ANSYS®

**Workshop 01
Mixing Tee**

14. 5 Release

Fluid Dynamics   Structural Mechanics   Electromagnetics   Systems and Multiphysics

**Introduction to ANSYS
CFX**

Welcome!

- **Introductory tutorial for CFX**

- **Starting from existing mesh**
  - generated in earlier tutorial during the DM / Meshing session

- **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.
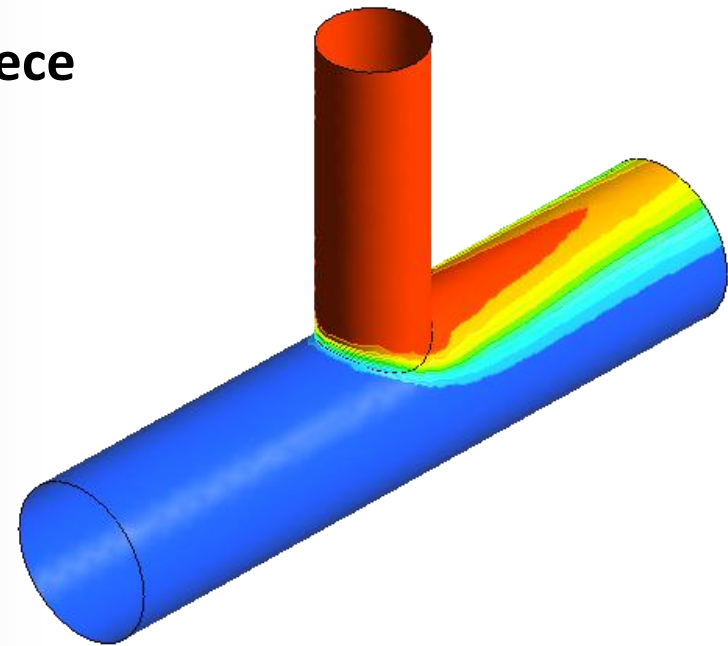
| Introduction | Setup | Solving | Postprocessing | Summary |

Goals

- **Launch CFX-Pre from Workbench**

- **Use pre-defined materials**

- **Define the fluid models in a domain**

- **Create & 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**

| Introduction | Setup | Solving | Postprocessing | Summary |

Start in Workbench

- **Start Workbench**

- **From the *Analysis Systems* toolbox, drag a *Fluid Flow (CFX)* system into the *Project Schematic***
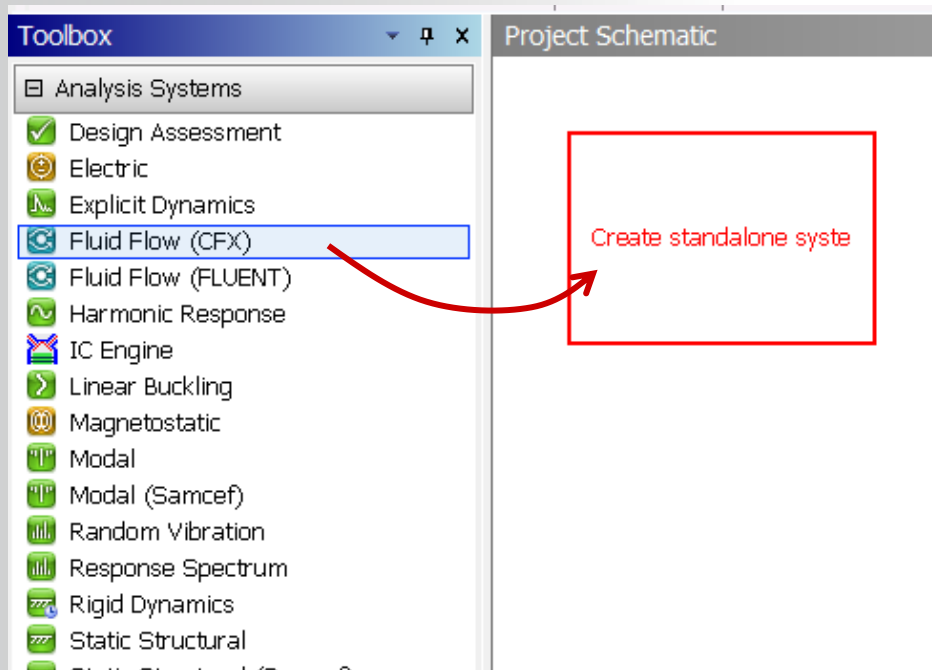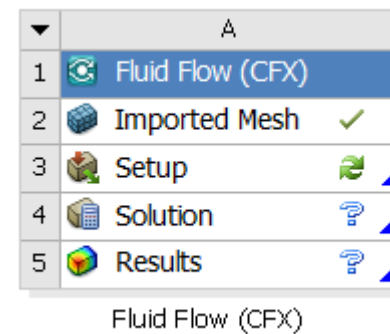
Start a CFX case

- **Right-click on *Mesh* and select *Import Mesh File***
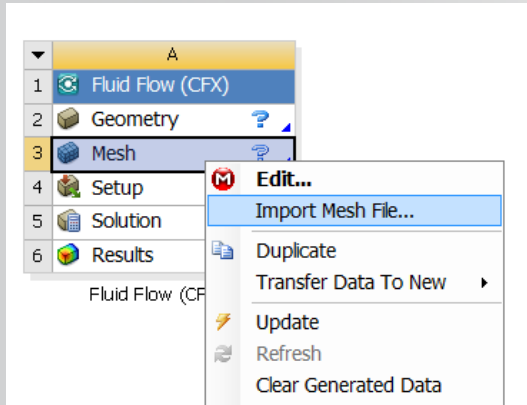
- **Select *mixing_tee.msh (workshop_input_files\WS_01_Mixing Tee)***
  - Note: Change the filter on the right side of the file selection box to read *Fluent Files (*.cas,*.msh)*

- **You can now double-click *Setup* to open *CFX-Pre***

Fluid Flow (CFX)

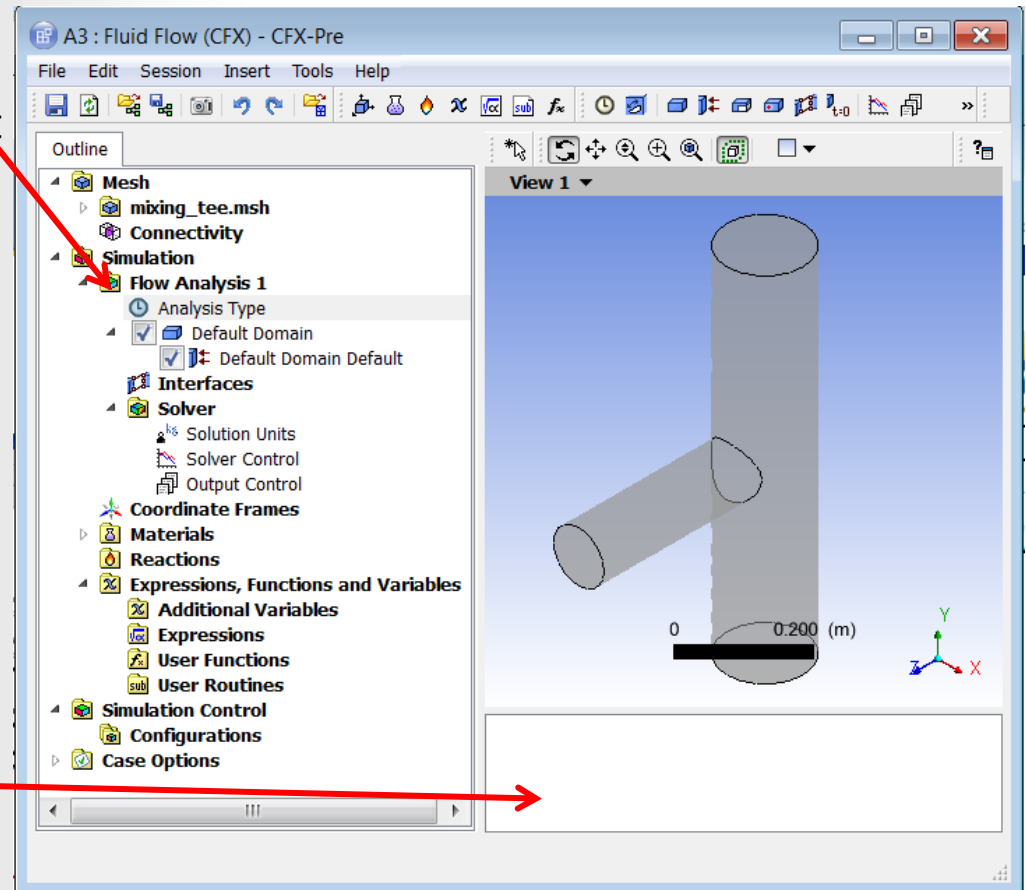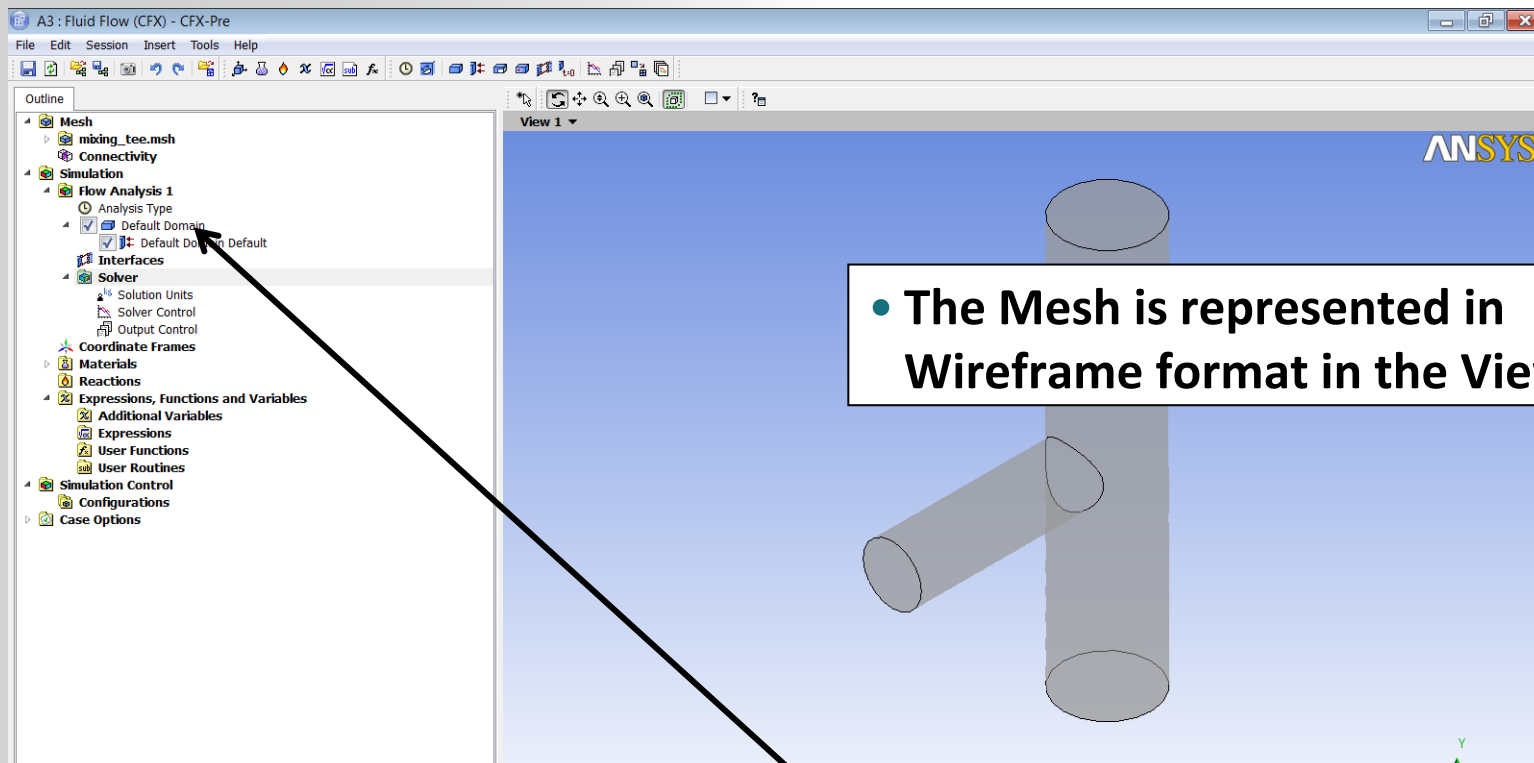| Introduction | Setup | Solving | Postprocessing | Summary |

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

| Introduction | Setup | Solving | Postprocessing | Summary |

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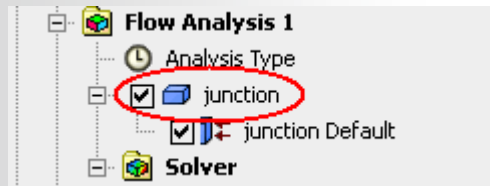
The first step is to change the domain name to something more meaningful

