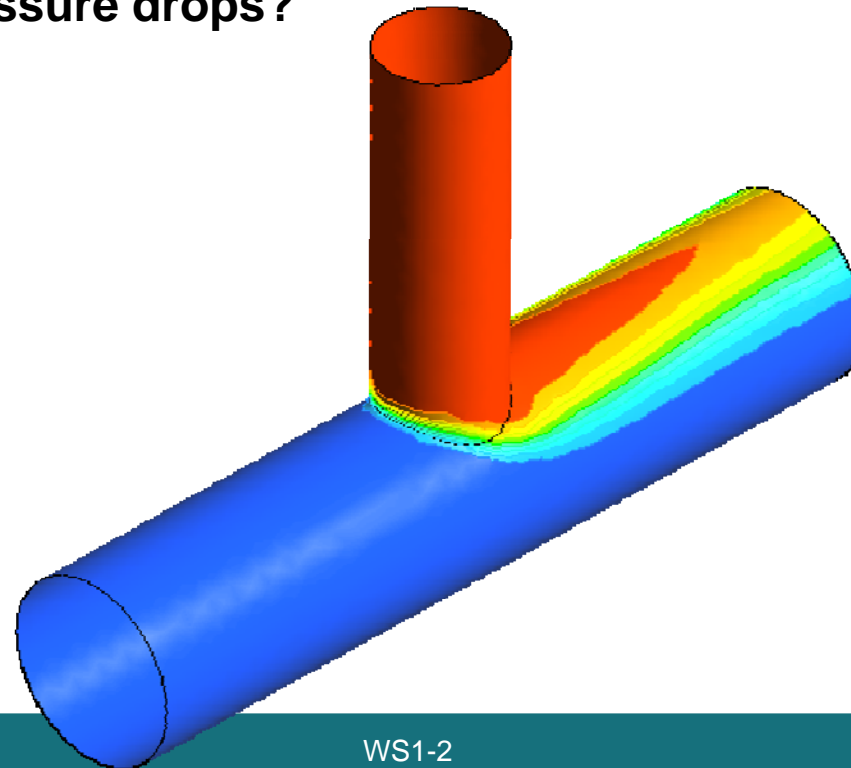
Introductory FLUENT Training



Welcome!

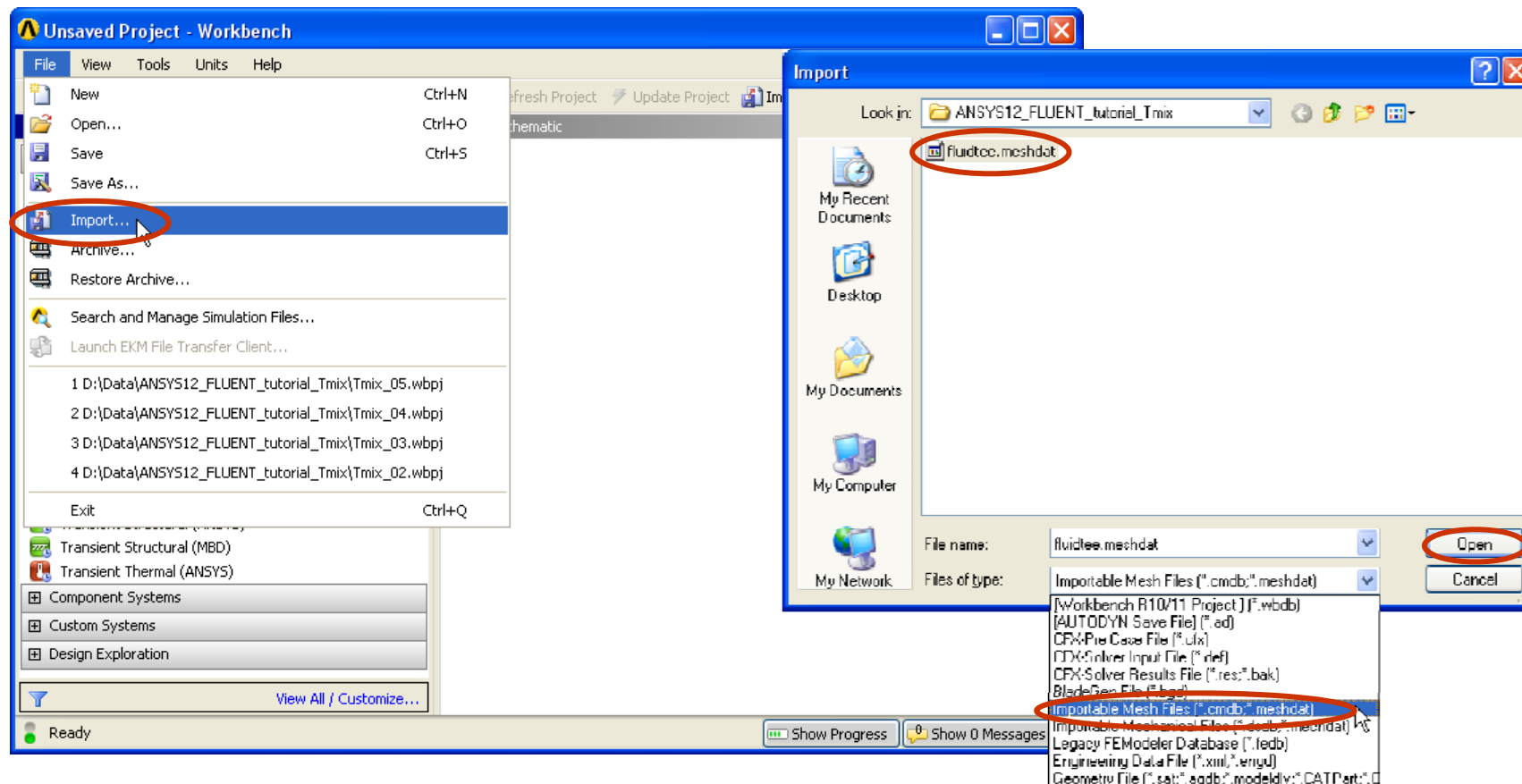
- **Introductory tutorial for FLUENT**
 - starting from existing mesh (generated in earlier tutorial)
 - model set-up, solution and post-processing
- **Mixing of cold and hot water in a T-piece**
 - how well do the fluids mix?
 - what are the pressure drops?

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

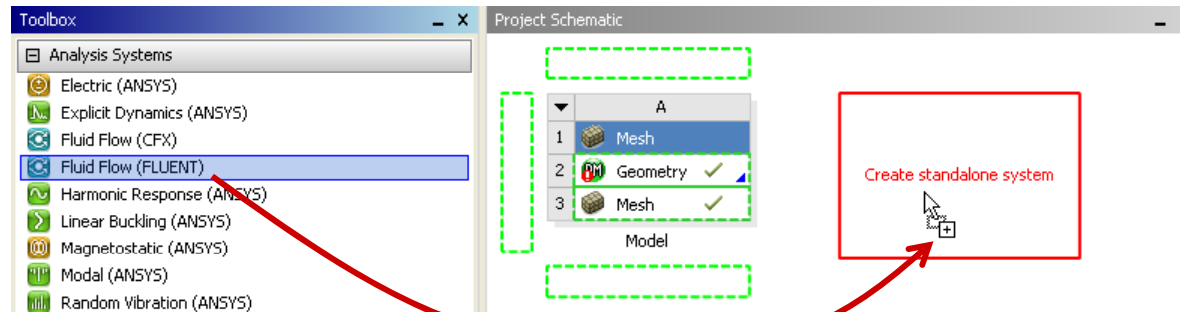


Start in Workbench

- 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

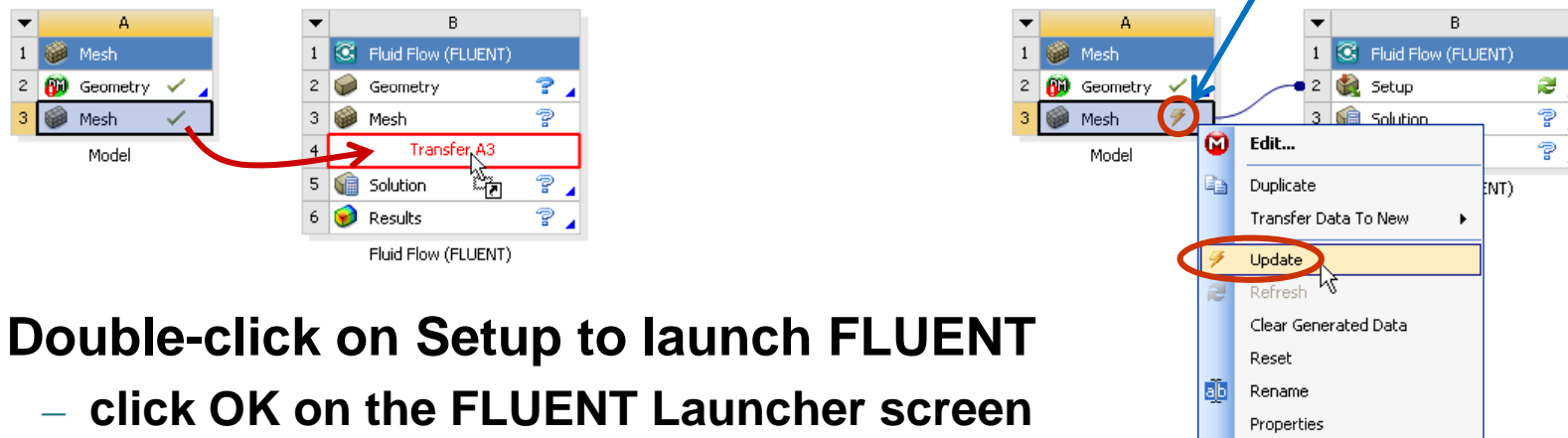


- Drag a FLUENT analysis into the project



You can see that the mesh needs to be updated, because its status icon changes.

- Drag the existing mesh into the FLUENT analysis
 - then Update the mesh (via Right-click) to convert the mesh format



- 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**

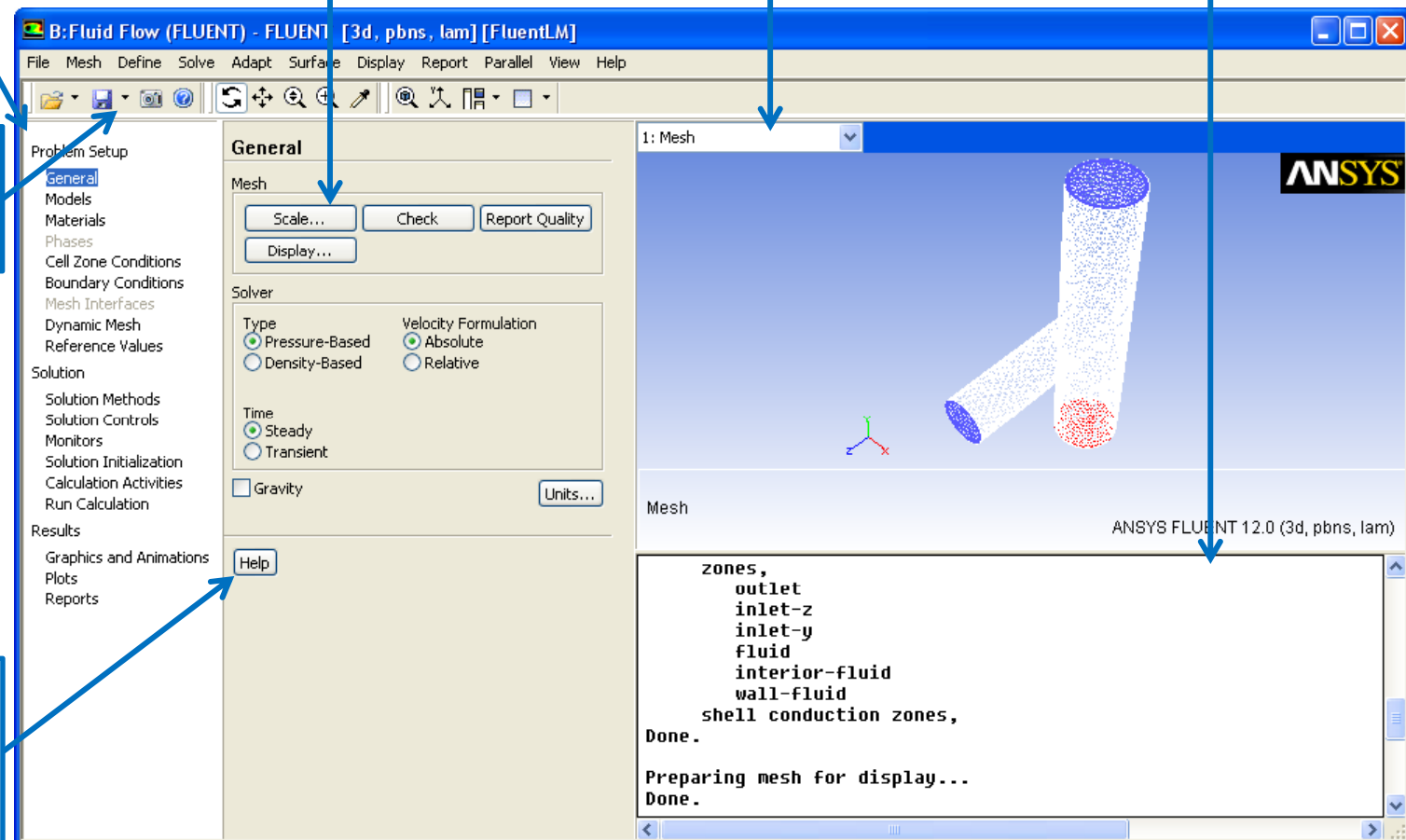
Each item in the navigation pane brings up a new **task page**. A typical workflow will tackle these in order

