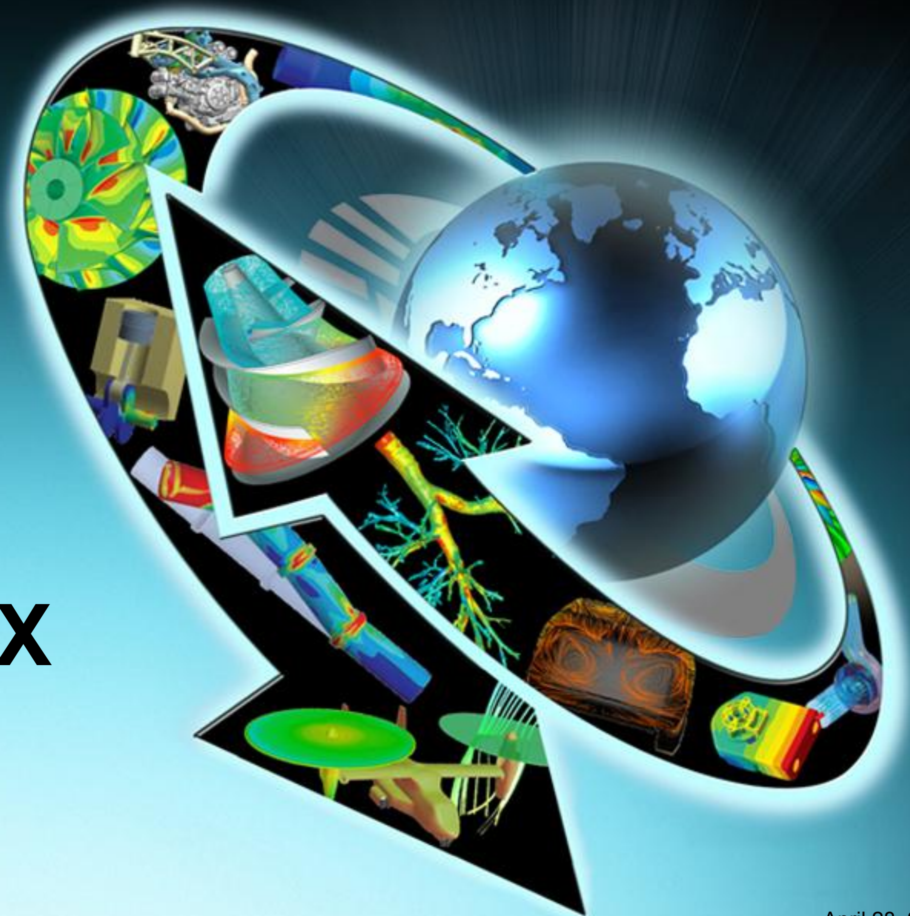
Workshop 1

# Mixing T-Junction



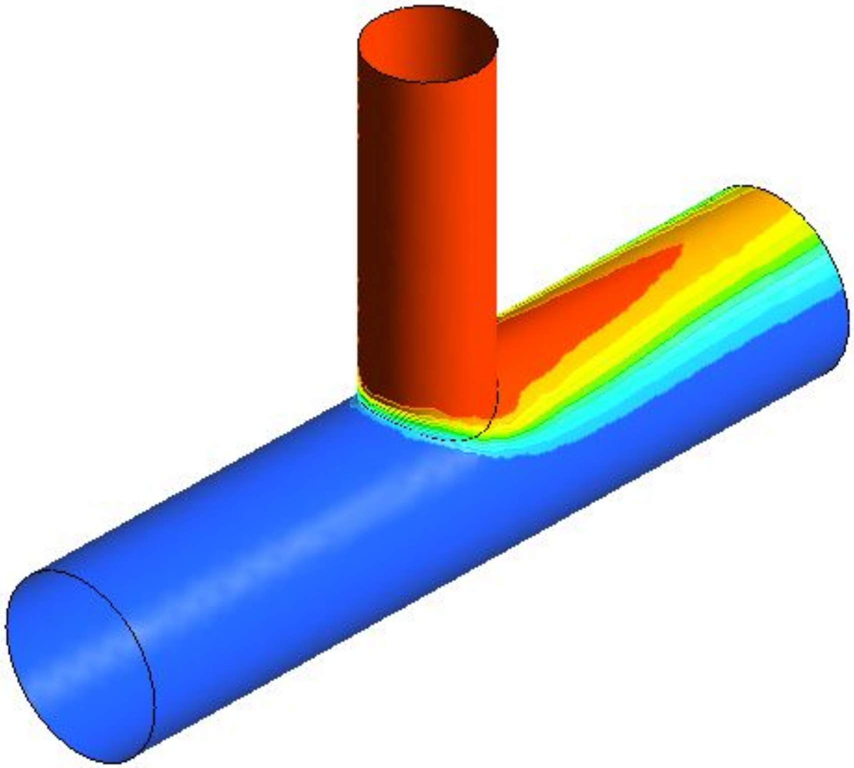
# Introduction to CFX

# Welcome!



- This introductory tutorial models mixing of hot and cold water streams
- The workshop starts from an existing mesh and applies boundary conditions to model a cold main inlet and a hot side inlet
- Analysis goals for this type of problem could be to determine:
  - how well do the fluids mix?
  - what are the pressure drops?

*Note: It's a good idea to identify the quantities of interest from the start. You can use these to monitor the progress of the solution*



# Pre-processing Goals

ANSYS

*Workshop Supplement*

- Launch CFX-Pre from Workbench
- Use pre-defined materials
- Define the fluid models in a domain
- Create and edit objects in CFX-Pre
- Define boundary conditions
- Set up monitor points using simple expressions

- Launch the CFX Solver Manager from Workbench
- Monitor convergence

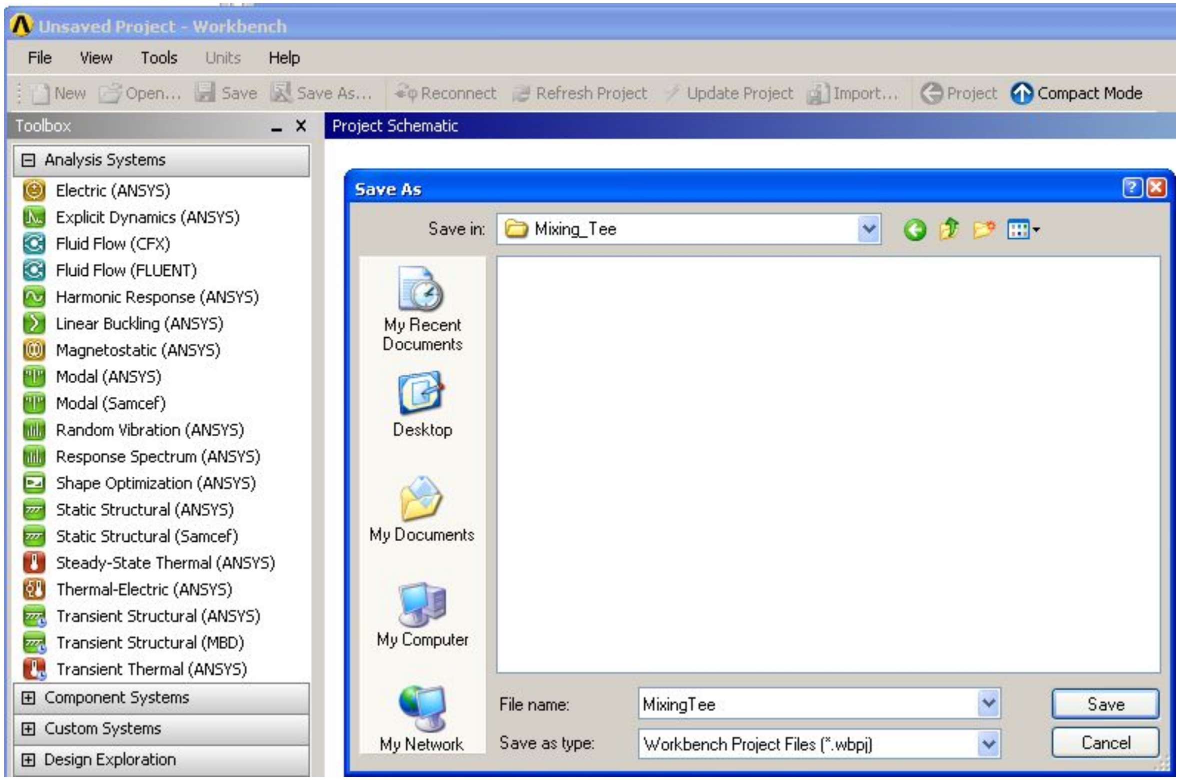
- Launch CFD-Post from an existing CFX simulation in Workbench
- Rotate, zoom and pan the view
- Create contour plots
- Create a plane for use as a locator
- Create a velocity vector plot
- Use pre-defined views
- Create streamlines of velocity
- Create an isosurface, coloured by a separate variable

# Start in Workbench



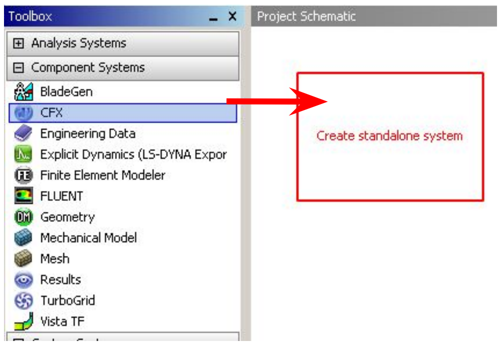
The first step is to start Workbench:

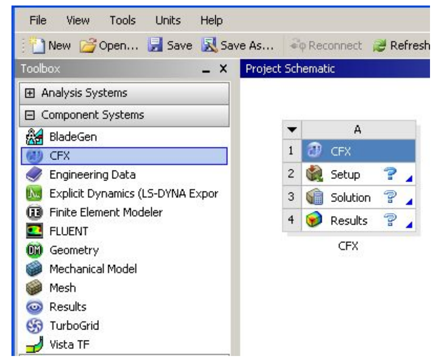
1. From the windows Start menu, select *Programs > Ansys 12.0 > Workbench*
2. When Workbench opens, select *File > Save* and save the project as *MixingTee.wbprj*



# Start a CFX case

3. Next, expand the *Component Systems* toolbox and drag a *CFX* analysis into the top left area of the *Project Schematic*

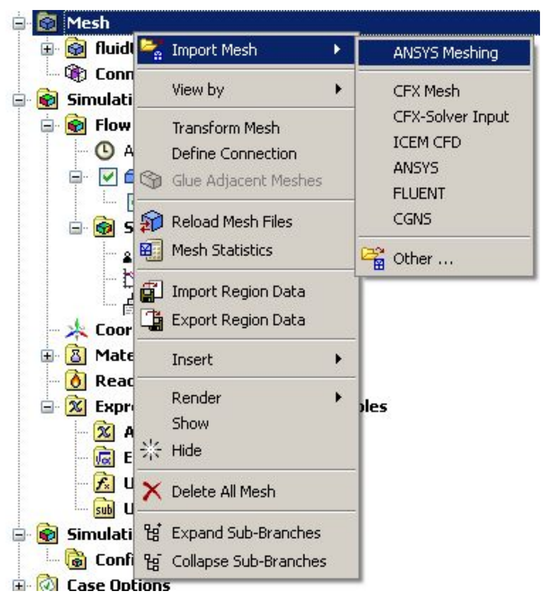



4. Double-click on *Setup* to launch CFX

5. When CFX-Pre opens, right-click on *Mesh* in the Outline tree and select *Import Mesh > ANSYS Meshing*

6. Select the file *fluidtee.cmdb* and click *Open*



# CFX-Pre GUI Overview

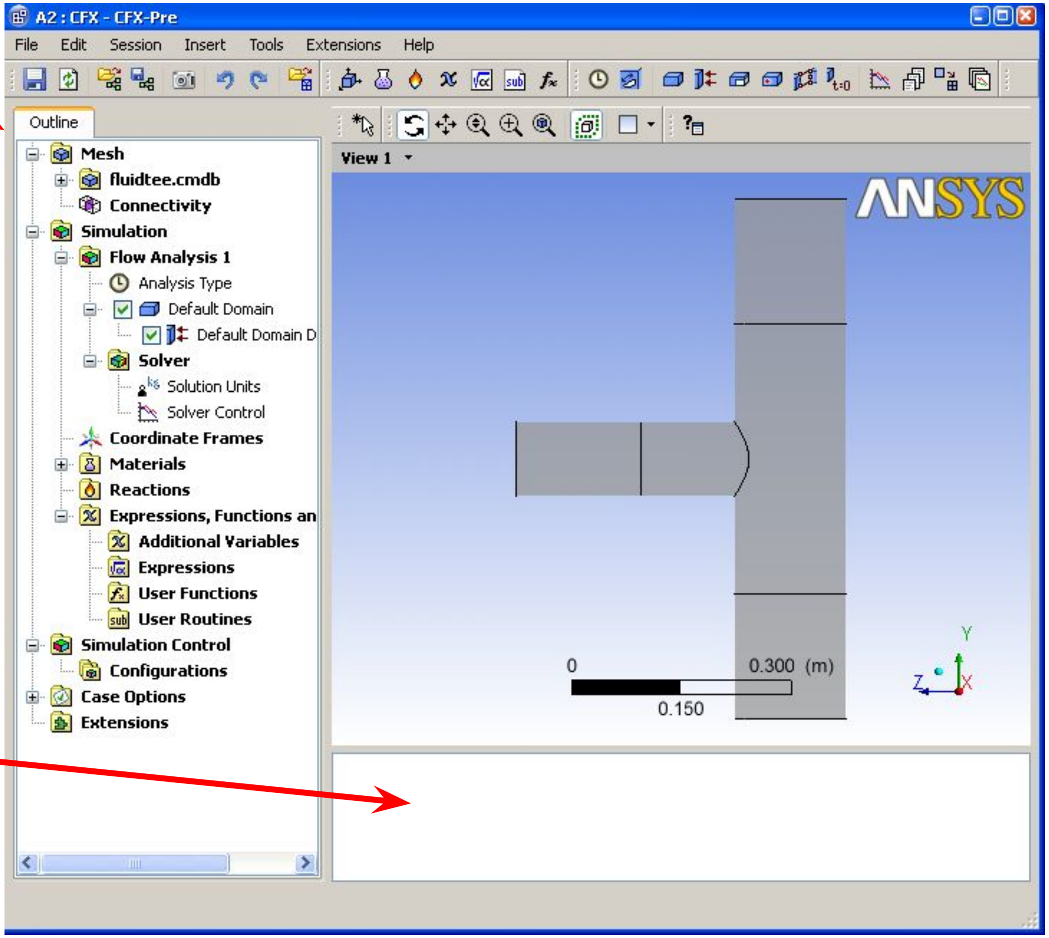


## Outline Tree

- New objects appear here as they are created
- Double-click to edit existing object
- New objects are often inserted by right-clicking in the Outline tree

