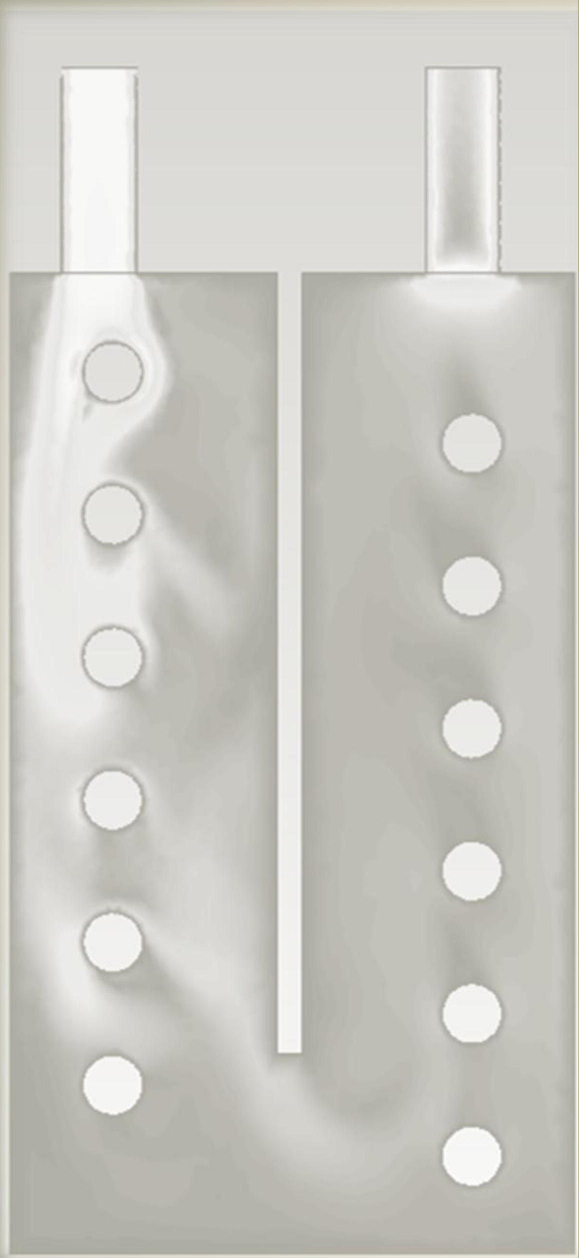


# Flow Modelling in Heat-Exchangers using ANSYS CFX

By Ahmed Al Makky

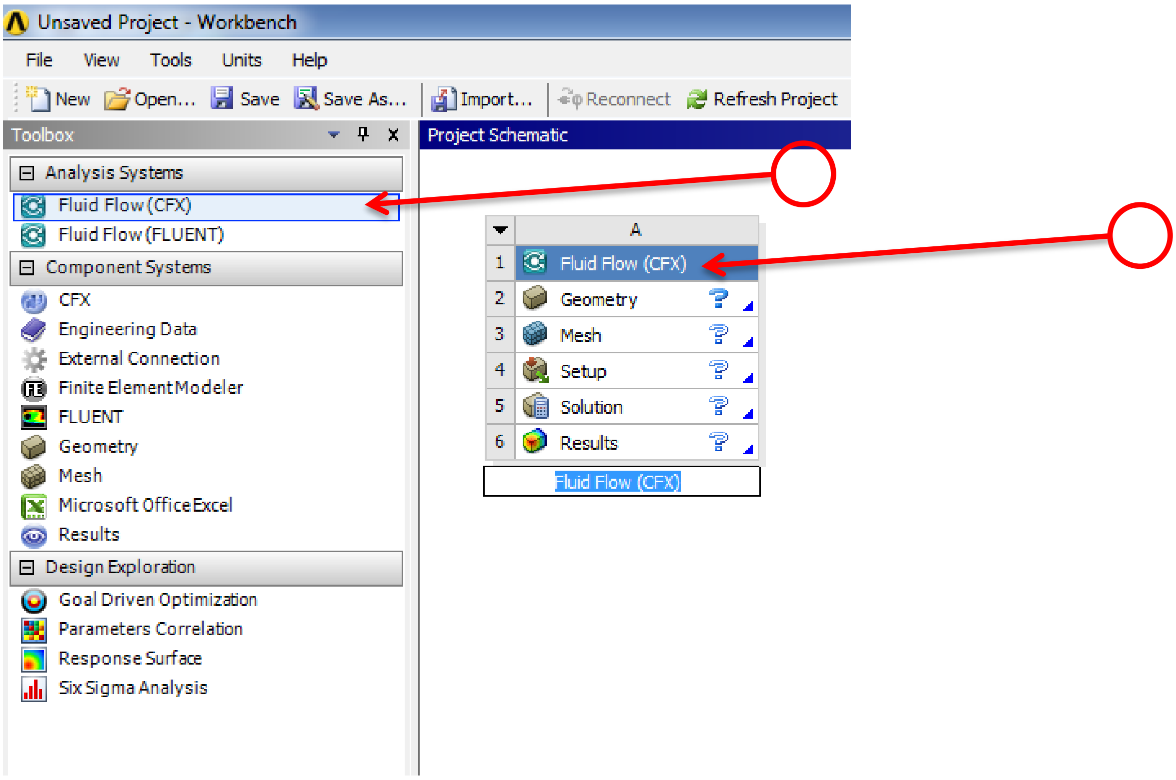
# Flow Modelling in Heat-Exchangers using ANSYS CFX

## Introduction

The tutorial was written in a rush so it has spelling mistakes never go the time to correct them, feedback would much appreciated to improve the tutorials. A mesh file is provided with this tutorial in order to focus on the flow modelling side of the problem. Heat exchangers are encountered in lots of engineering applications, in cars , in planes , in home boilers, ….etc.

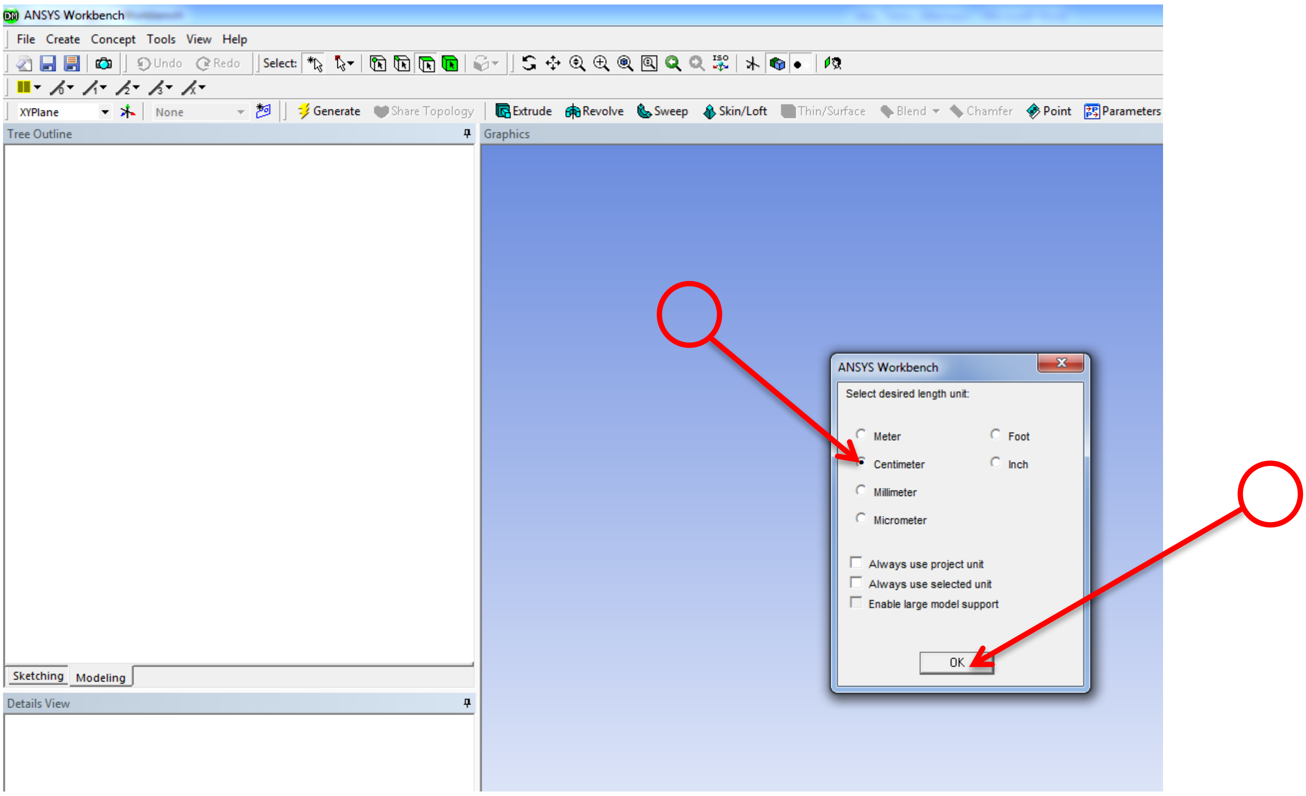
**Step 1:**

Double click on Fluid Flow (CFX) under the Analysis sytems, then double click on the Fluid Flow (CFX) under A.



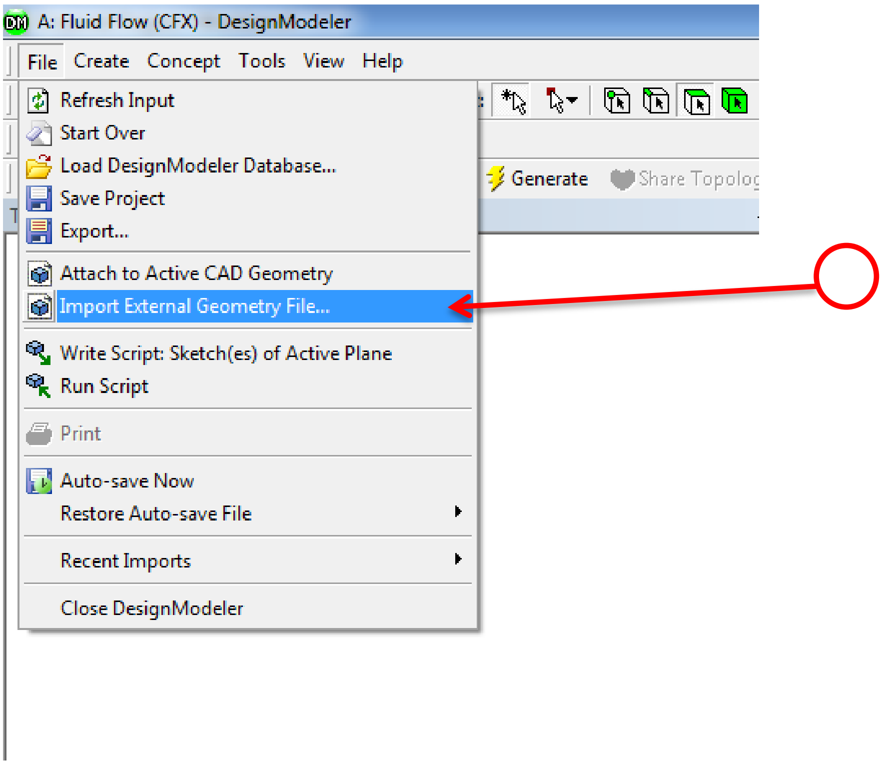
**Step 2:**

A window will open up asking you at what length scales do you want to work on, click on centimeter tick box and click Ok.



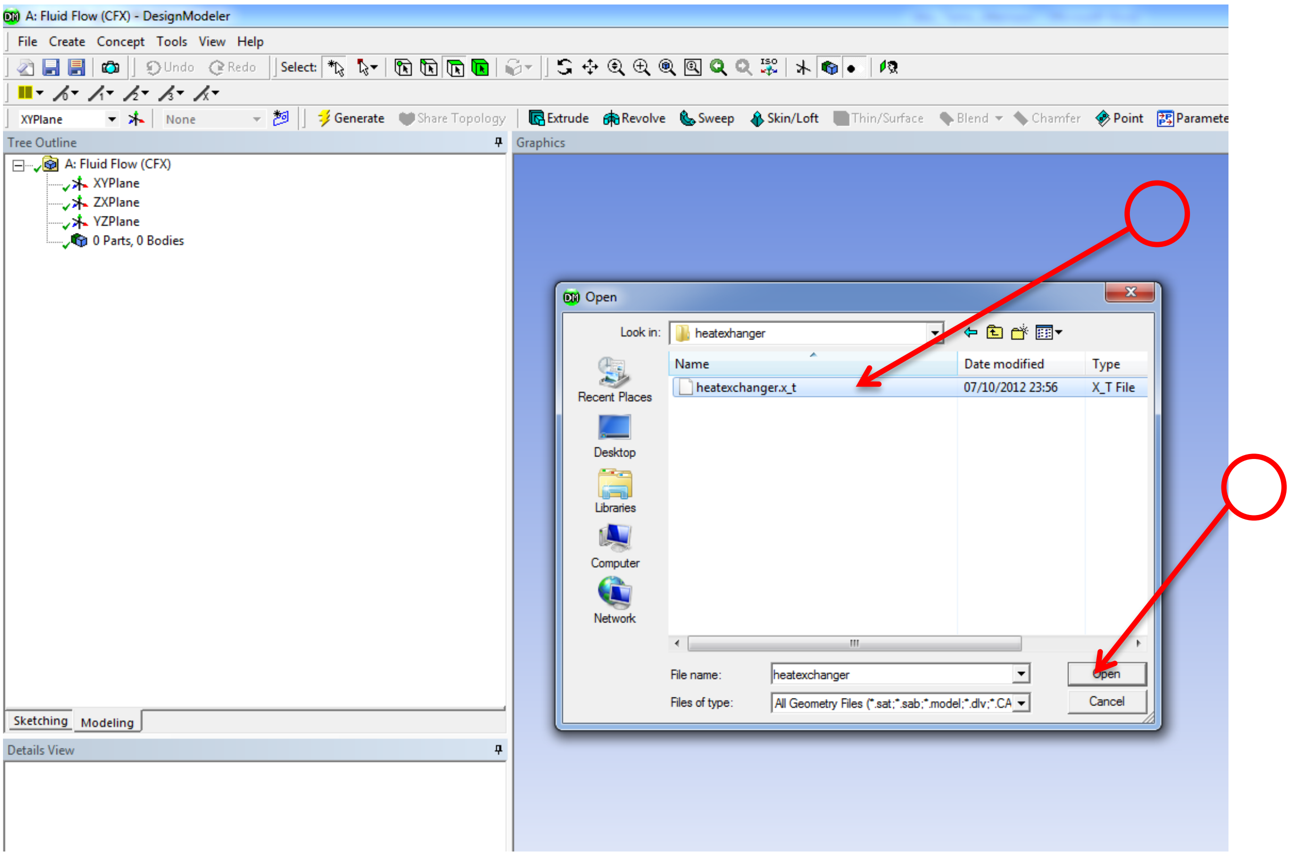
**Step 3:**

Go to File and select Import External Geometry file.