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

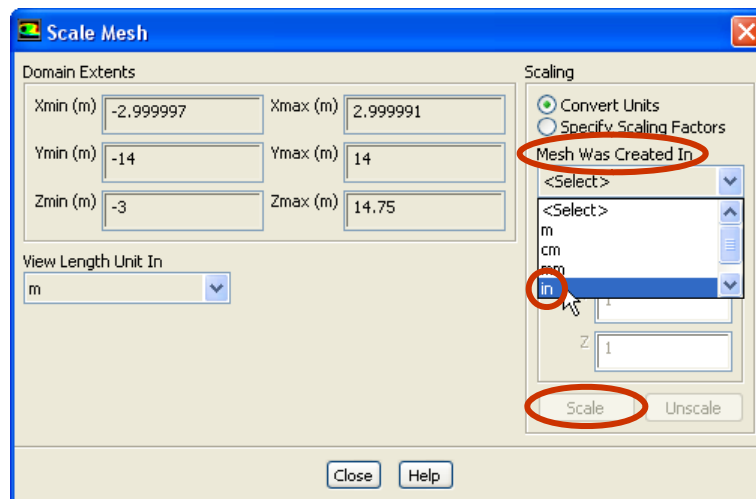
Some useful commands have **toolbar buttons**

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



Mesh scale and check

- In the 'General' task page, press 'Scale'
 - select 'Mesh Was Created In' to be 'in' (inches)
 - press the 'Scale' button (once only!) and 'Close'



The **mesh check** ensures that each cell is in a correct format, 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.

- Press 'Check' and 'Report quality'
 - review the text output

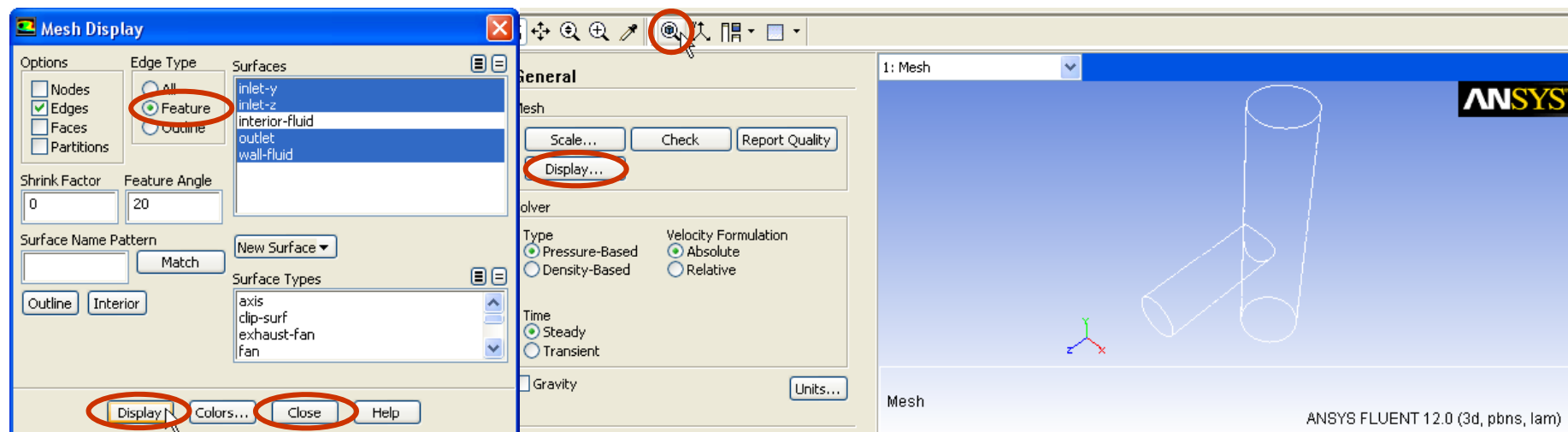
```
Checking node count.
Checking nosolve cell count.
Checking nosolve face count.
Checking face children.
Checking cell children.
Checking storage.
Done.


Applying quality criteria for tetrahedra/mixed cells.
Maximum cell squish = 6.68356e-001
Maximum cell skewness = 7.41971e-001
Maximum aspect ratio = 3.42422e+001
```

Mesh quality is very important to getting a converged, accurate solution. The User Guide suggests that maximum cell squish and skewness should be below 0.95, which the mesh here obeys. The maximum aspect ratio is 34, which is high, but acceptable in inflation layers. If the mesh quality is unacceptable, it is best to remesh the problem before proceeding. There are other possible remedies in FLUENT, such as conversion to polyhedral cells.

Display geometry

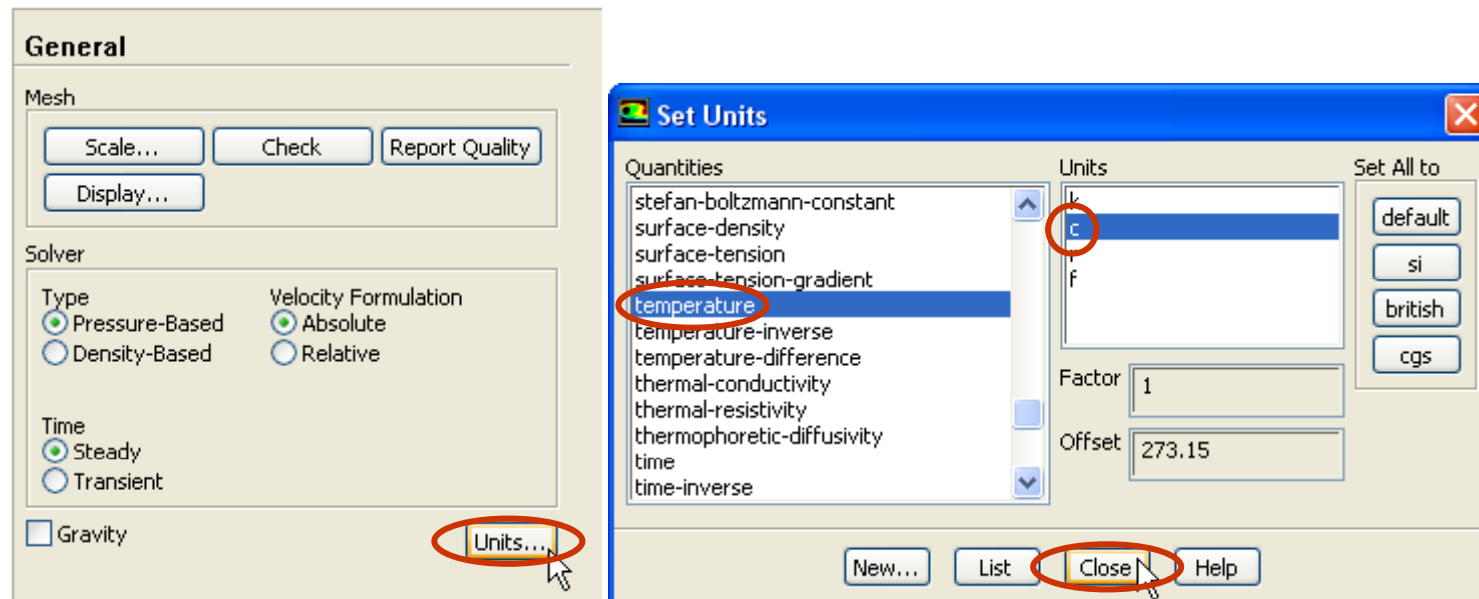
- Press 'Display'
 - select 'Edge Type' to be 'Feature', and press 'Display' and then 'Close'
 - mesh has scaled, so press 'Fit to Window'



- 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 on click

Change units of temperature

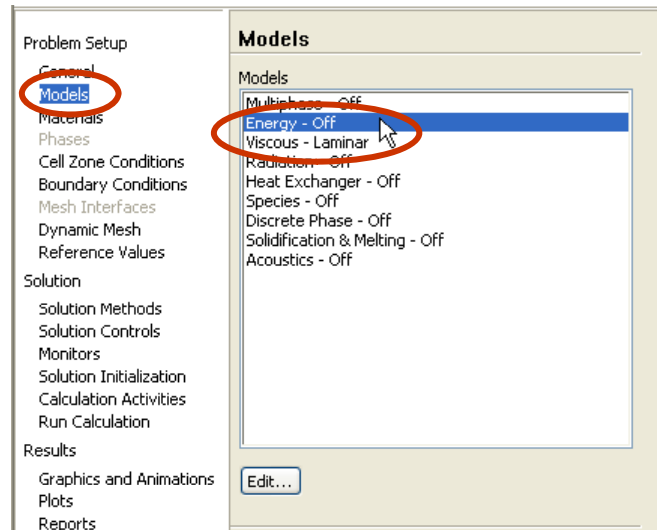
- Click 'Units'
 - select 'Temperature' to be 'c' (Celsius)
 - press 'Close'



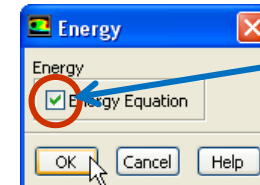
FLUENT stores values in **SI units**. Most postprocessing can be converted to other units.