- **Right-click on *Default Domain* in the *Outline* tree**

- **Select *Rename***
  - **The domain name can now be edited**

- **Change the domain name to *junction***
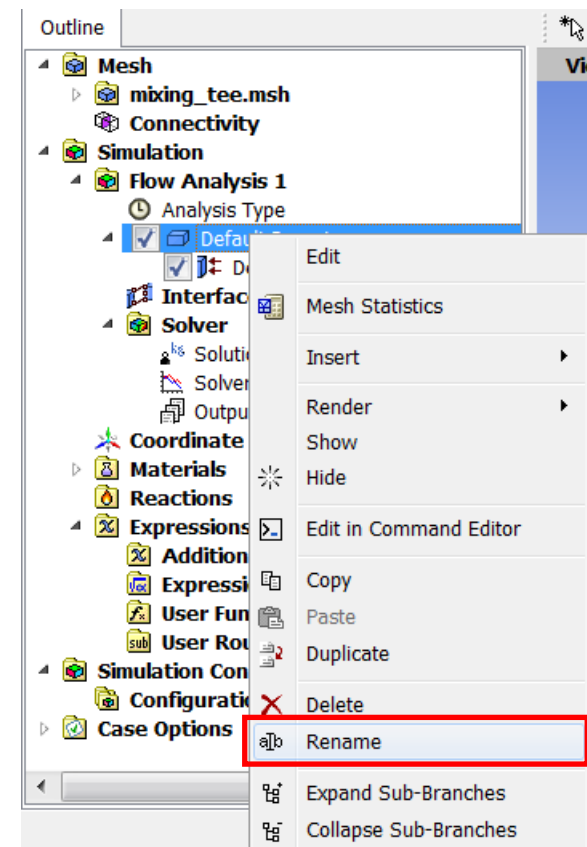
CFX-Pre – Domain settings (continued)

- **Double-click on the domain *junction***

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

- **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.
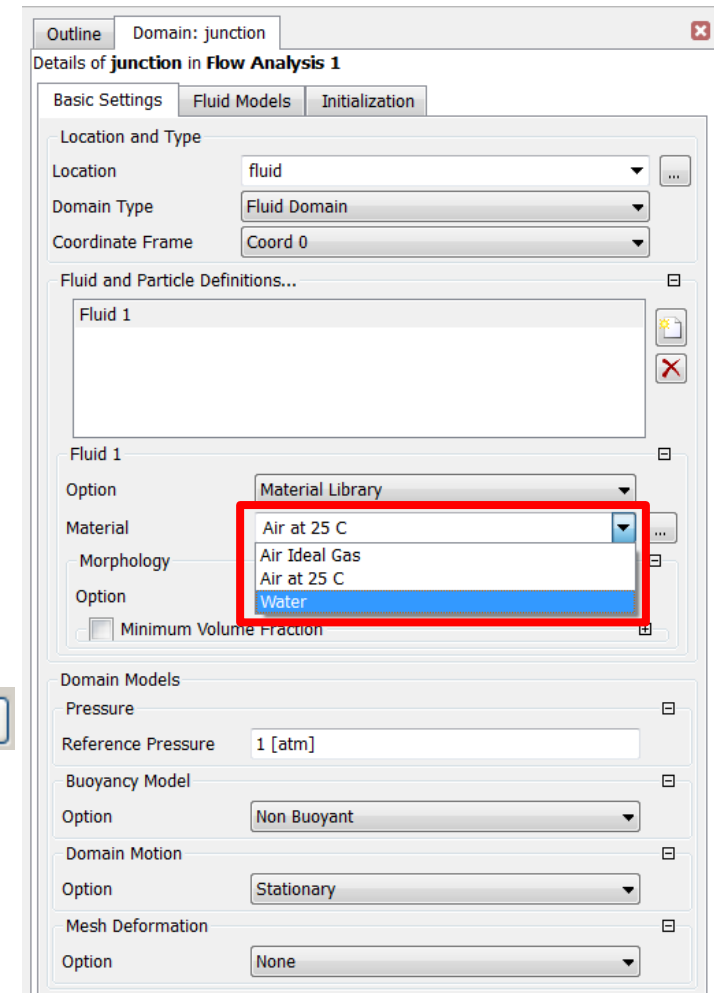
CFX-Pre – Domain settings (continued)

- **Click the *Fluid Models* tab**

- **In the *Heat Transfer* section, change *Option* to *Thermal Energy***
  - Heat transfer will be modelled. This model is suitable for incompressible flows

- **Leave all other settings as they are**
  - The k-Epsilon turbulence model will be used, which is the default

- **Click *OK* to apply the new settings and close the domain form**
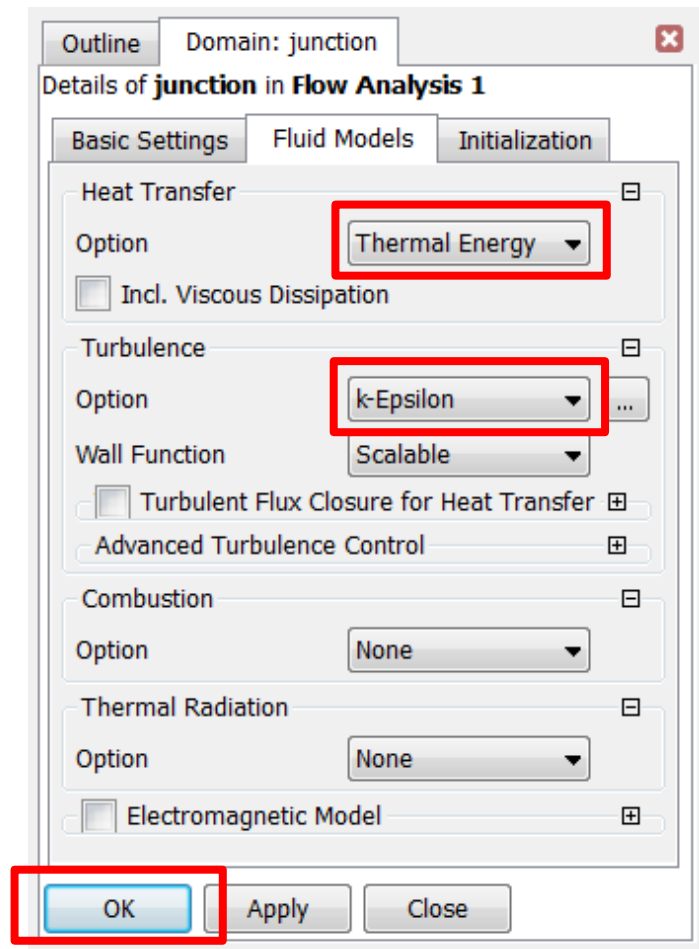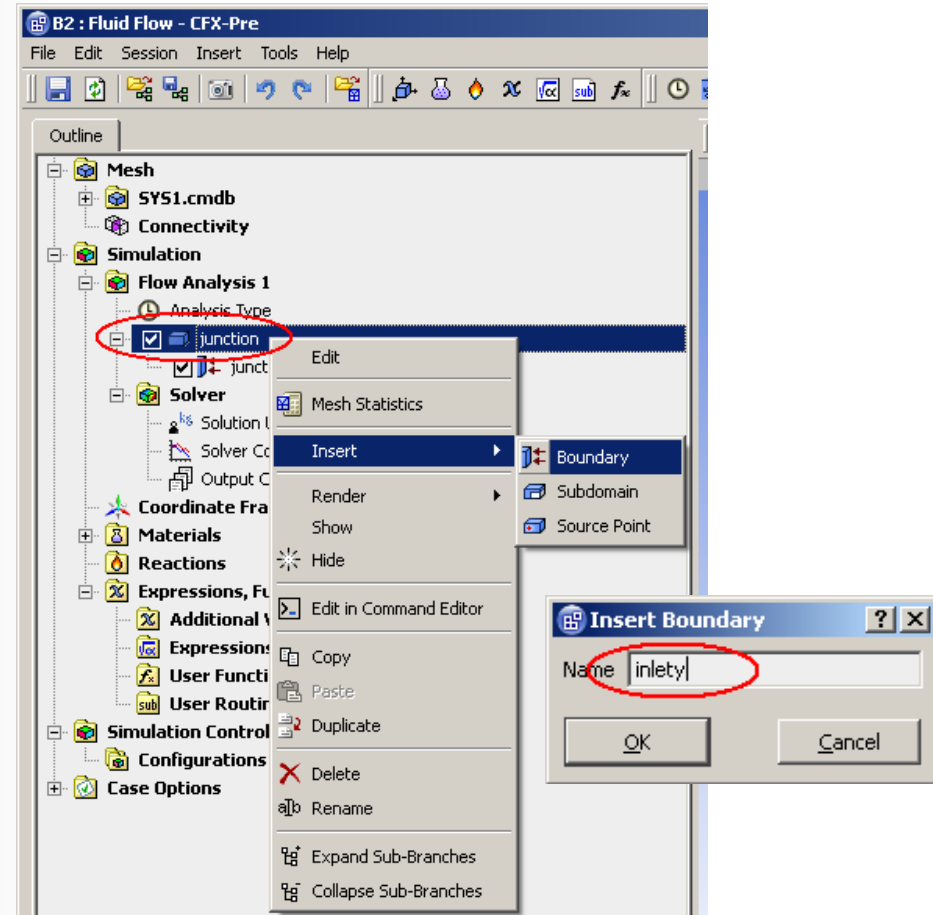
Boundary Conditions

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.
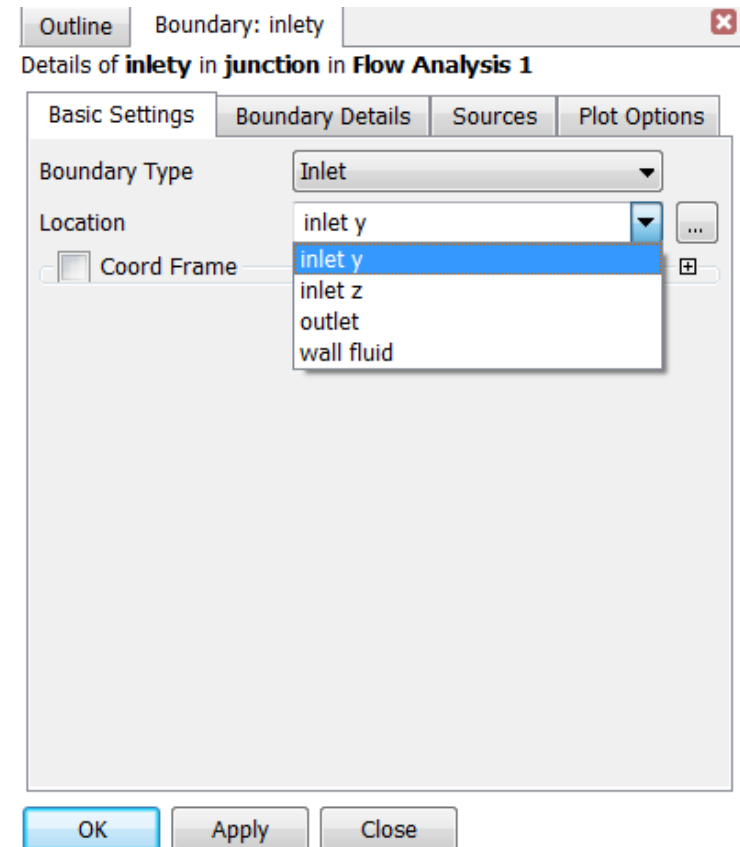
CFX-Pre – Inlet boundary conditions

Now that the domain is set up, boundary conditions can be added

- **Right-click on the *junction* domain**

- **Select *Insert > Boundary***

- **Set the *Name* to *inlety***

- **Click *OK***
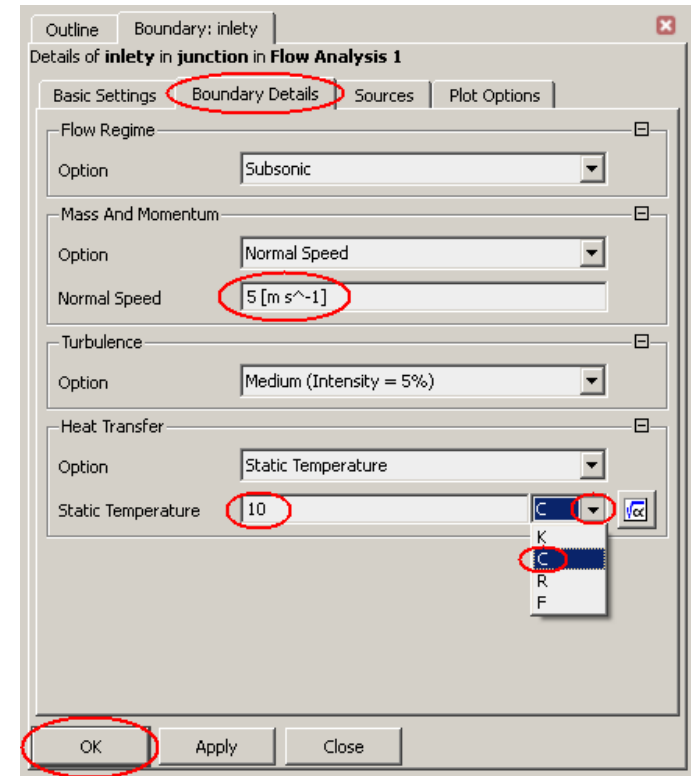
CFX-Pre – Inlet boundary conditions