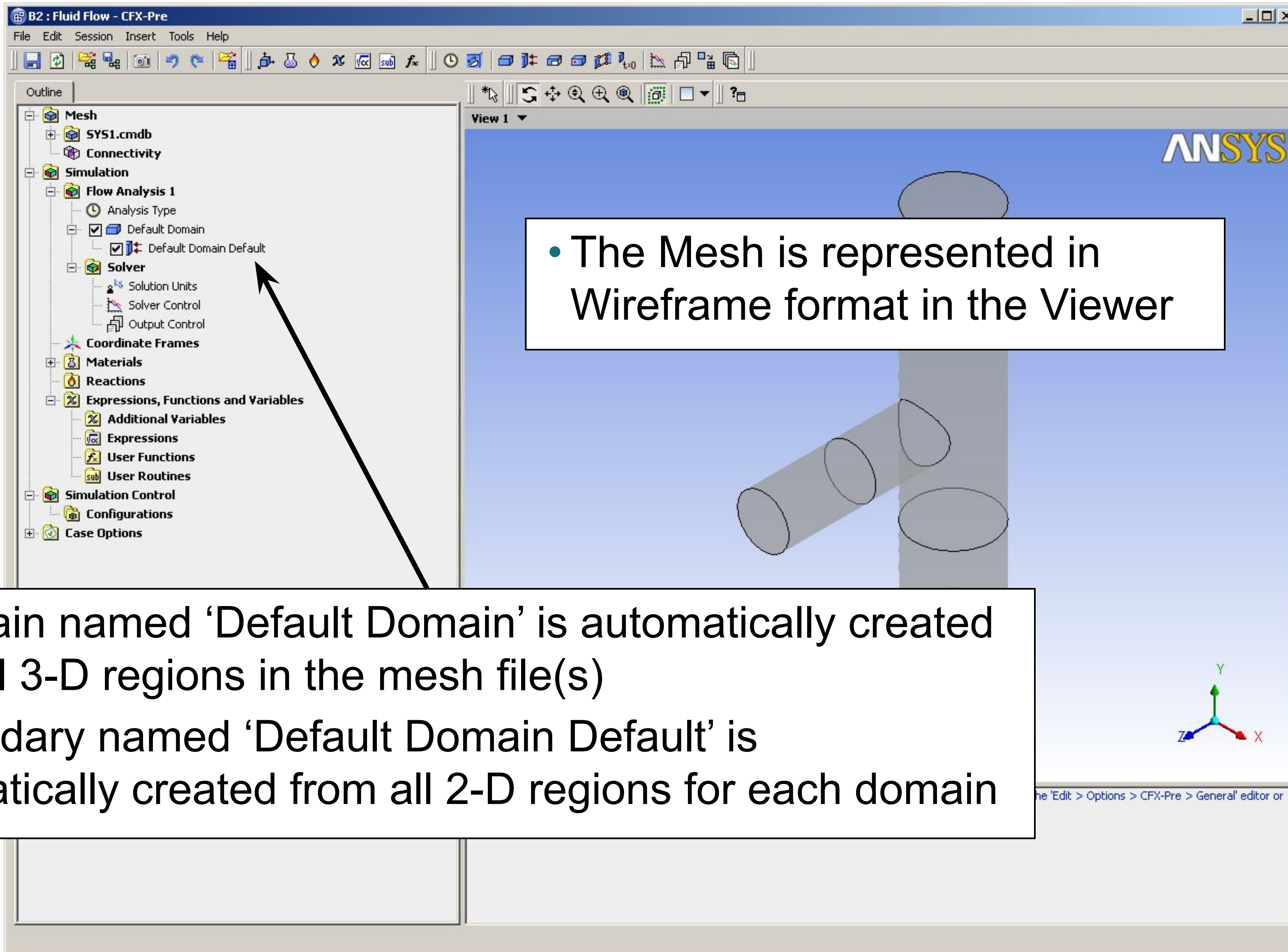
## Message Window

- Warnings, errors and messages appear here



# CFX-Pre Mesh and Regions






- The Mesh is represented in Wireframe format in the Viewer

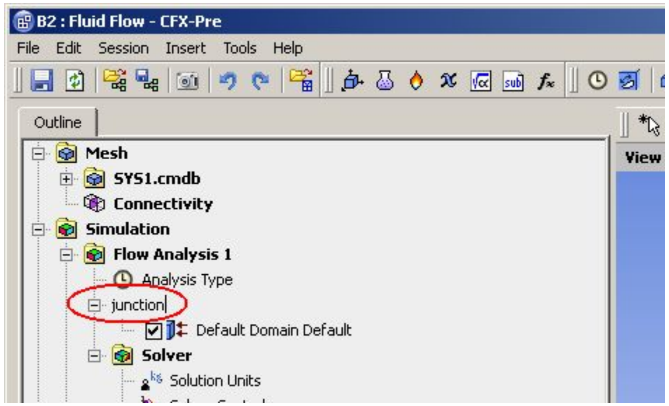
- A domain named ‘Default Domain’ is automatically created from all 3-D regions in the mesh file(s)
- A boundary named ‘Default Domain Default’ is automatically created from all 2-D regions for each domain

# CFX-Pre – Domain settings



**The Default Domain contains all 3D mesh regions that are imported. If you create new domains, those regions are automatically removed from the Default Domain. The Default Domain is automatically deleted if no unassigned 3D regions remain.**

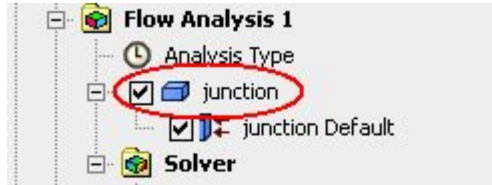
The first step is to change the domain name to something more meaningful.




1. Right-click on *Default Domain* in the *Outline* tree
2. Select *Rename*
   – The domain name can now be edited
3. Change the domain name to *junction*

# CFX-Pre – Domain settings (continued)

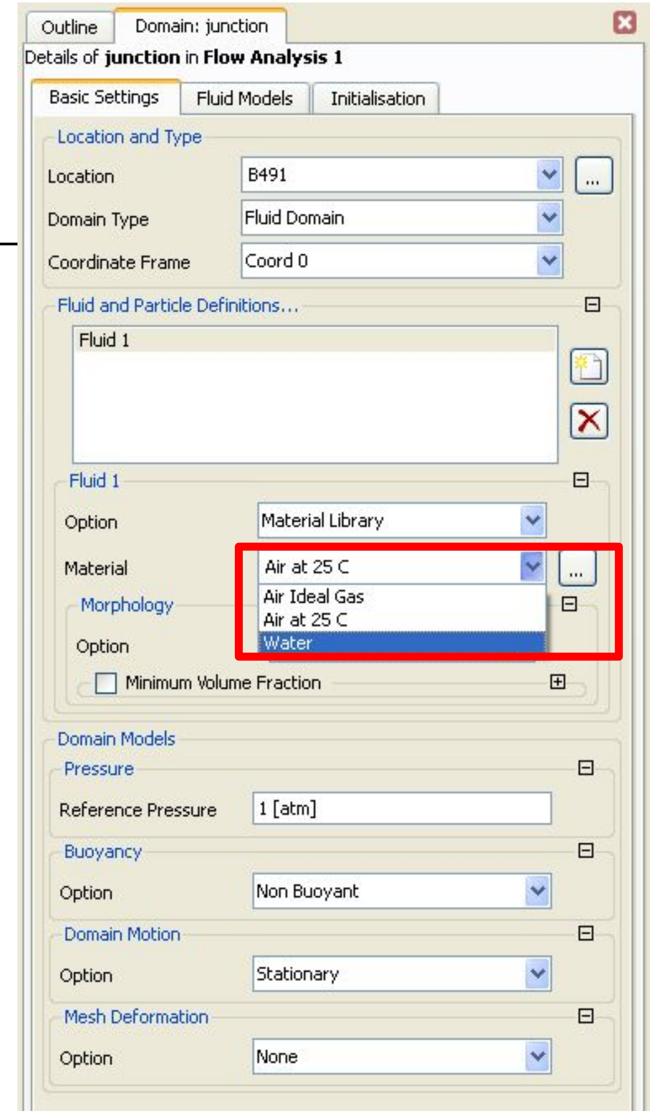
4. Double-click on the renamed domain *junction*




**The Domain panel contains three tabs named *Basic Settings*, *Fluid Models* and *Initialisation*. For more complex simulations additional tabs may appear.**

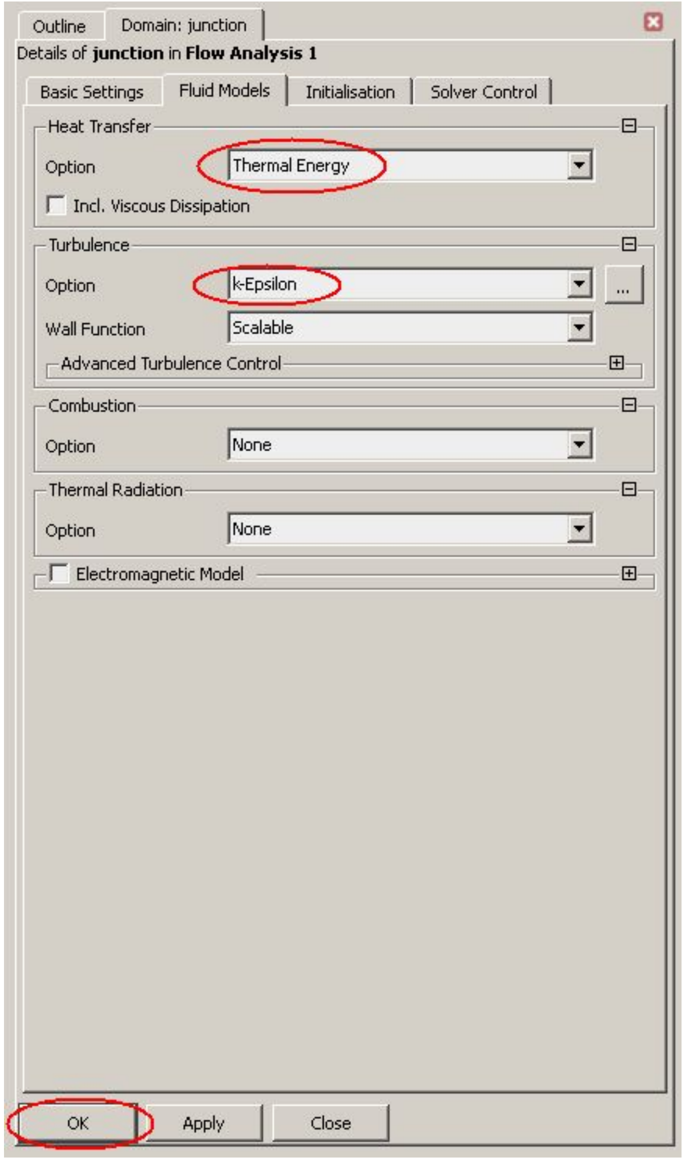
5. Set the *Material* to *Water*.
   - The available materials can be found in the drop-down menu

Note that CFX has a comprehensive library of materials. These can be accessed by using the ... icon and then selecting the *Import Library Data* icon.



# CFX-Pre – Domain settings (continued)

6. Click the *Fluid Models* tab
7. In the *Heat Transfer* section, change *Option* to *Thermal Energy*
   – Heat Transfer will be modelled. This model is suitable for incompressible flows
8. Leave all other settings as they are
   – The k-Epsilon turbulence model will be used, which is the default
9. Click *OK* to apply the new settings and close the domain form



# Boundary Conditions



The next step is to create the boundary conditions. You will create a cold inlet, a hot inlet and an outlet. The remaining faces will be set to adiabatic walls. Currently all external 2D regions are assigned to the *junction Default* boundary condition.

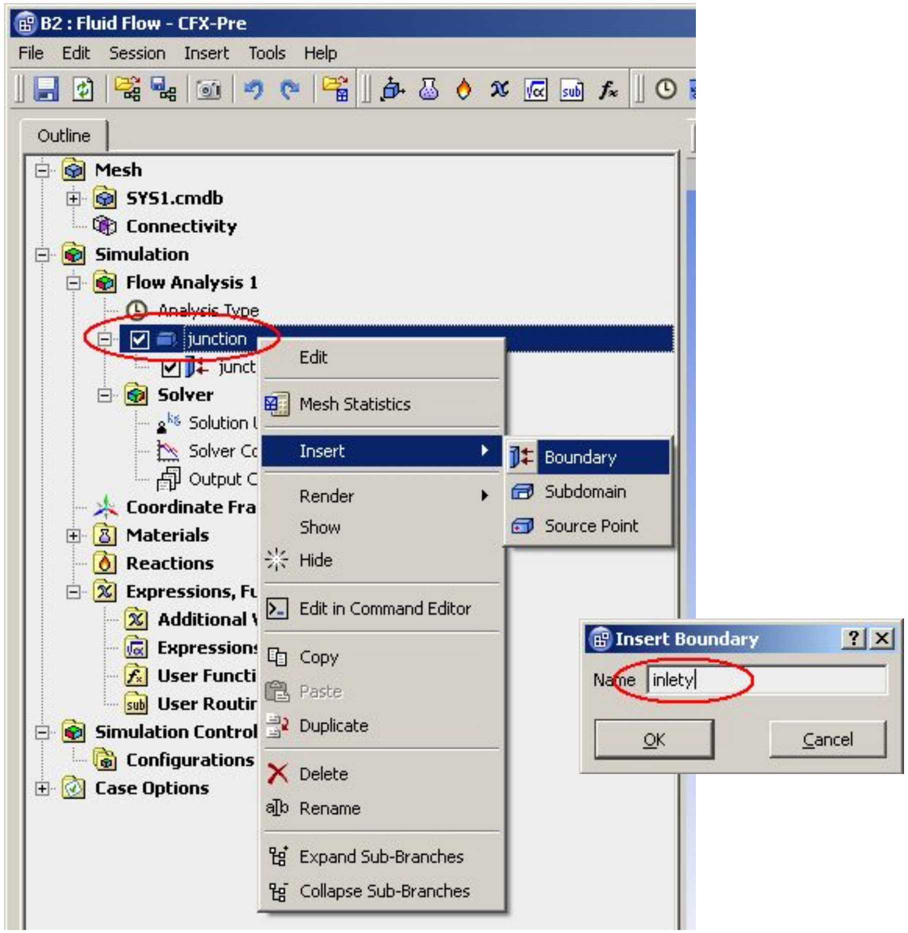
**Each domain has an automatic default boundary condition for *external* surfaces. The default boundary condition is a No Slip, Smooth, Adiabatic wall. As you create new boundary conditions, those regions are automatically removed from the default boundary condition.**