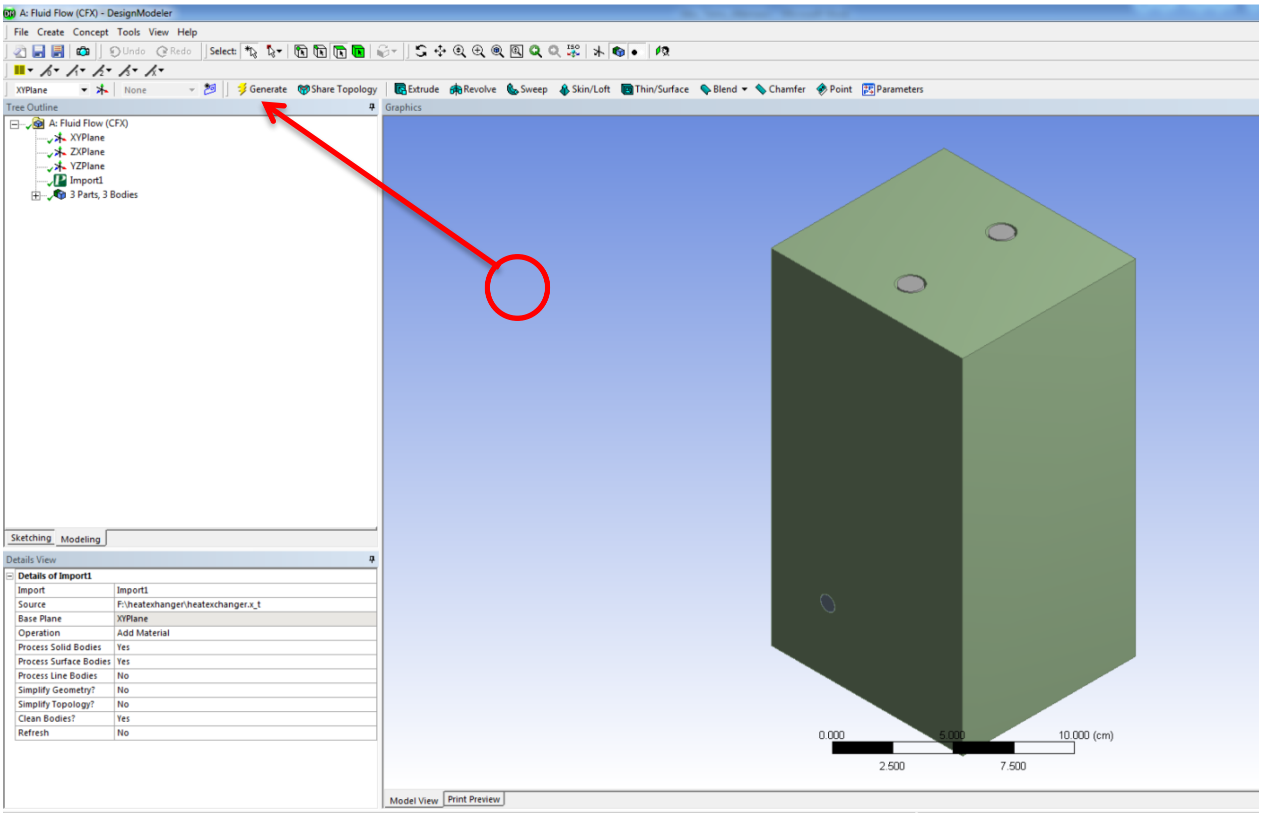
**Step 4:**

A window will open up click on the file called heatexchanger.x_t and then press open.



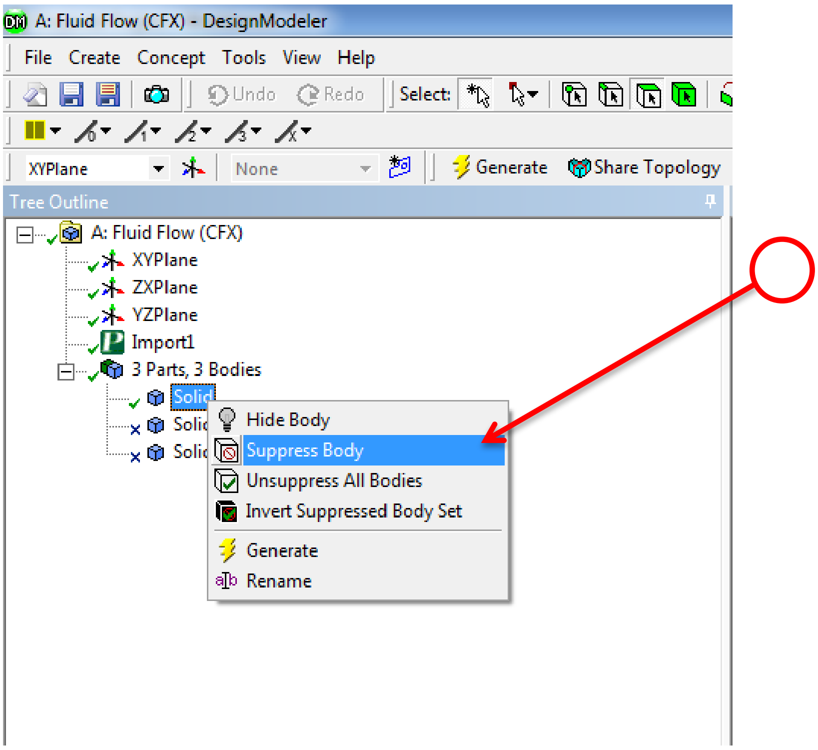
**Step 5:**

Press the genrate button and what you will see is the read in geometry into design modeller.



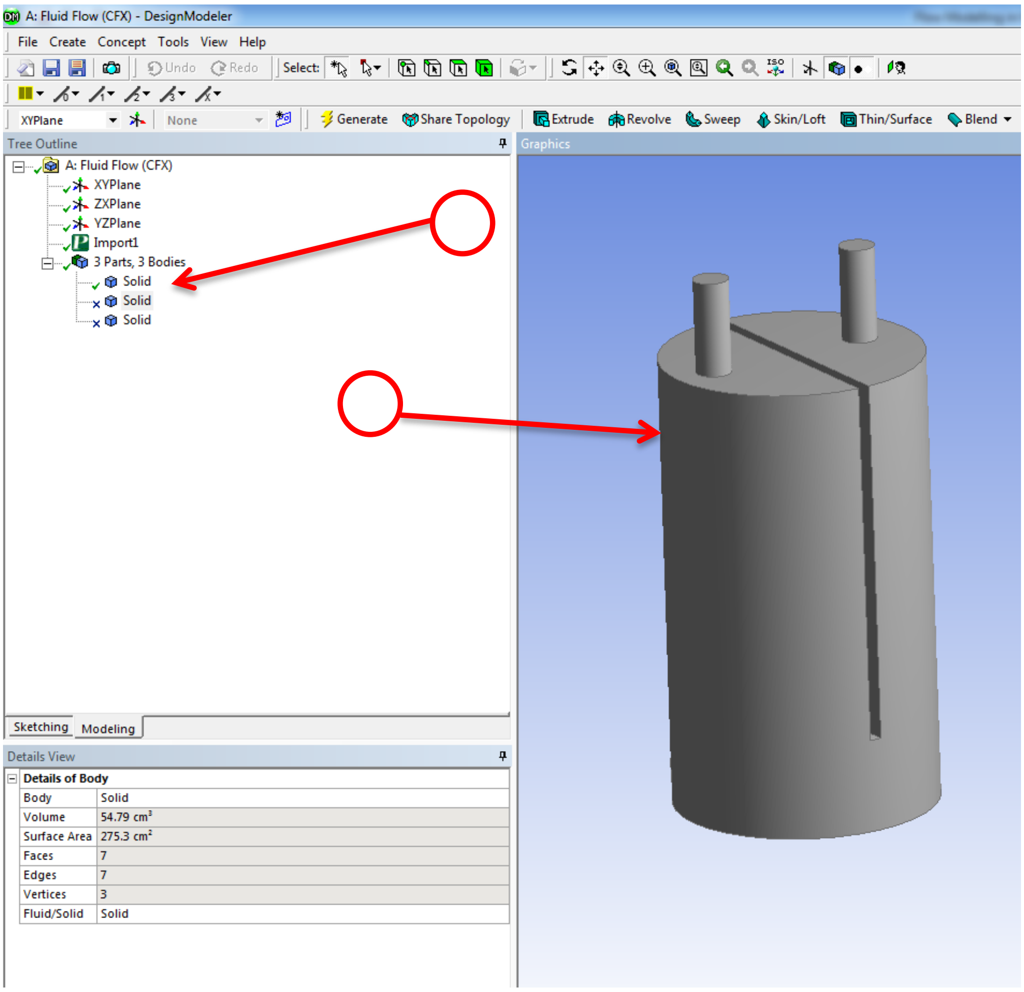
**Step 6:**

Postion the cursour on the icon (3 Parts, 3Bodies) and then press the left button on it a subtree of three directoreis would occur. Position the cursour on the the last two solids ( doing each solid seperatly) and then click the left mouse button and select the suppress body option.



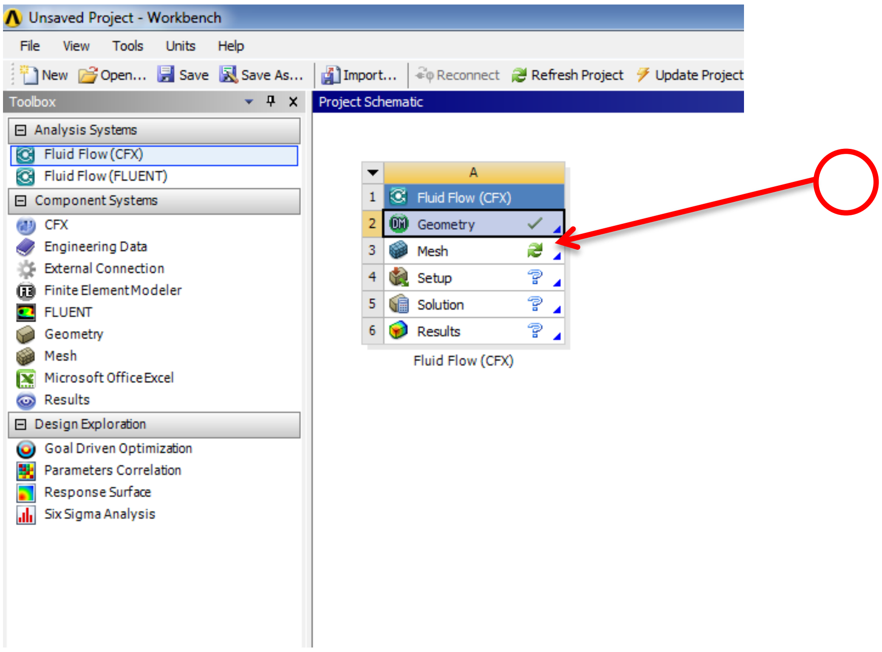
**Step 7:**

The need geometry should look like this. Notice that there is a green tick near the solid meaning the soild is visible while the two suppressed solids have a green x sign beside it.



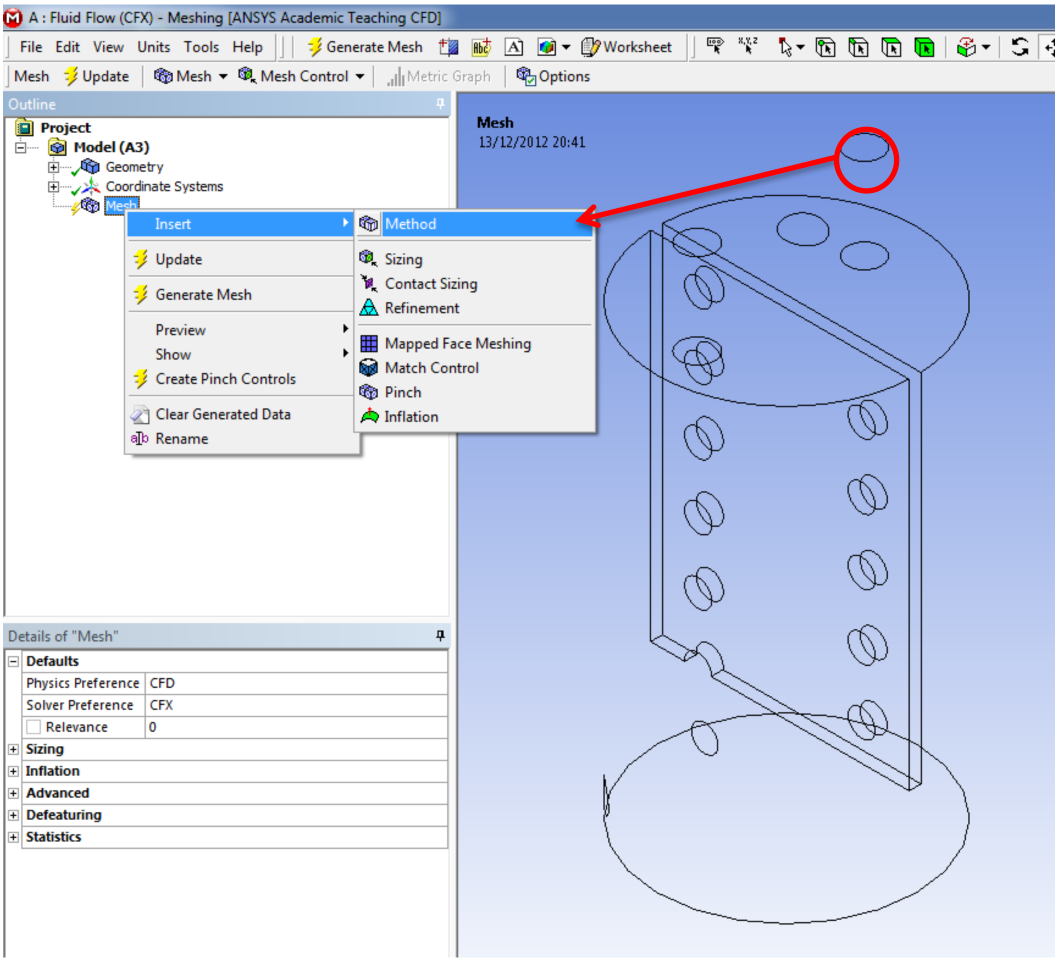
**Step 8:**

Once you see there is a green tick sign beside geometry, double click on the Mesh icon.