- **Leave the *Boundary Type* field set to *Inlet***

- **Set *Location* to *inlet y***
  - The available locations can be found in the drop-down menu
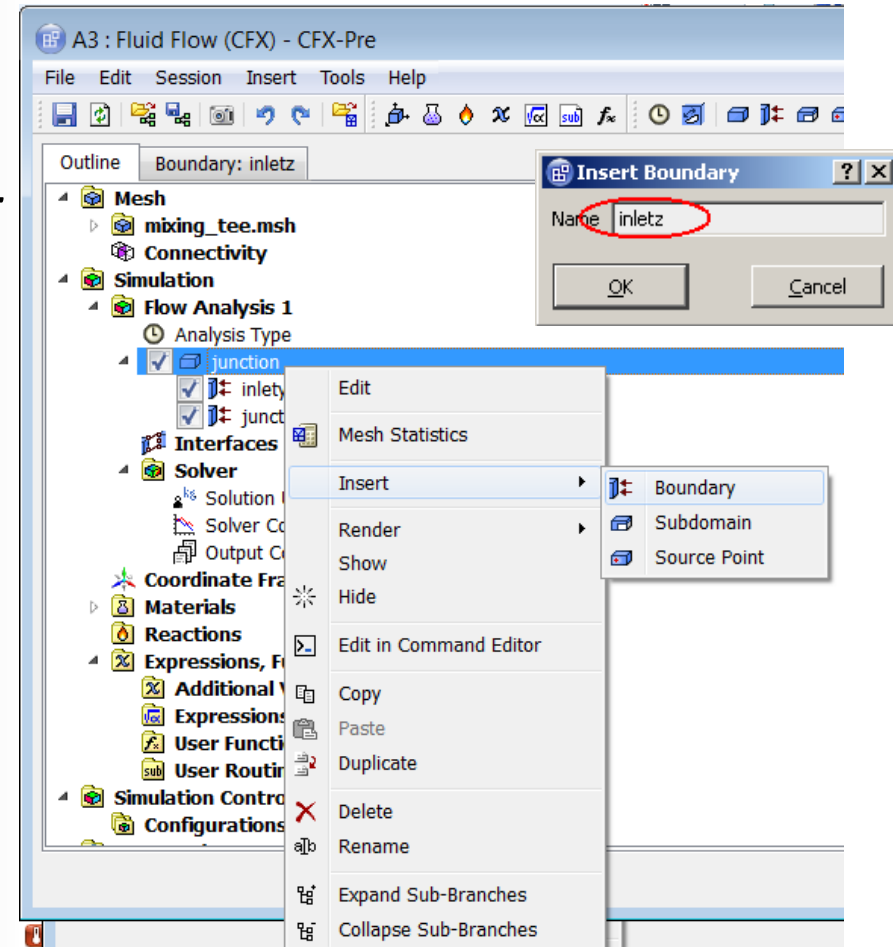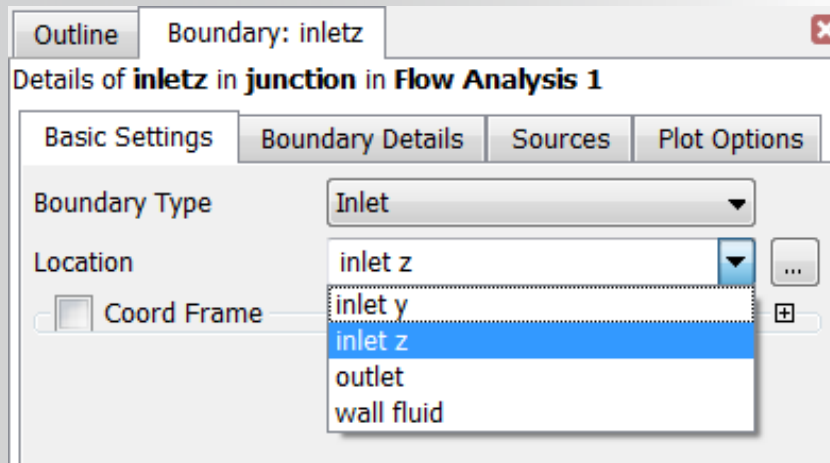
CFX-Pre – Inlet boundary conditions

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

- **Click the *Boundary Details* tab**

- **Enter a value of 5 for *Normal Speed*. The default units are [m s^-1]**

- **Enter a value of 10 for *Static Temperature*.**
  - You will need to change the units

- **Use the drop-down menu to the right of the field to change the units to C (Celsius)**

- **Click *OK* to apply the boundary and close the form**
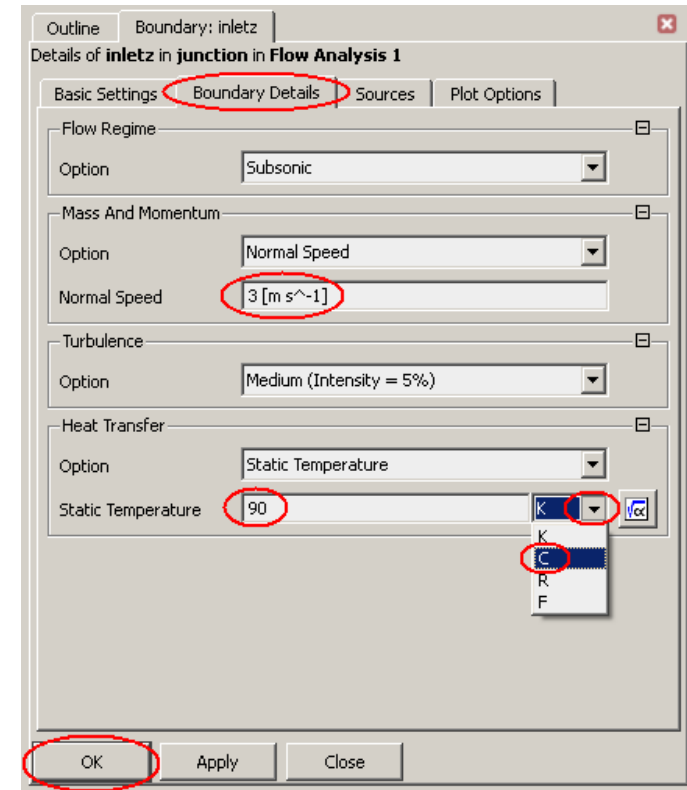
CFX-Pre – Inlet boundary conditions

- **Right-click on the *junction* domain and select *Insert > Boundary***

- **Set the Name to *inletz* and click *OK***

- **Leave the *Boundary Type* field set to *Inlet***

- **Set *Location* to *inlet z***

CFX-Pre – Inlet boundary conditions

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

- **Click the *Boundary Details* tab**

- **Enter a *Normal Speed* of 3 [m s^-1]**

- **Set the *Static Temperature* to 90 [C]**
  - (make sure the units are correct!)

- **Click *OK***

CFX-Pre – Outlet boundary conditions

- **Insert another boundary in the same way as before and name it *outlet***

- **Set the *Boundary Type* to *Outlet***

- **Set *Location* to *outlet***

- **Click the *Boundary Details* tab**

- **Set *Relative Pressure* to 0 [Pa]**
  - This is relative to the domain *Reference Pressure*, which is 1 [atm]

- **Leave all other settings at their default values**

- **Click *OK***

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

| Introduction | Setup | Solving | Postprocessing | Summary |
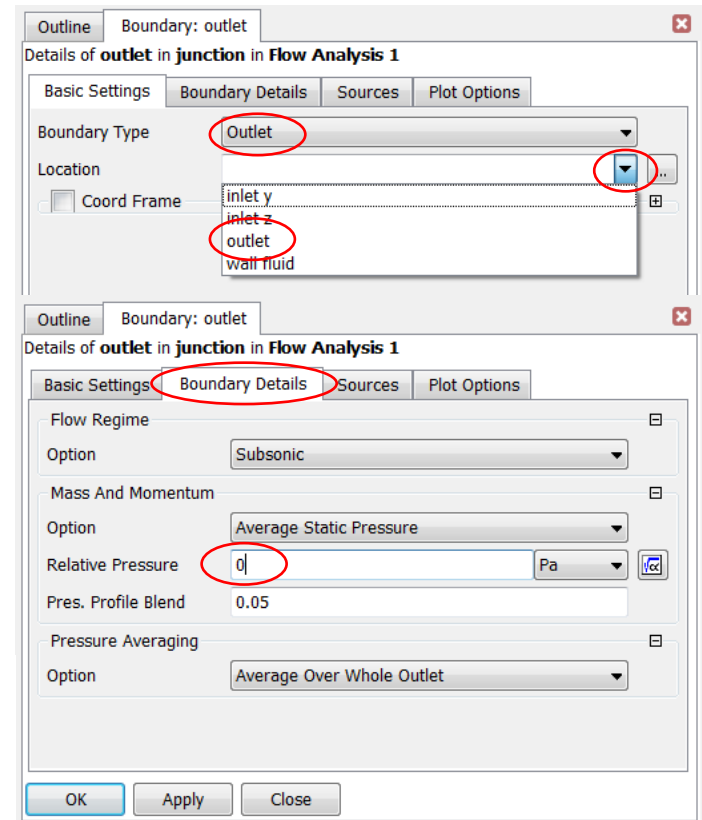
CFX-Pre – Wall boundary conditions

The default boundary condition, *junction Default,* comprises all the 2-D regions not yet assigned to a boundary condition

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

The default boundary type is an adiabatic wall, which is appropriate here. To check, double-click on wall and select the *Boundary Details* tab
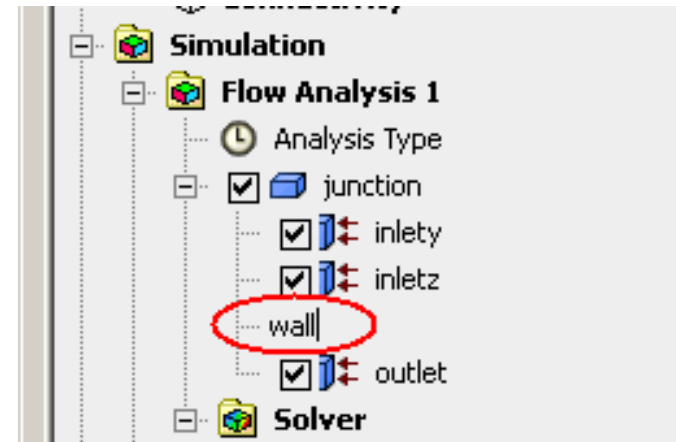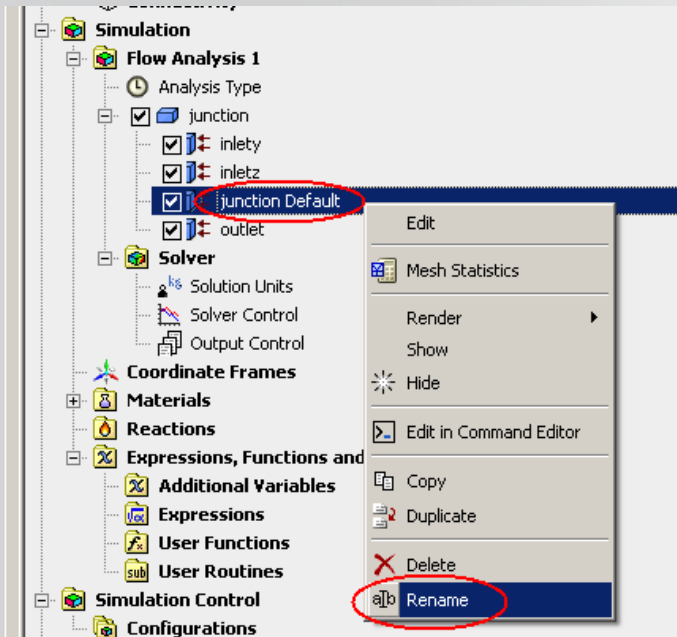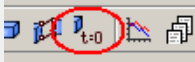
CCL at a Glance

- **Before proceeding you will now take a quick look at CCL (CFX Command Language). CCL will be covered in detail in a later lecture.**

- **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**

- **Right-click on *inlety* and select *Edit in Command Editor***
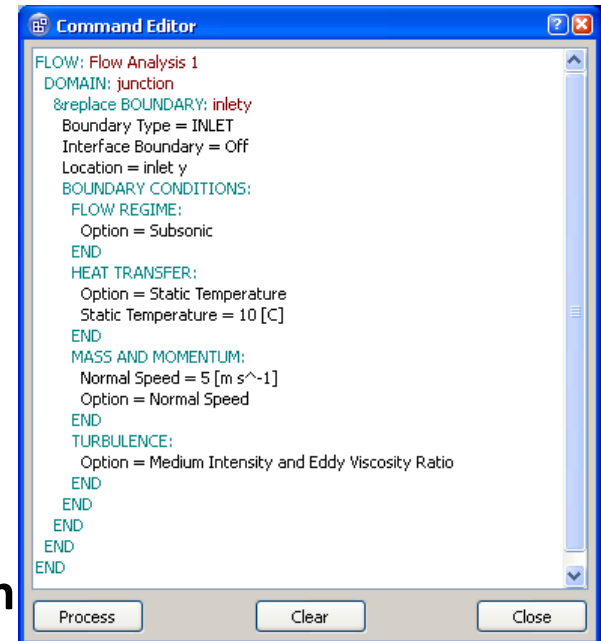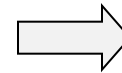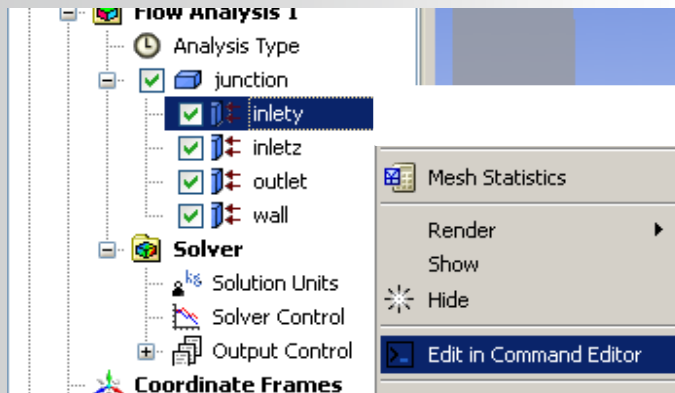
- **Close the *Command Editor* after taking a quick look at the CCL definition of the inlet boundary condition**

| Introduction | Setup | Solving | Postprocessing | Summary |

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.**

- **You will use *Automatic* initial conditions and so there is no need to set any values. Click the *Initialization* icon** **to view 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 speed of convergence and the accuracy of the results. For this model the default settings are reasonable but will not be suitable for all simulations**