# CFX-Pre – Inlet boundary conditions



Now that the domain exists, boundary conditions can be added

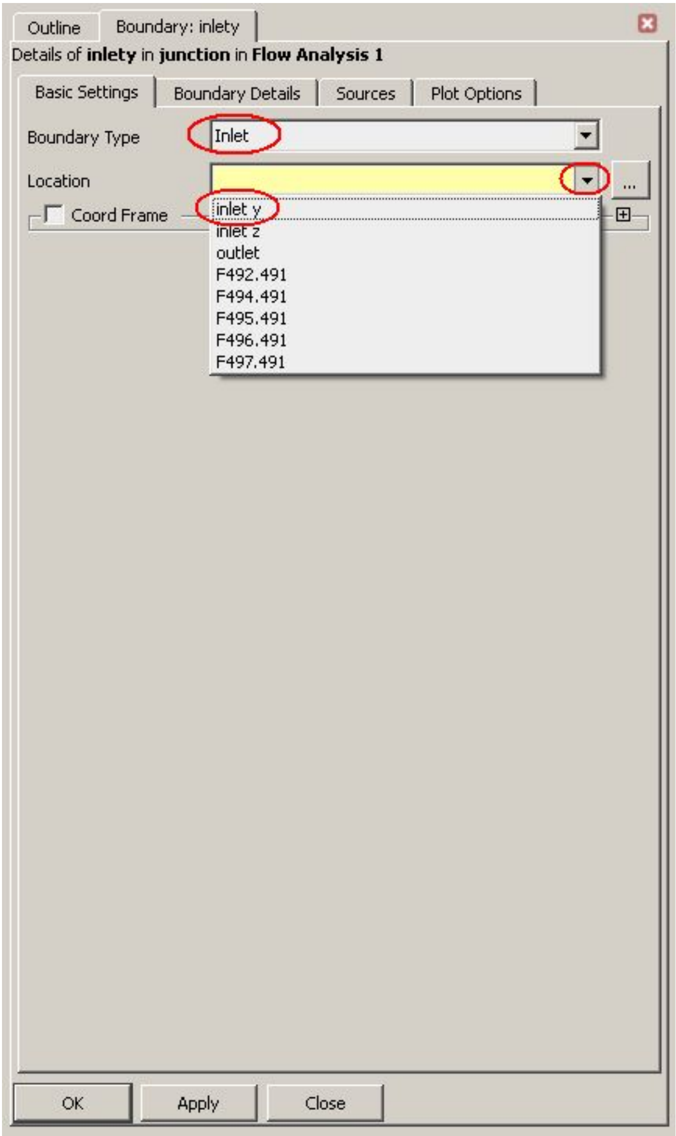
1. Right-click on the *junction* domain
2. Select *Insert > Boundary*
3. Set the *Name* to *inlety*
4. Click *OK*



# CFX-Pre – Inlet boundary conditions (contd.)



5. Leave the *Boundary Type* field set to *Inlet*
6. Set *Location* to *inlet y*
   – The available locations can be found in the drop-down menu of the extended "…" menu

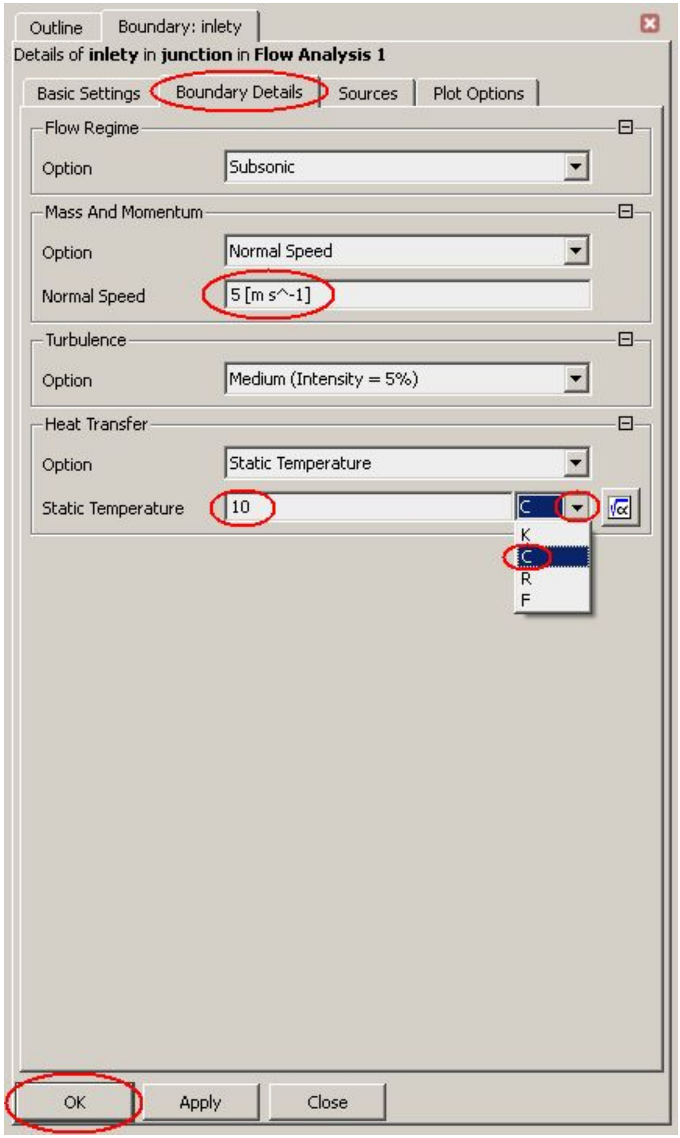


# CFX-Pre – Inlet boundary conditions (contd.)



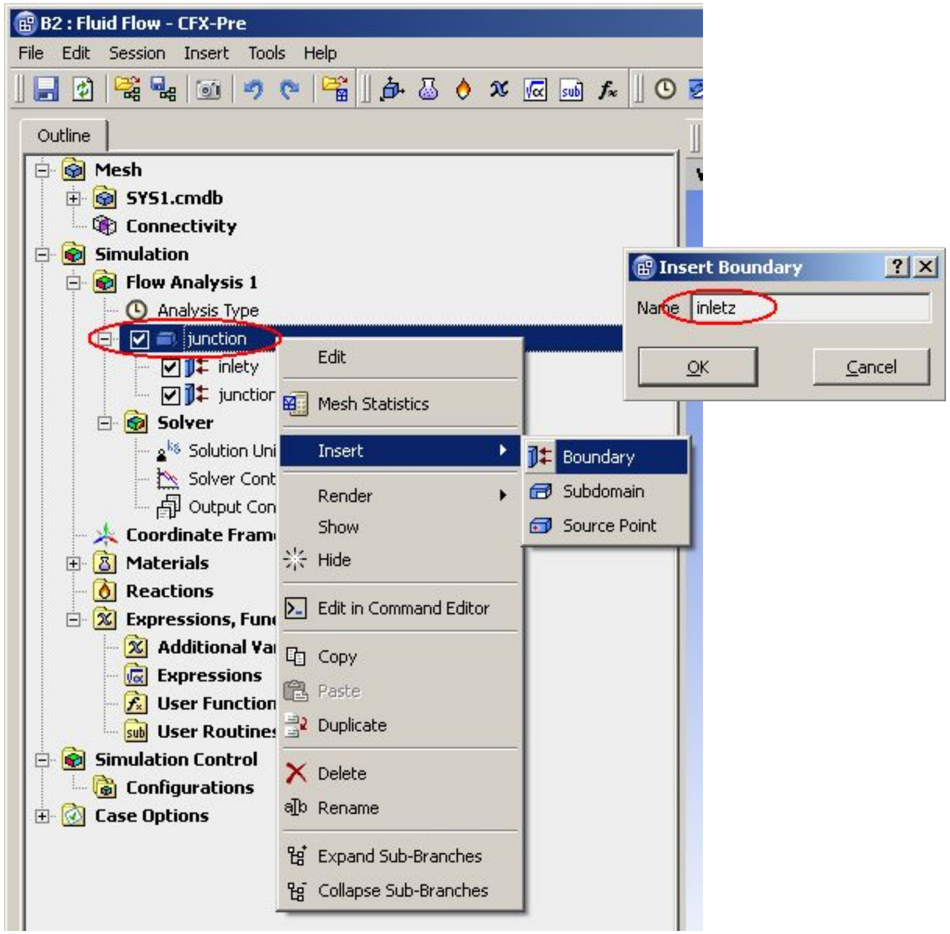
This inlet will have a normal speed of 5 m/s and temperature of 10°C.

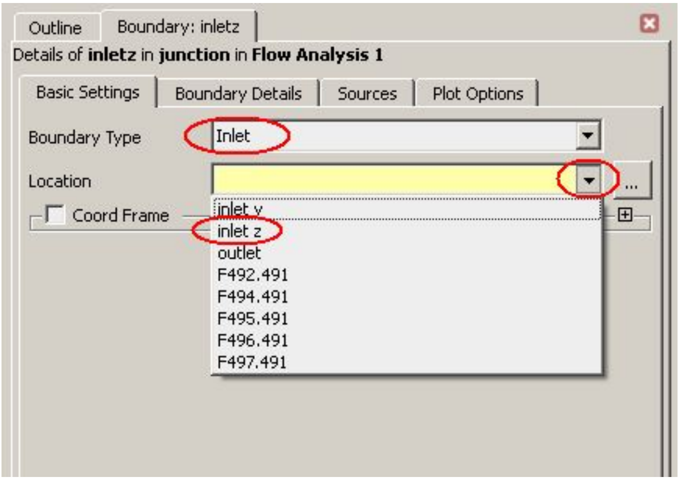
7. Click the *Boundary Details* tab
8. Enter a value of *5* for *Normal Speed*. The default units are *[m s^-1]*
9. Enter a value of *10* for *Static Temperature*. Use the drop-down menu to the right of the field to change the units to *C* (Celcius)
10. Click *OK* to apply the boundary and close the form



# CFX-Pre – Inlet boundary conditions (contd.)

1. Right-click on the *junction* domain and select *Insert > Boundary*
2. Set the *Name* to *inletz* and click *OK*
3. Leave the *Boundary Type* field set to *Inlet*
4. Set *Location* to *inlet z*



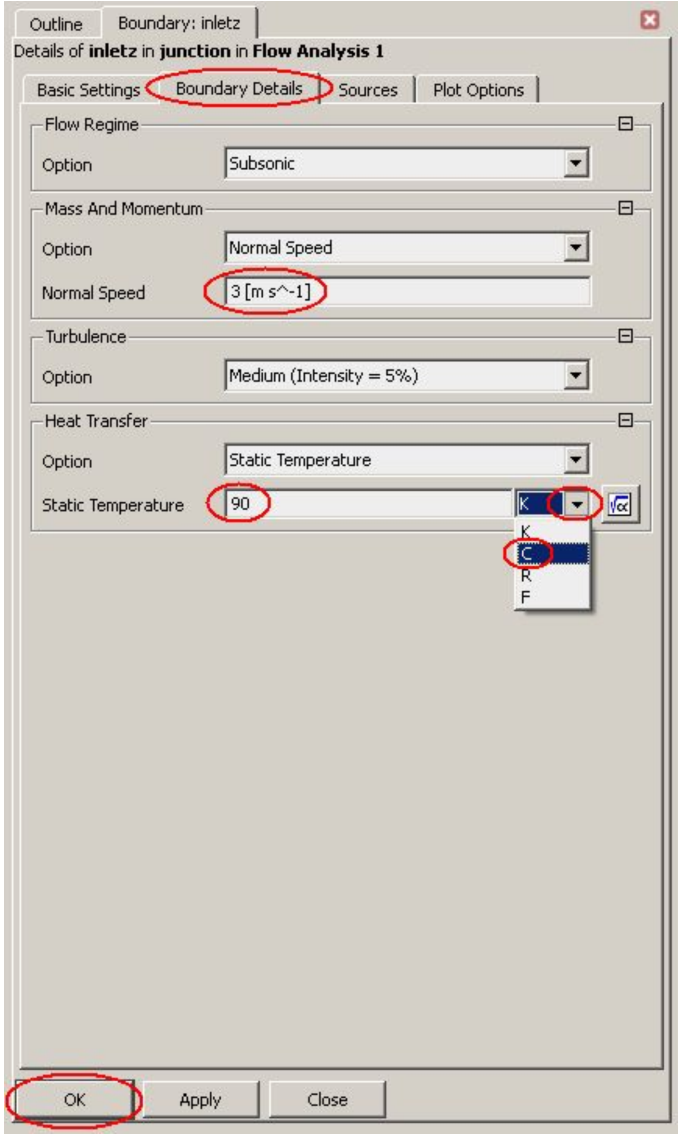



# CFX-Pre – Inlet boundary conditions (contd.)

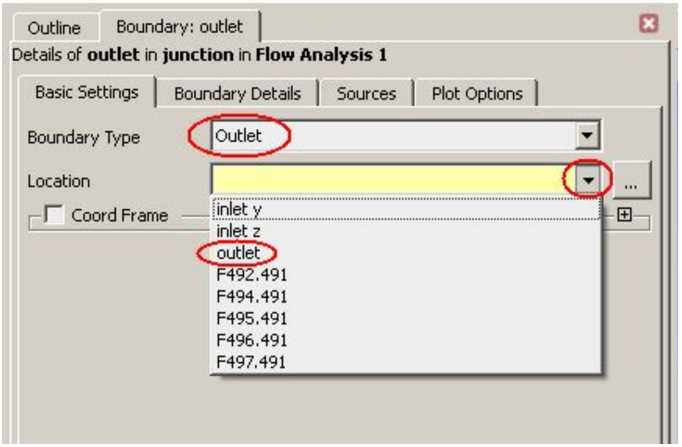


This inlet will have an inlet speed of 3 m/s and temperature of 90°C.