**Step 9:**

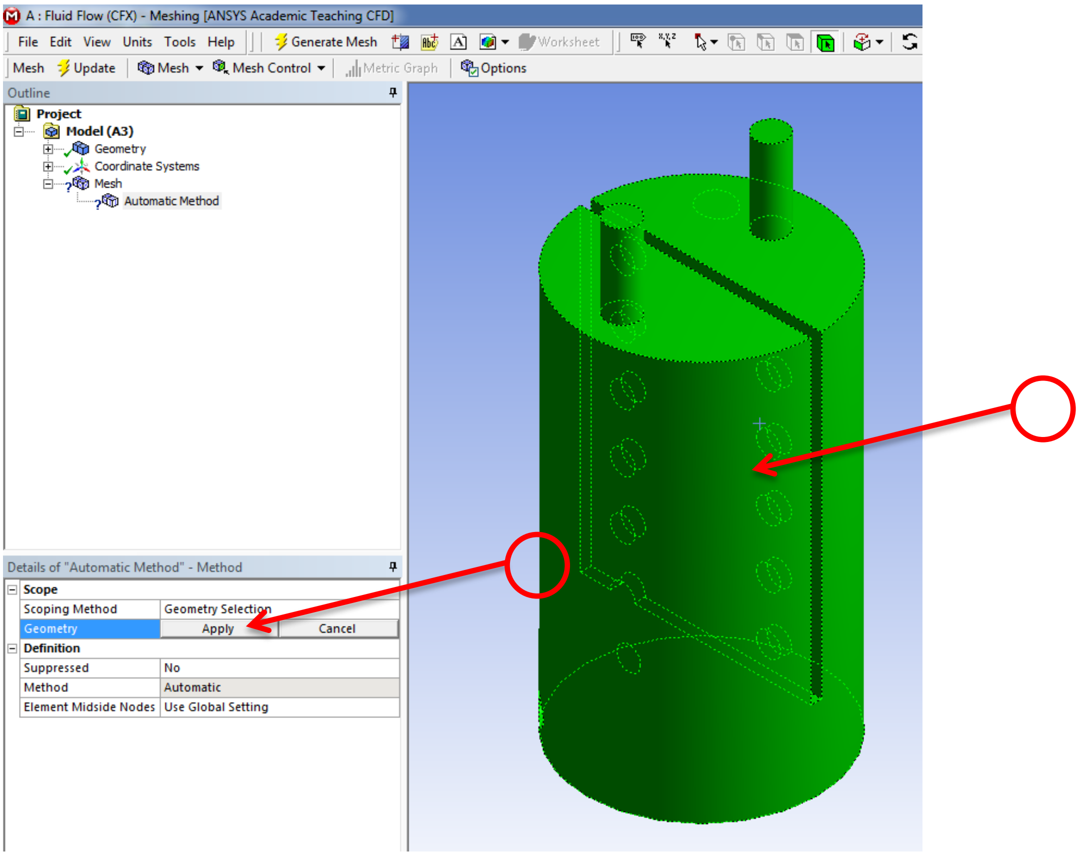
Position the cursor on the Mesh icon and press on the right button go to insert and select Method.



**Step 10:**

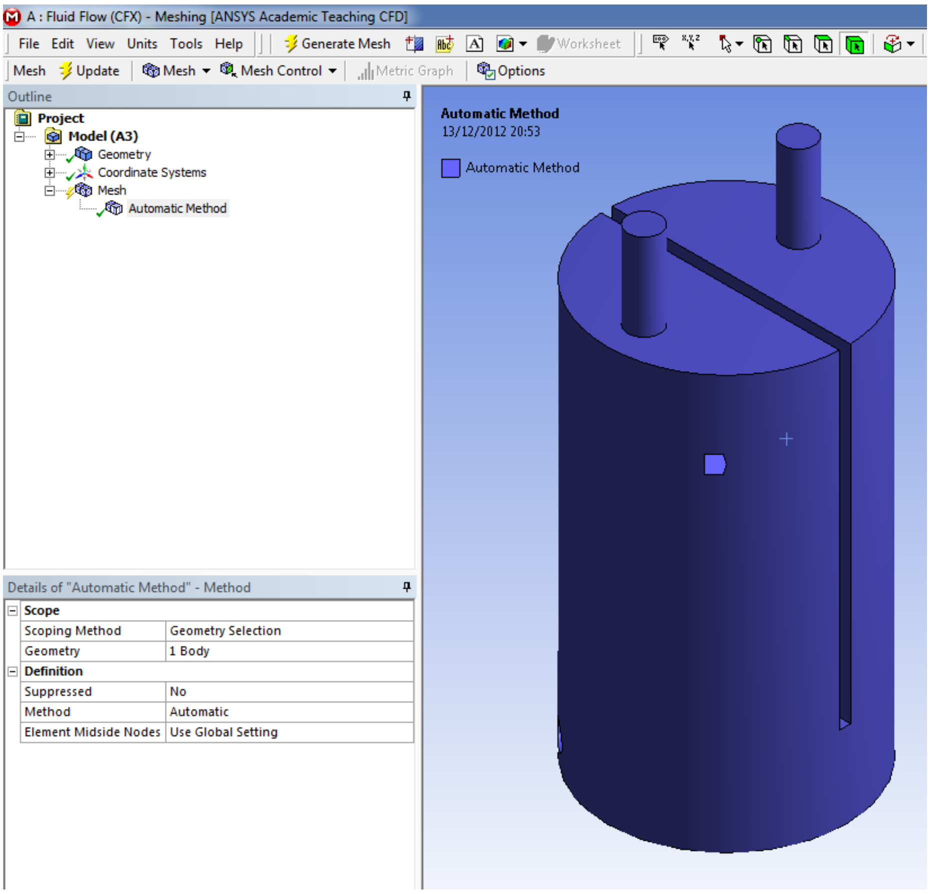
Position the cursor on the geometry then press the left mouse button to select the geometry.

As a result the geometry turns green. The next step then to press apply.



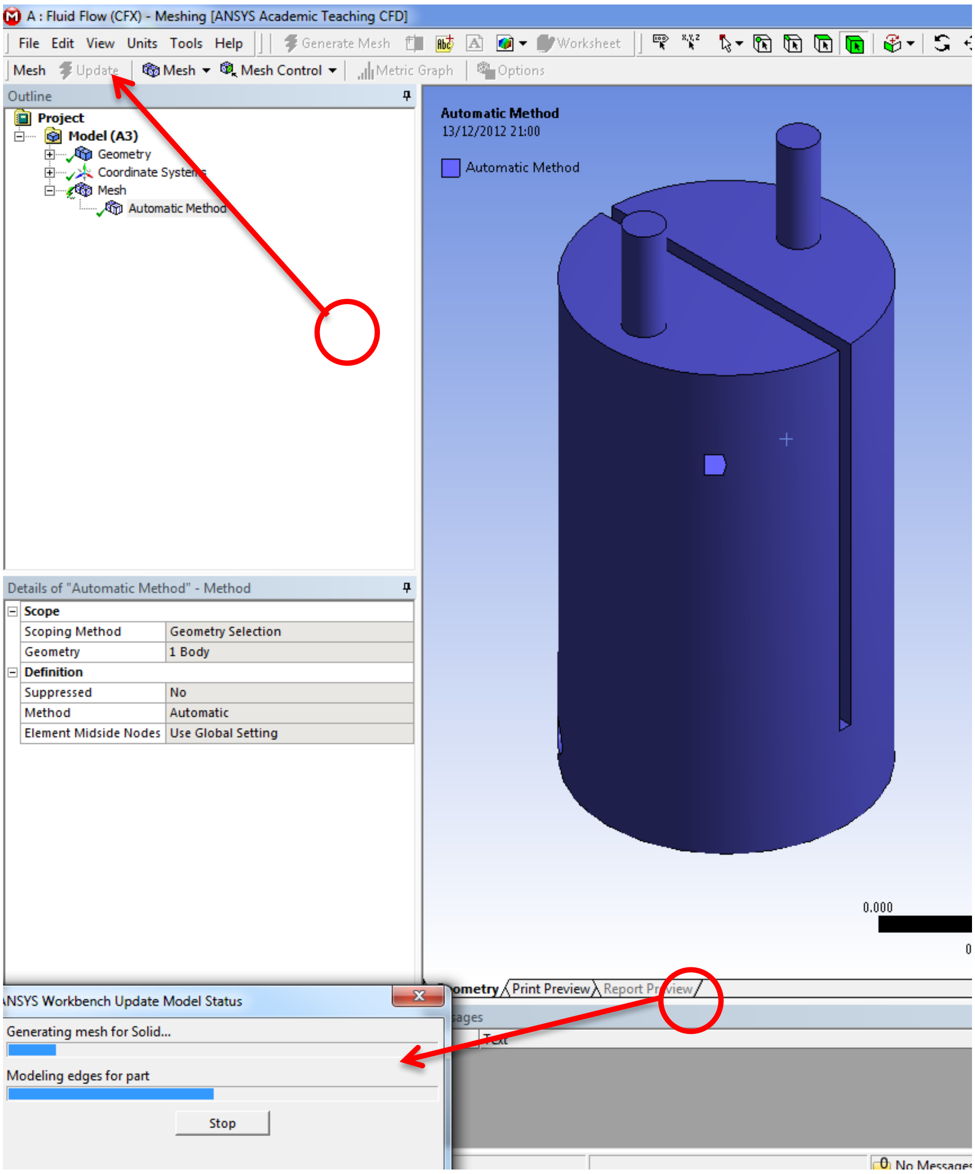
**Step 11:**

The geometry turns to a light purple color.



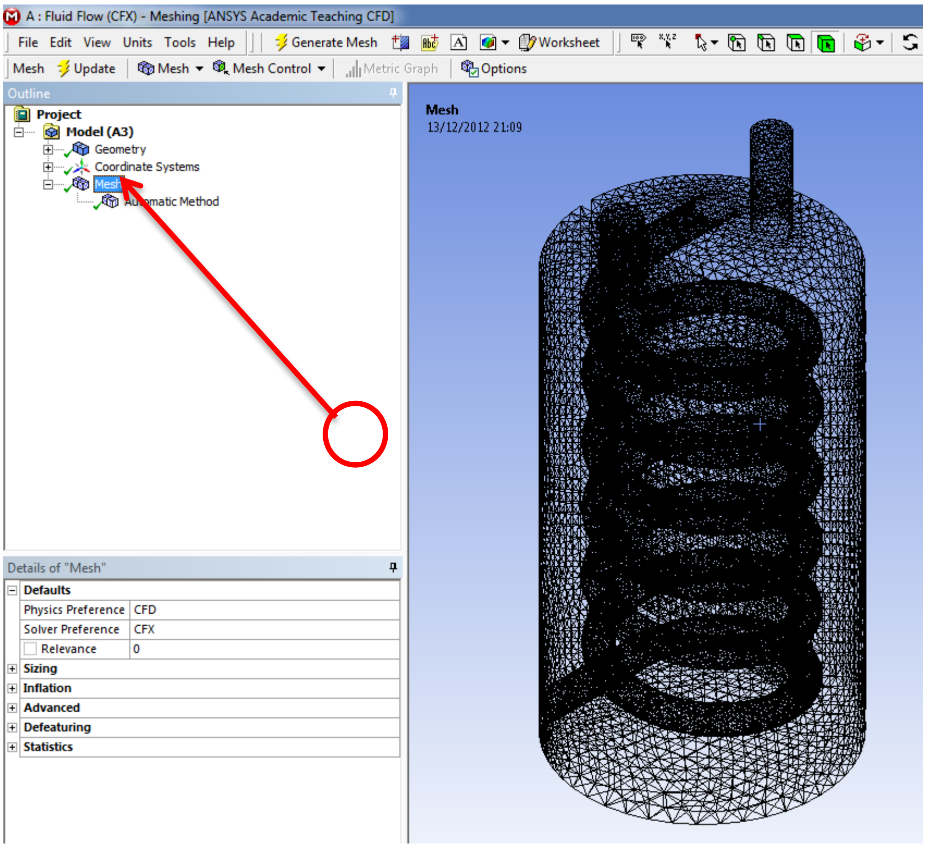
**Step 12:**

After pressing the update button a window at the bottom will show you the stage at which the Mesher packedge is in the meshgeneration process.



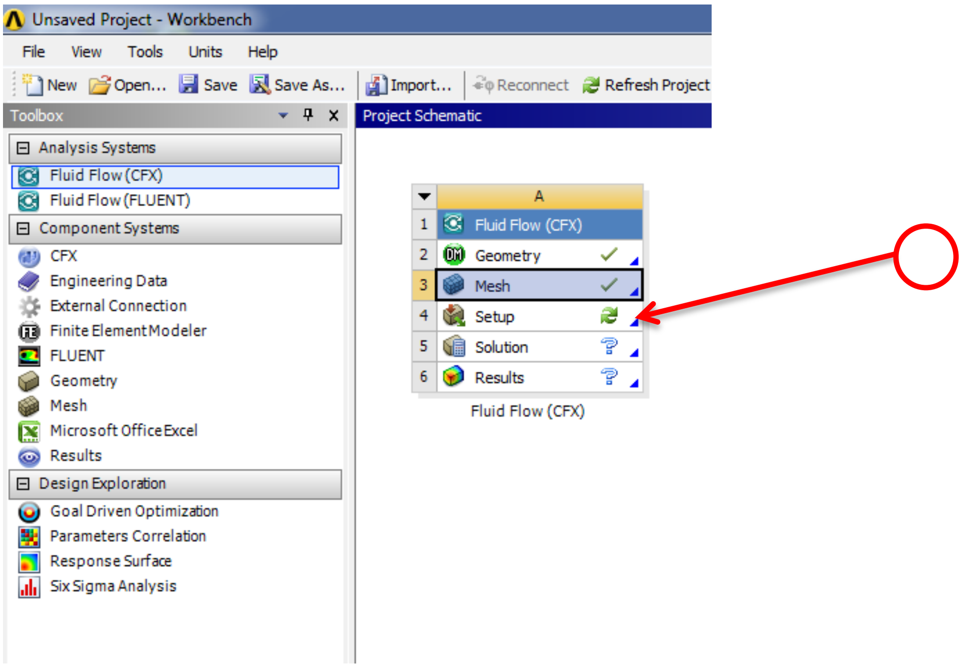
**Step 13:**

By pressing the left button onto the Mesh Icon the generated mesh should look something like this.