Activate models

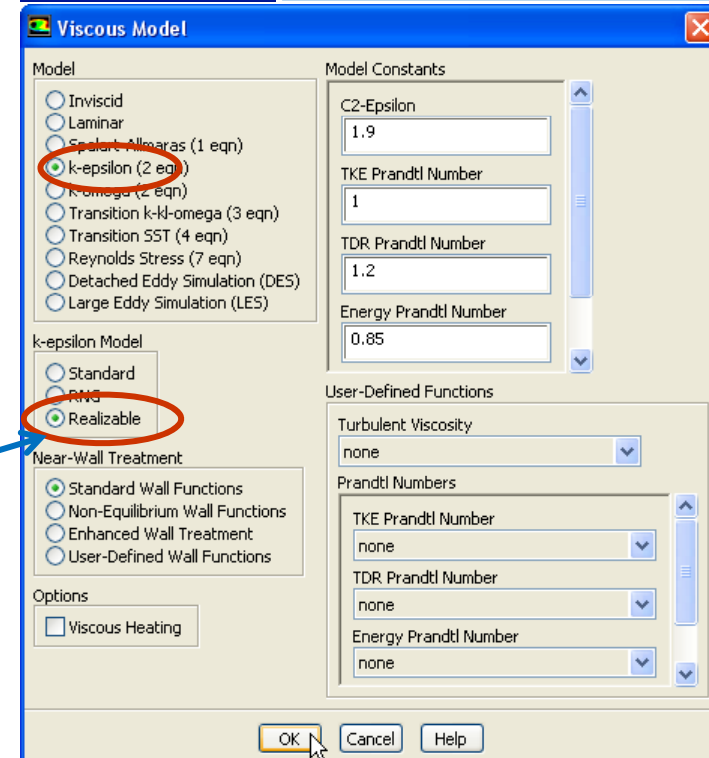
- Double-click (or click and press 'Edit...') these models:
 - Energy Equation: On
 - Viscous model: 'k-epsilon', 'Realizable'



Turbulence modeling is a complicated 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.

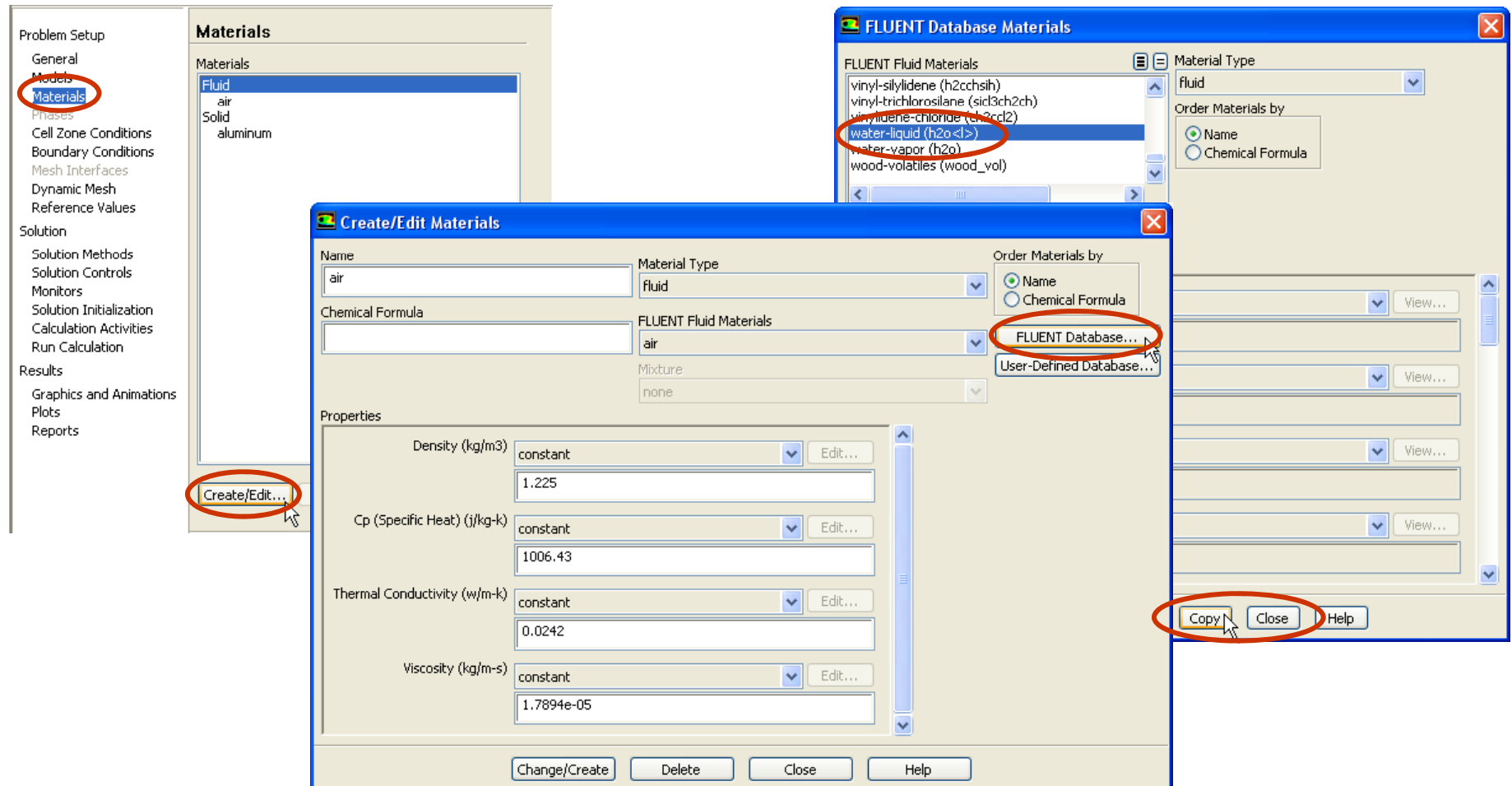


Activating the **Energy equation** simply says that temperature changes should be simulated in the model.



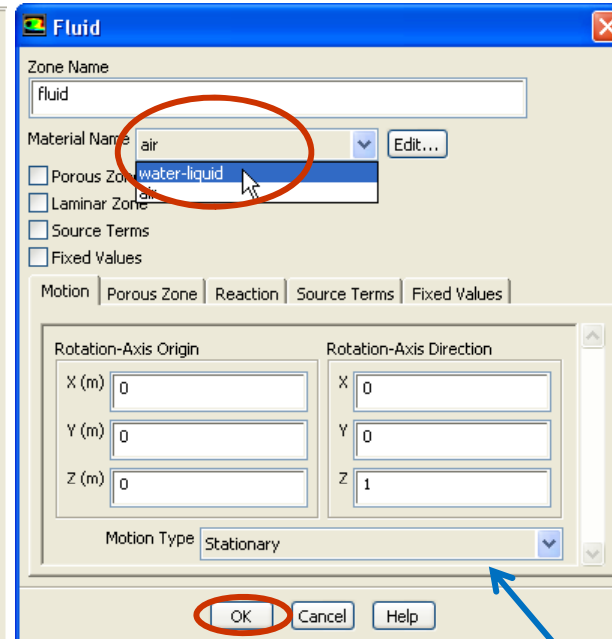
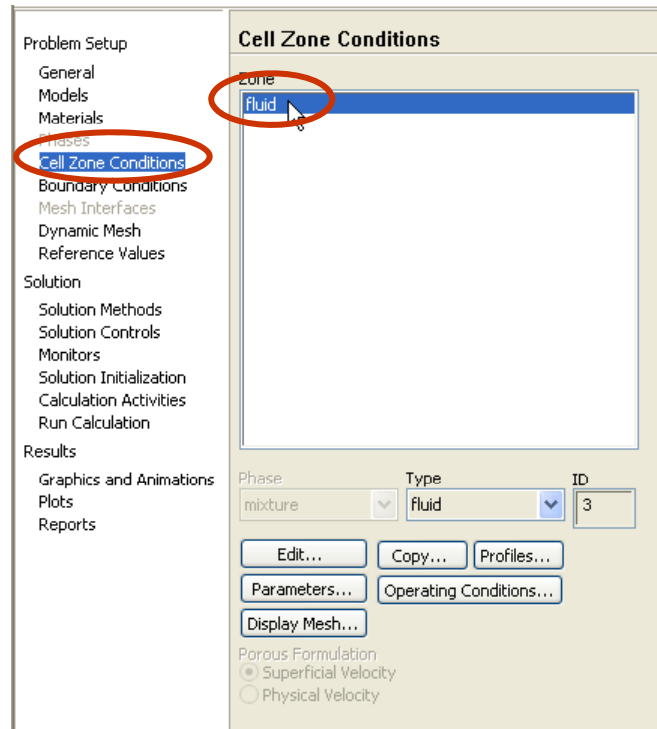
Define a new material

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



Cell Zone Conditions

- In 'Cell Zone Conditions', double-click the zone called 'fluid'
 - change the material it contains to 'water-liquid'
 - accept all other settings

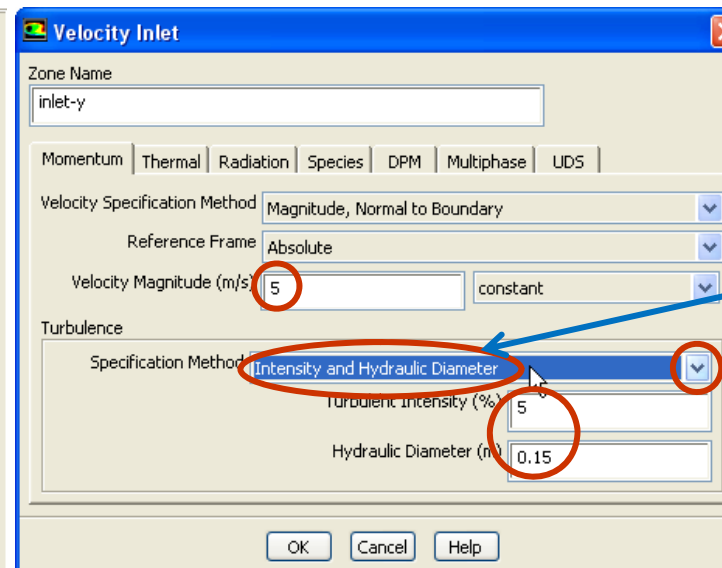
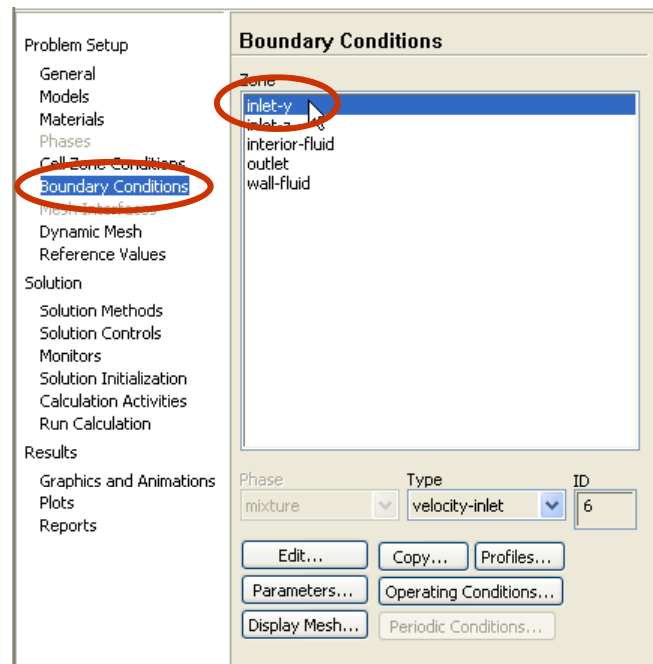


Alternatively, click once on 'fluid' to highlight it, and then click 'Edit...'.

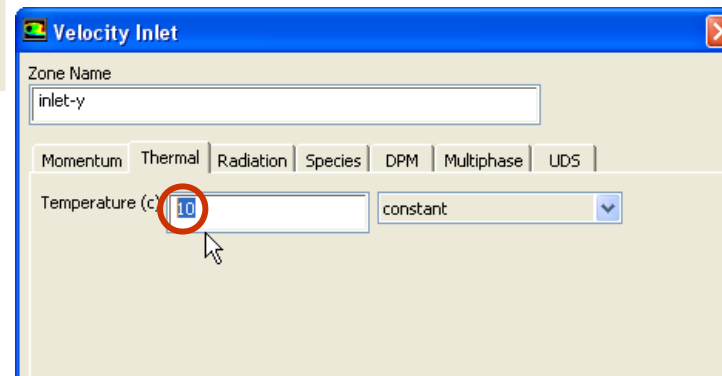
Throughout the problem setup, there are **many options and default settings** that will not be investigated in this tutorial.