5. Click the *Boundary Details* tab
6. Enter a *Normal Speed* of *3 [m s^-1]*
7. Set the *Static Temperature* to *90 [C]* (make sure the units are correct!)
8. Click *OK*



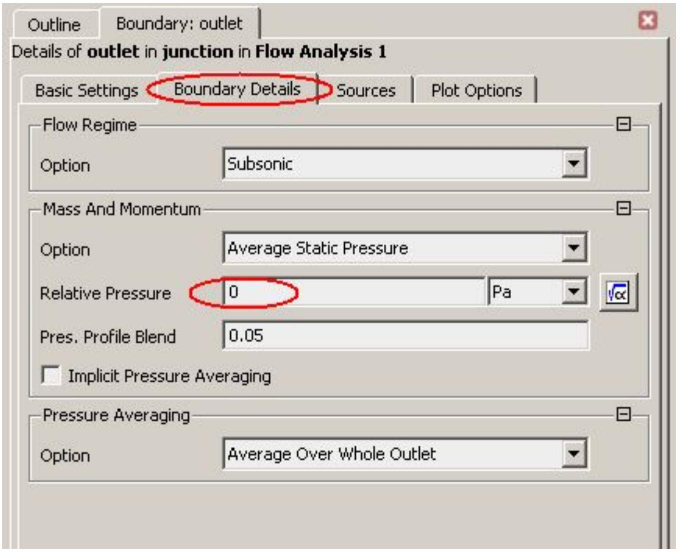
# CFX-Pre – Outlet boundary conditions




1. Insert a boundary named *outlet*
2. Set the *Boundary Type* to *Outlet*
3. Set *Location* to *outlet*
4. Click the *Boundary Details* tab
5. Set *Relative Pressure* to *0 [Pa]*
   - This is relative to the domain Reference Pressure, which is 1 [atm]
6. Leave all other settings at their default values

The Average Static Pressure boundary condition allows pressure to float locally on the boundary while preserving an specified average pressure. If “Pressure” had been chosen a fixed Pressure would be applied at every nodal location on the outlet boundary

7. Click *OK*

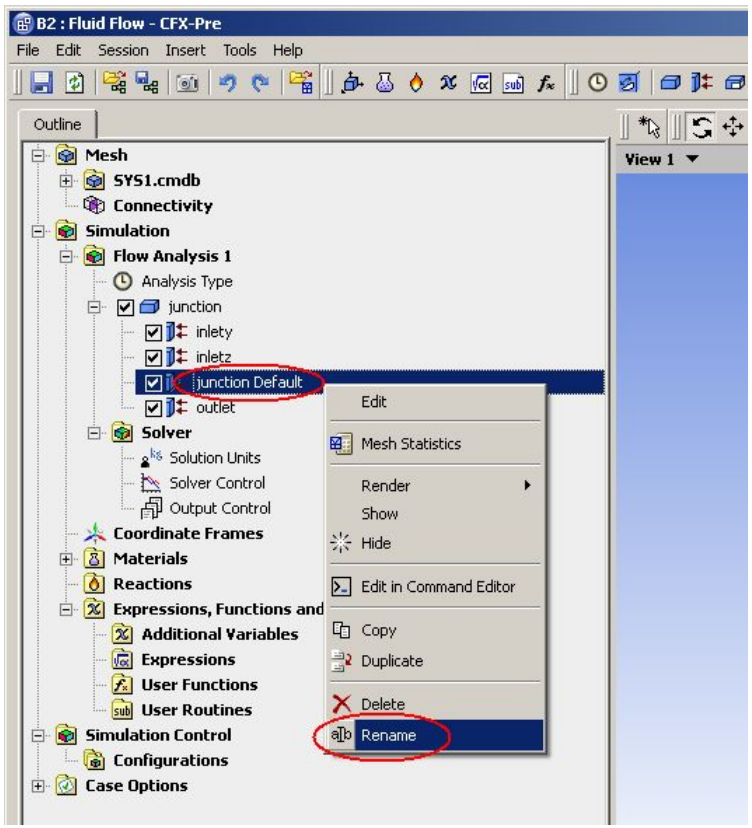


# CFX-Pre – Wall boundary conditions

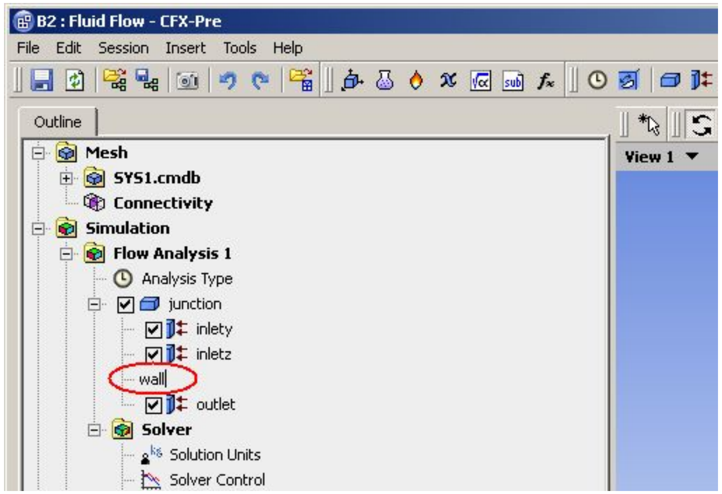


The default boundary condition (*junction Default* in this case) comprises of all the 2-D regions not yet assigned a boundary condition.

1. Right-click *junction Default,* select *Rename* and change the boundary name to *wall*




- The default boundary type is an adiabatic wall and is appropriate here

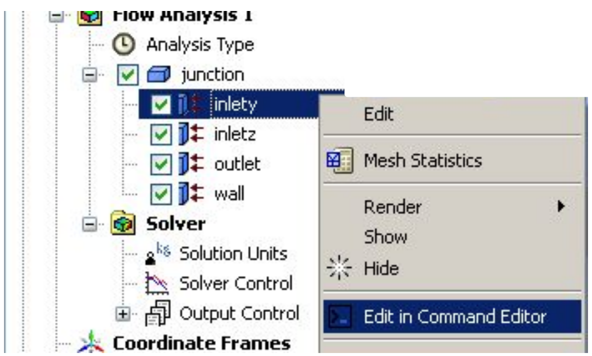


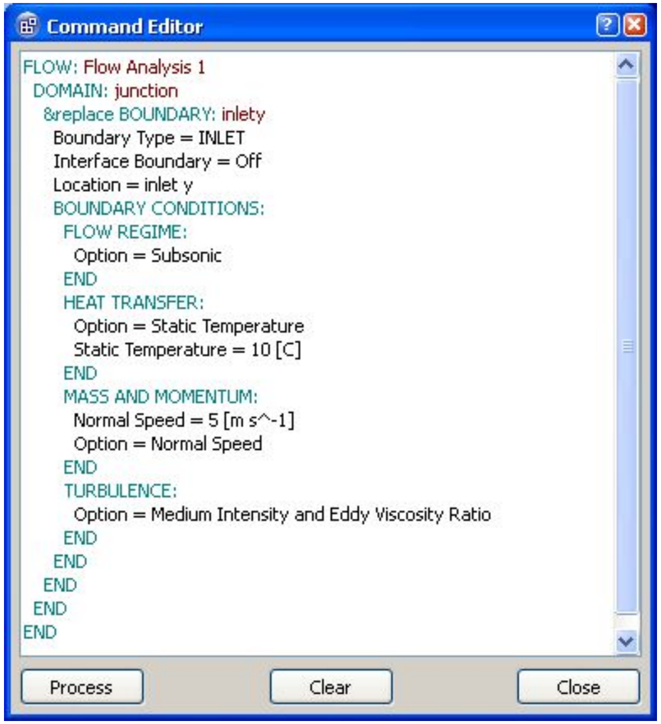
# CCL at a Glance



Before proceeding you will now take a quick look at CCL (CFX Command Language). CCL describes objects in a command language format. You will come across CCL in all CFX modules. Among other things, CCL allows you to perform batch processing and scripting.

1. Right-click on *inlety* and select *Edit in Command Editor*




2. Close the *Command Editor* after taking a quick look at the CCL definition of the Inlet boundary condition

# Initialisation

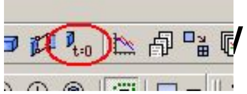


Initial values must be provided for all solved variables. This gives the solver a starting point for the solution. There are two options when setting an initial value for a variable:

- Automatic: This will use a previous solution if provided, otherwise the solver will generate an initial guess based on the boundary conditions
- Automatic with Value: This will use a previous solution if provided, otherwise the value you specify will be used

**The solver generated initial conditions are often good enough as a starting point. However, in some cases you will need to provide a better starting point to avoid solver failure**

Initial conditions can be set on a per-domain basis, or on a global basis.

1. Since you will use Automatic Initial Conditions, there is no need to set any values, but click the *Initialisation* icon  the settings, and then close the form

# Solver Control



The Solver Control options set various parameters that are used by the solver and can affect the accuracy of the results. The default settings are reasonable, but will not be correct for all simulations. In this case the default settings will be used, but you will still look at what those defaults are.

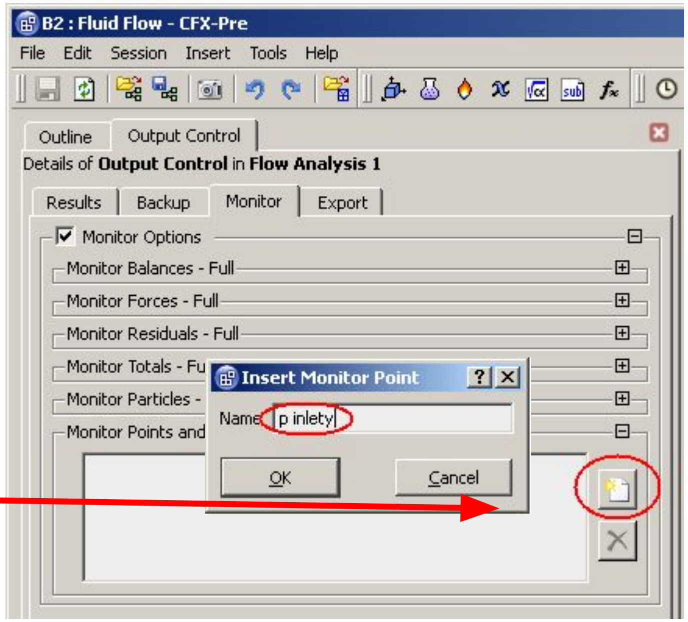
1. Double-click on *Solver Control* from the *Outline* tree
   - The solver will stop after *Max. Iterations* regardless of the convergence level
   - Advection Scheme and Timescale Control will be discussed later
   - Residuals are a measure of how well the posed equations have been solved. In this case the solver will stop when the RMS (Root Mean Squared) residuals have reached 1.E-4. Tighter convergence is achieved with lower residuals.
2. Click *Close*

# CFX-Pre – Monitor points



In all engineering flows, there are specific variables or quantities of interest. Sometimes, these establish themselves in a different way from other variables and do not reach a satisfactory value at the same time as the overall solution converges, so it is always a good idea to monitor them as the solution progresses.  In this simulation, pressure will be monitored at both inlets.

1. Double-click *Output Control* from the Outline tree
2. On the *Output Control* form, select the *Monitor* tab
3. Check the *Monitor Options* box
4. Click the *New* icon
5. Set the Name to *p inlety* and click *OK*



# CFX-Pre – Monitor points (continued)

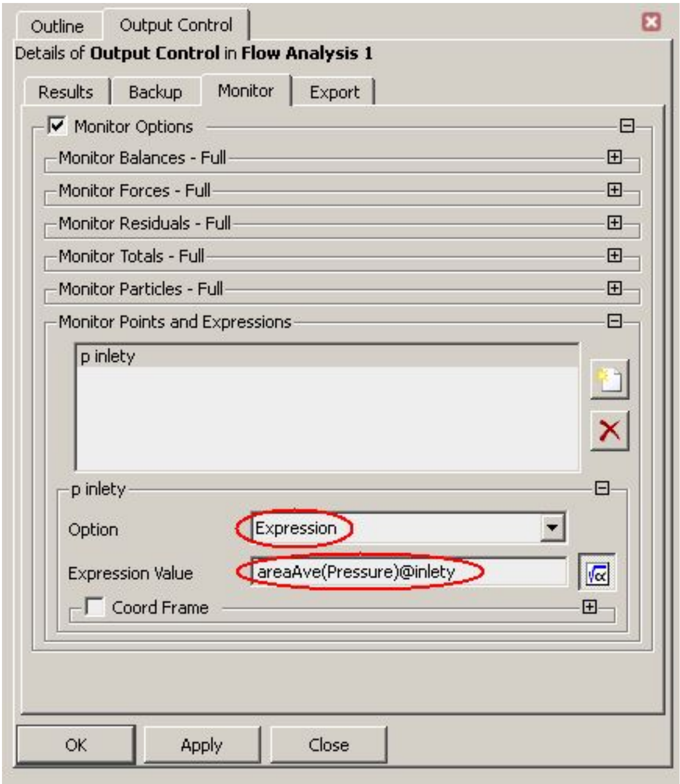


An expression will be used to define the monitor point.

7. Set *Option* to *Expression*
8. Enter the expression:

   **`areaAve(Pressure)@inlety`**

   in the *Expression Value* field




The expression calculates the area weighted average of pressure at the boundary *inlety*.

Note that expressions and expression language will be covered in more detail elsewhere.

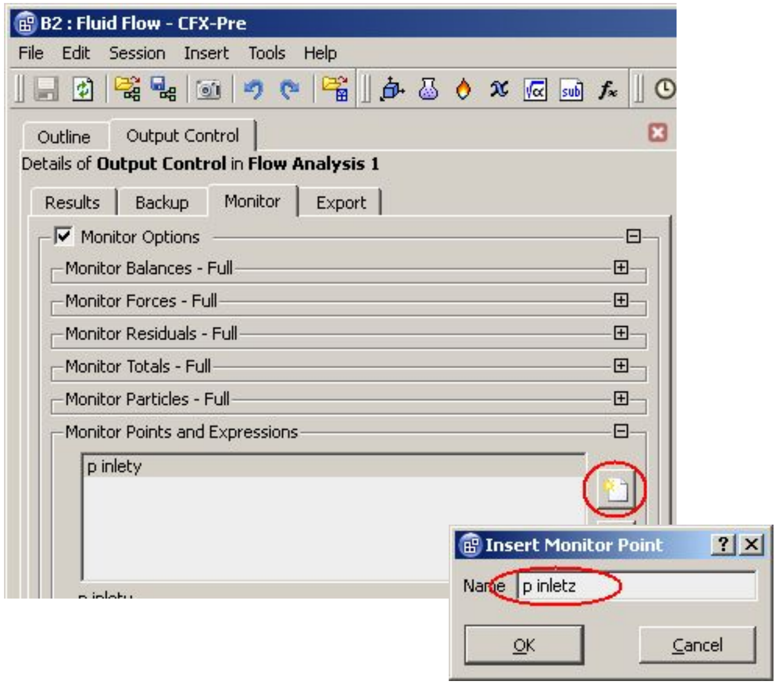
# CFX-Pre – Monitor points (continued)



A second monitor point will be used to monitor the pressure at the second inlet, *inletz*.