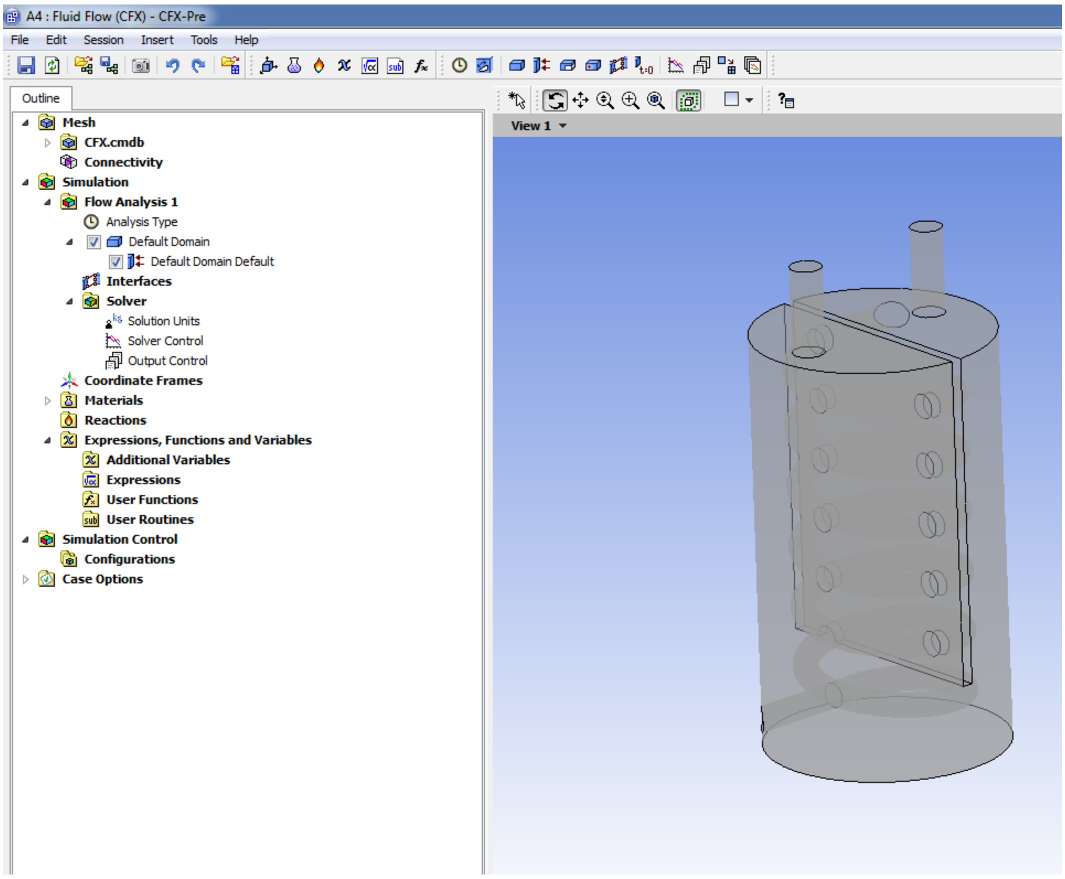
**Step 14:**

Check that there is a green tick sign beside the Mesh icon then double click on the Setup icon.



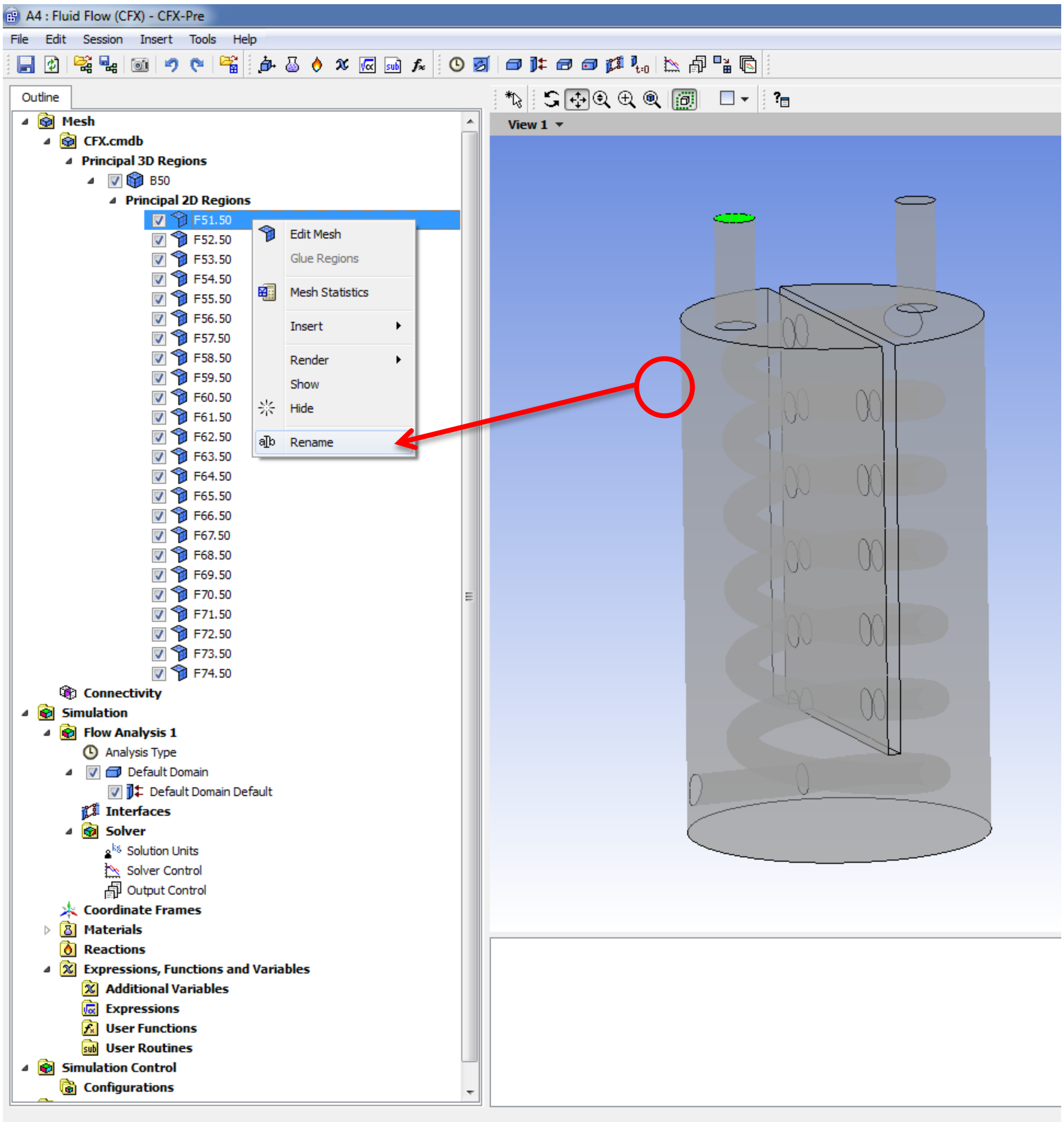
**Step 15:**

The CFX pre once it is lunched should look something like this.



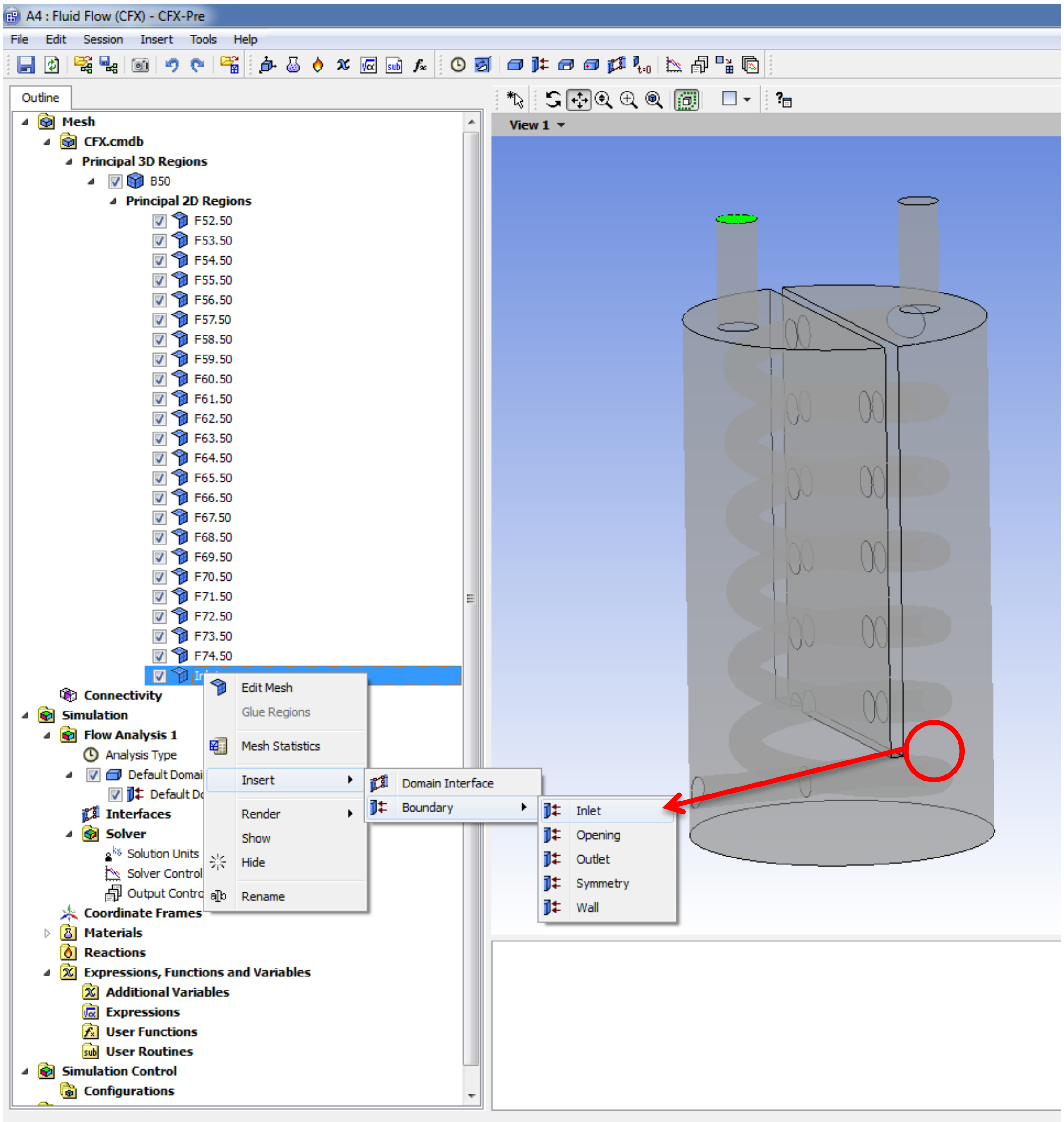
**Step 16:**

Go to the CFX.cmdb and from the drop down list for the Principial 2D Regions select the region numbered F51.50. Then right click and select the Rename option, this action will help us in applying a name to the inflow face.



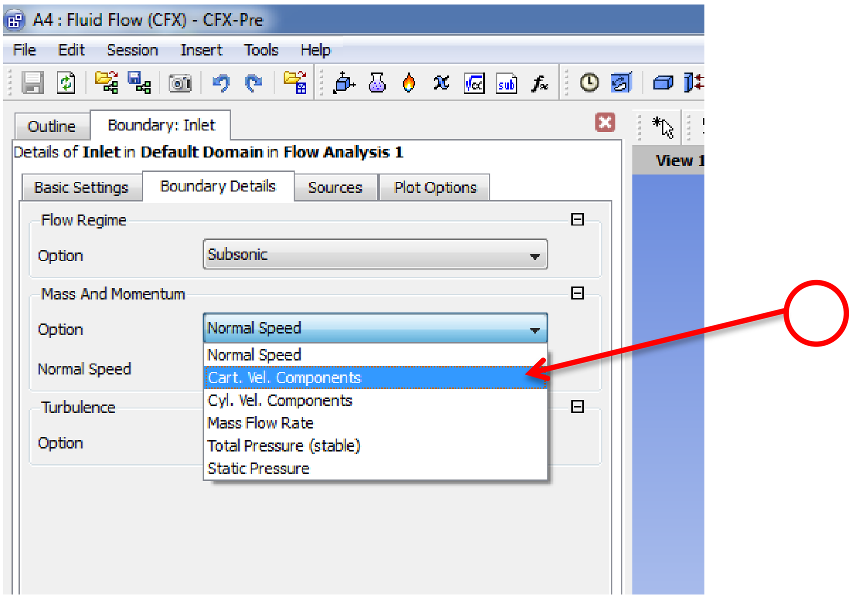
**Step 17:**

Select the renamed surface called inlet and the right click the mouse button and select Insert, then boundary then Inlet.