- **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 in a later lecture

    - Residuals are a measure of how well the 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

CFX-Pre – Monitor points

In all engineering flows there are specific variables or quantities of interest. Sometimes they do not reach a satisfactory value by the time the overall solution has reached the convergence criteria. So it is always a good idea to monitor them as the solution progresses. In this simulation pressure will be monitored at both inlets

- **Double-click *Output Control* from the *Outline* tree**
- **On the *Output Control* form, select the *Monitor* tab**
- **Check the *Monitor Objects* box**
- **Click the *New* icon**
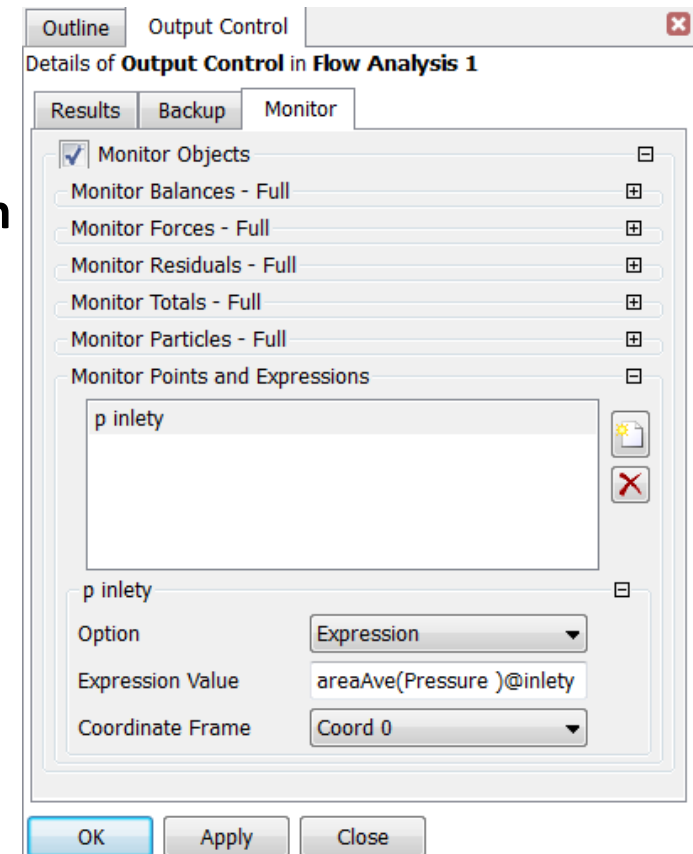- **Set the *Name* to *p inlety* and click *OK***
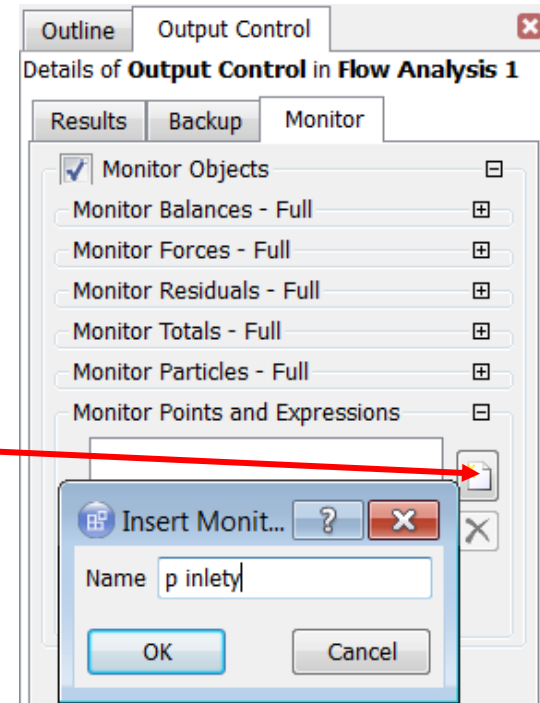
CFX-Pre – Monitor points (continued)

- **An expression will be used to define the monitor point**

- **Set 'Option' to 'Expression'**

- **Enter the following expression in the Expression Value field :**

areaAve(Pressure)@inlety

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

- **Note that expressions are case sensitive.**

- **Expressions and expression language will be covered in more detail later**

| Introduction | Setup | Solving | Postprocessing | Summary |

CFX-Pre – Monitor points (continued)

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

- **Click the *New* icon**

- **Set the *Name* to *p inletz* and click *OK***

- **Set *Option* to *Expression***

- **For *Expression Value* enter:**

$$areaAve(Pressure)@inletz$$
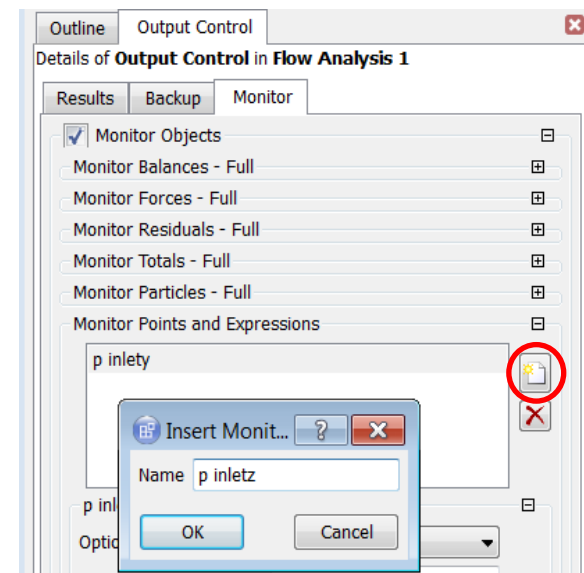
- **Click *OK* to apply the settings and close the *Output Control* form**

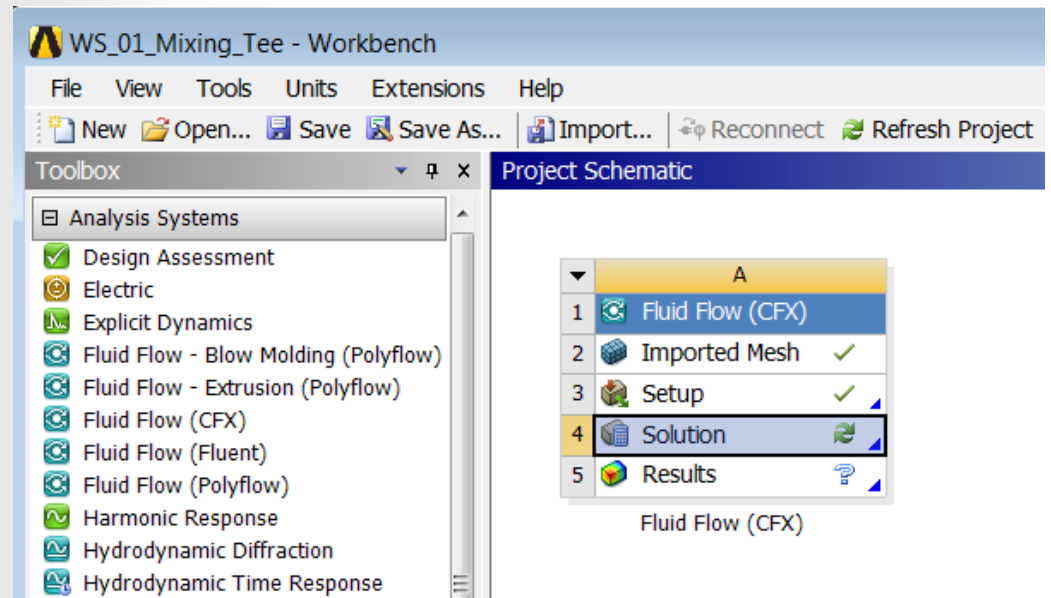The monitor points will be utilised during the solution in a later in the workshop

Solution Goals

- **Launch CFX-Pre from Workbench**

- **Use pre-defined materials**

- **Define the fluid models in a domain**

- **Create & 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

- **Close CFX-Pre**
  - When running in WB the CFX-Pre case will be saved automatically

- **Save the Workbench project to MixingTee.wbpj in your working directory**
  - *File > Save As*

- **In Workbench double-click *Solution* to launch the *CFX-Solver Manager***
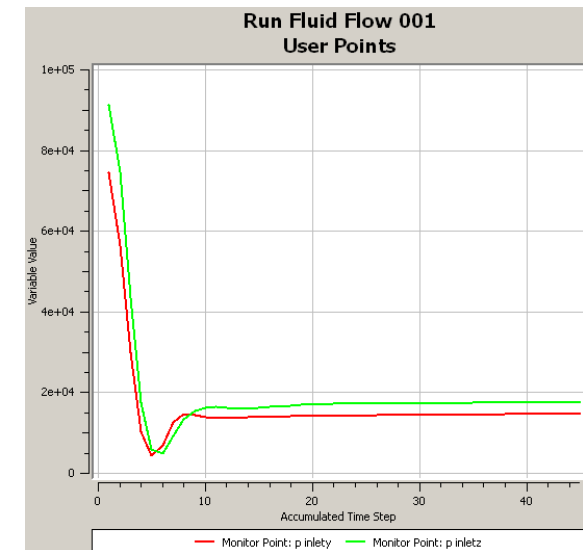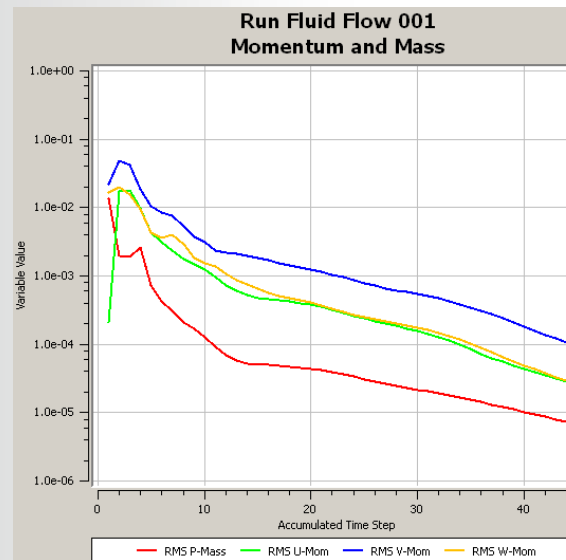
Obtaining a solution

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

- **Click *Start Run* to begin the solution process**
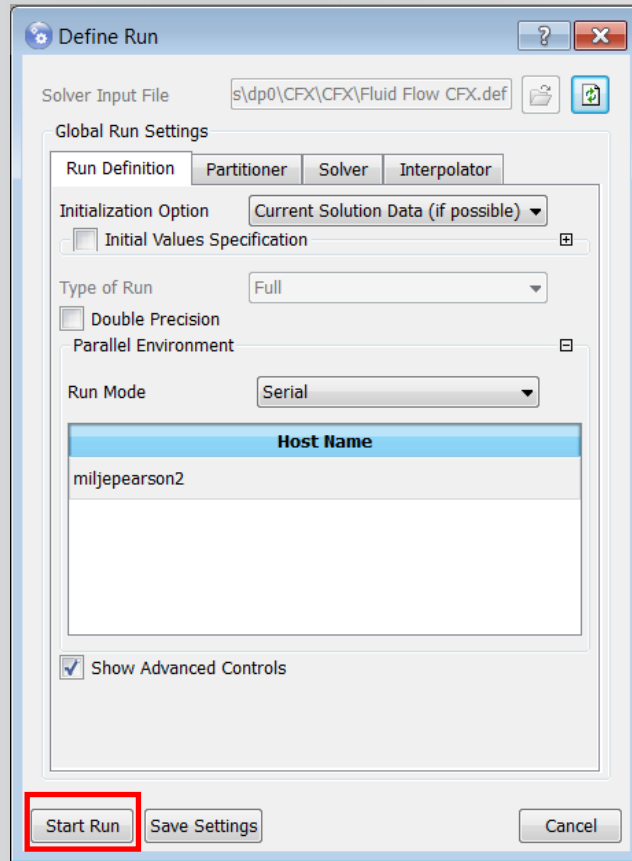
46 iterations are required to reduce the RMS residuals to below the target of 1.0×10^{-4}

The pressure monitor points approach steady values. To view select the User Points tab.