9. Click the *New* icon

10. Set the *Name* to *p inletz* and click *OK*

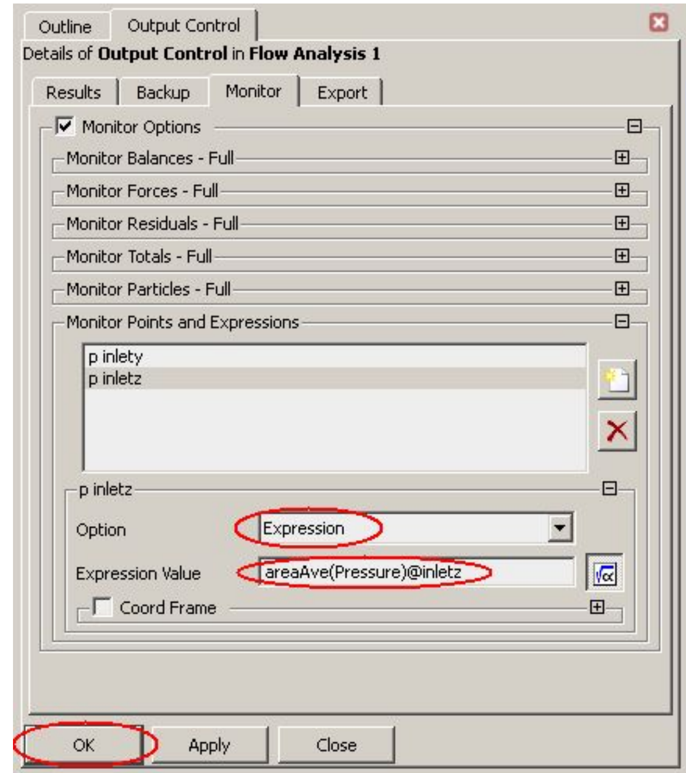


# CFX-Pre – Monitor points (continued)



An expression will be used to define the monitor point:

12. Set *Option* to *Expression*
13. Enter the expression **`areaAve(Pressure)@inletz`** in the *Expression Value* field
14. Click *OK* to apply the settings and close the Output Control form




The expression calculates the area weighted average of pressure at the boundary ‘inletz’.

These monitor points will be utilised during the solution process in a later part of this tutorial.

# Solution Goals



- Launch CFX-Pre from Workbench.
- Use pre-defined materials.
- Define the fluid models in a domain.
- Create and edit objects in CFX-Pre.
- Define boundary conditions.
- Set up monitor points using simple expressions.

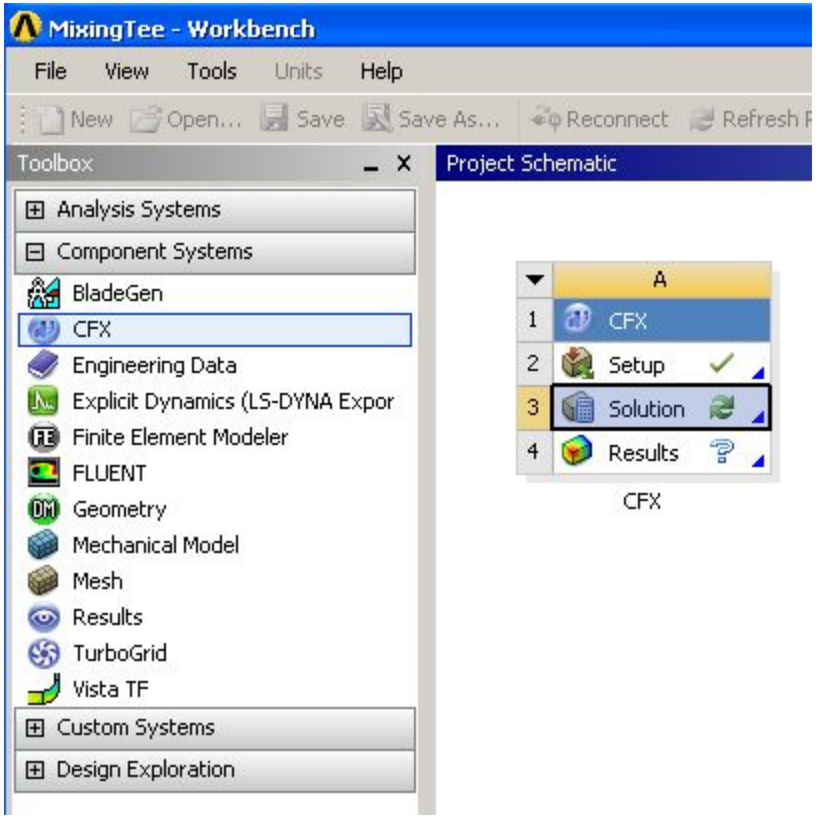
- Launch the CFX Solver Manager from Workbench.
- Monitor convergence.

- Launch CFD-Post from an existing CFX Simulation in Workbench.
- Rotate, zoom and pan the view.
- Create contour plots.
- Create a plane for use as a locator.
- Create a velocity vector plot.
- Use pre-defined views.
- Create streamlines of velocity.
- Create an isosurface, coloured by a separate variable.

# Obtaining a solution



1. Exit CFX-Pre
   – When running in WB the CFX-Pre case will be saved automatically
2. Save the Workbench project
3. In Workbench, double-click *Solution* to launch the CFX Solver Manager

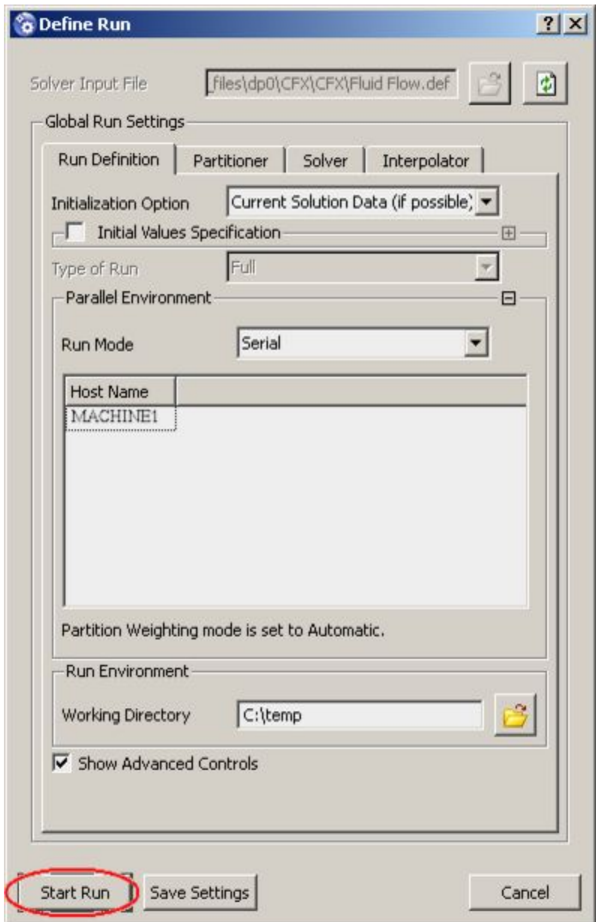


# Obtaining a solution (continued)

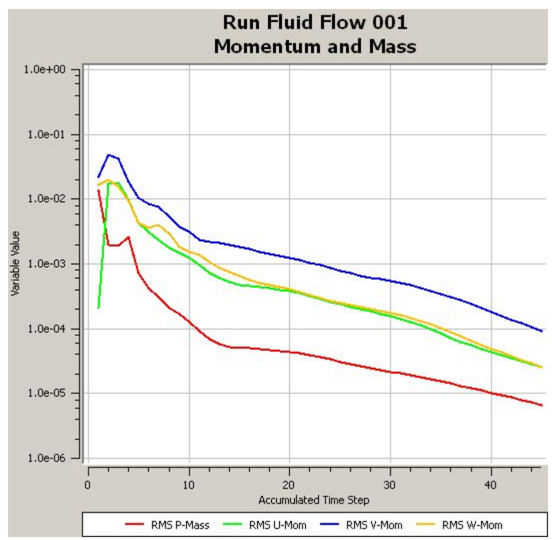


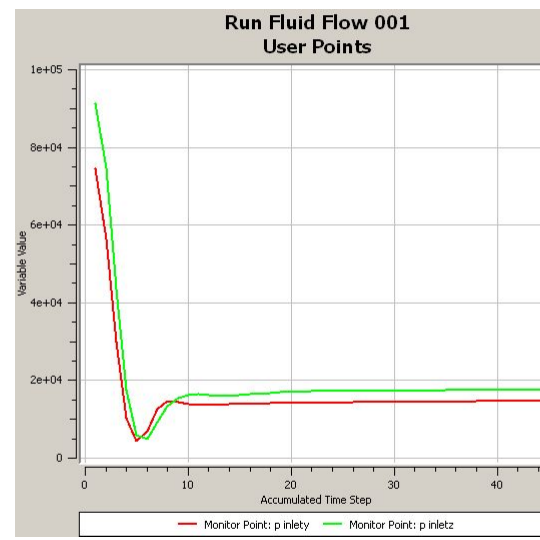
The CFX Solver Manager will start with the simulation ready to run.

3. Click *Start Run* to begin the solution process




- 45 iterations are required to reduce the RMS residuals to below the target of $1.0\text{x}10^{-4}$
- The pressure monitor points approach steady values



# Post-processing Goals



- Launch CFX-Pre from Workbench.
- Use pre-defined materials.
- Define the fluid models in a domain.
- Create and edit objects in CFX-Pre.
- Define boundary conditions.
- Set up monitor points using simple expressions.

- Launch the CFX solver manager from Workbench.
- Monitor convergence.

- **Launch CFD-Post from an existing CFX Simulation in Workbench.**
- **Rotate, zoom and pan the view.**
- **Create contour plots.**
- **Create a plane for use as a locator.**
- **Create a velocity vector plot.**
- **Use pre-defined views.**
- **Create streamlines of velocity.**
- **Create an isosurface, coloured by a separate variable.**

# Launching CFD-Post



1. Exit the CFX Solver Manager
2. Save the project
3. Double click *Results* to launch CFD-Post

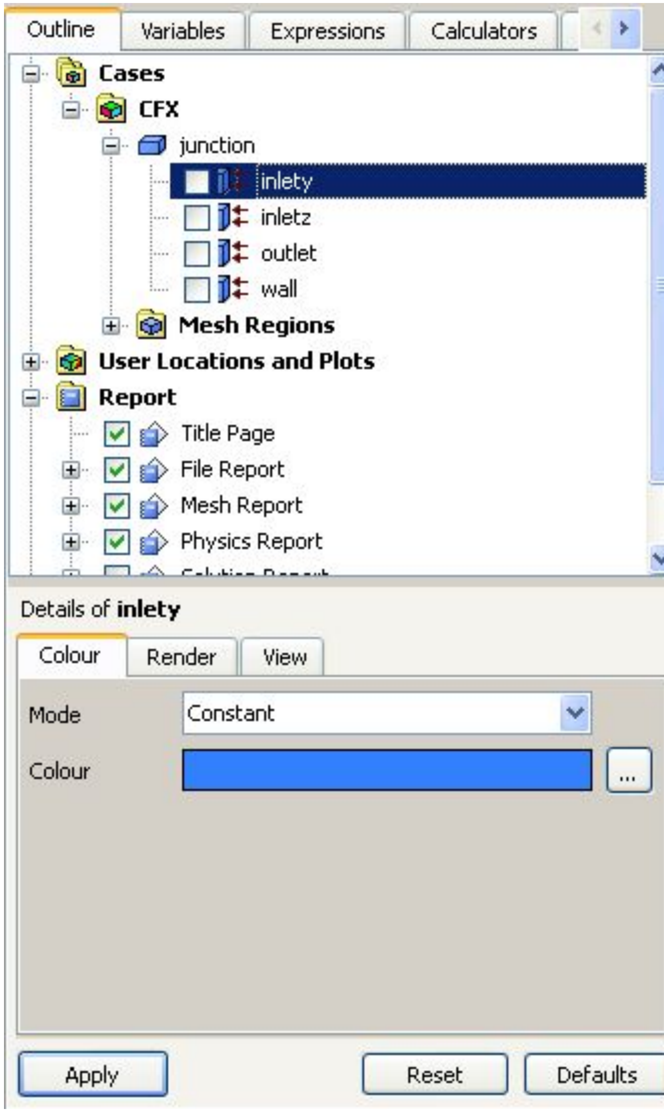
# CFD-Post Overview



When CFD-Post opens, you will see that the layout is similar to CFX-Pre

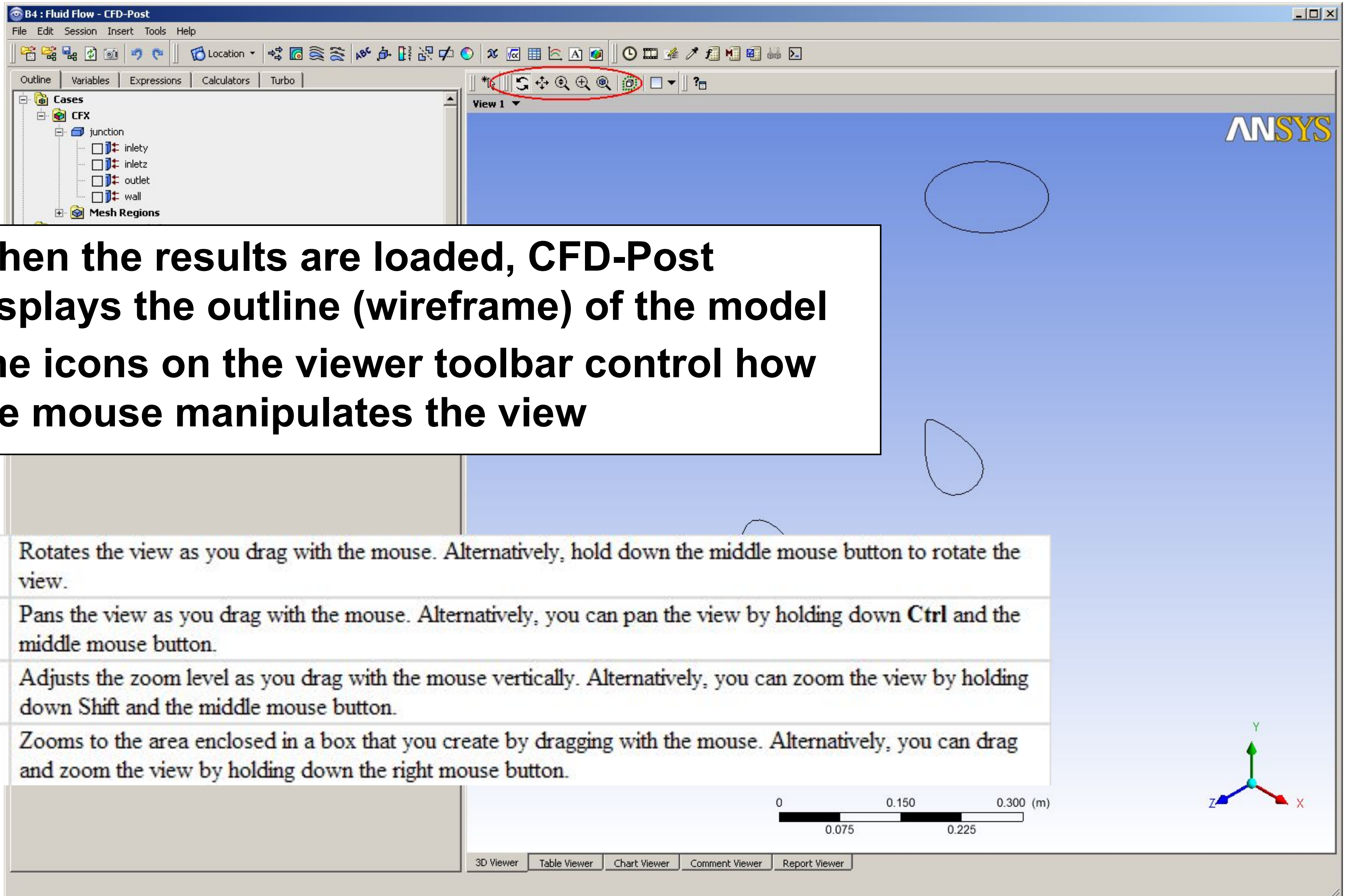
There are two windows on the left side:

- Selector Window
  - Lists currently defined graphics objects. Object for each boundary condition are created automatically
  - Object are edited by double-clicking or right-clicking on the object
  - The check boxes next to each object turn the visibility on or off in the Viewer
- Details Window
  - When you edit an object the Details window shows the current object status



# CFD-Post – Manipulating the view






- **When the results are loaded, CFD-Post displays the outline (wireframe) of the model**
- **The icons on the viewer toolbar control how the mouse manipulates the view**

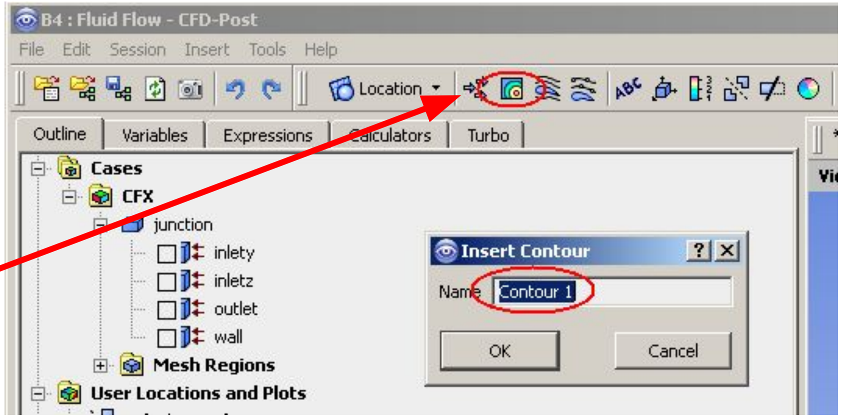
| | |
|---|---|
| | Rotates the view as you drag with the mouse. Alternatively, hold down the middle mouse button to rotate the view. |
| | Pans the view as you drag with the mouse. Alternatively, you can pan the view by holding down **Ctrl** and the middle mouse button. |
| | Adjusts the zoom level 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. |

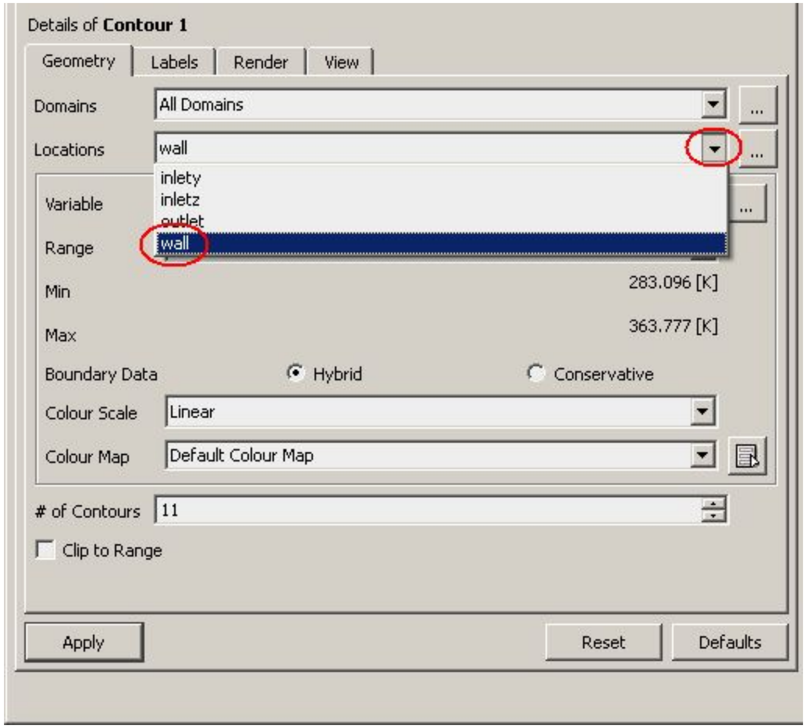
# CFD-Post – Temperature contour plot



In the first step, you will plot contours of temperature on the exterior walls of the model

1. Click the *Contour* icon from the toolbar
2. Click *OK* to accept the default name *Contour 1*
3. Set *Locations* to *wall*

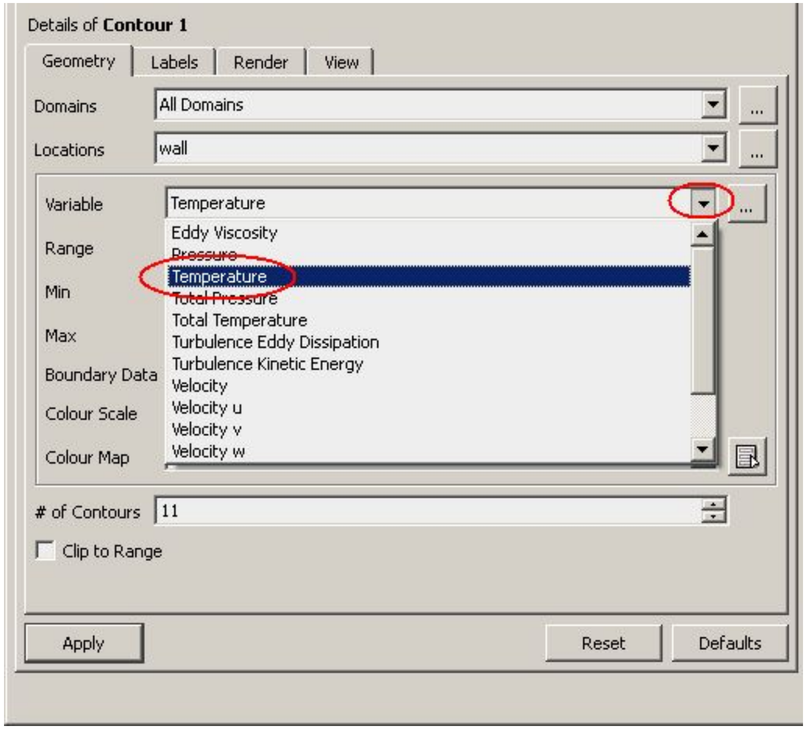


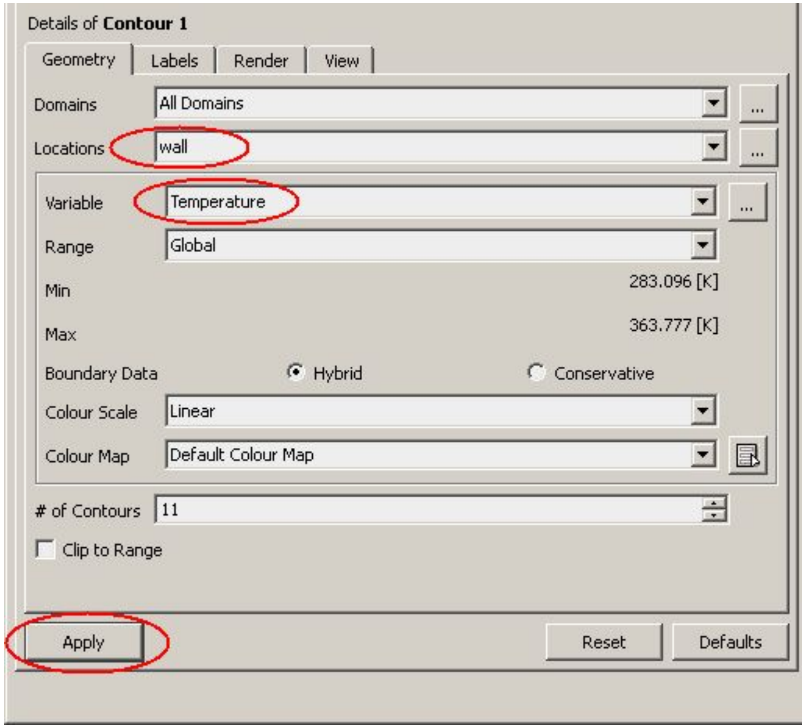



# CFD-Post – Temperature contour plot (contd.)



4. Set the *Variable* to *Temperature*
   - The drop-down menu provides a list of common variables. Use the "…" icon to access a full list
5. Leave the other settings unchanged
6. Click *Apply* to generate the plot



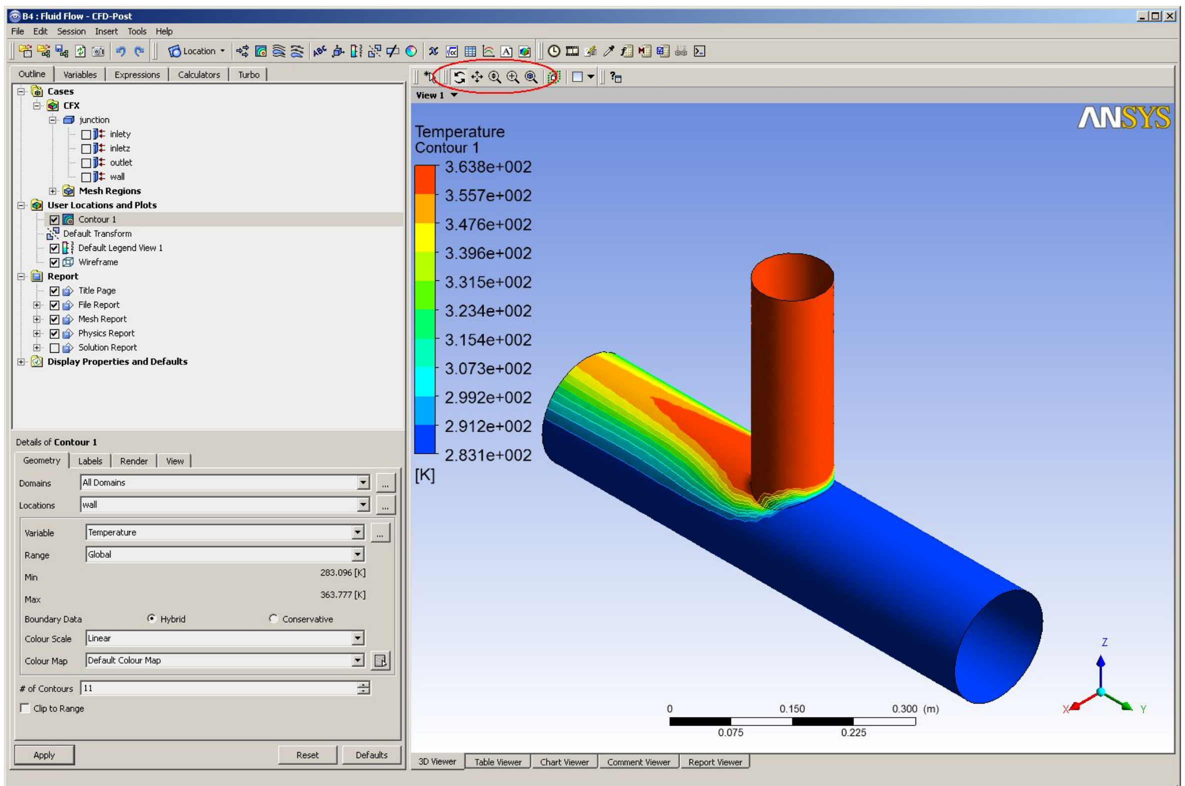



# CFD-Post - Temperature contour plot (contd.)



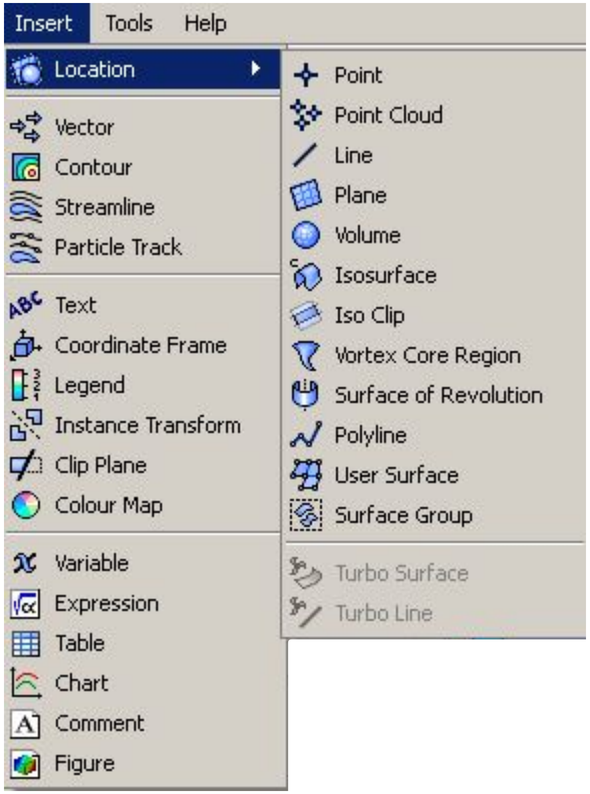
A temperature contour plot on the walls should now be visible.

7. Try changing the view using rotate, zoom and pan. You may find it easier to use the middle mouse button in combination with <Ctrl> and <Shift>
8. Also try clicking on the axes in the bottom right corner of the Viewer



You can create many different objects in CFD-Post. The *Insert* menu shows a full list, but there are toolbar shortcuts for all items. Some common object are:

- Location: Points, Lines, Planes, Surfaces, Volumes
- Vector Plots
- Contour Plots
- Streamline Plots
- Particle Track (if enabled in CFX-Pre)




For turbo machinery cases there are additional objects available that will be discussed later.