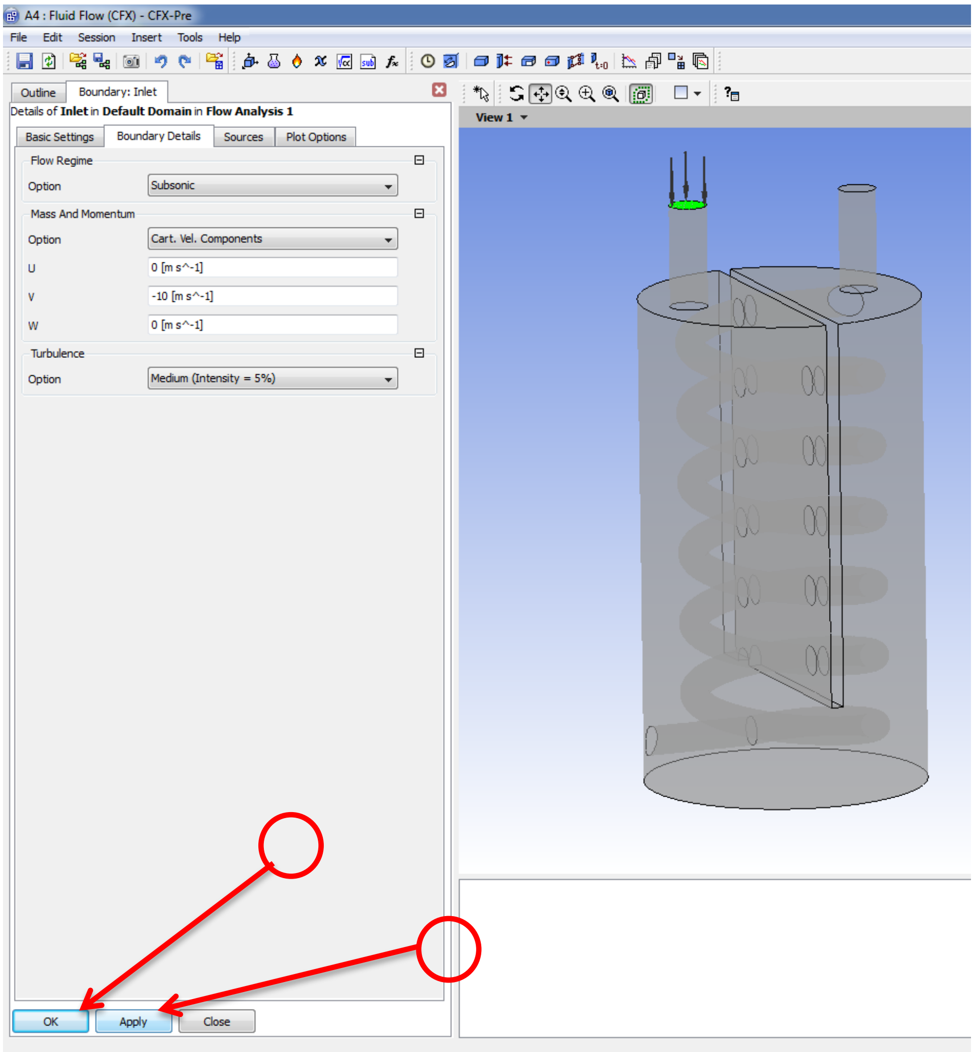
**Step 18:**

Once the inlet option is selected go to boundary details and under Mass and Momentum select the Cart. Vel. Components. In the main view look at the coordinate system shown on the bottom right hand side this will clarify that the directions of the selected velocities is correct.



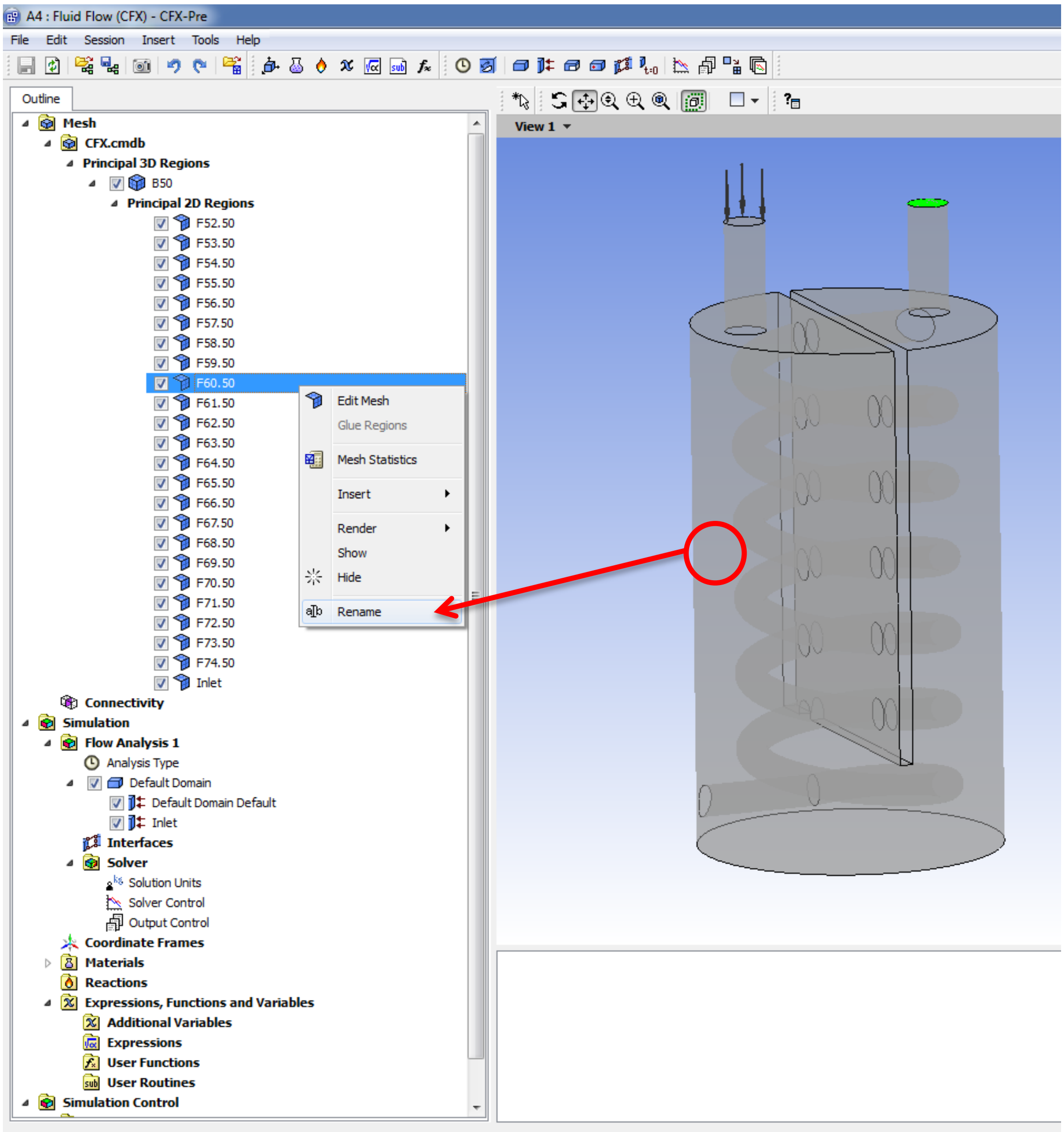
**Step 19:**

Enter -10 into the V velocity cell, and enter the value of zero to the U and W cell, then press apply and Ok.



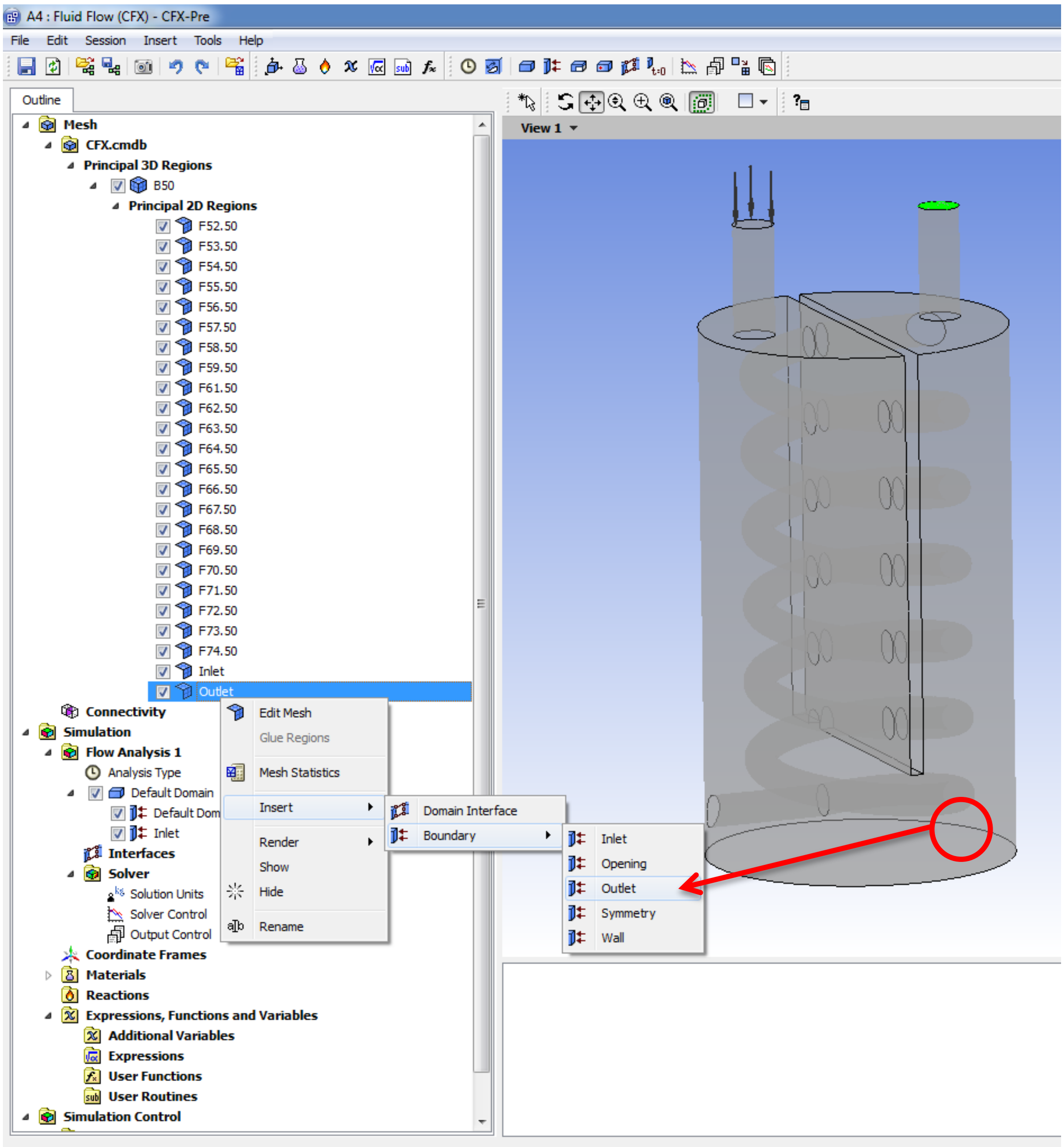
**Step 20:**

Go to the CFX.cmdb and from the drop down list for the Principial 2D Regions select the region numbered F60.50. Then right click and select the Rename option, this action will help us in applying a name to the outflow face.



**Step 21:**

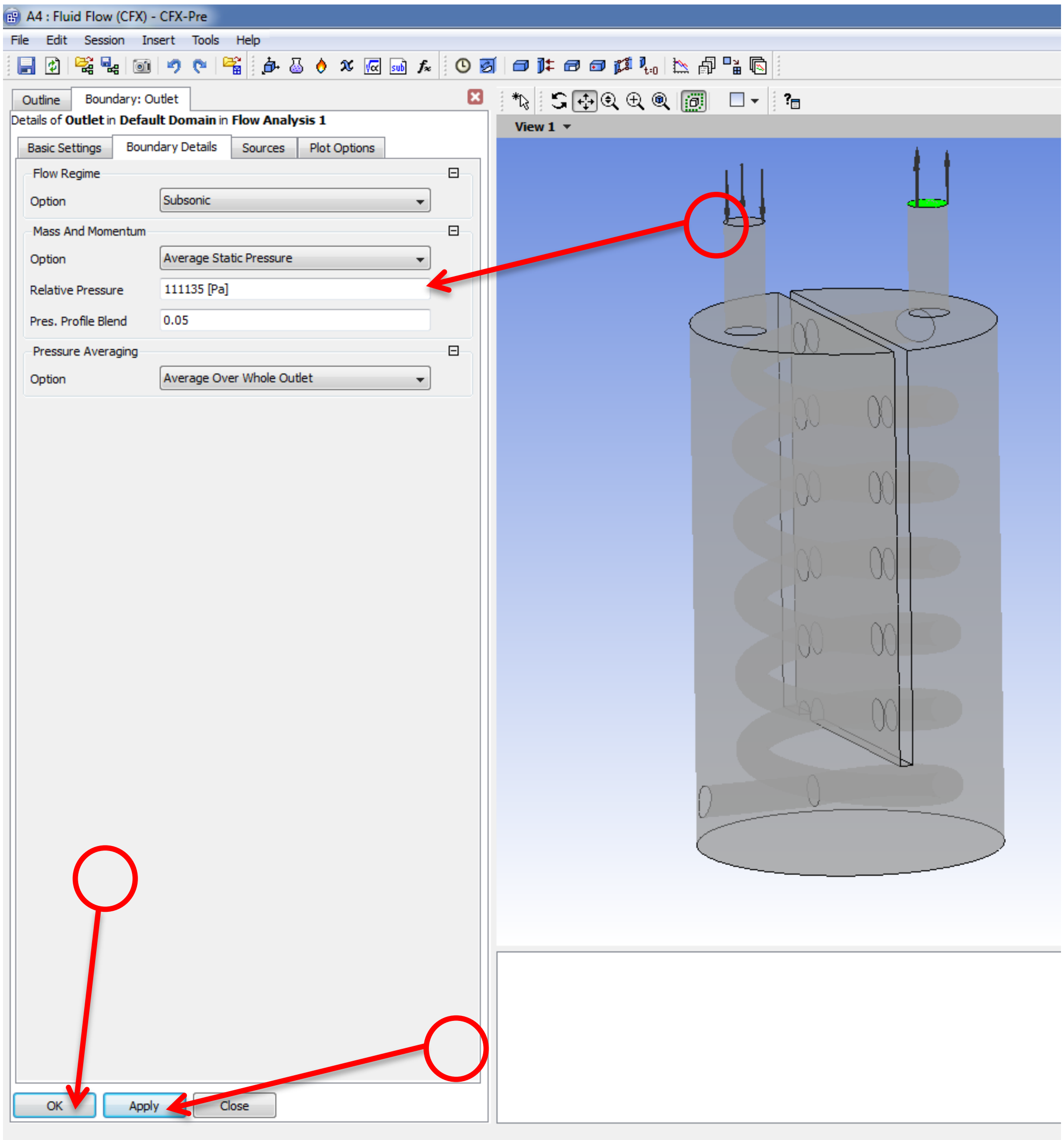
Select the renamed surface called outlet and the right click the mouse button and select Insert, then boundary then select outlet.