| Introduction | Setup | Solving | Postprocessing | Summary |

Post-processing Goals

- **Launch CFX-Pre from Workbench**

- **Use pre-defined materials**

- **Define the fluid models in a domain**

- **Create & 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**

| Introduction | Setup | Solving | Postprocessing | Summary |

Launching CFD-Post

- **Close the *CFX-Solver Manager***

- **Save the project**

- **Double-click *Results* to launch *CFD-Post***

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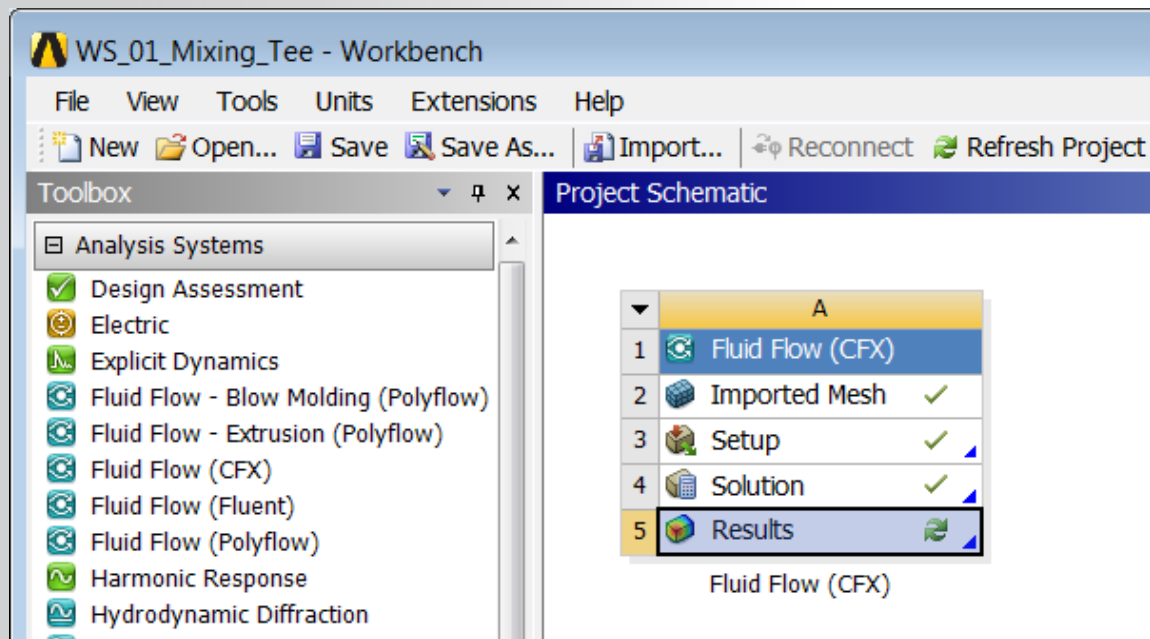
- **When *CFD-Post* opens you will see that the layout is similar to *CFX-Pre***

- **There are two windows on the left side:**

- **Selector Window**
  - Lists currently-defined graphics objects. Objects for each boundary condition are created automatically
  - An objects is edited by double-clicking or right-clicking on the icon
  - 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 status of the object

Introduction   Setup   Solving   Postprocessing   Summary
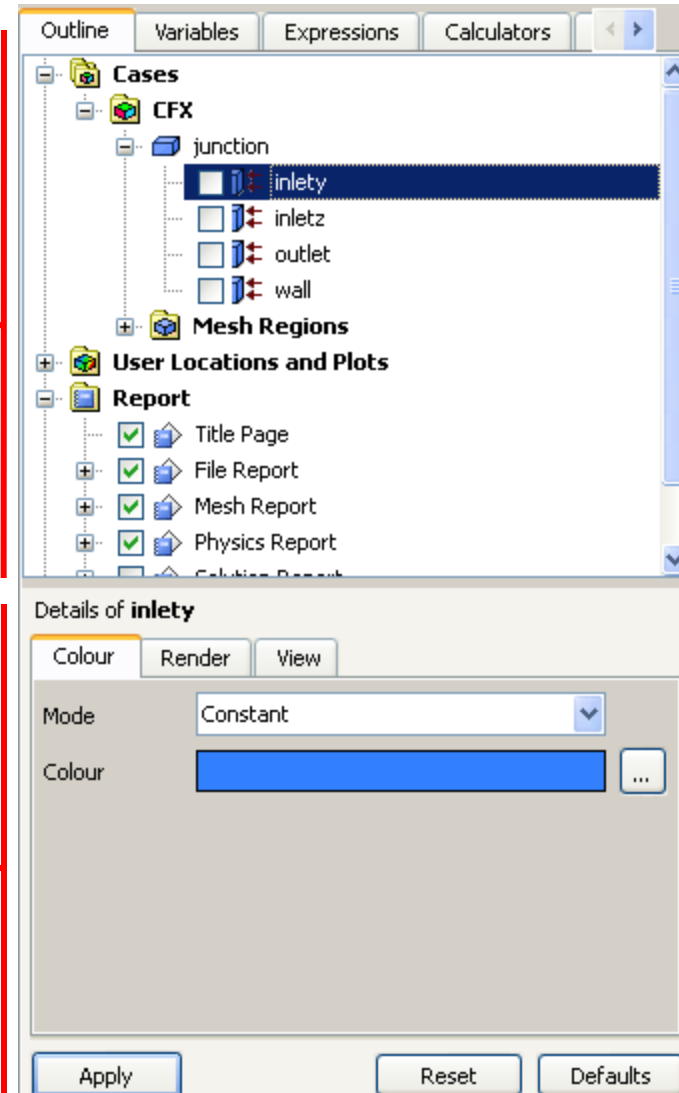
CFD-Post

- **The results are loaded**
- **CFD-Post initially displays the outline (wireframe) of the model**
- **Viewer toolbar buttons allow you to manipulate the view**
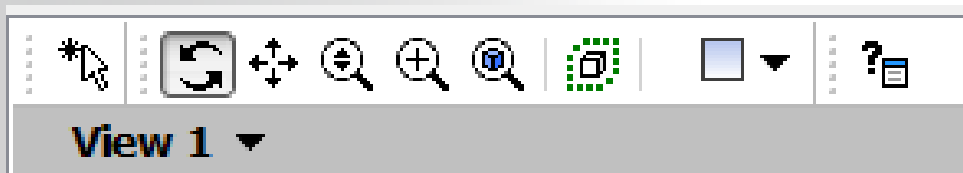
- **Descriptions and alternative short cuts are given below:**

|	
↻	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**

- **Click the *Contour* icon from the toolbar**

- **Click *OK* to accept the default name, *Contour 1***

- **Set *Locations* to *wall***
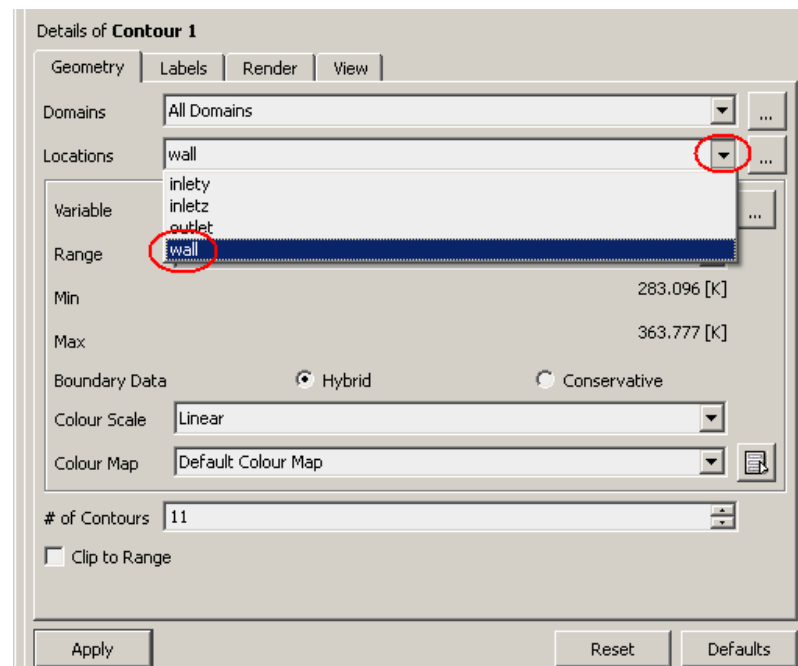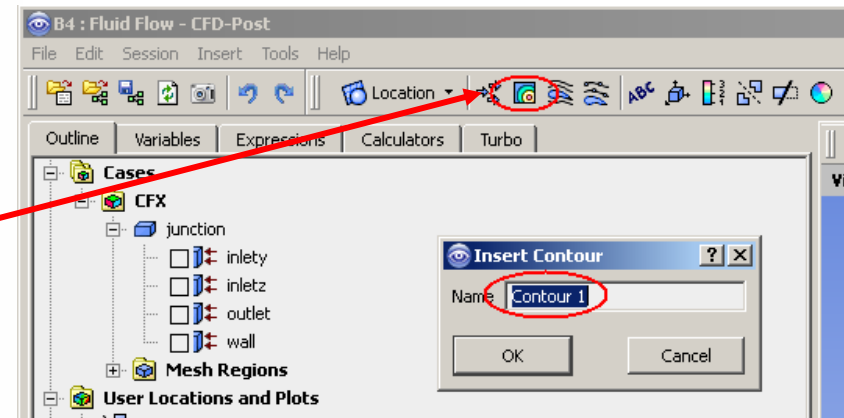
CFD-Post – Temperature contour plot

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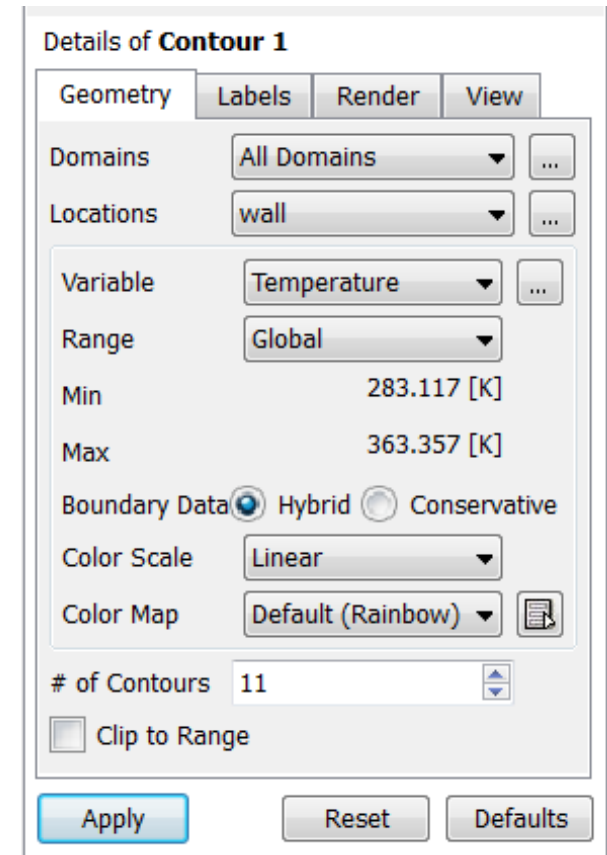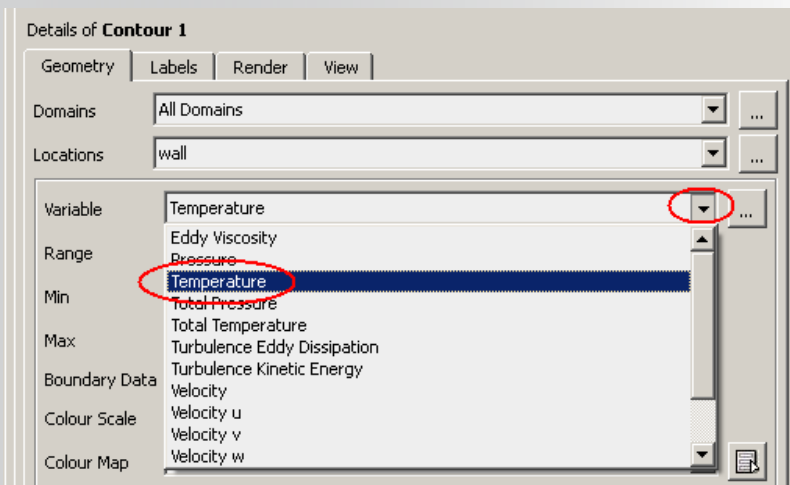
CFD-Post - Temperature contour plot