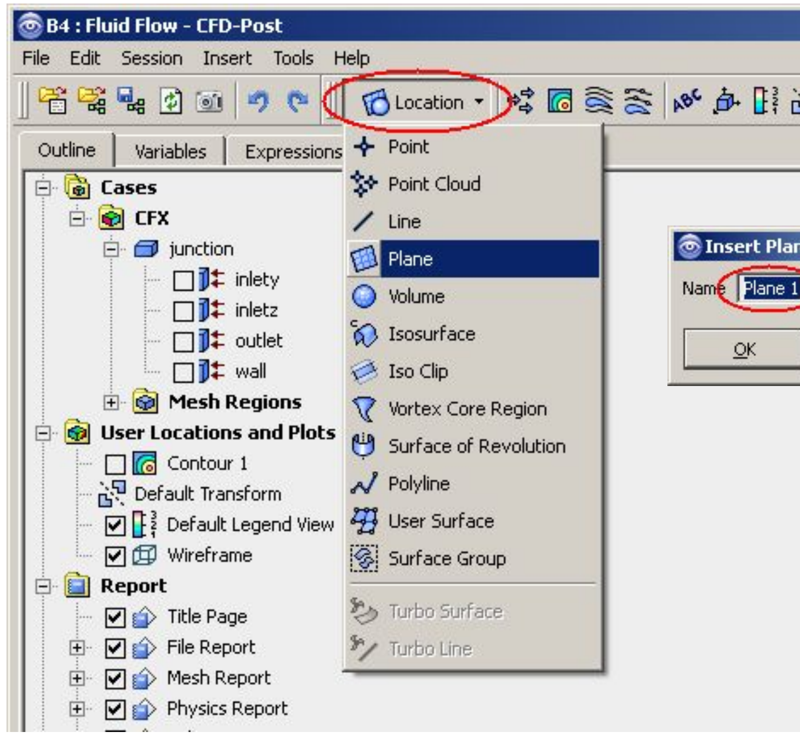
# CFD-Post – Creating a plane at x = 0

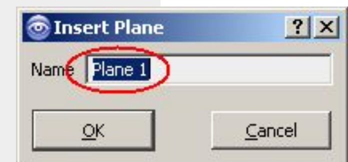


1. First, hide the previously created contour plot, by un-checking the associated box in the tree view


2. Click the *Location* button on the toolbar and select *Plane* from the drop-down menu

3. Click *OK*, accepting the default name of *Plane 1*

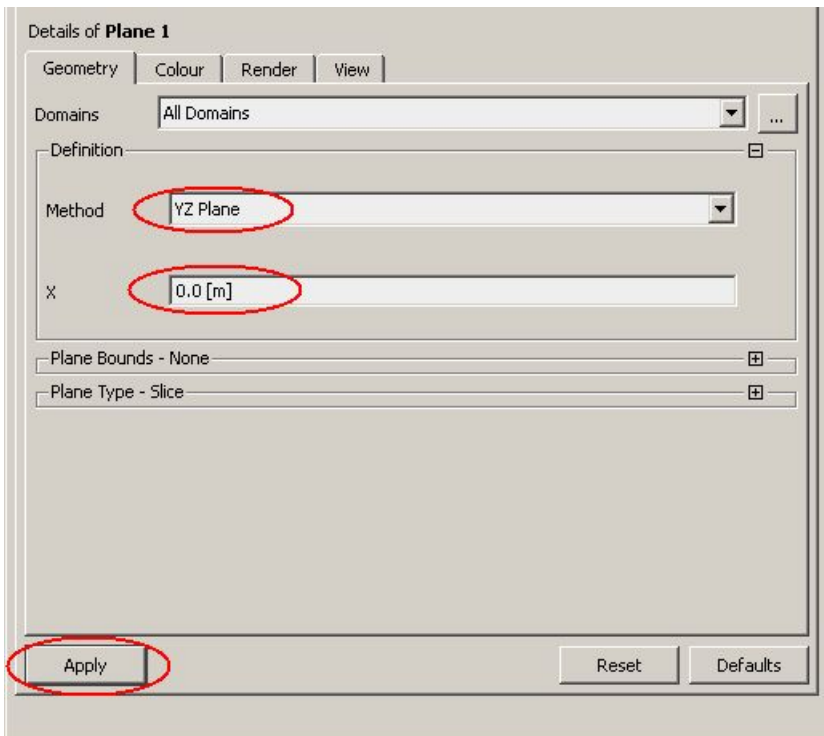


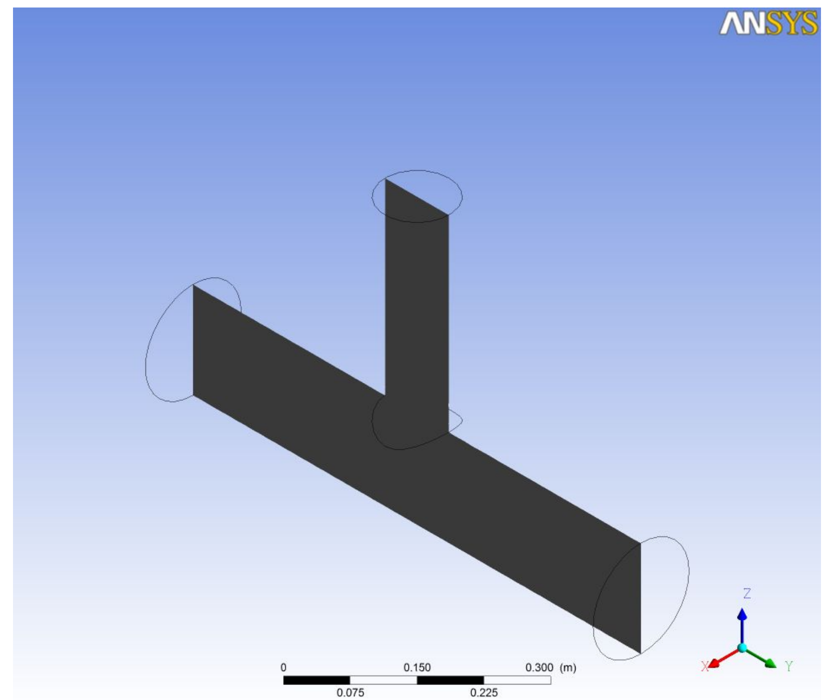



# CFD-Post – Creating a plane at x = 0 (contd.)



4. Set *Method* to *YZ Plane*
5. Leave *X* set to *0 [m]*
6. Click *Apply* to generate the plane



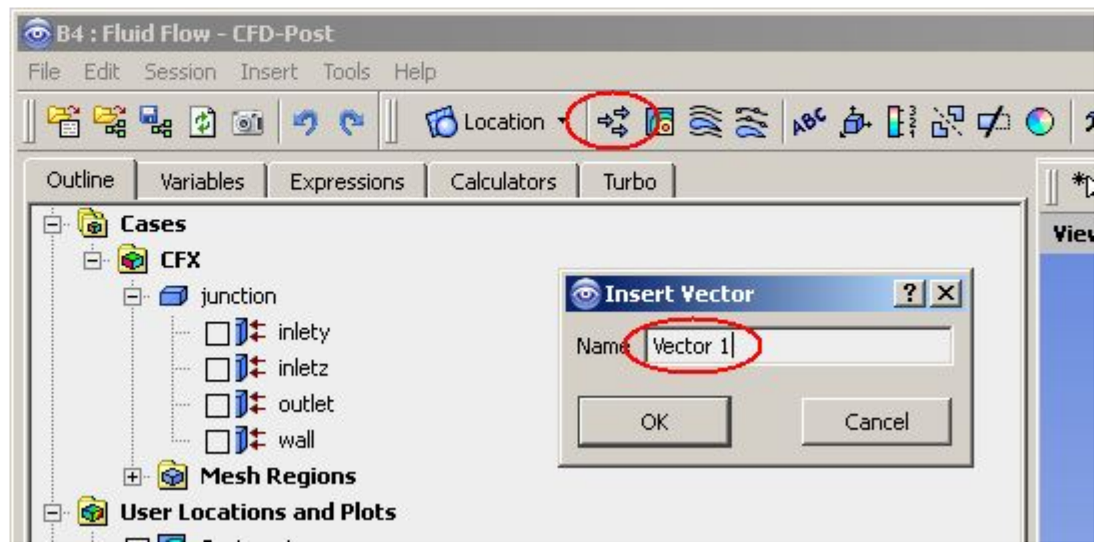



# CFD-Post – Creating a velocity vector plot



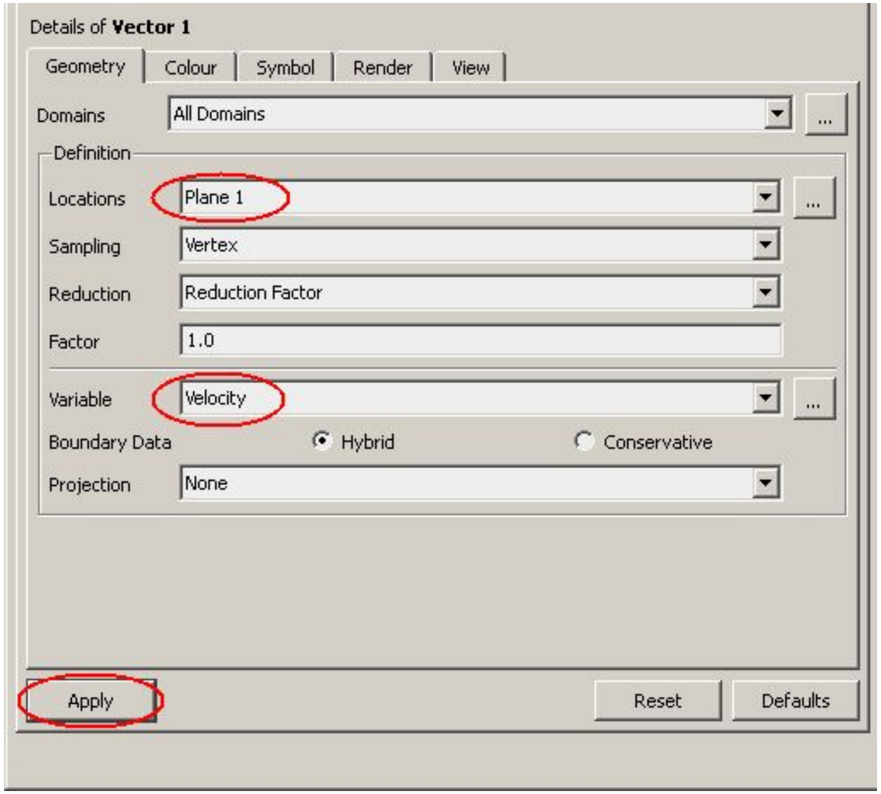
While planes can be coloured by variables, in this case the plane will be used only as a locator for a vector plot.

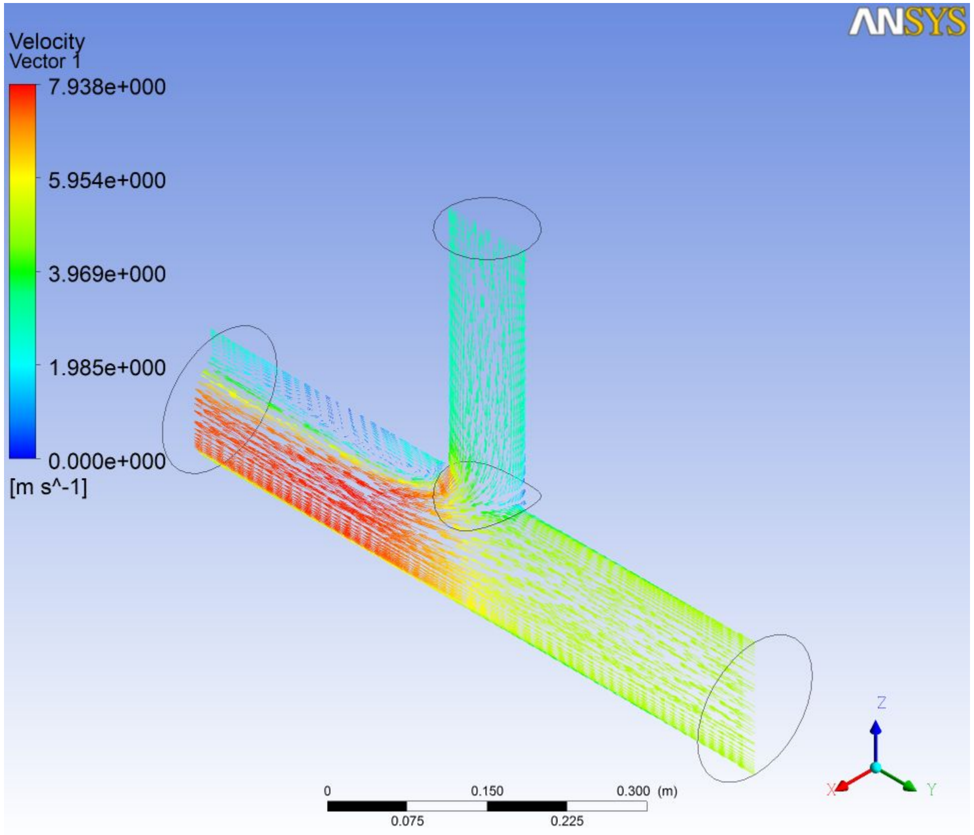

1. Hide the plane by un-checking the associated box in the tree view
2. Click the *Vector* icon from the toolbar
3. Click *OK*, accepting the default name of *Vector 1*



# CFD-Post – Velocity vector plot (continued)

4. Set *Locations* to *Plane 1*
5. Leave the *Variable* field set to *Velocity*
6. Click *Apply*



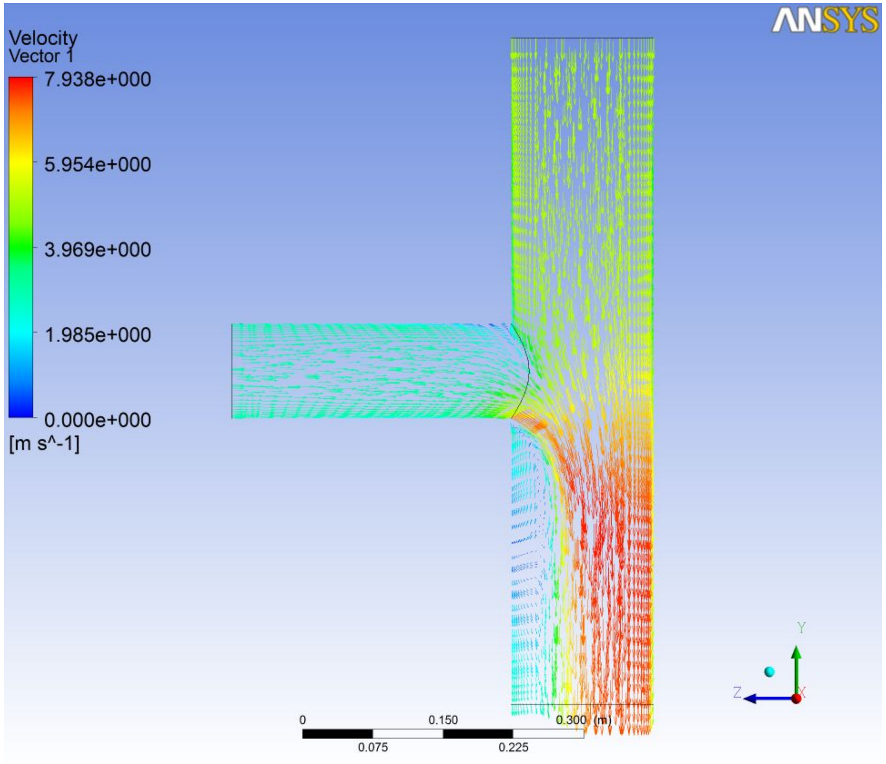



# CFD-Post – Aligning the view



Given that the vector plot is on a 2-D Y-Z plane, you might want to view the plot normal to that axis (i.e. aligned with the X axis).