**Step 22:**

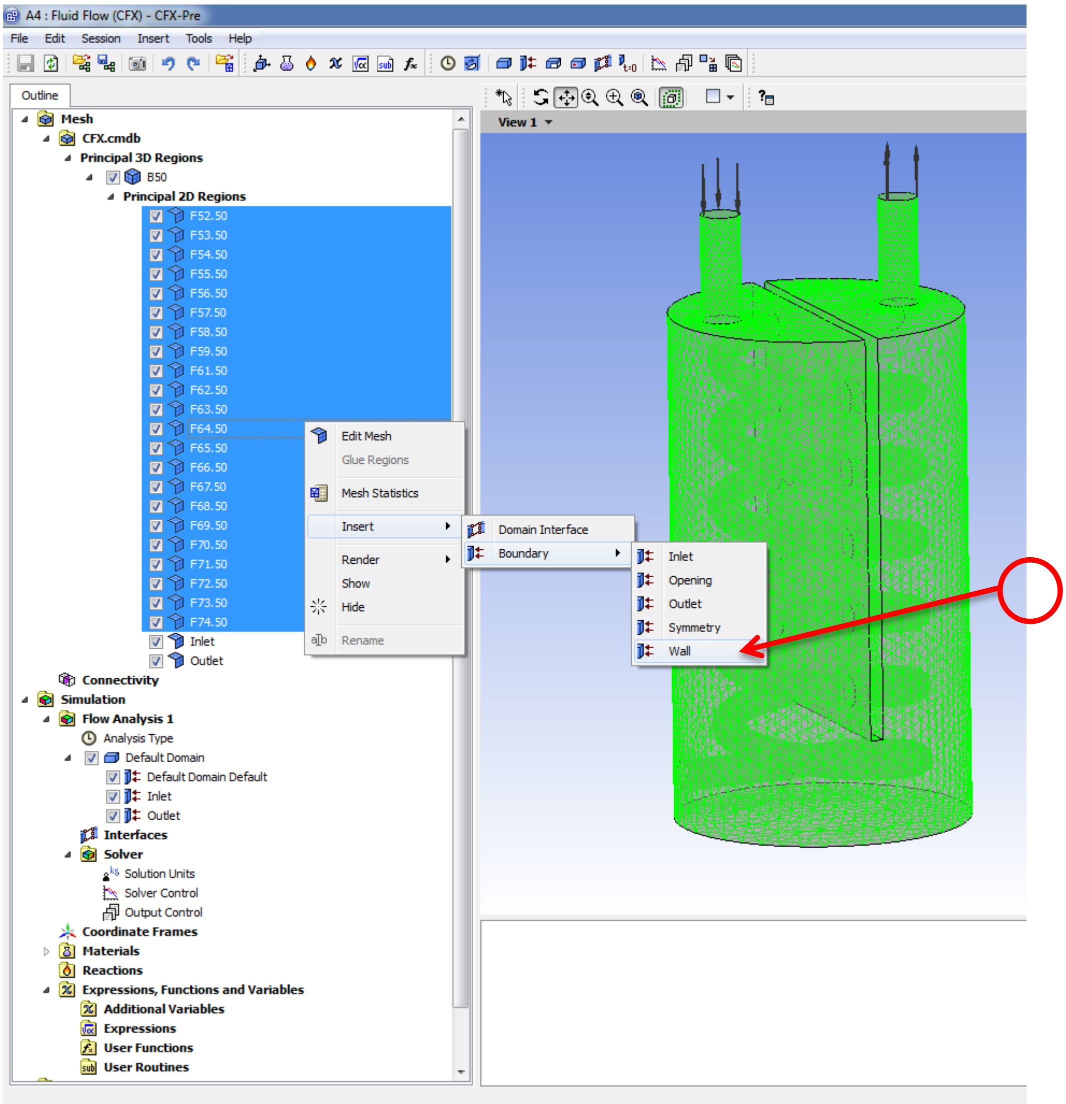
Under boundary Details, Under the Mass and Momentum go to option and select from the drop down list Average Static Pressure, then enter the value of 111135 value into the Relative Pressure cell, then press apply then Ok.

The value is calculated $P = \rho g h + P_{atm} = 1000 \times 9.81 \times 1 + 101325 = 111135$ Pa



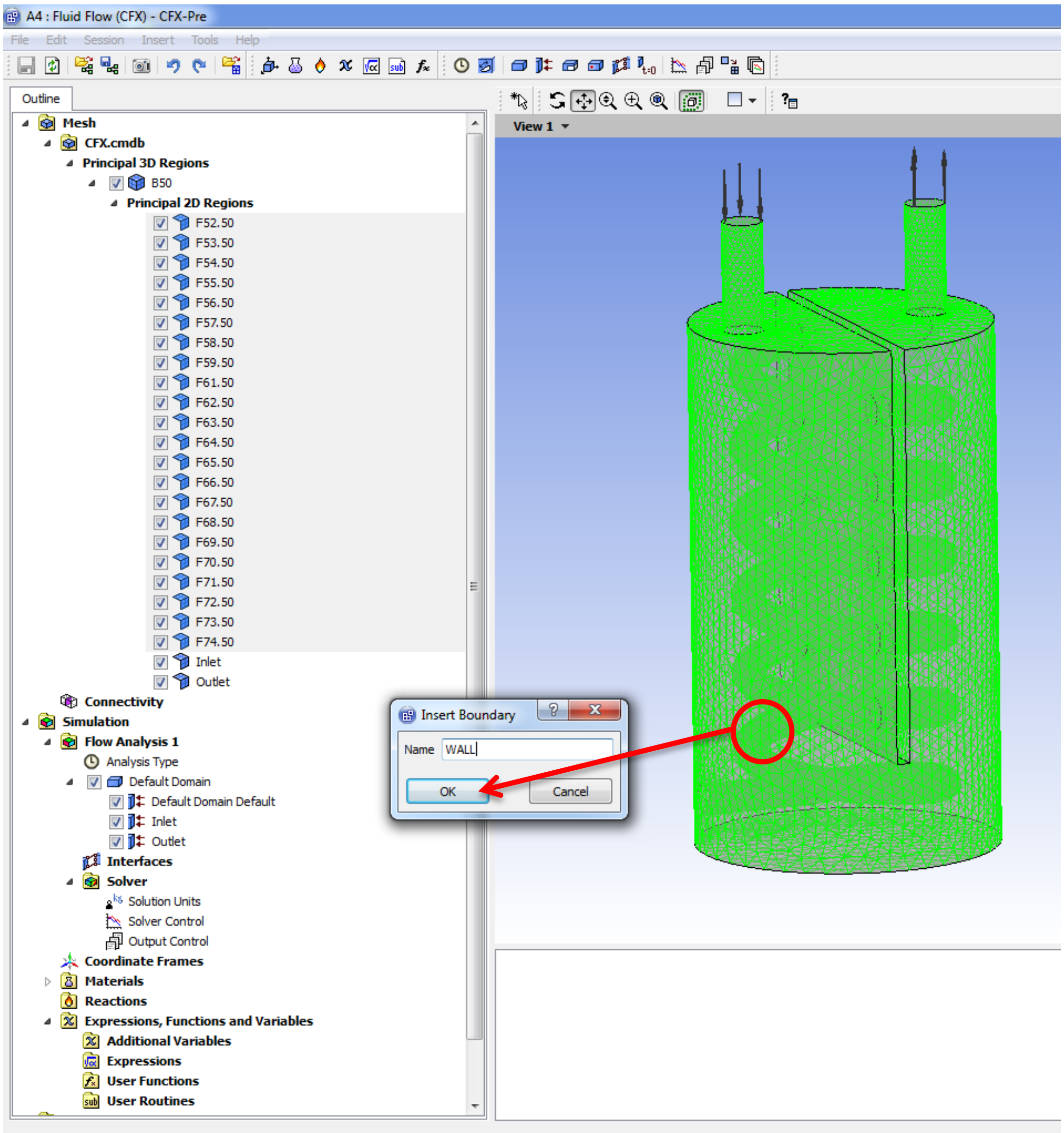
**Step 23:**

Select the rest of the surfaces then then right click, go to insert ,then go to boundary, then select Wall.



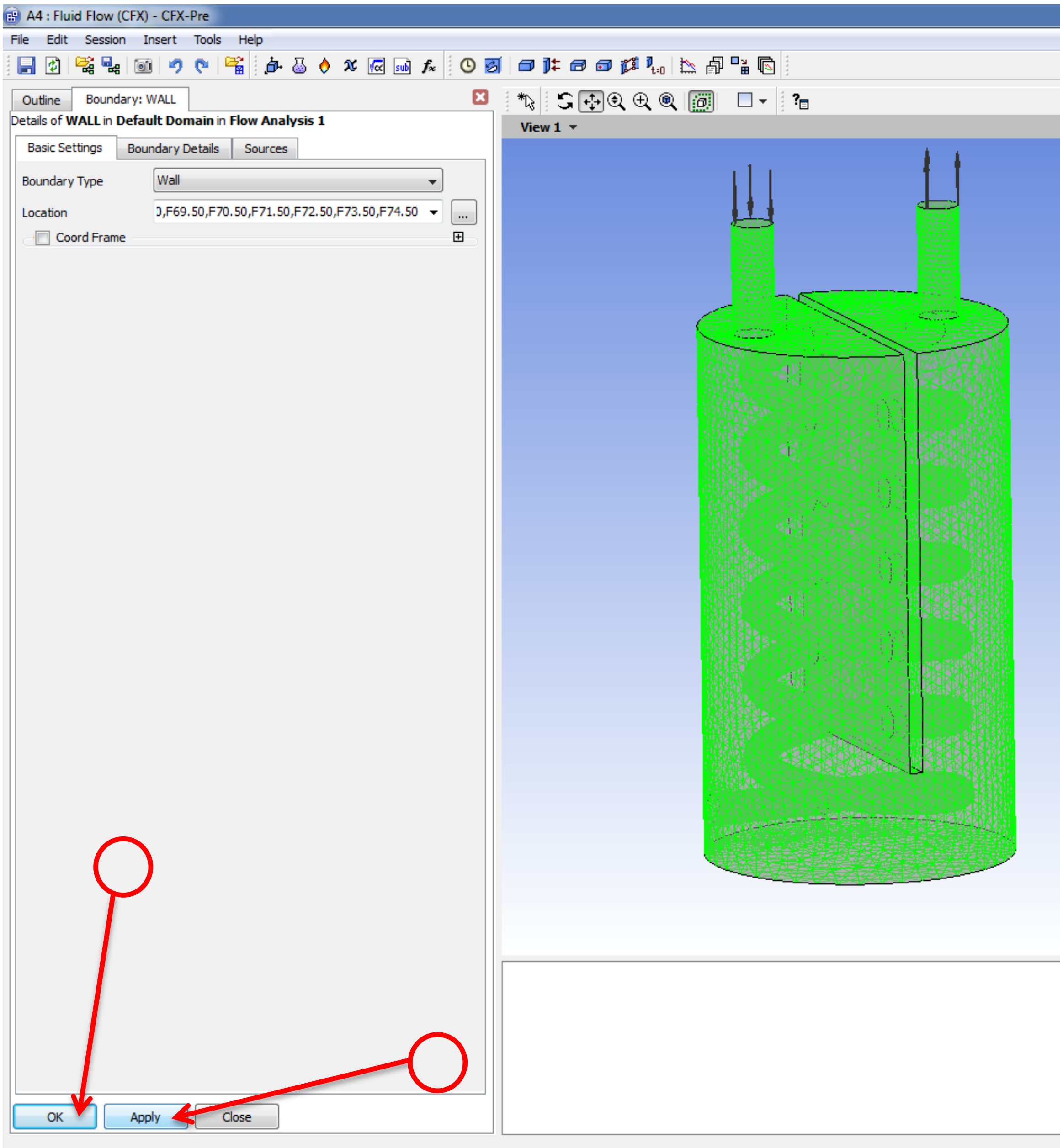
**Step 24:**

Enter in the name cell: WALL and press Ok.



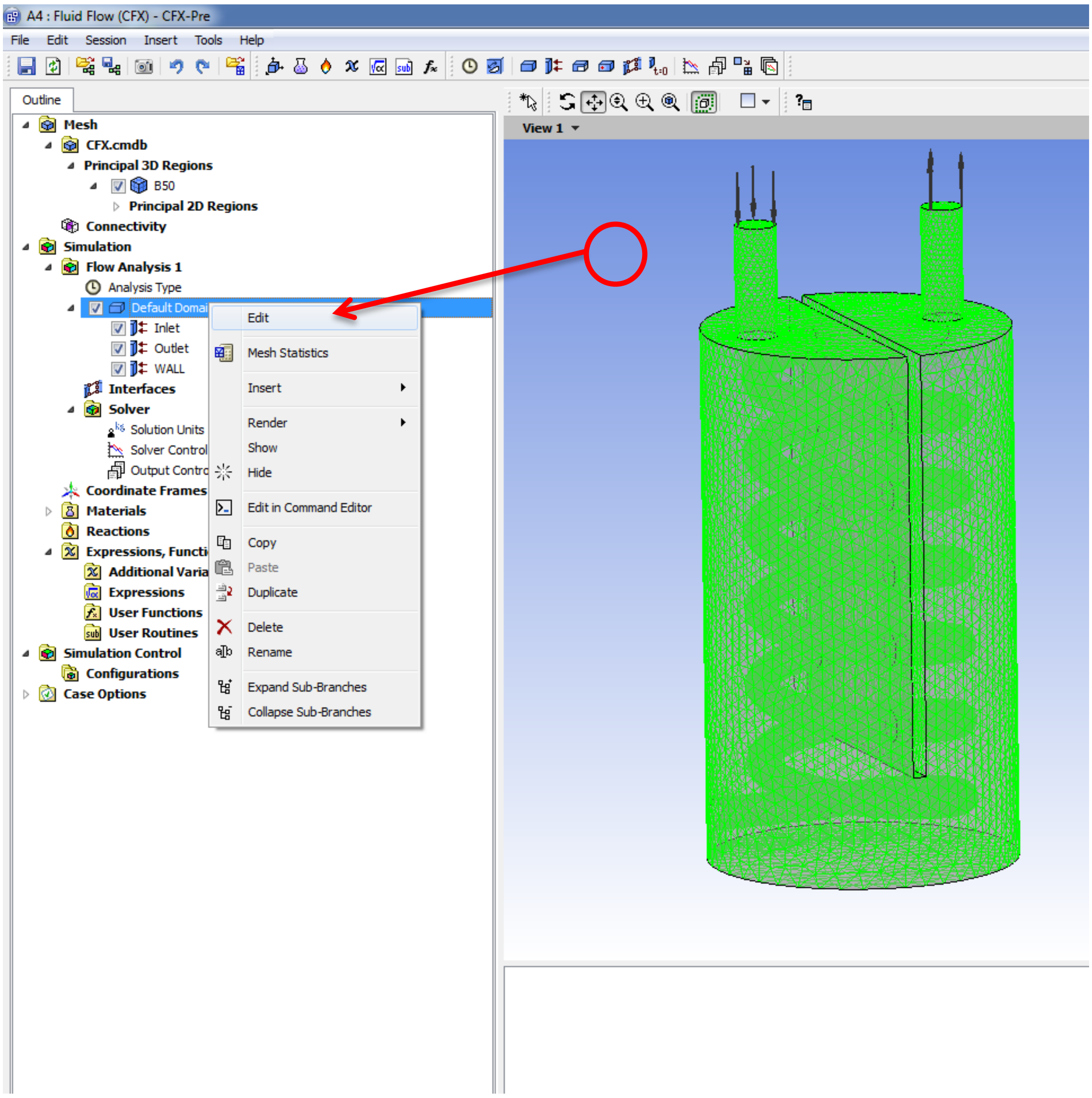
**Step 25:**

The selected surfaces are highligted in green press Apply then press Ok.