Boundary Conditions

- In 'Boundary Conditions', double-click the zone called 'inlet-y'
 - Velocity Magnitude 5m/s Turbulent Intensity 5%
 - Hydraulic Diameter 0.15m Temperature 10°C

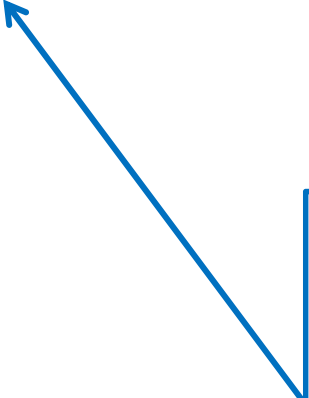


Inlet flows bring turbulence with them. The quantities depend on the upstream conditions, so they are user inputs. For flow in pipes, turbulent intensity is typically 5% to 10%, and the length-scale of the turbulence can be deduced from the pipe diameter.



Boundary Conditions

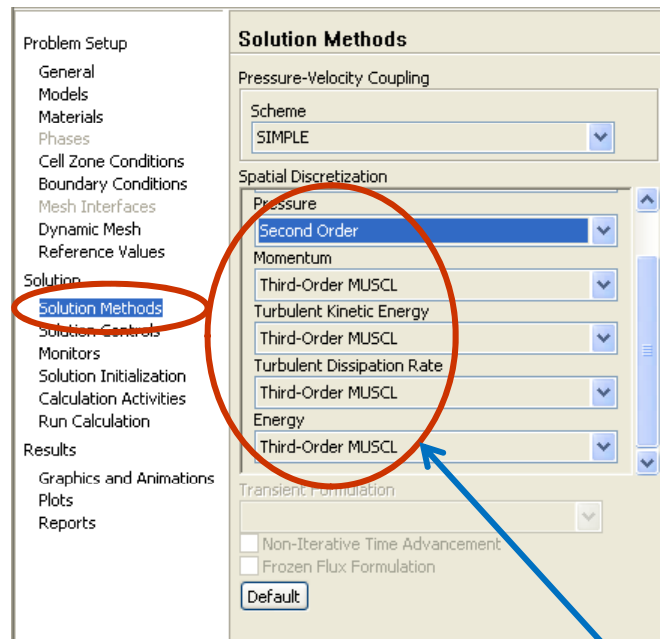
- Still in 'Boundary Conditions':
- Double-click the zone called 'inlet-z'
 - Velocity Magnitude 3m/s Turbulent Intensity 5%
Hydraulic Diameter 0.10m Temperature 90°C
- Double-click the zone called 'outlet'
 - for backflow: Turbulent Intensity 5%
Hydraulic Diameter 0.15m Temperature 30° C



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, but the model cannot predict how much (because the flow is coming from outside of the model). So, it is 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'
 - Discretization 'Second Order' for pressure
 - Discretization '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

- In 'Monitors', press 'Create...' for a Surface Monitor

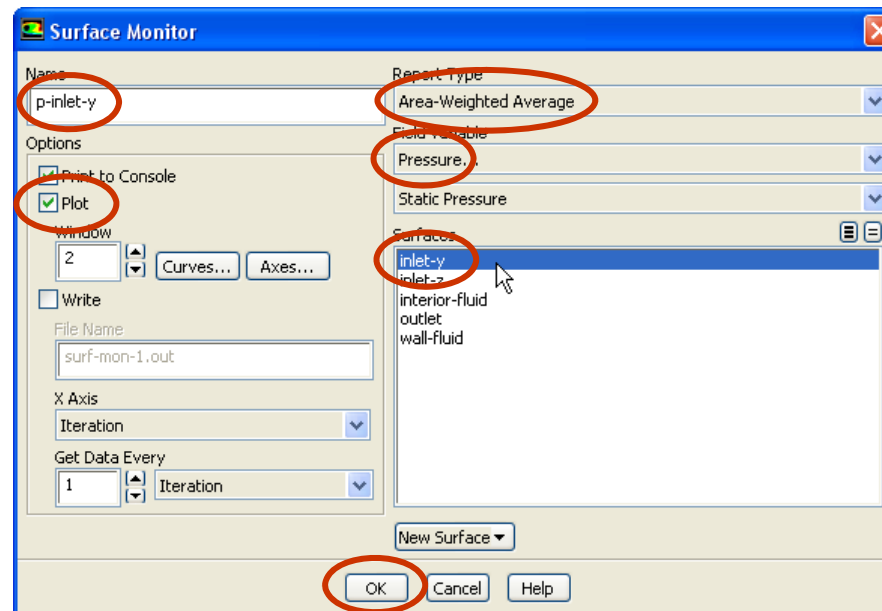
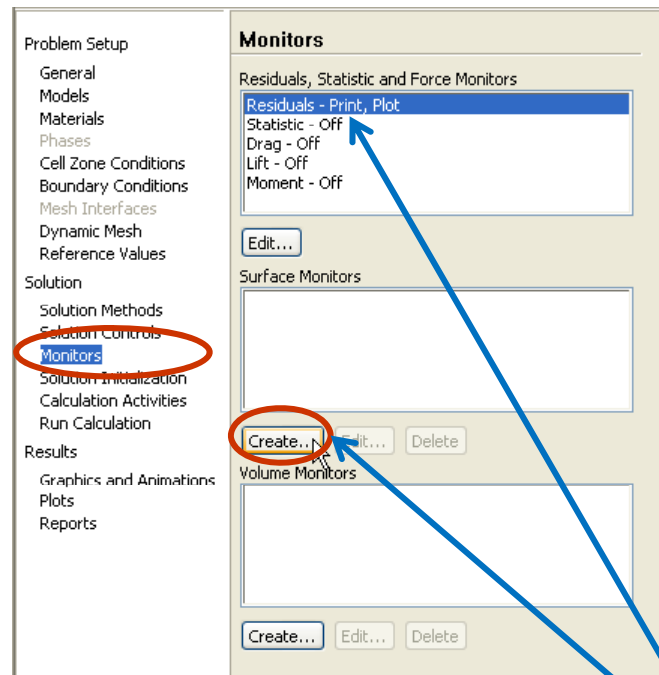
- Name 'p-inlet-y'
- Area-Weighted Average

Plot in window 2

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 calculation. There are guidelines on the reductions that indicate a solution is 'converged'. It is also recommended to observe other quantities, chosen to be important in the simulation. In the current case, we will look at pressure drops and temperature as **monitors**.

- In Monitors, press 'Create...' for a Surface Monitor

- Name 'p-inlet-z'
 - Area-Weighted Average

Plot in window 2

Pressure

inlet-z

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

- In Monitors, press 'Create...' for a Surface Monitor

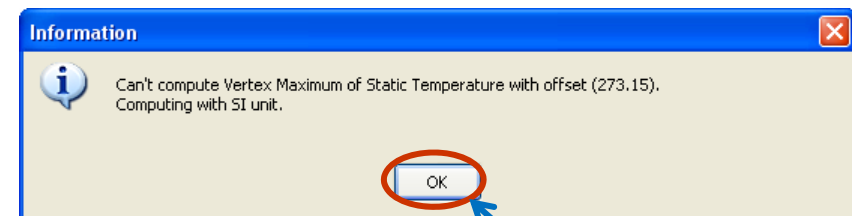
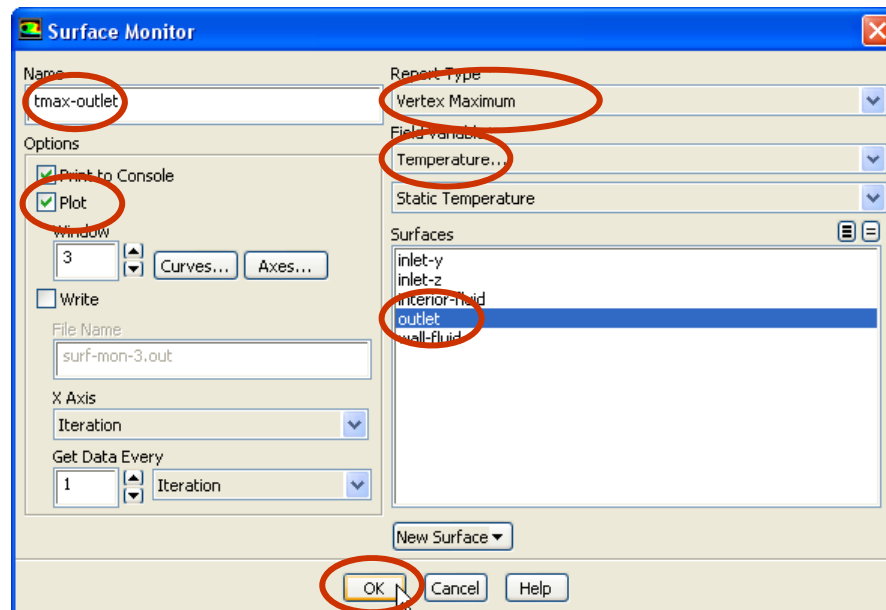
- Name 'tmax-outlet'
 - Vertex maximum

Plot in window 3

Temperature

outlet

Accept 'Static pressure' and 'Static temperature' in the sub-menu.



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

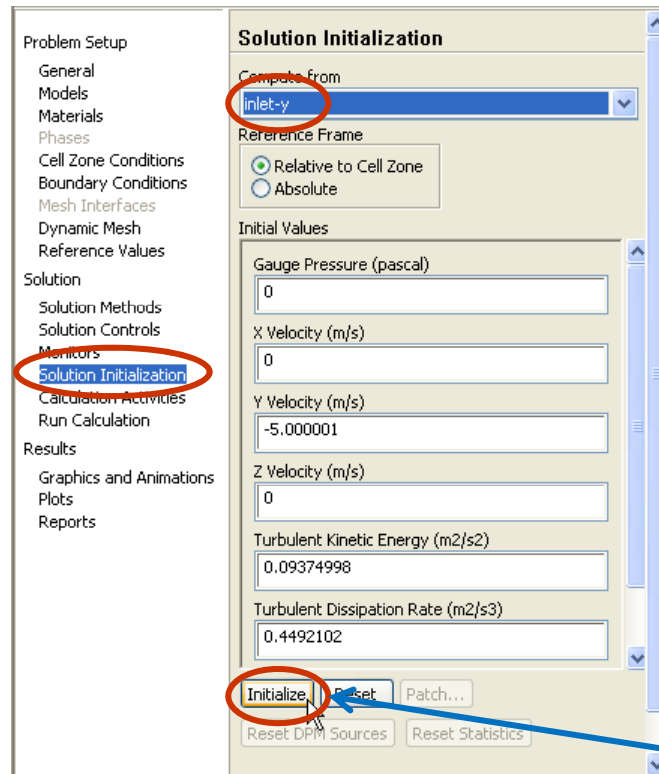
WS1: Mixing Tee

Initialization



- In 'Solution Initialization'
 - select 'Compute from' to be 'inlet-y' ←
 - press 'Initialize' ←

This computes a value for each variable, based on average conditions in the select zone. This value is used in every cell when you press '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.