A temperature contour plot on the walls is now visible

- **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>**

- **Also try clicking on the axes in the bottom right corner of the *Viewer***
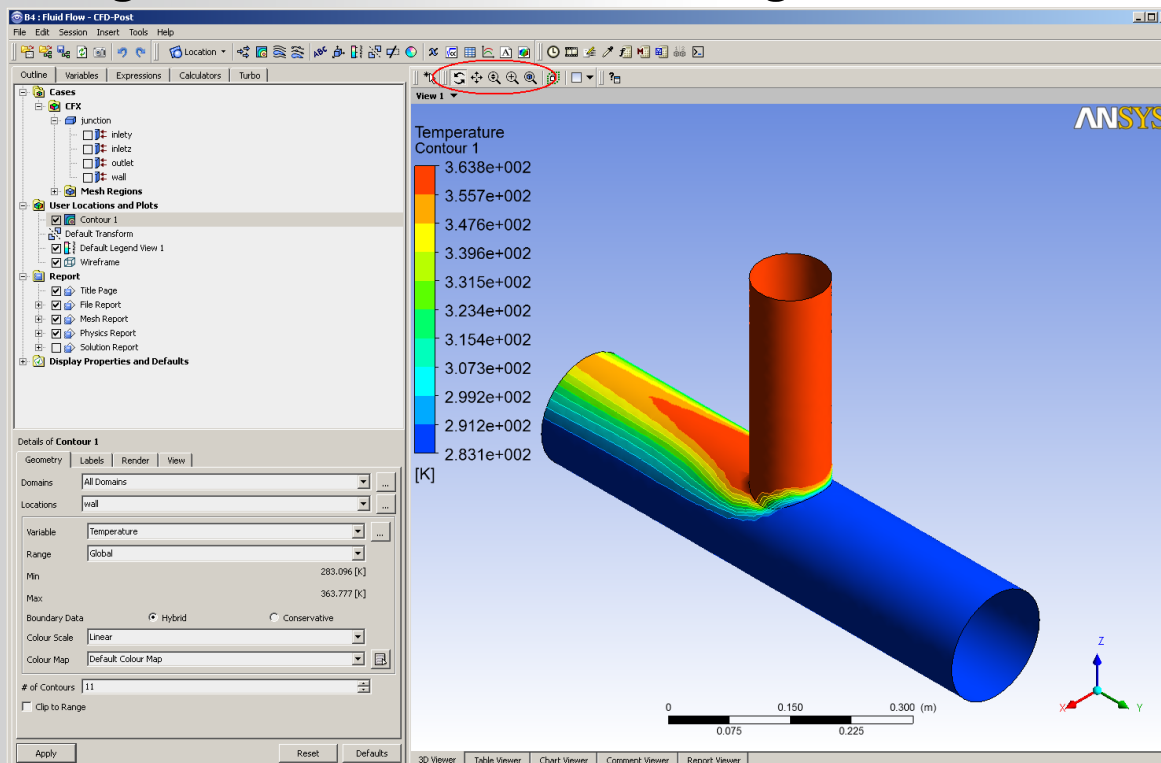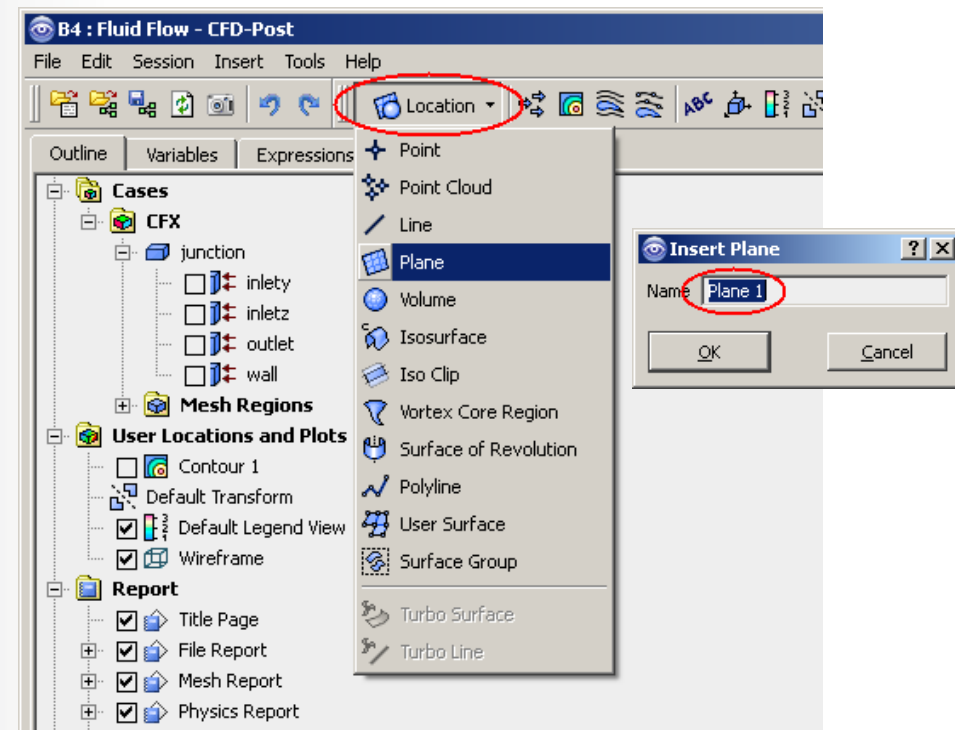
CFD-Post

- **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 Lagrangian Particle tracking has been enabled in *CFX-Pre*)
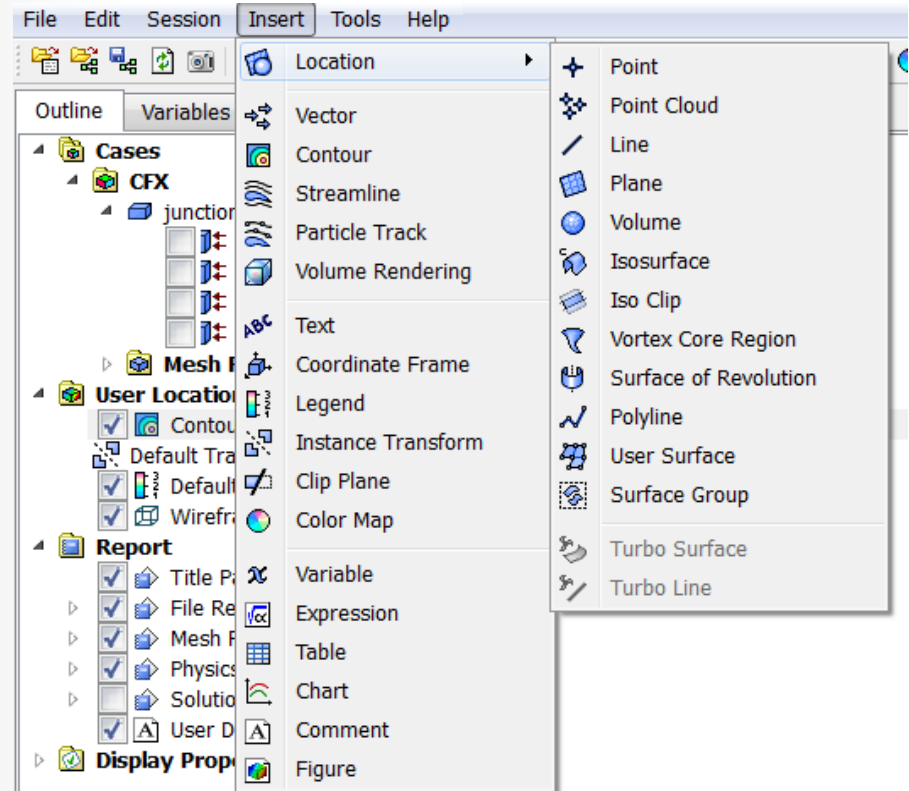
CFD-Post – Creating a plane at x = 0

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

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

- **Click *OK*, accepting the default name of *Plane 1***
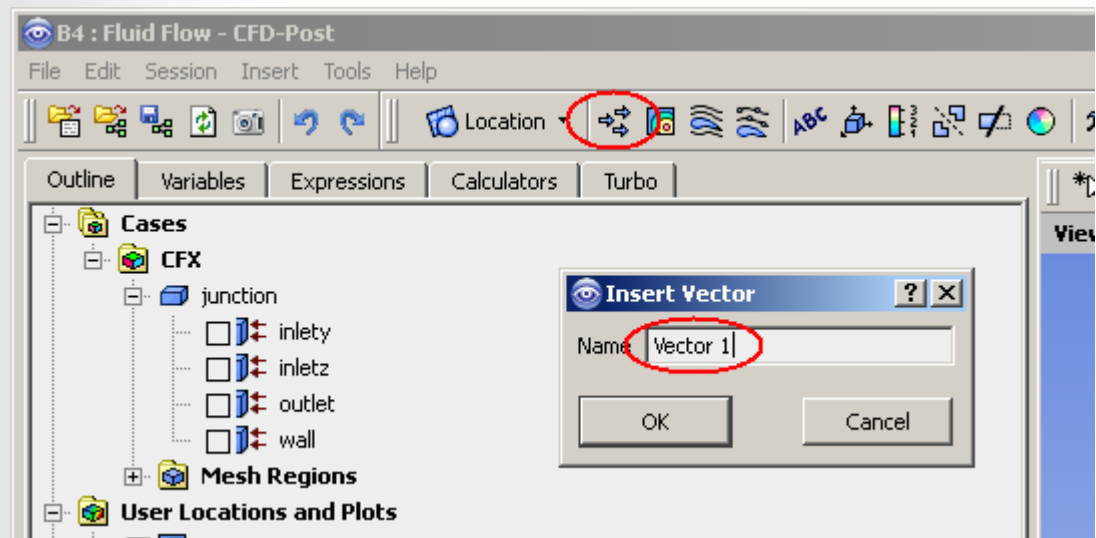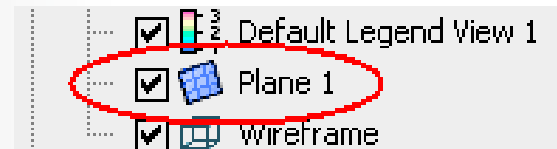
CFD-Post – Creating a plane at x = 0

- **Set *Method* to *YZ Plane***

- **Leave *X* set to 0 [m]**

- **Click *Apply* to generate the plane**
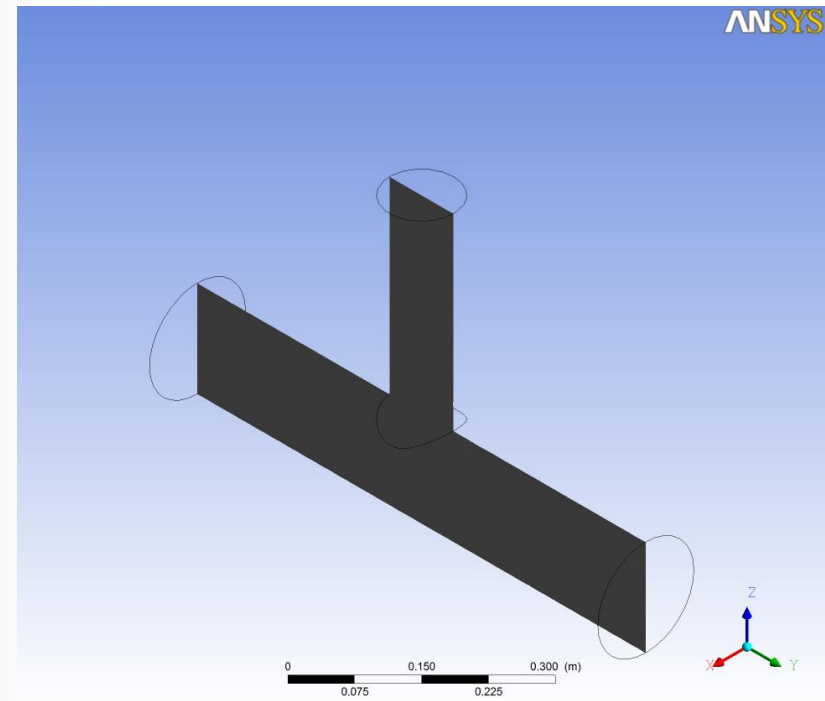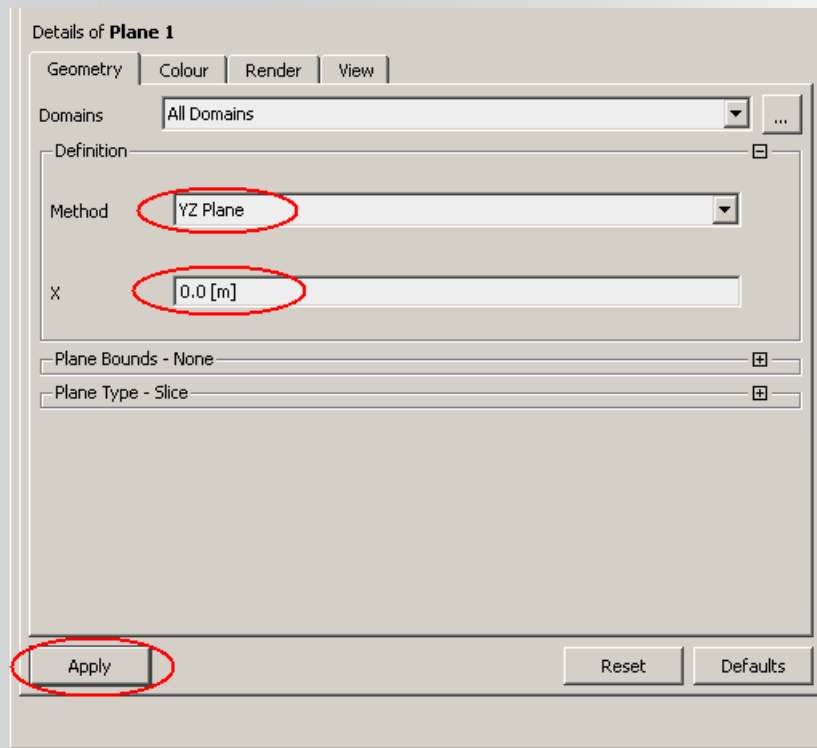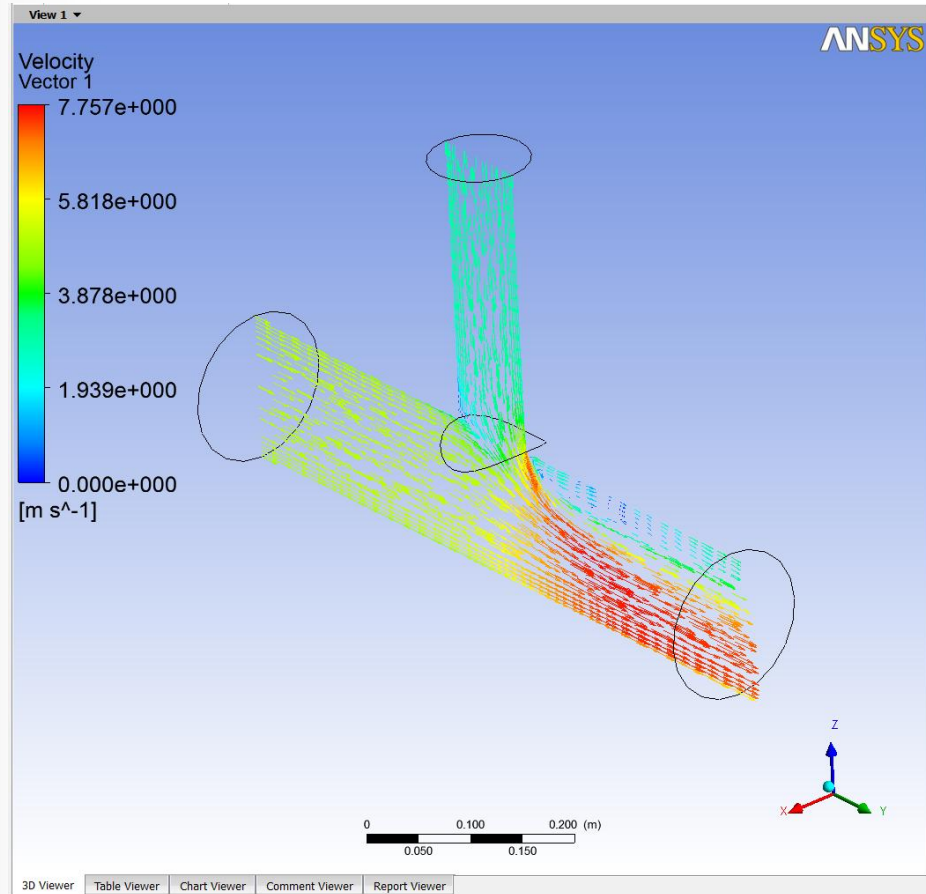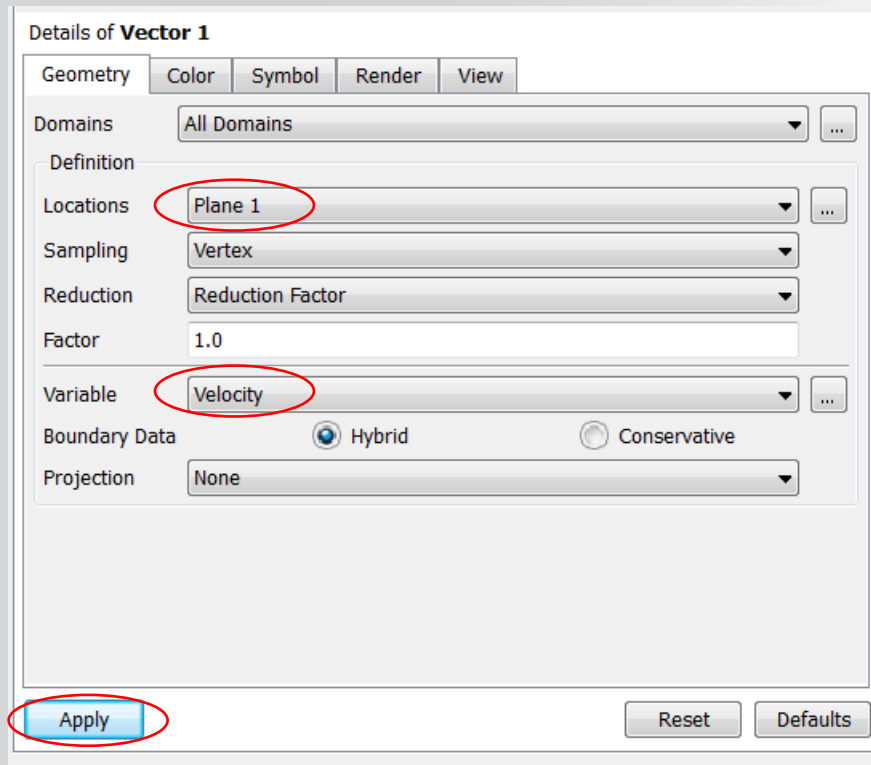
CFD-Post – Creating a velocity vector plot