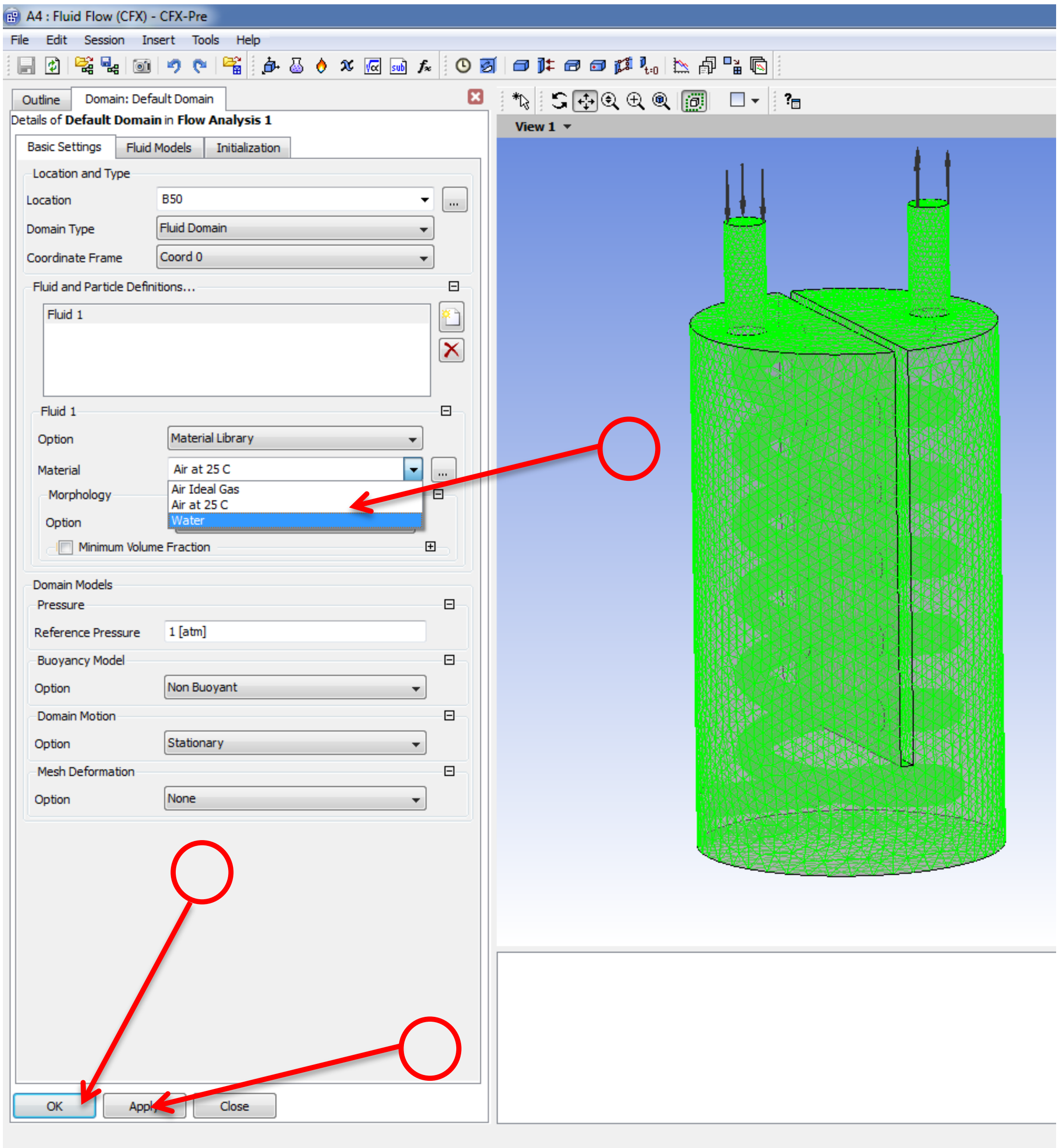
**Step 26:**

Position the cursor over the Default Doman then select it using the left mouse button. Right click and then select the Edit option.



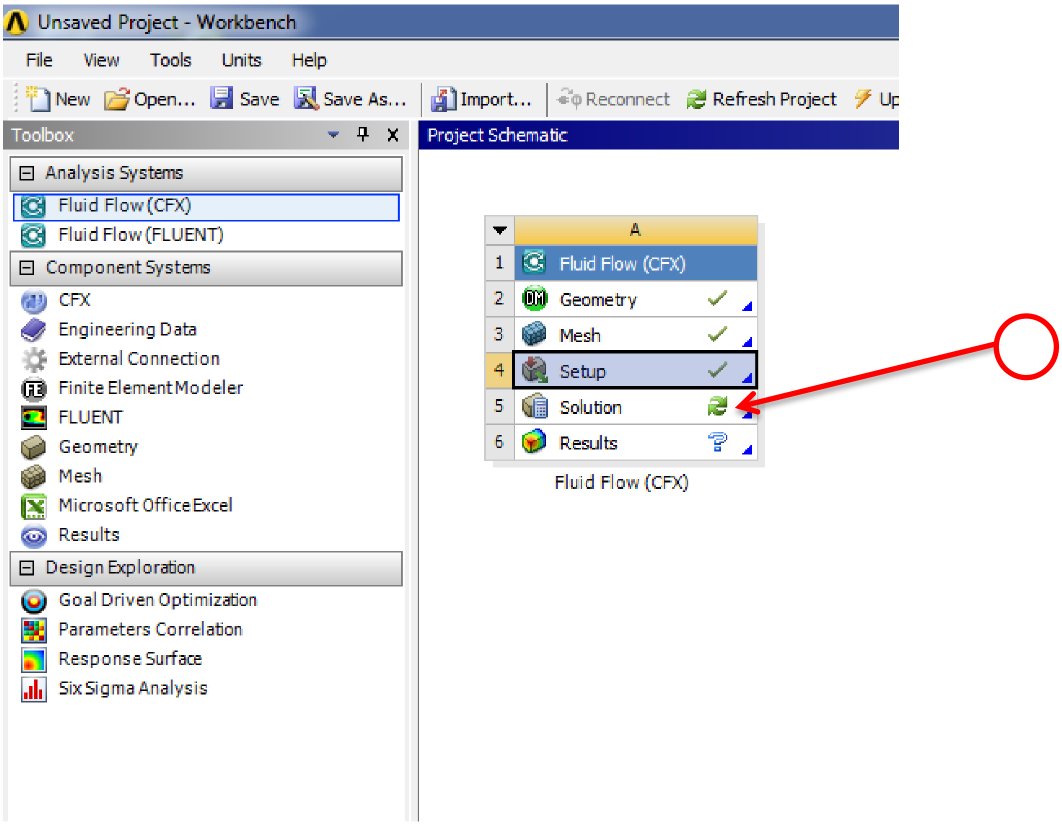
**Step 27:**

Go to materials select from the drop down list Water, then press apply, then press Ok.



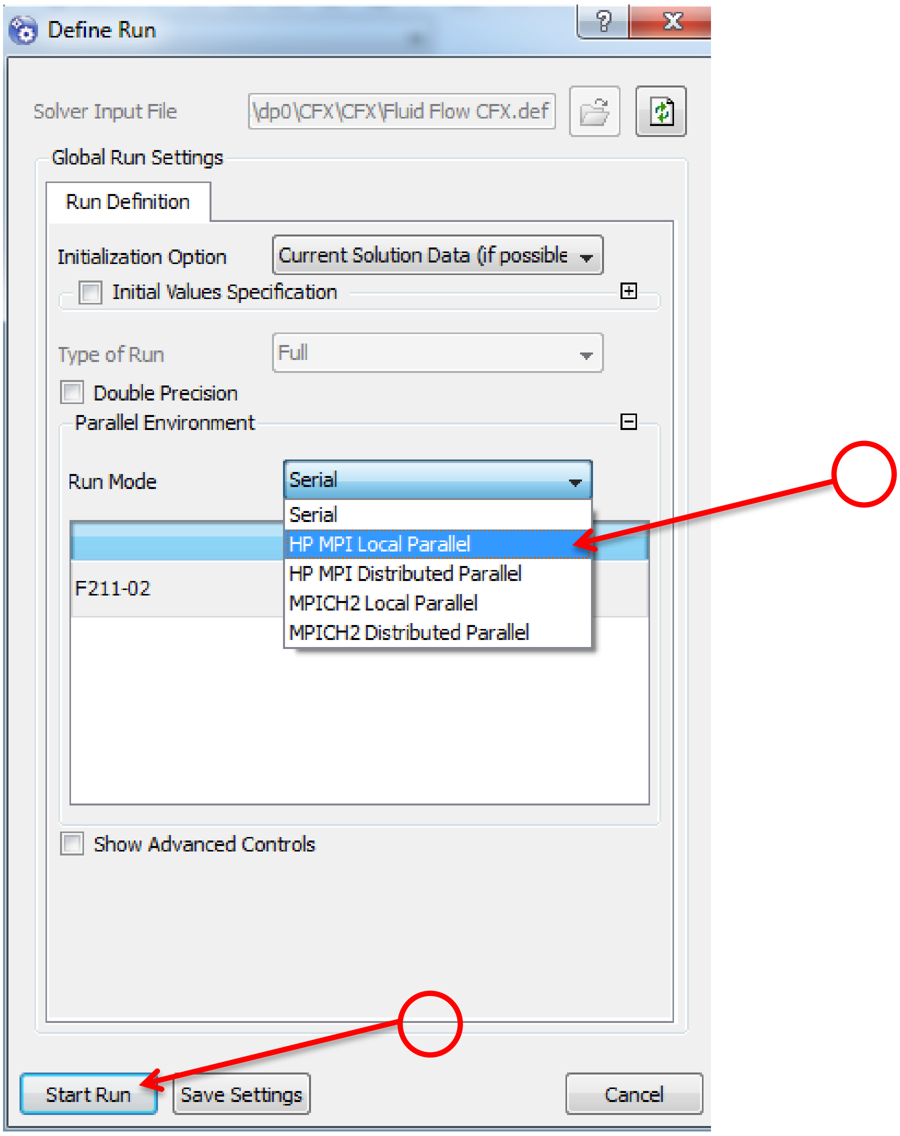
**Step 28:**

Check there is a green tick sign beside Setup, then double clip on the Solution icon.



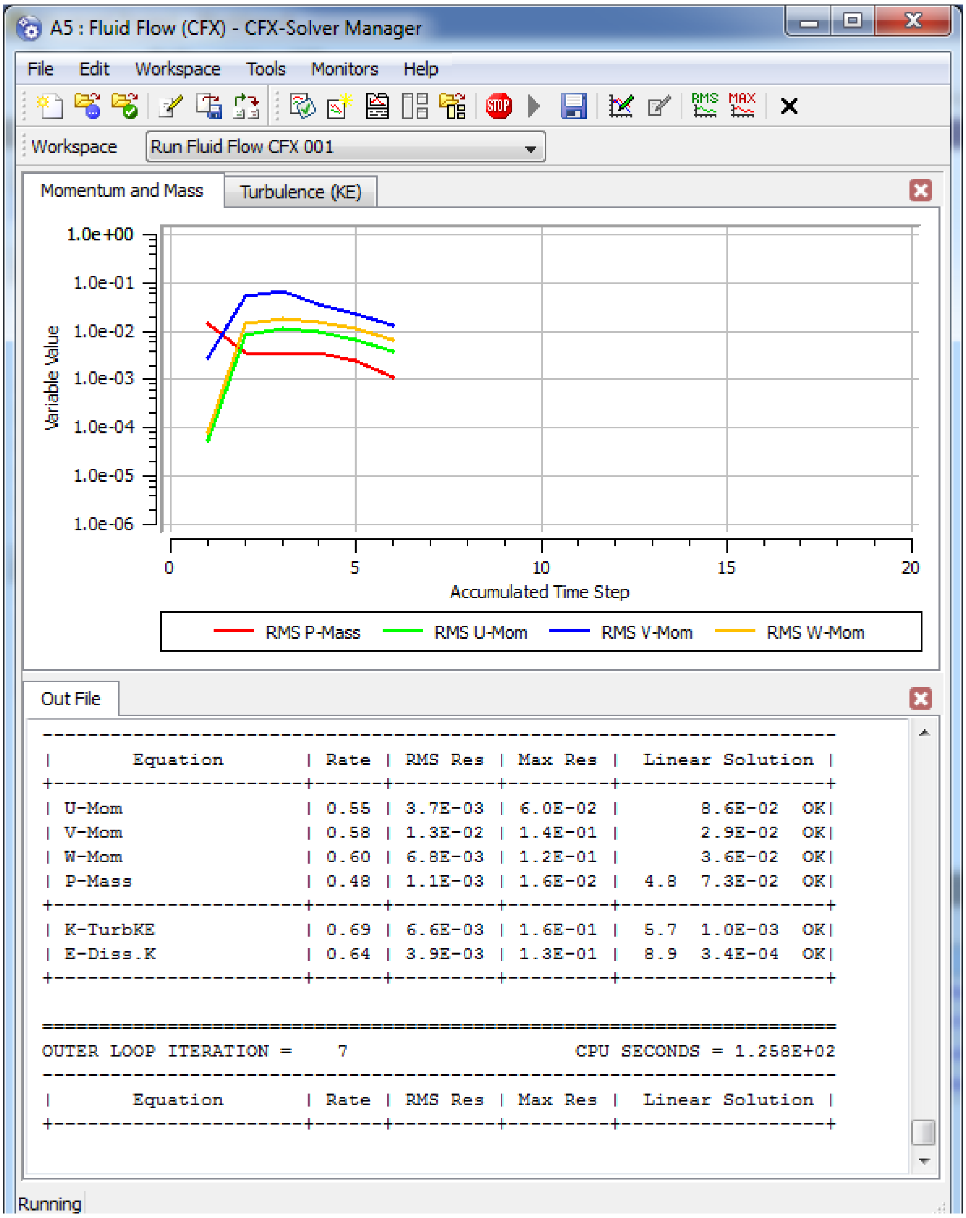
**Step 29:**

Select the HP MPI Local Parallel, this will enable the user to conduct a parallel core calculation on the desktop he is using, then press on Start Run.