FMG initialization

- Click in the console window, then:
 - press RETURN to see the TUI (Text User Interface) command menu
 - to enter the 'solve' sub-menu, type 'solve' and RETURN
 - to go up a level, type 'q' and RETURN
 - to issue a command starting from top level, start the command with '/'
 - many abbreviations are allowed (try it!)
 - type '/solve/initialization/fmg-initialization' and RETURN
 - override the default by typing 'yes' and RETURN

```
adapt/      file/      report/
close-fluent mesh/      solve/
define/     parallel/ surface/
display/    plot/      view/

> solve

/solve> q

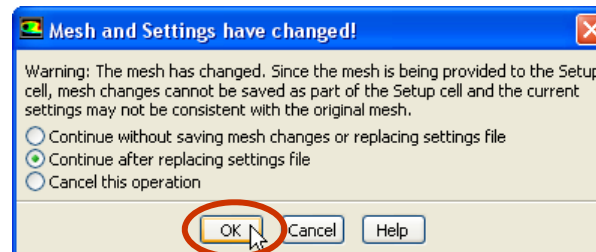
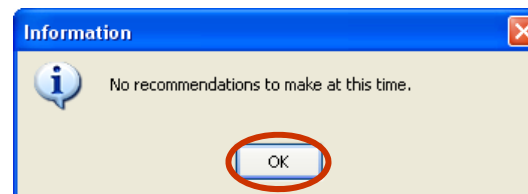
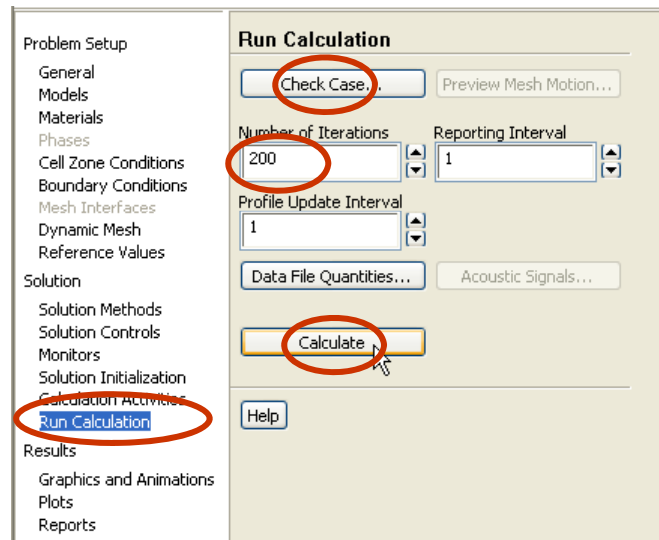
/solve/initialize/fmg-initialization
Enable FMG initialization? [no] yes
```

Here we use an advanced feature to improve on the basic initialization:

Full Multi-Grid (FMG) initialization.

This solves a very simplified set of flow equations, initially considering the geometry at a crude level and then building up detail.

- In 'Run Calculation'
 - press 'Check Case...'
 - see 'No recommendations to make at this time'
 - set 'Number of Iterations' to 200
 - press 'Calculate'
 - OK to 'Continue after replacing settings file'



Problem setup has **changed the mesh** – for example, the coordinates changed by scaling. There are many other changes that FLUENT can make – for example, adapting the mesh to increase the number of cells where the solution requires it.

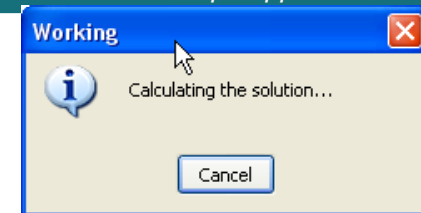
The link from Mesh to FLUENT in Workbench needs care – are you starting a new Problem Setup with a new mesh, or are you finding a new Solution on the old mesh?

WS1: Mixing Tee

Calculating

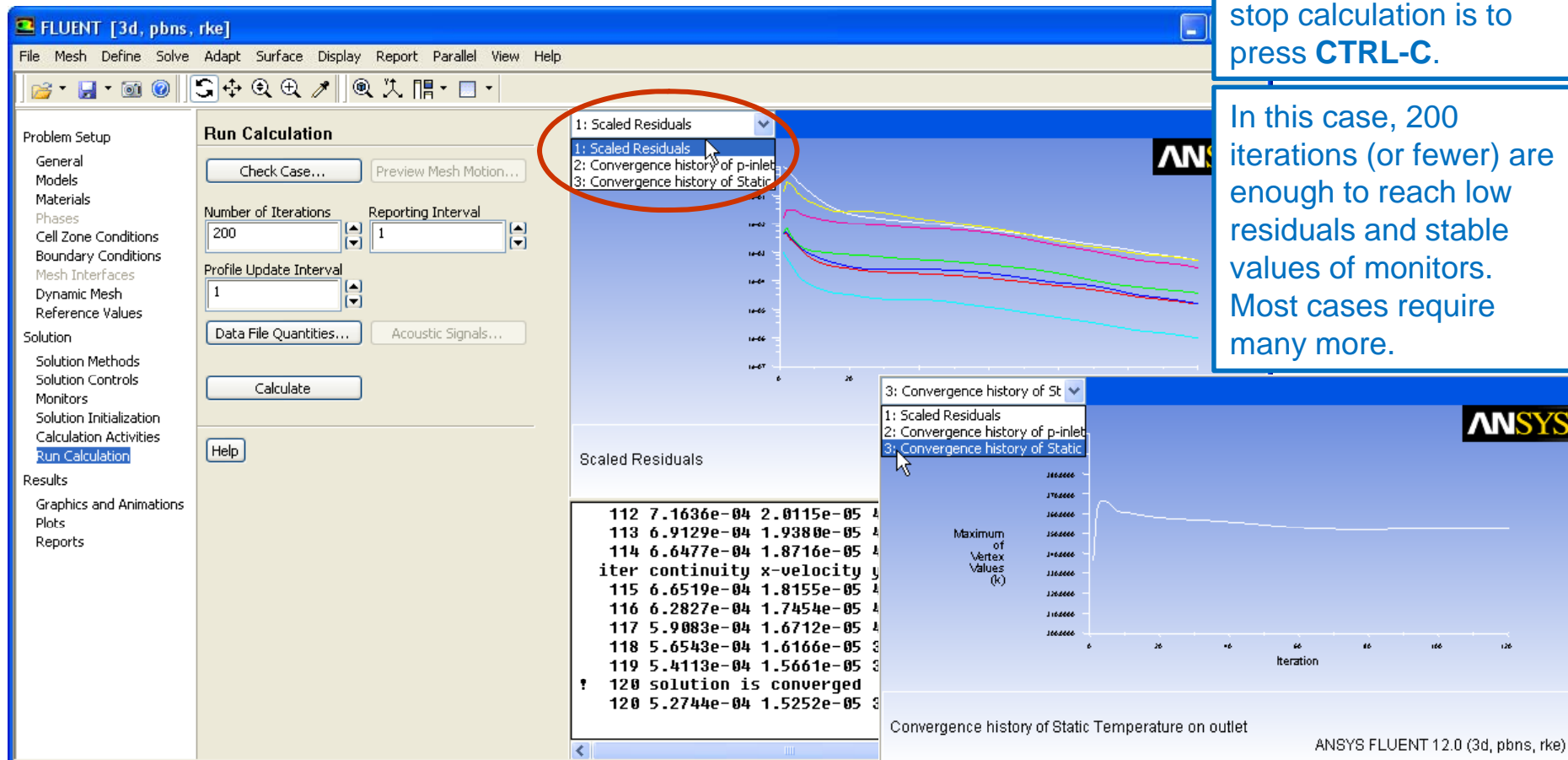


- While calculating, review residuals and monitors
 - change graphic windows using the drop-down box

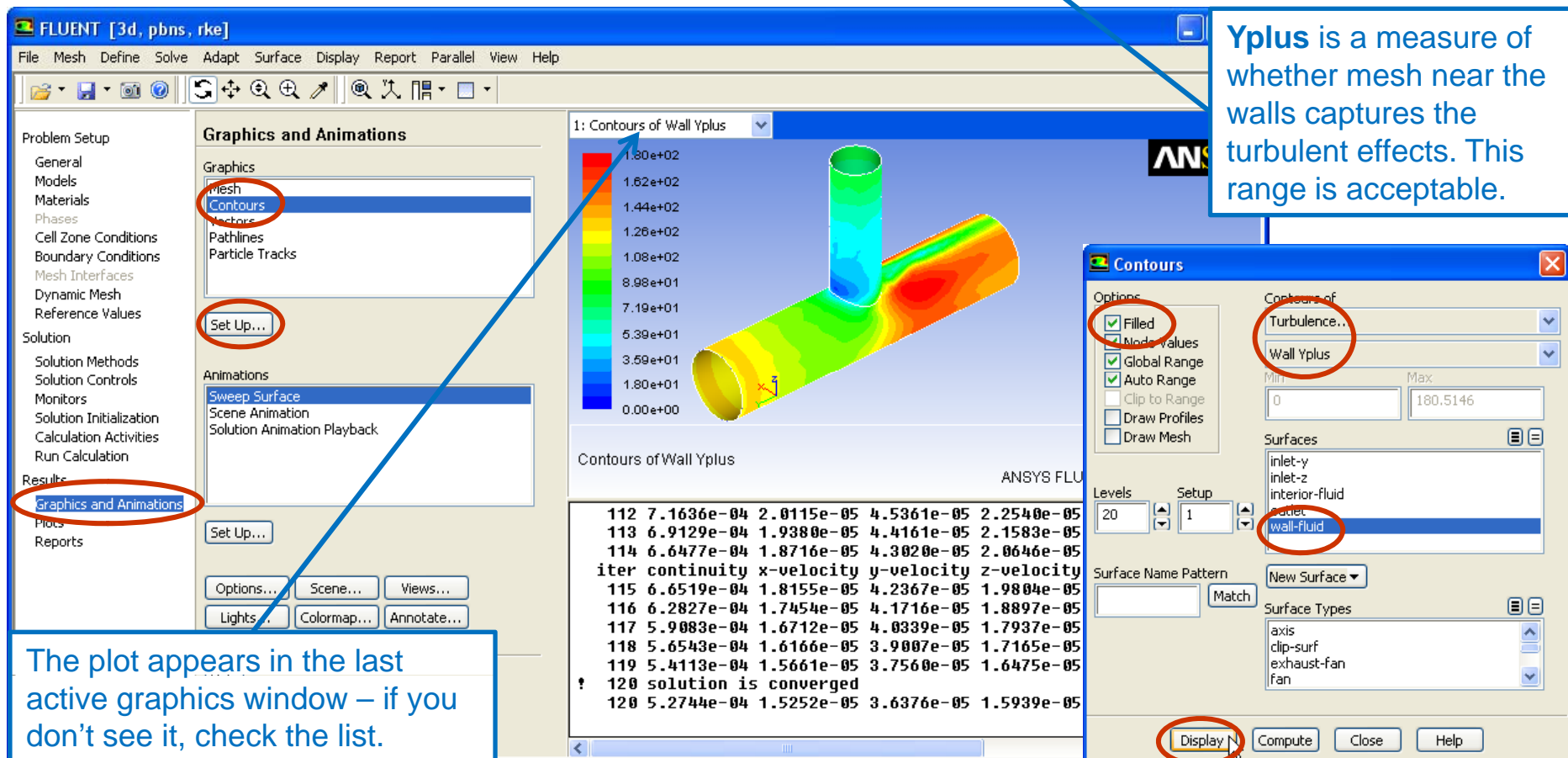


An alternative way to stop calculation is to press **CTRL-C**.

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



- In 'Graphics and Animations', select Contours, press 'Set Up...'
 - select 'Filled' contours of 'Turbulence...Wall Yplus' on 'wall-fluid'
 - press 'Display' – note almost all values are between 30 and 200



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.

Flux Reports

Options

- ☒ Mass Flow Rate
- ☐ Total Heat Transfer Rate
- ☐ Radiation Heat Transfer Rate

Boundary Types

- axis
- exhaust-fan
- fan
- inlet-vent

Boundary Name Pattern

Save Output Parameter...