The plane will be used as a locator for a vector plot

- **Hide the plane by un-checking the associated box in the tree view**
- **Click the *Vector* icon from the toolbar**
- **Click *OK*, accepting the default name of *Vector 1***
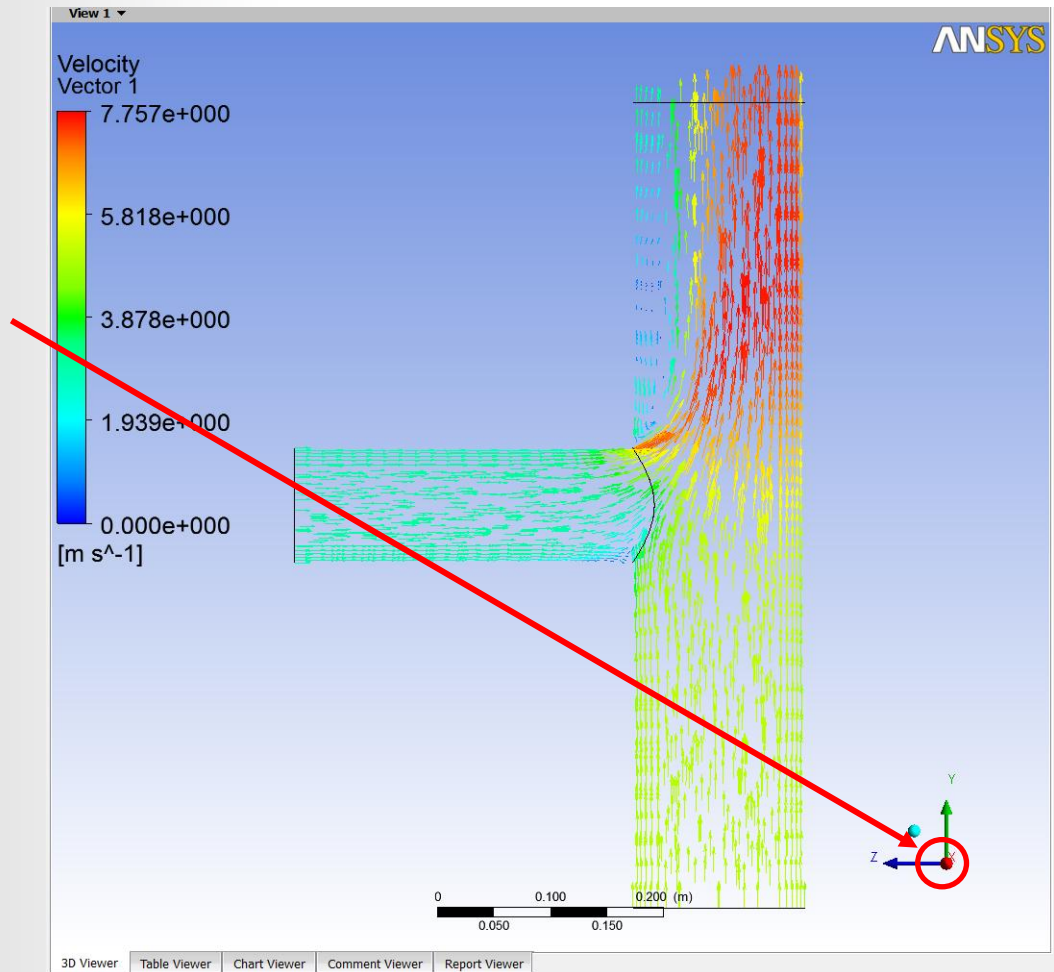
CFD-Post – Velocity vector plot

- **Set *Locations* to *Plane 1***

- **Leave the *Variable* field set to *Velocity***

- **Click *Apply***

CFD-Post – Aligning the view

Given that the vector plot is on a Y-Z plane, you might want to view the plot along the X axis

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

CFD-Post – Creating velocity streamlines

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

- **Click the *Streamline* icon from the toolbar**

- **Click *OK*, accepting the default name of *Streamline 1***
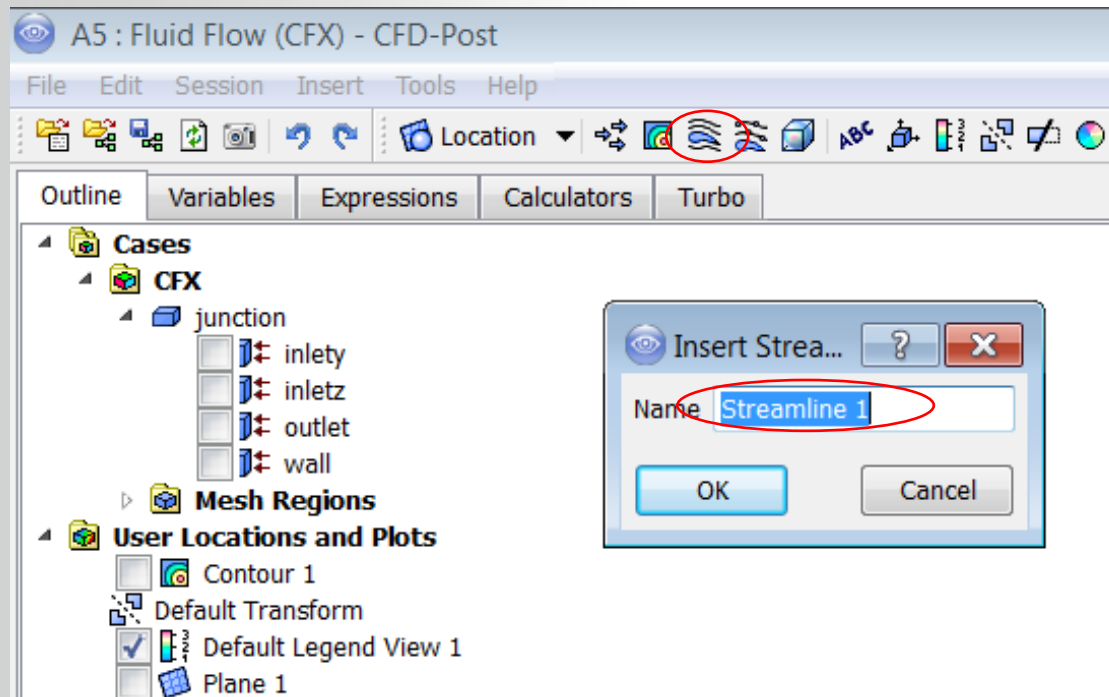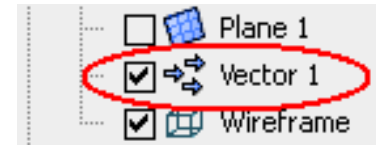
CFD-Post – Velocity streamlines