7. Click on the red x-axis in the bottom right corner of the Viewer to orientate the view



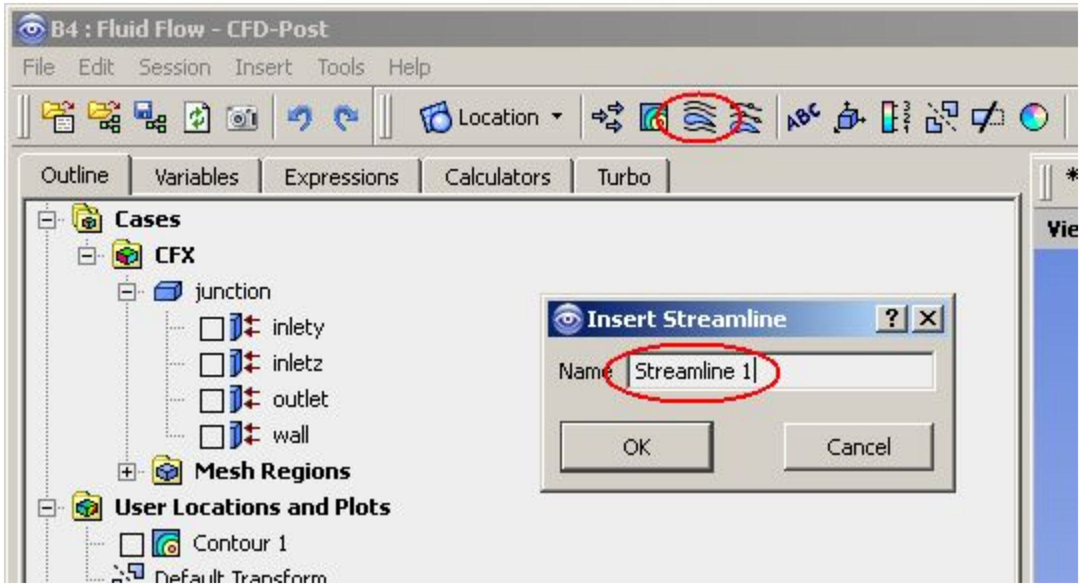
# CFD-Post – Creating velocity streamlines



1. Hide the previously created vector plot, by un-checking the associated box in the tree view


2. Click the *Streamline* icon from the toolbar

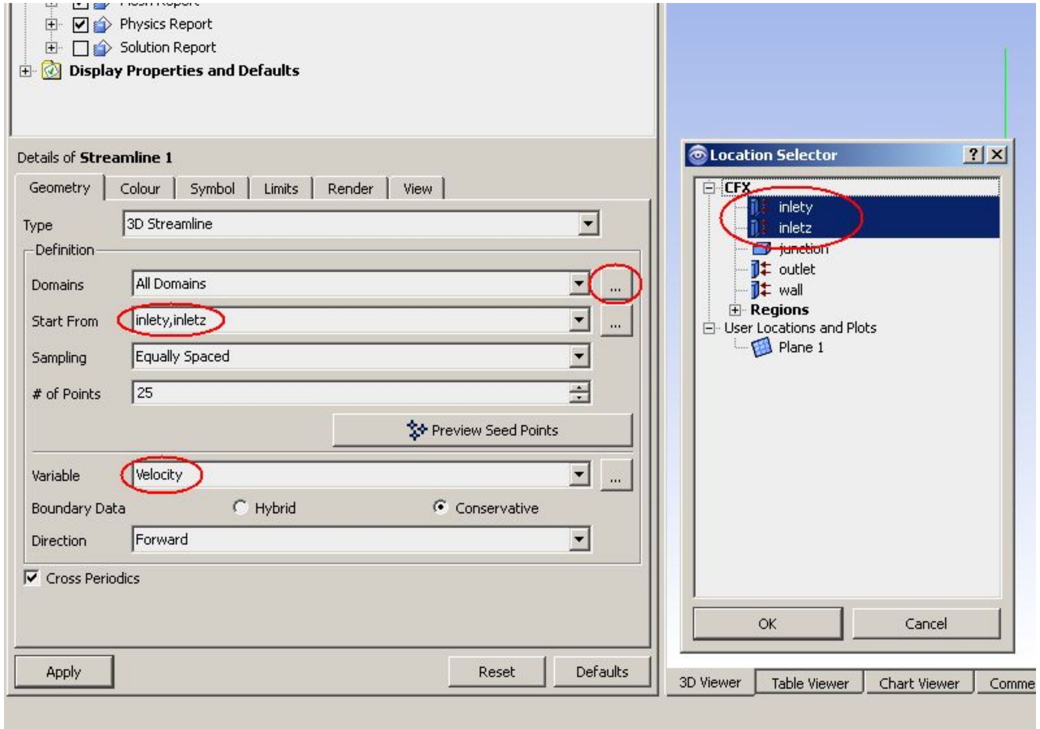
3. Click *OK*, accepting the default name of *Streamline 1*



# CFD-Post – Velocity streamlines (continued)



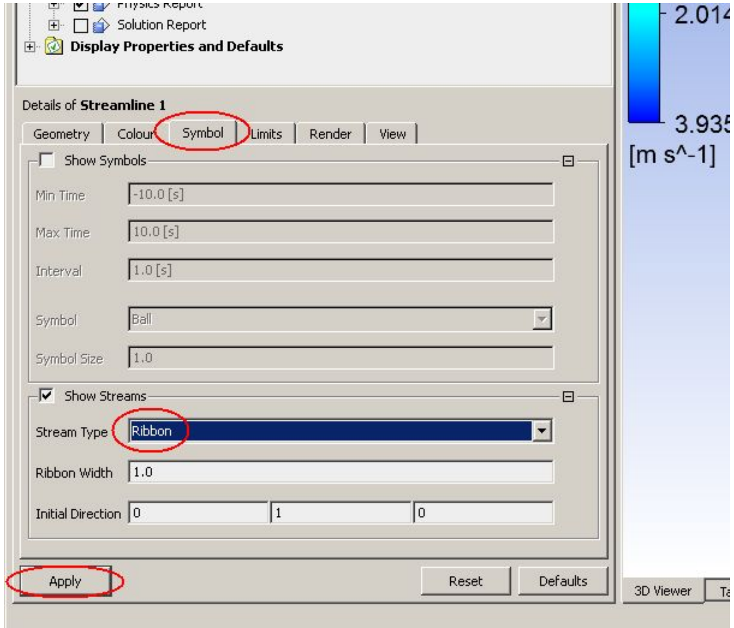
4. In the *Start From* field, select both *inlety* and *inletz*. Use the '…' icon to the right of the field and select both locations using the CTRL key.
5. Leave the *Variable* field set to *Velocity*

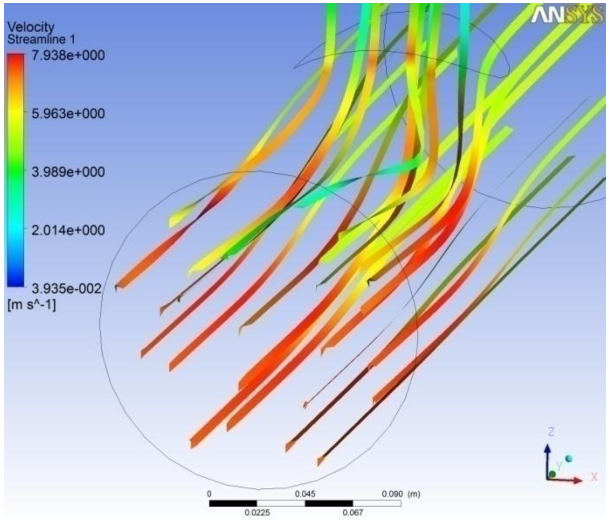


# CFD-Post – Velocity streamlines (continued)



6. Click the *Symbol* tab
7. Change the *Stream Type* to *Ribbon*
8. Click *Apply*
9. Examine the streamlines from different views using rotate, zoom and pan
   - The ribbons give a 3-D representation of the flow direction
   - Their colour indicates the velocity magnitude
   - Velocity streamlines may be coloured using other variables e.g. temperature

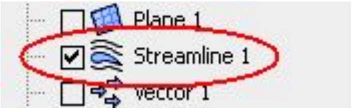


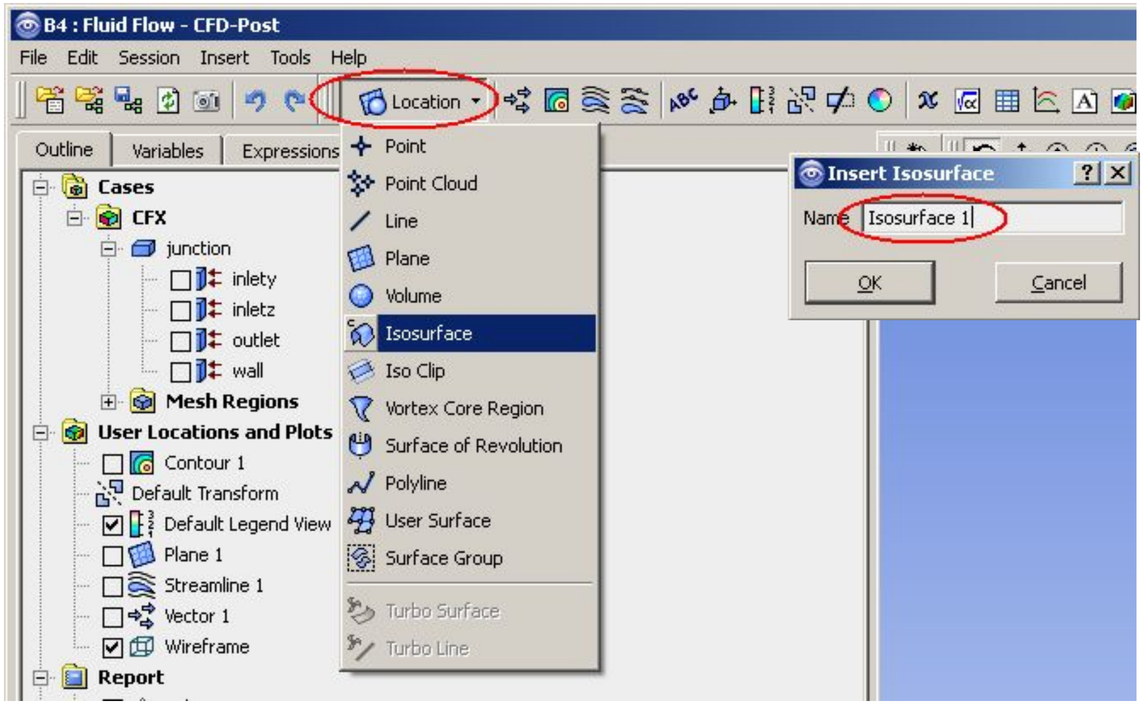
# CFD-Post – Creating a velocity isosurface



1. Hide the previously created streamlines, by un-checking the associated box in the tree view



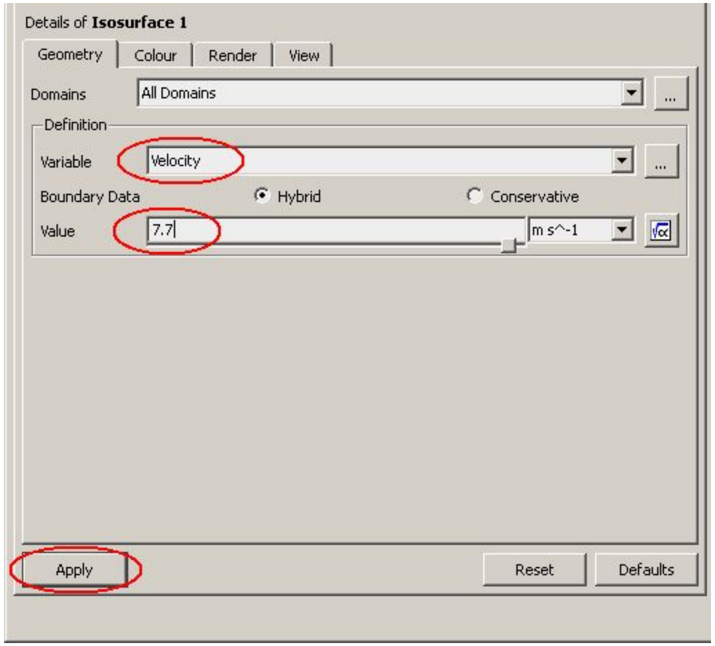

2. Click the *Location* button on the toolbar and select *Isosurface* from the drop-down menu
3. Click *OK*, accepting the default name of *Isosurface 1*

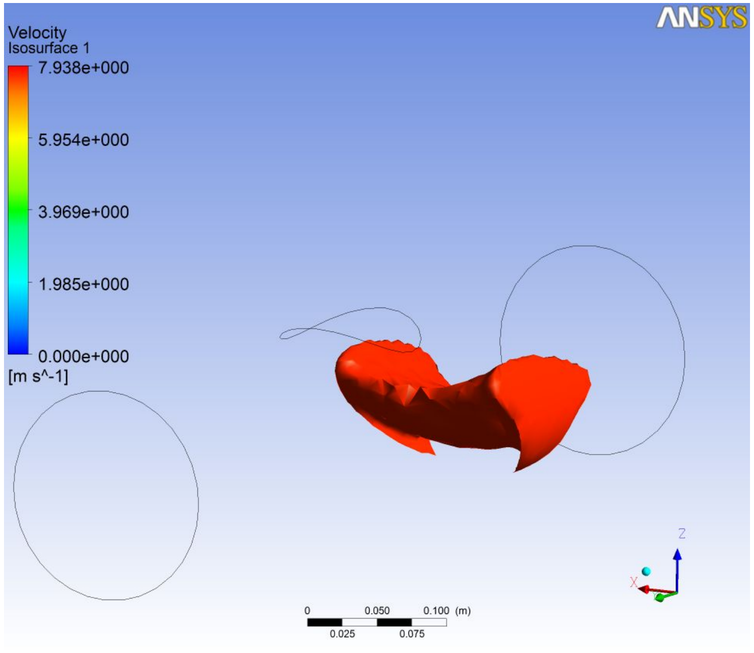


# CFD-Post – Velocity isosurface (continued)



4. Set the *Variable* to *Velocity* (magnitude used in this context)
5. Enter a value of *7.7 [m s^-1]* in the *Value* field (note: there is nothing special about this value – other values can be tried)
6. Click *Apply*
   - The speed is > 7.7 m/s inside the isosurface and < 7.7 m/s outside. Isosurfaces in general are useful for showing pockets of highest velocity, temperature, turbulence, etc.



# CFD-Post – Velocity isosurface (continued)



By default, an isosurface is coloured by the variable used to create it (speed in this case), but a different variable can be used.

7. Click the *Colour* tab
8. Set the *Mode* to *Temperature*
9. Set the *Range* to *Local*
10. Click *Apply*