Compute

Results

Boundaries	Results
inlet-y	90.459167
inlet-z	23.917273
interior-fluid	-114.37637
outlet	
wall-fluid	

Net Results (kg/s)

6.67572e-05

Contours of Static Temperature (c)

1: Contours of Static Temper

90.0
80.0
70.0
60.0
50.0
40.0
30.0
20.0
10.0

Net

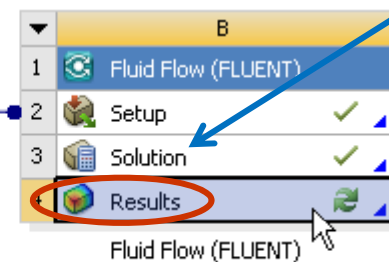
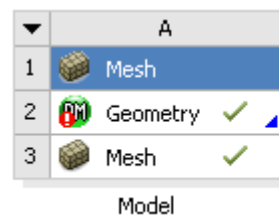
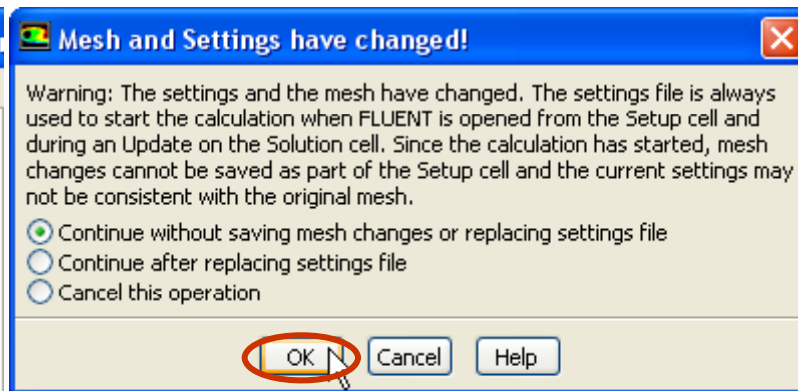
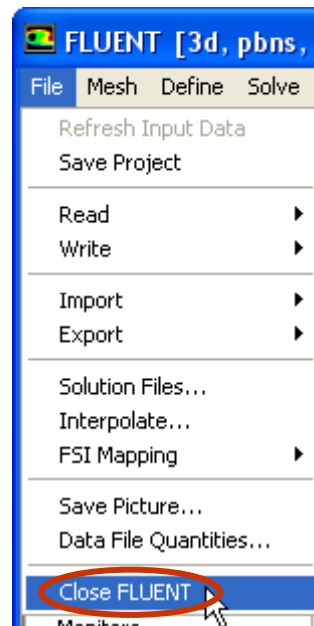
Total Heat Transfer Rate

inlet-y
inlet-z
outlet
wall-fluid

Net

Exit FLUENT

- **Exit FLUENT**
 - accept the default, 'Continue without saving'
- **In Workbench, double-click 'Results' to launch CFD-Post**
 - in the FLUENT session, we have completed Setup *and* Solution



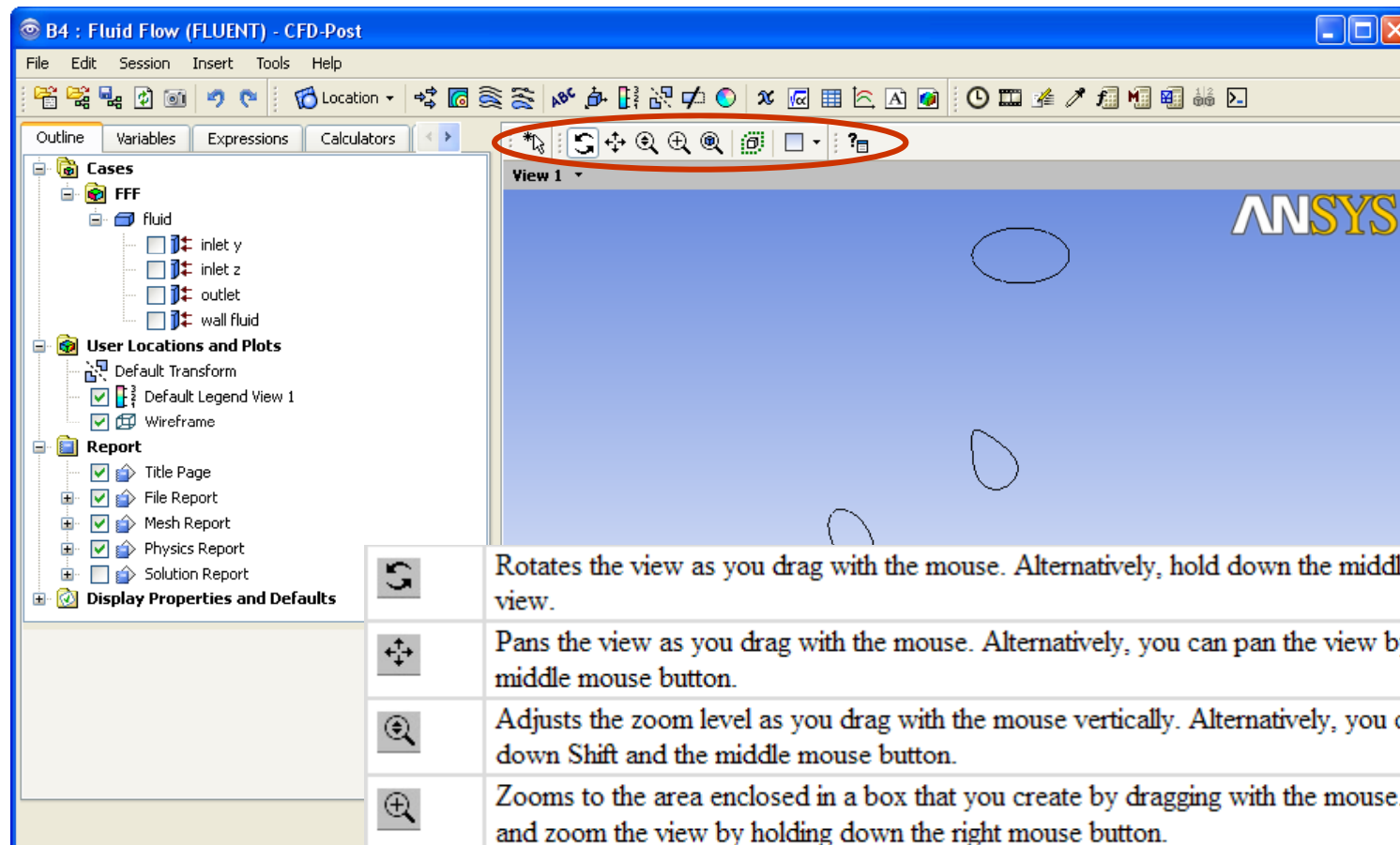
To adjust conditions in FLUENT, **double-click on Solution**.
If you re-open Setup, the link with the old, unscaled mesh is loaded back in – a window will warn you of this.

WS1: Mixing Tee


CFD-Post

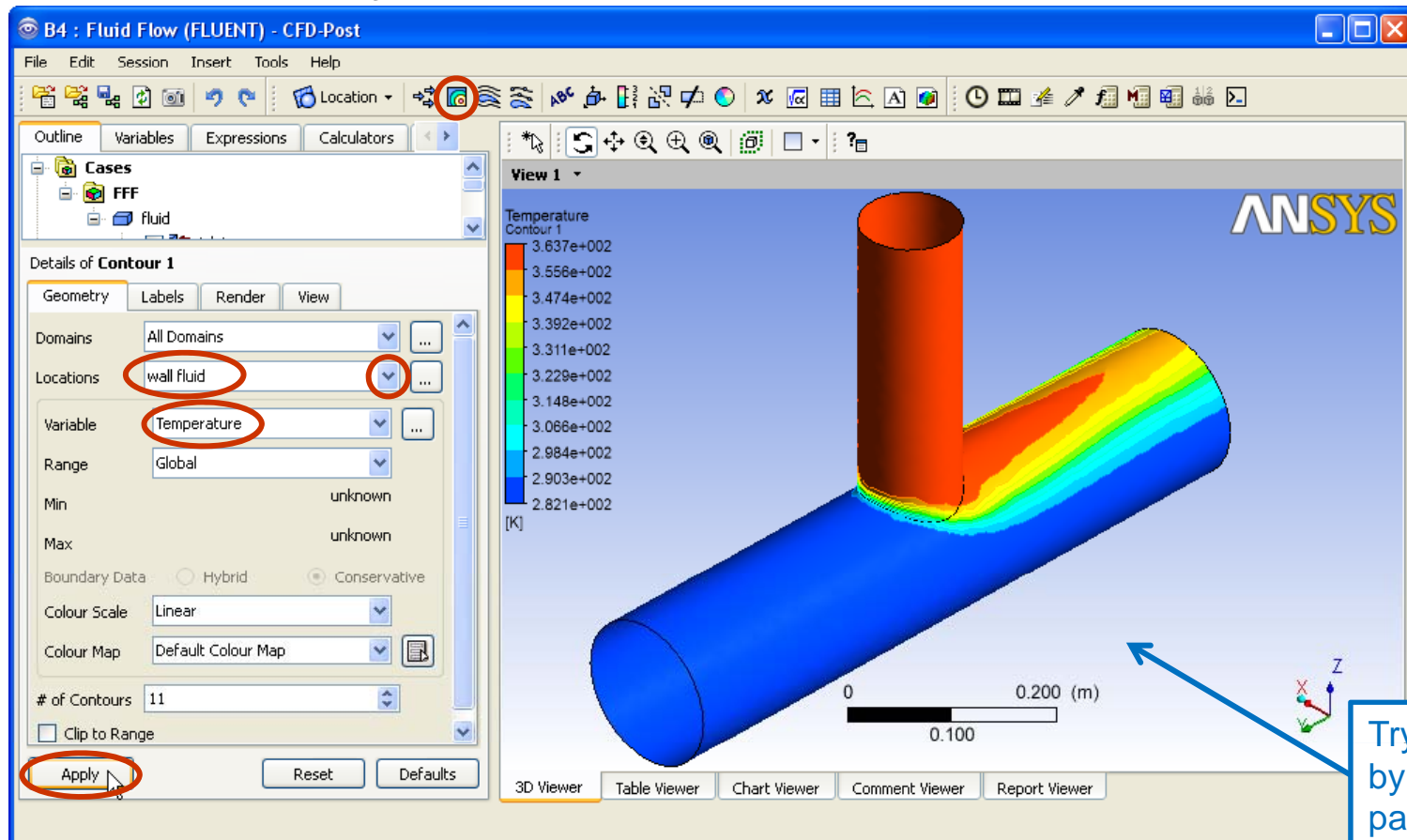


- The results are loaded
- CFD-Post displays the outline (wireframe) of the model
 - viewer toolbar buttons allow you to manipulate the view