- 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

- Leave the *Variable* field set to *Velocity*

# CFD-Post – Velocity streamlines
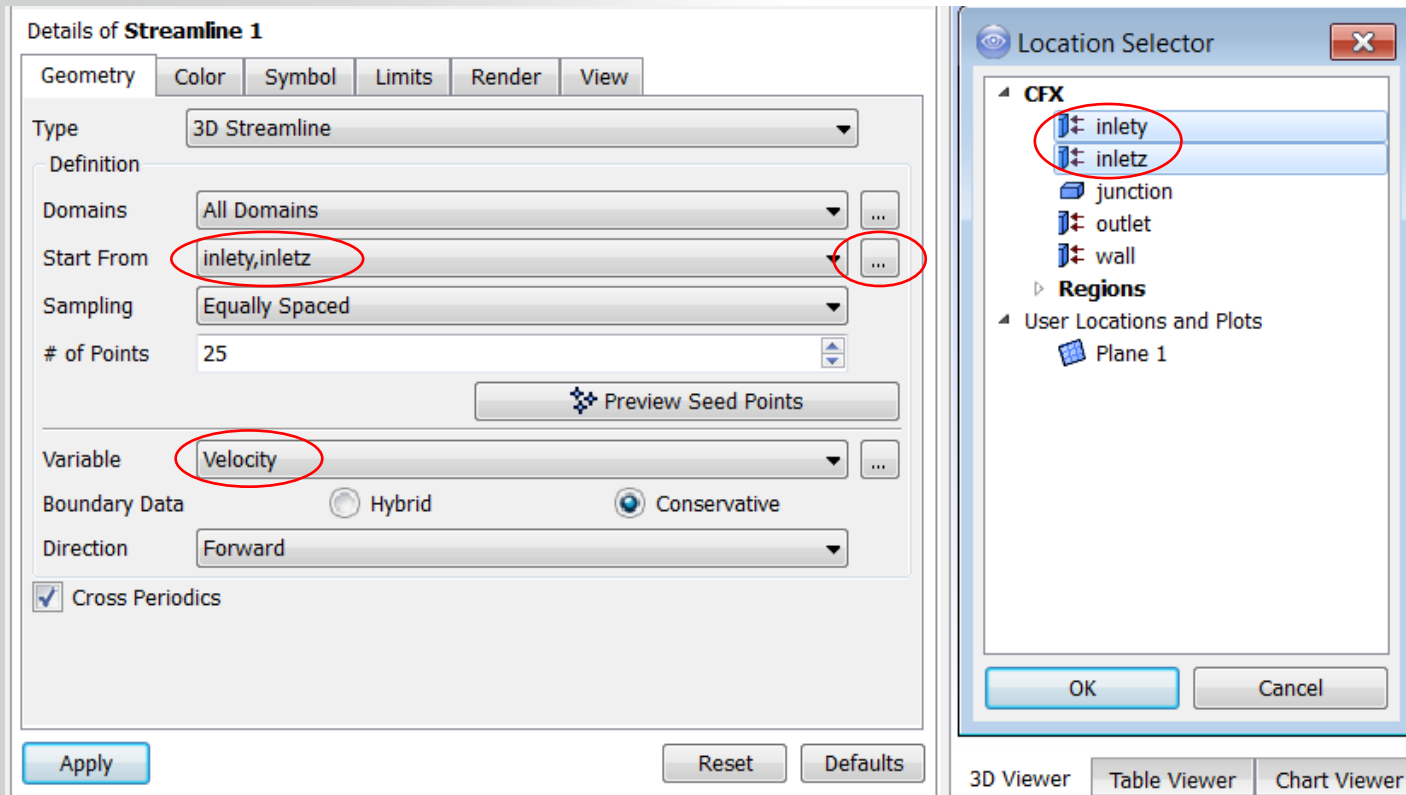
- **Click the *Symbol* tab**

- **Change the *Stream Type* to *Ribbon***

- **Click *Apply***

- **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
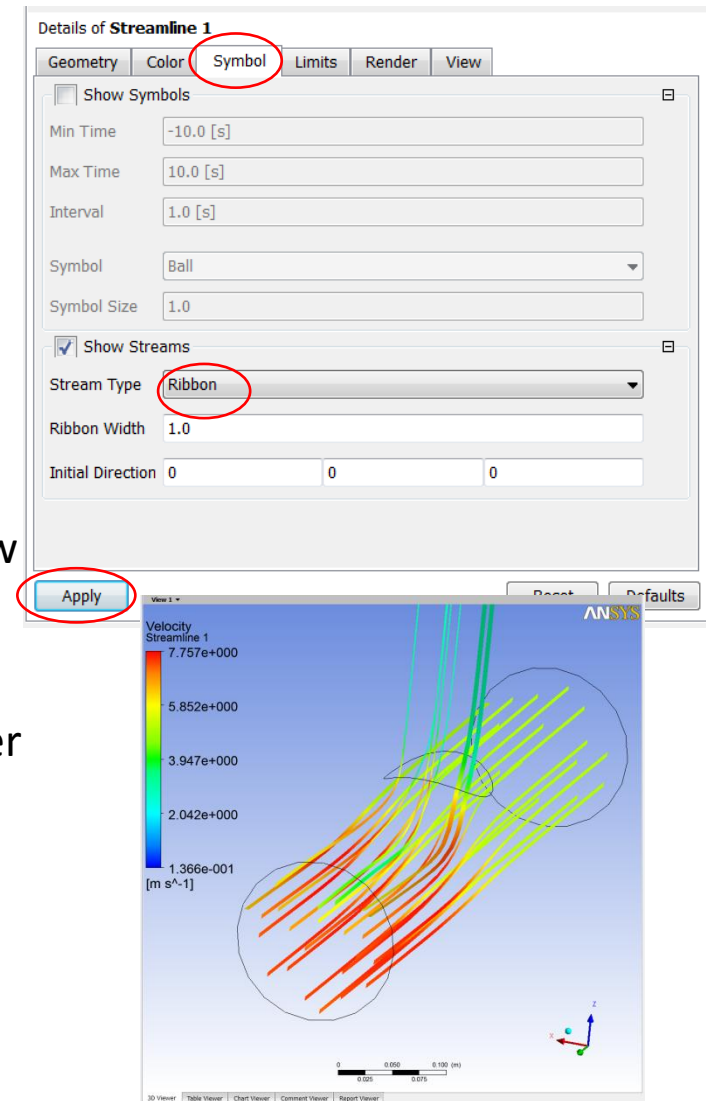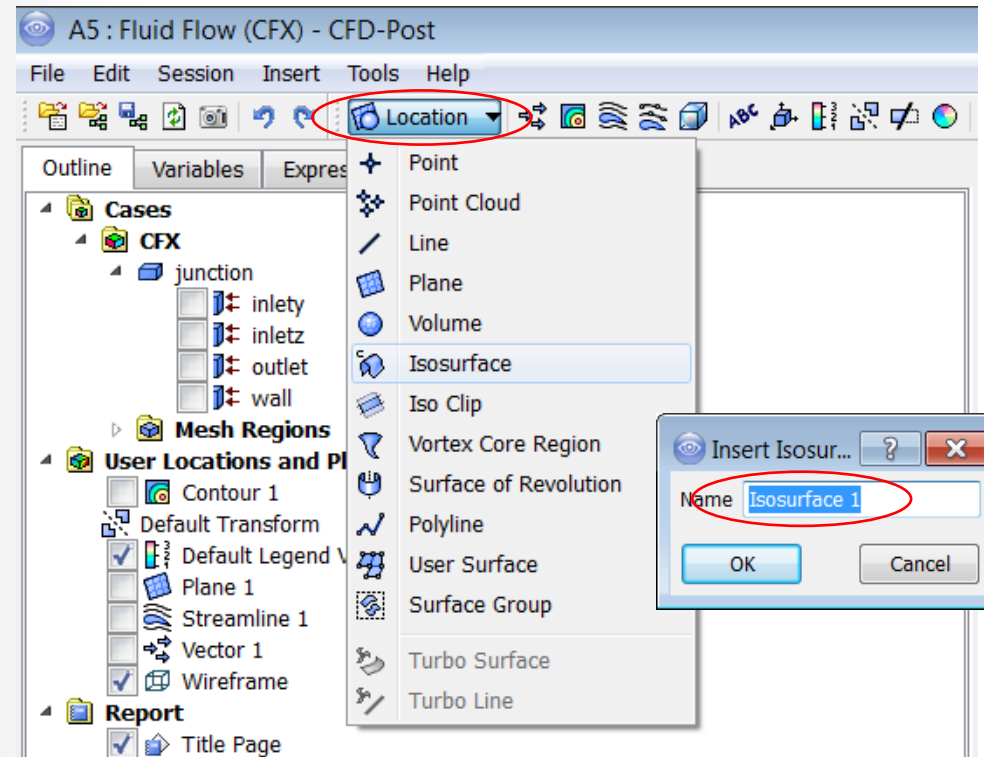
CFD-Post – Creating a velocity isosurface

- Hide the previously created streamlines by un-checking the associated box in the tree view
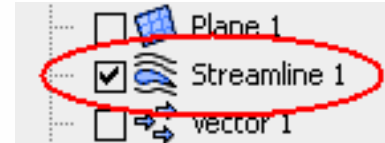
- Click the *Location* button on the toolbar and select *Isosurface* from the drop-down menu

- Click *OK*, accepting the default name of *Isosurface 1*

| Introduction | Setup | Solving | Postprocessing | Summary |

CFD-Post – Velocity isosurface

- **Set the *Variable* to *Velocity* (magnitude used in this context)**

- **Enter a value of 7.5 [m s^-1] in the *Value* field (this is an arbitrary value)**

- **Click *Apply***

  - The speed is > 7.5 m/s inside the isosurface and < 7.5 m/s outside. Isosurfaces in general are useful for showing pockets of high velocity, temperature, turbulence, etc.

  - Try values above and below 7.5 m/s

| Introduction | Setup | Solving | Postprocessing | Summary |
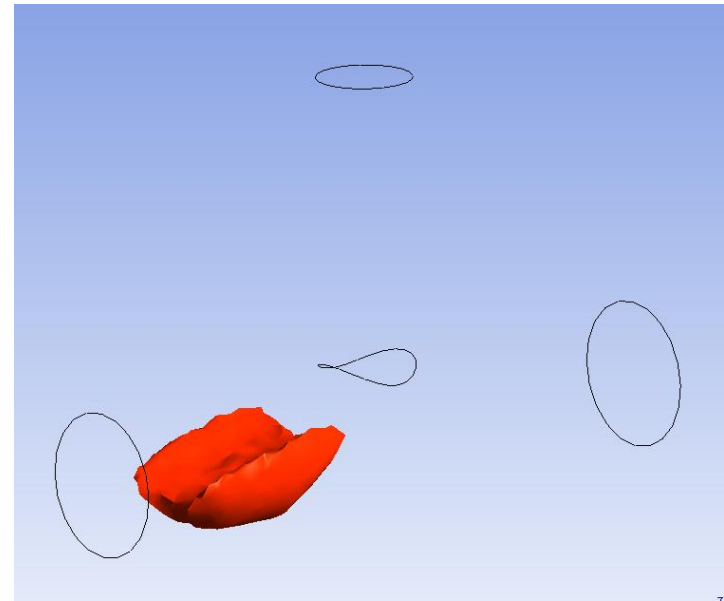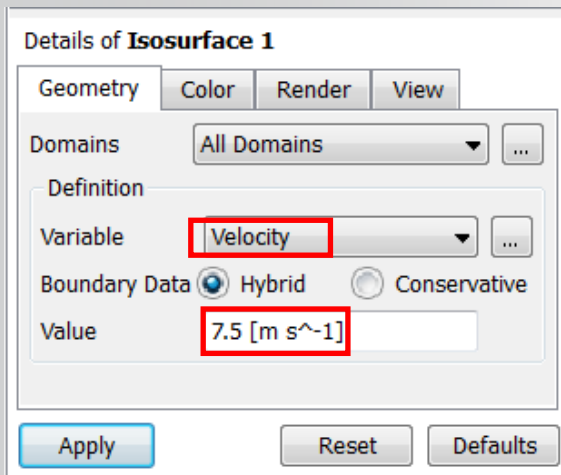
CFD-Post – Velocity isosurface (continued)

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

- **Click the *Color* tab**

- **Set the *Mode* to *Variable***

- **Set *Variable* to *Temperature***

- **Set the *Range* to *Local***

- **Click *Apply***

Further work
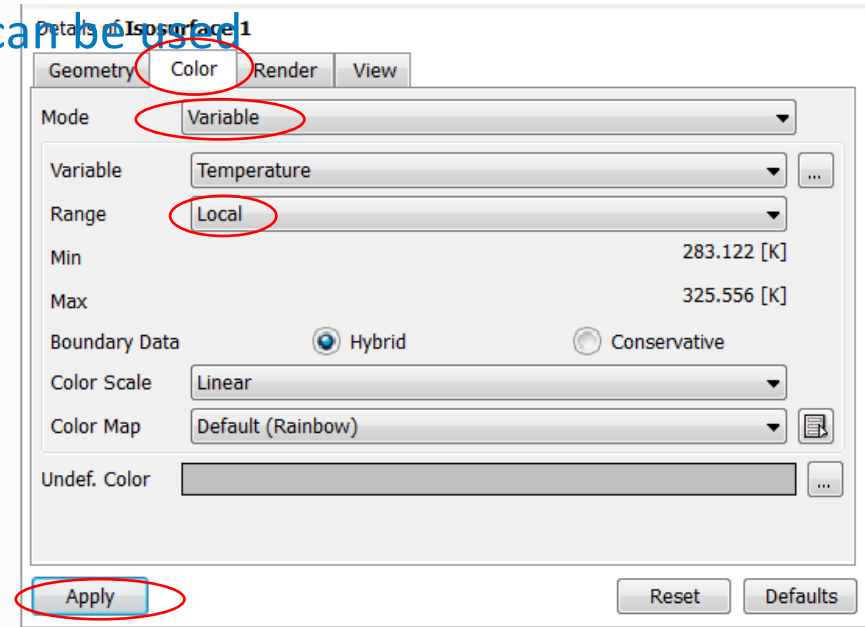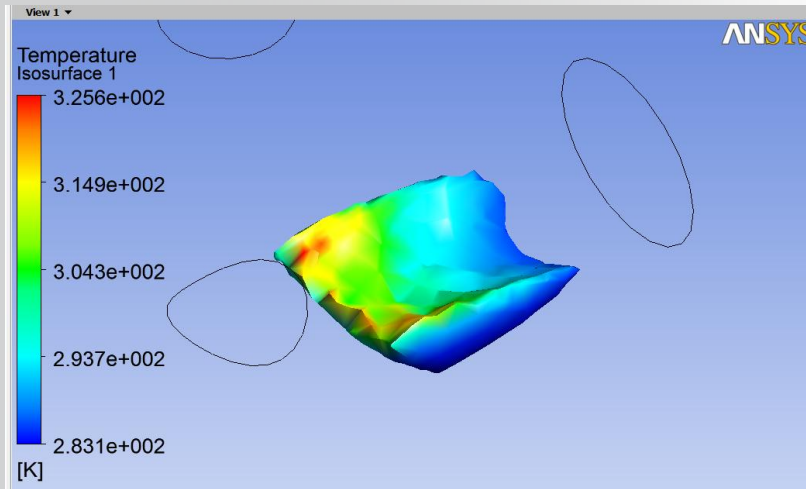
- **There are many ways the simulation in this workshop could be extended**

- **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)

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

- **Temperature-dependent physical properties**
  - Density
  - Viscosity, etc

Further work

- **This workshop has shown the basic steps that are applied in all CFD simulations:**
  - Defining Material Properties
  - Setting Boundary Conditions and Solver settings
  - Running a simulation whilst monitoring quantities of interest
  - Postprocessing the results in CFD-Post
- **One of the important things to remember in your own work is, before even starting the ANSYS software, is to think WHY you are performing the simulation:**
  - What information are you looking for
  - What do you know about the inlet conditions

  In this case we were interested in checking the pressure drop, and assessing the amount of mixing present around this T-piece.

- **Knowing your aims from the start will help you make sensible decisions of how much of the part to simulate, the level of mesh refinement needed, and which numerical schemes should be selected.**