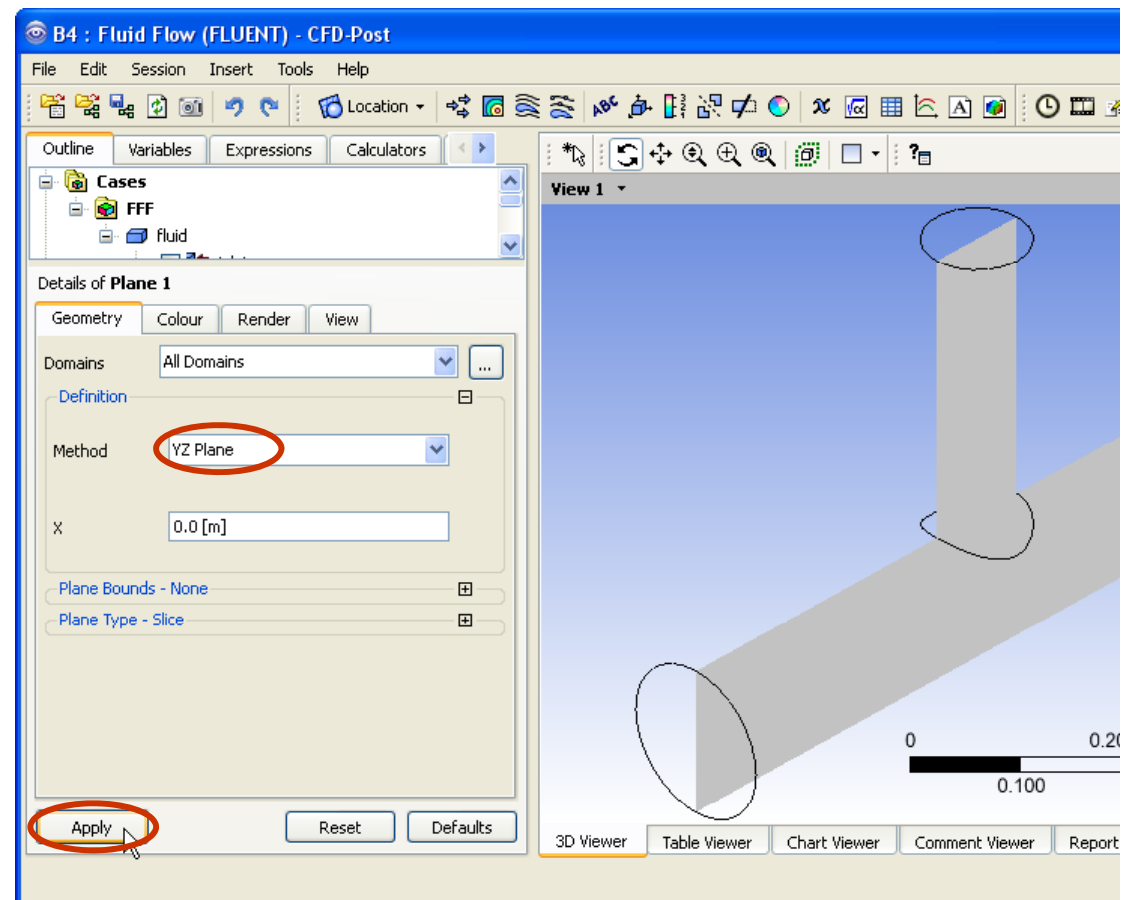
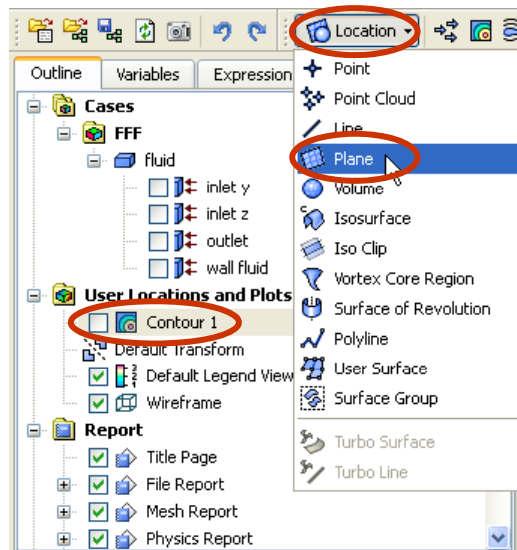
Temperature contour plot

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



Create a plane

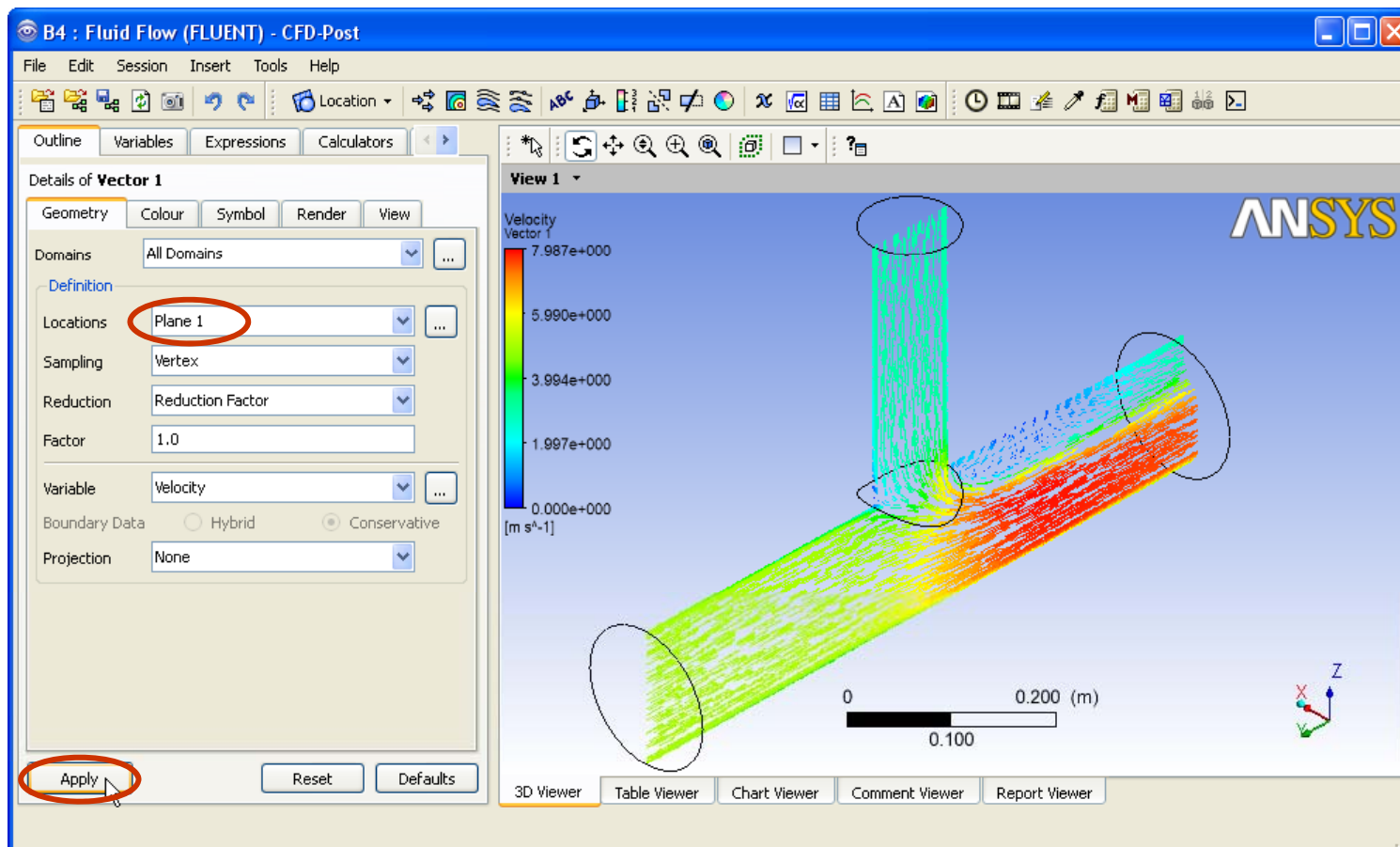
- Hide the contour plot by unchecking it in the tree view
- In the 'Location' menu, select 'Plane'
 - accept the default name 'Plane 1'
 - select 'Method' to be 'YZ Plane', accept 'X' as 0.0, and press 'Apply'



Velocity vector plot

- Hide the plane by unchecking it in the tree view
- Press the Vector button 
- select 'Locations' to be 'Plane 1', and press 'Apply'

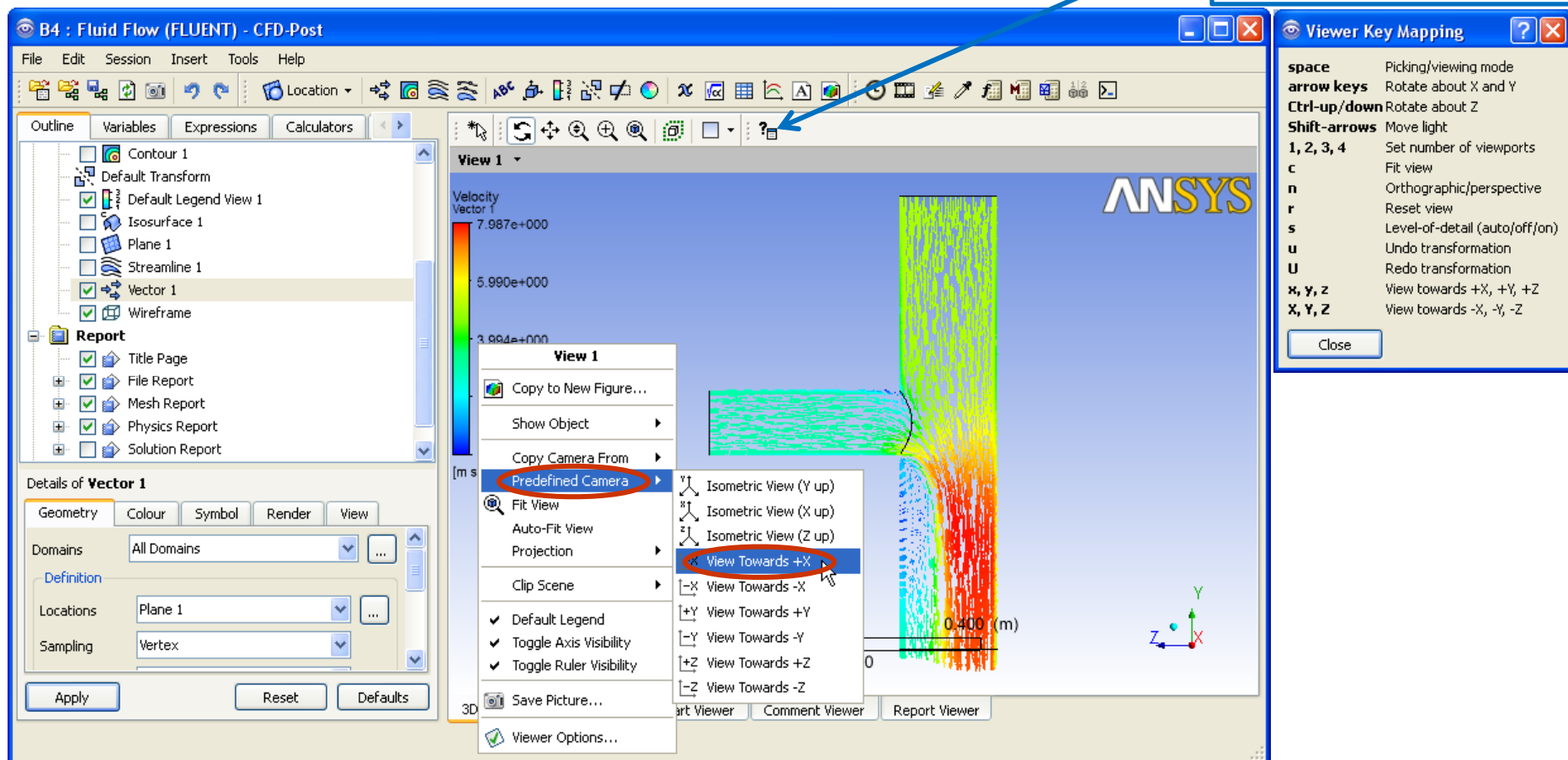
The plane is used only as a location for the vector plot.



Predefined Camera views and shortcuts


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

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



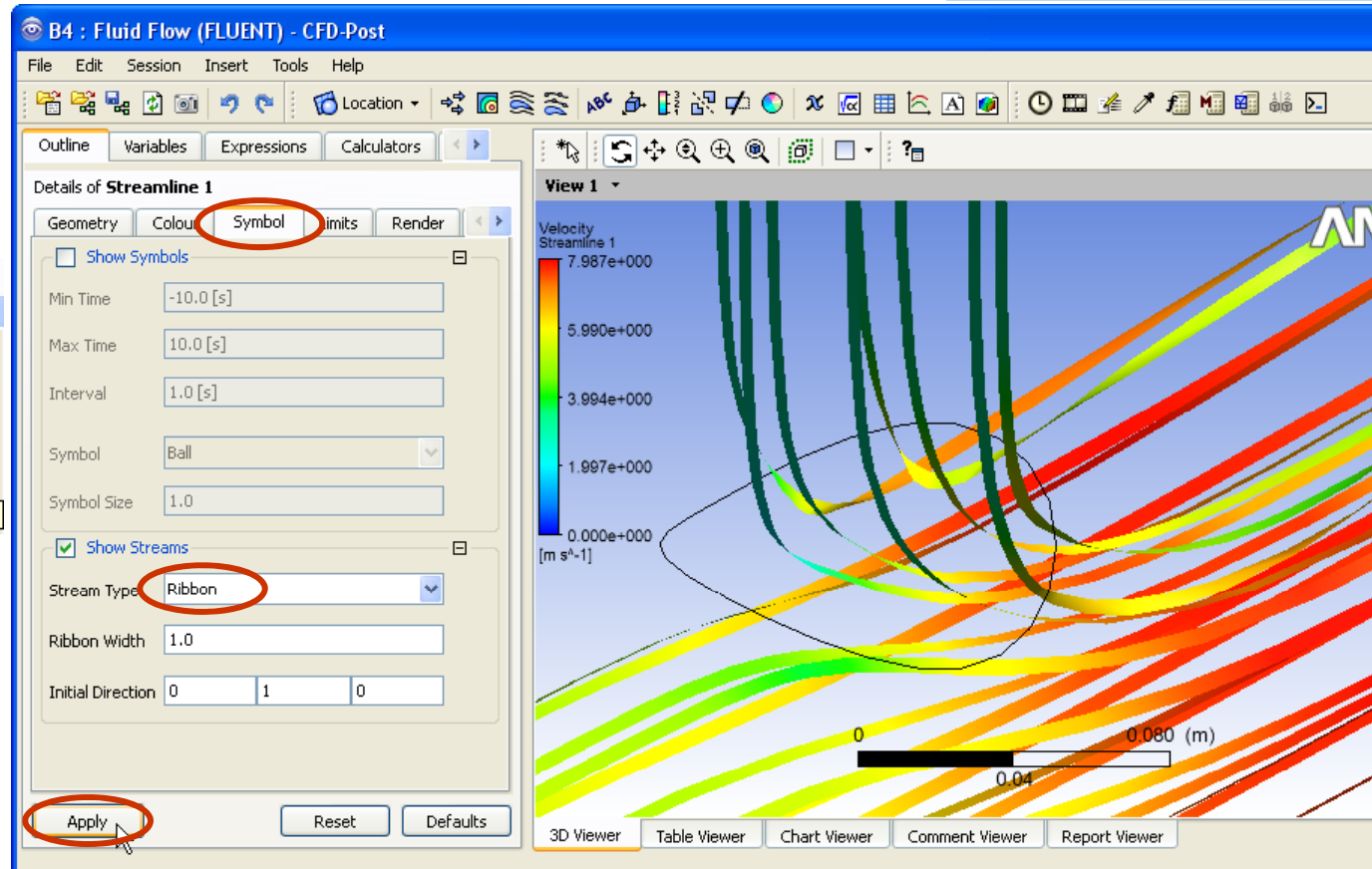
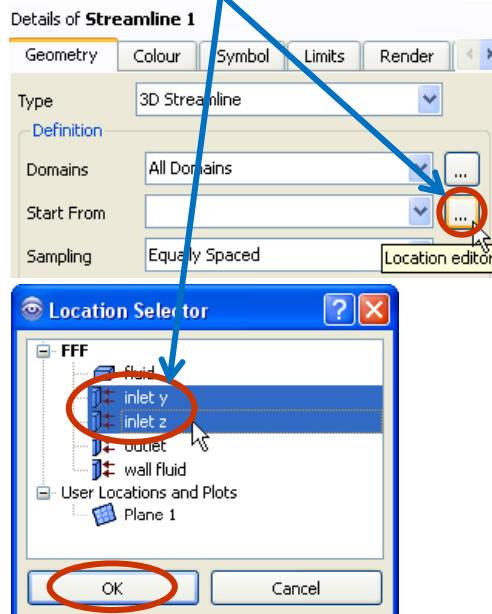
WS1: Mixing Tee

Streamline plot

- Hide the vector plot by unchecking it in the tree view
- Press the Streamline button 
 - select 'Start from' to be 'inlet y' and 'inlet z'
 - in the 'Symbol' tab, select 'Stream Type' to be 'Ribbon'

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

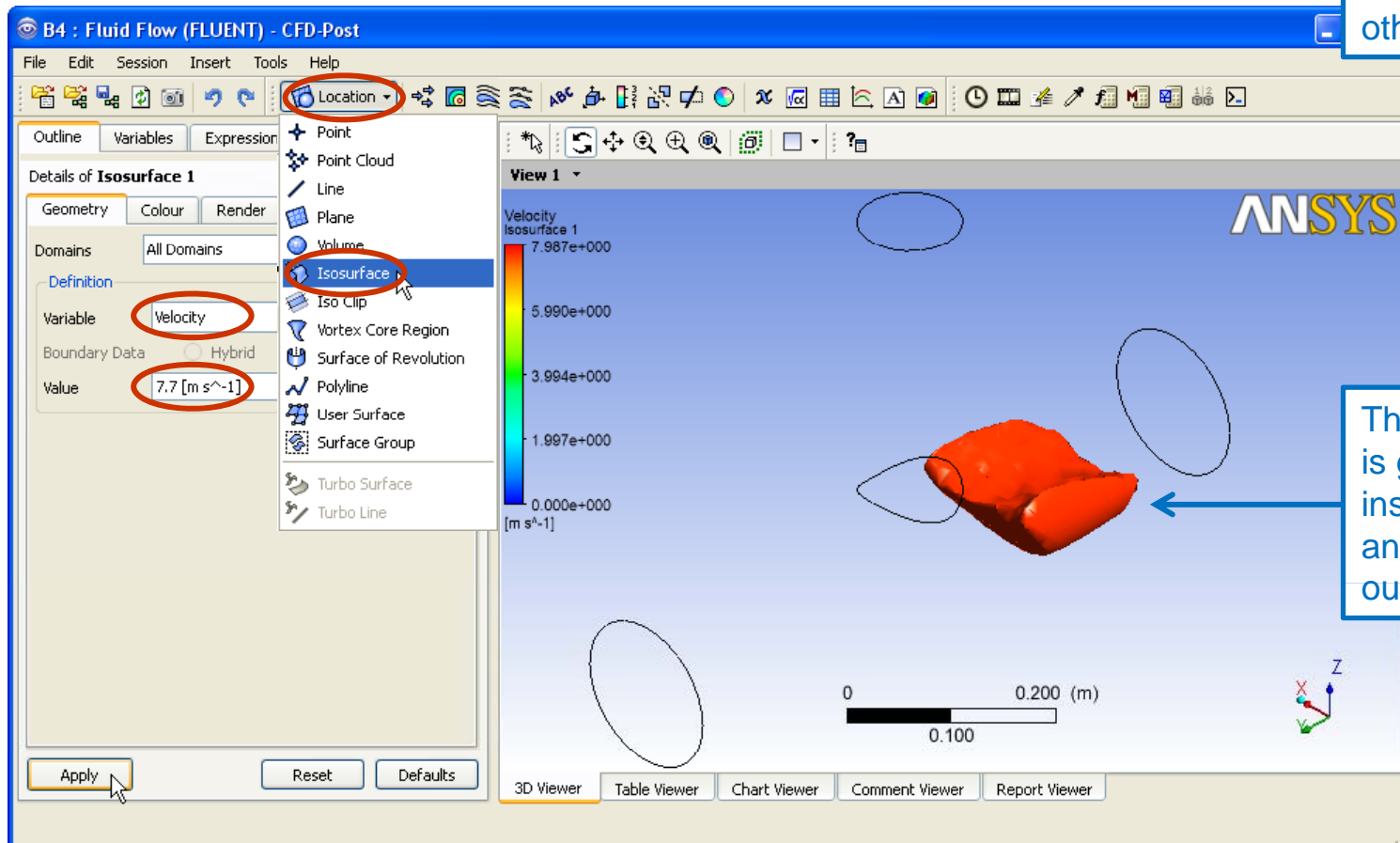
To select multiple locations, press the 'Location editor' button, and press CTRL while clicking.



Velocity isosurface

- Hide the streamline plot by unchecking it in the tree view
- In the 'Location' menu, select 'Isosurface'
 - in the 'Geometry tab', select 'Variable' to be 'Velocity' and 'Value' to be '7.7 [m s⁻¹]'

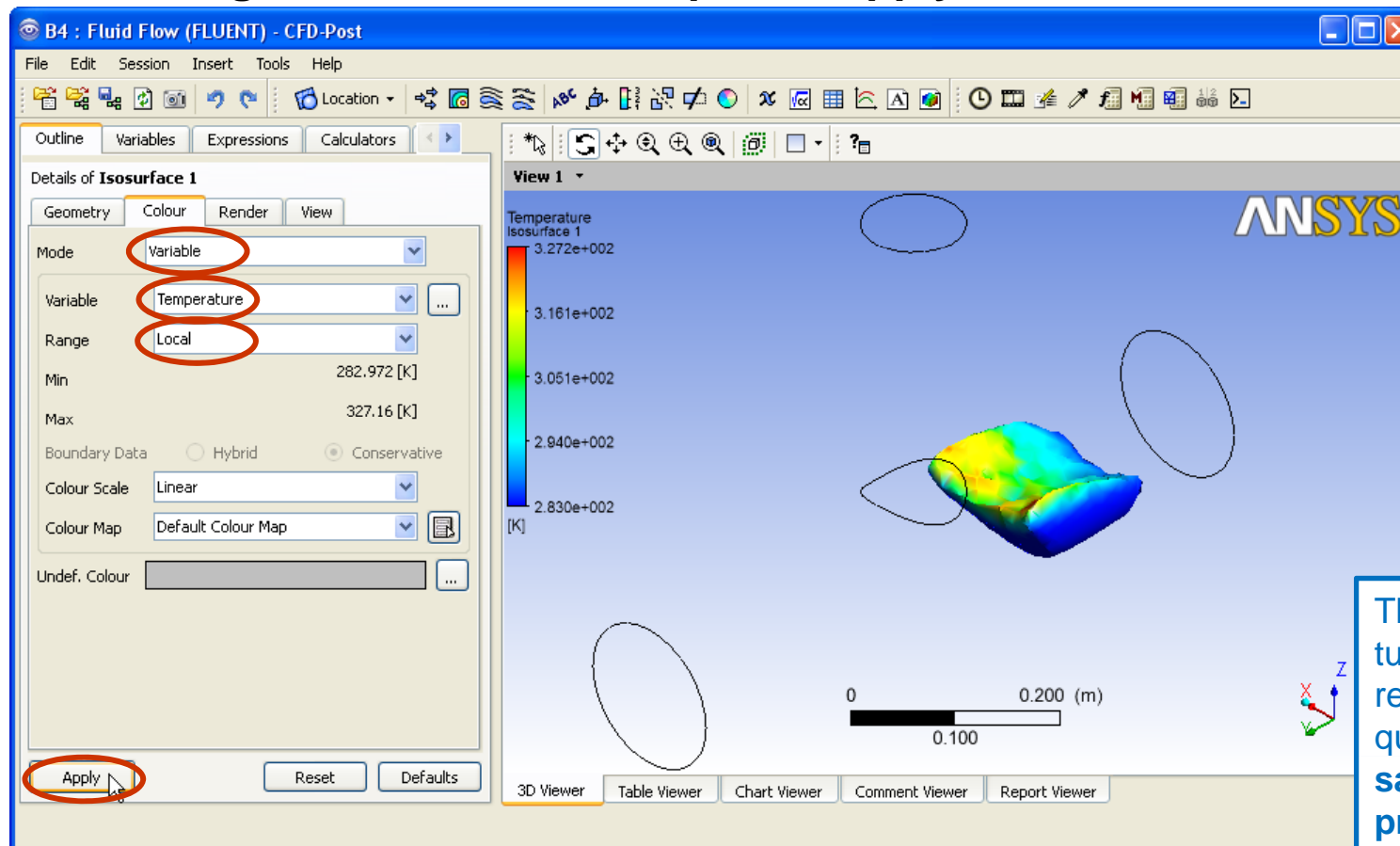
This is just one example – you can try other values.



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

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.