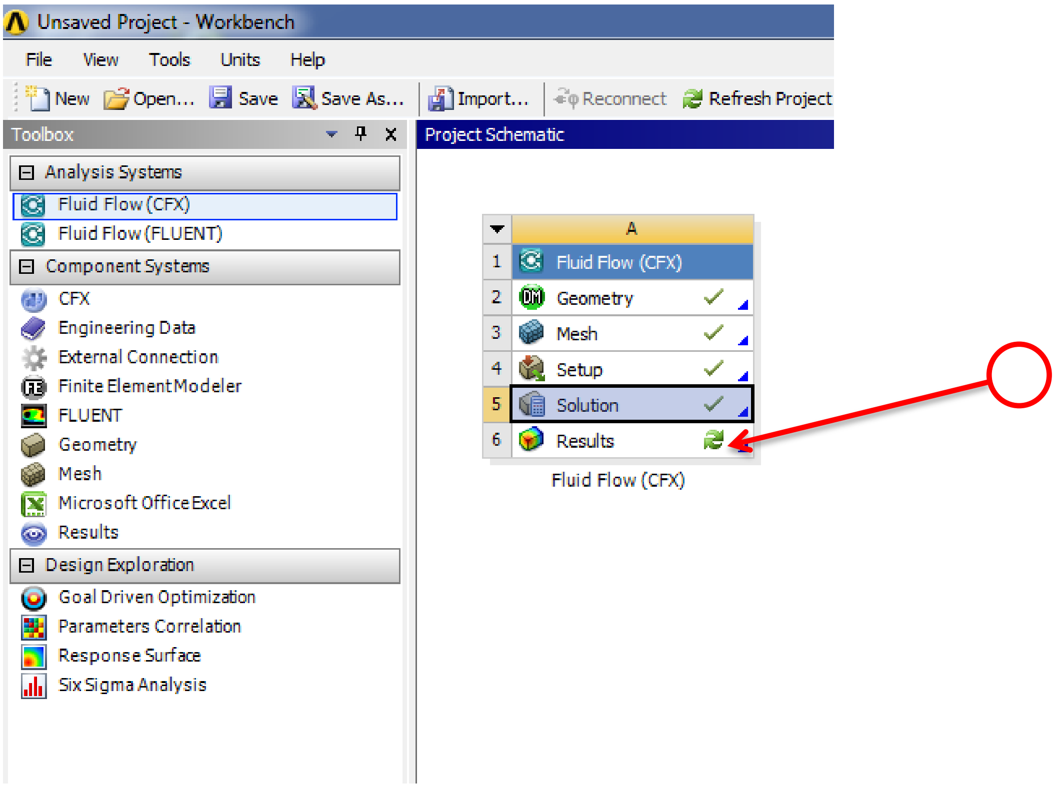
**Step 30:**

Once the calculation Kicks off, the user can follow up the progress of his calculation through the CFX-Solver Manager window. The calculation is setup by default to stp after 100 iterations or if reaches its cut-off criteria.



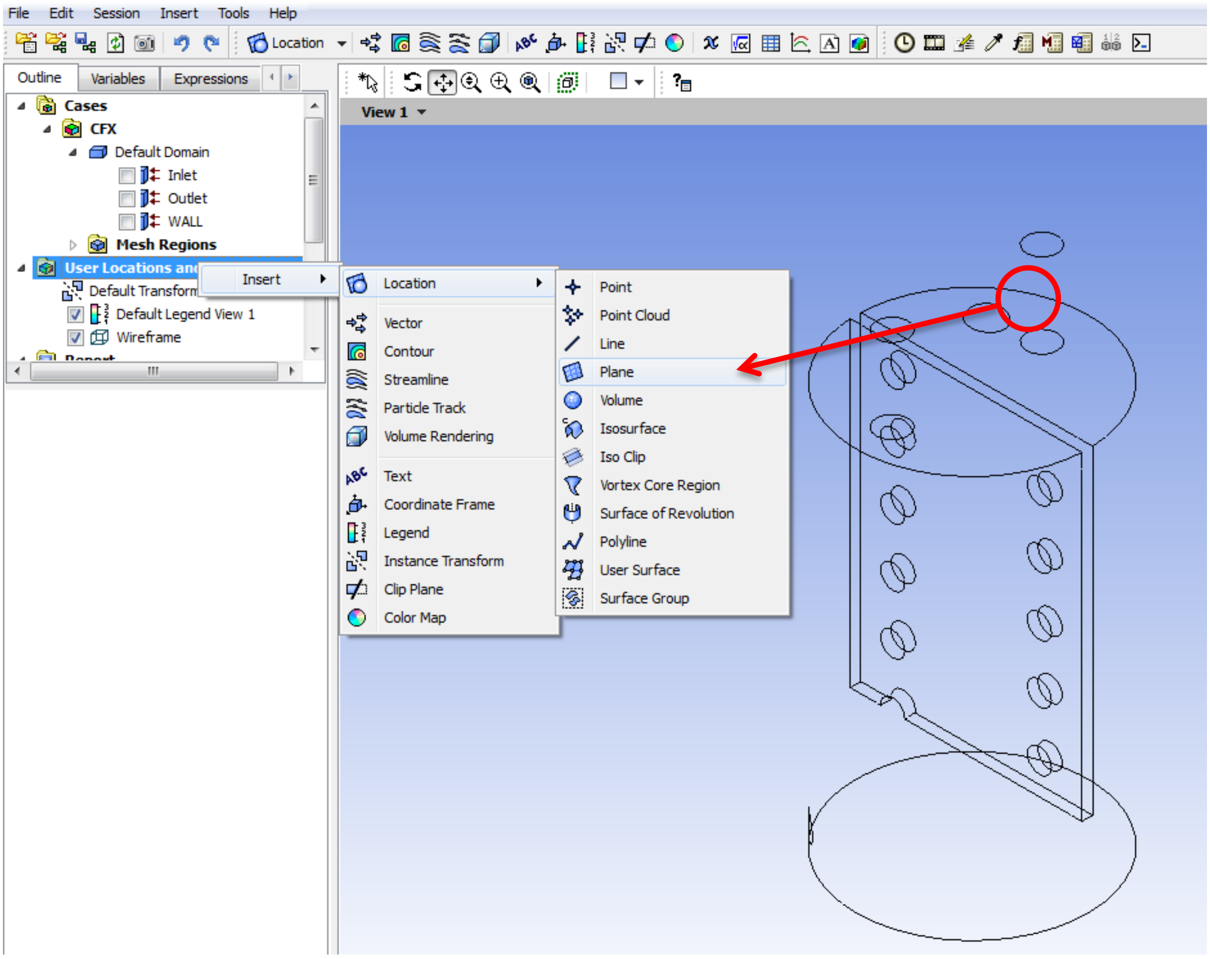
**Step 31:**

Check there is a green tick beside the Solution icon, then double click on the Results icon.



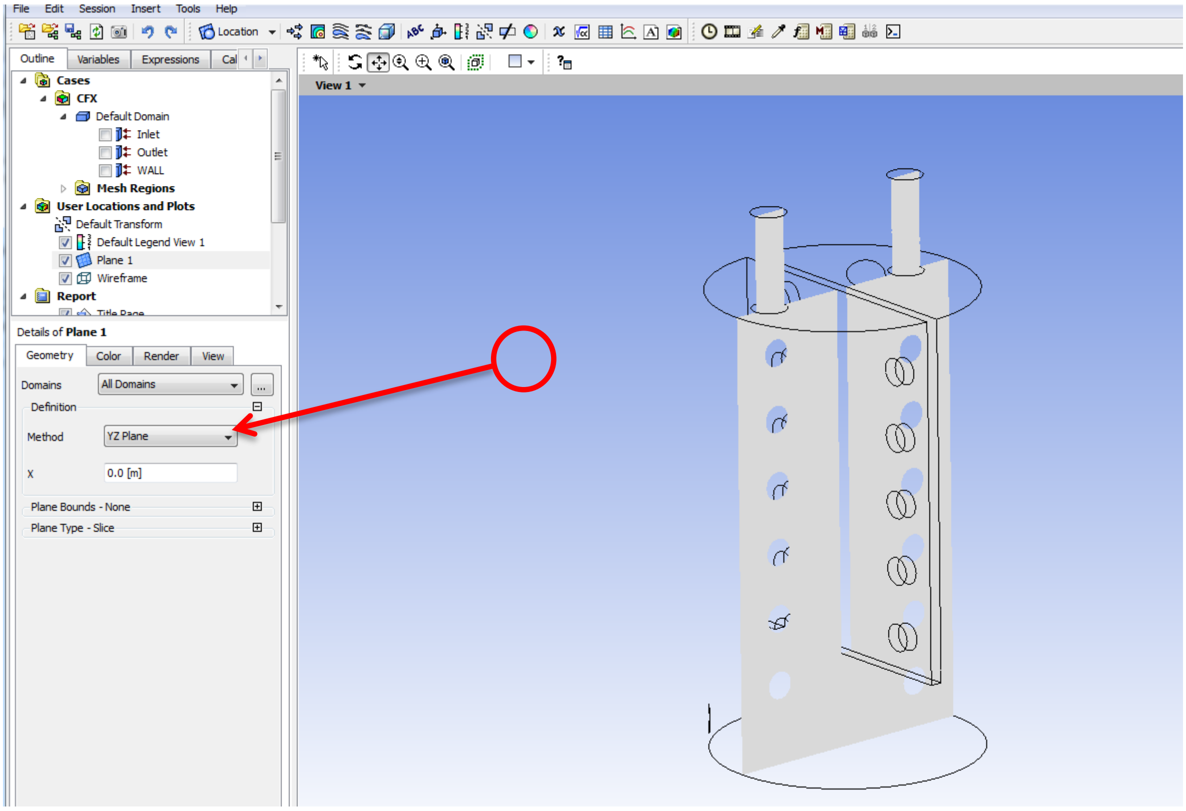
**Step 32:**

Left click the cursor on the User Locations and Plots icon, then go to insert then locations and select plane, a window will open with a plane default name of plane 1 highlighted in blue changing its name is up to the user.



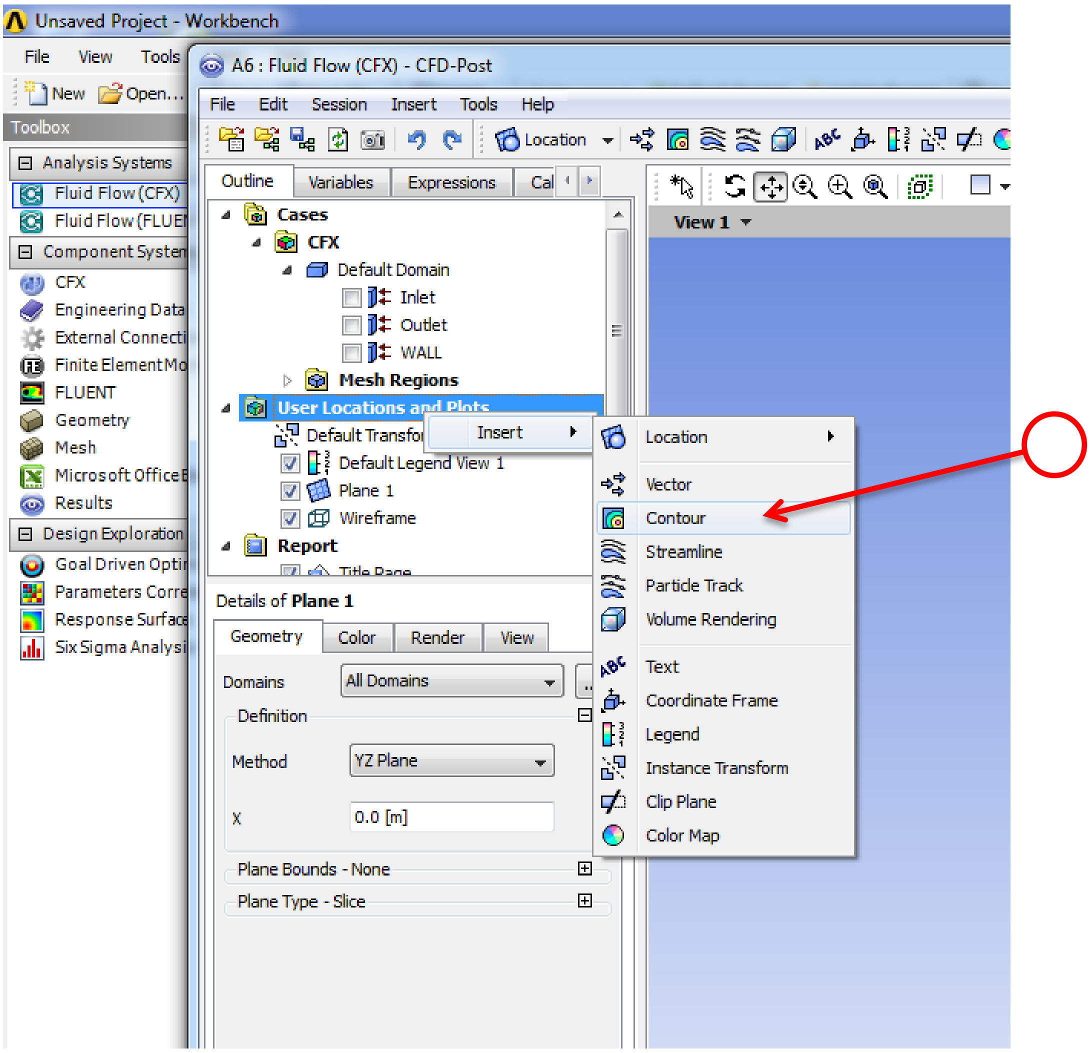
## Step 33:

After pressing the apply button you should see a cross sectional plane (grey in color) running through the domain in the yz plane.



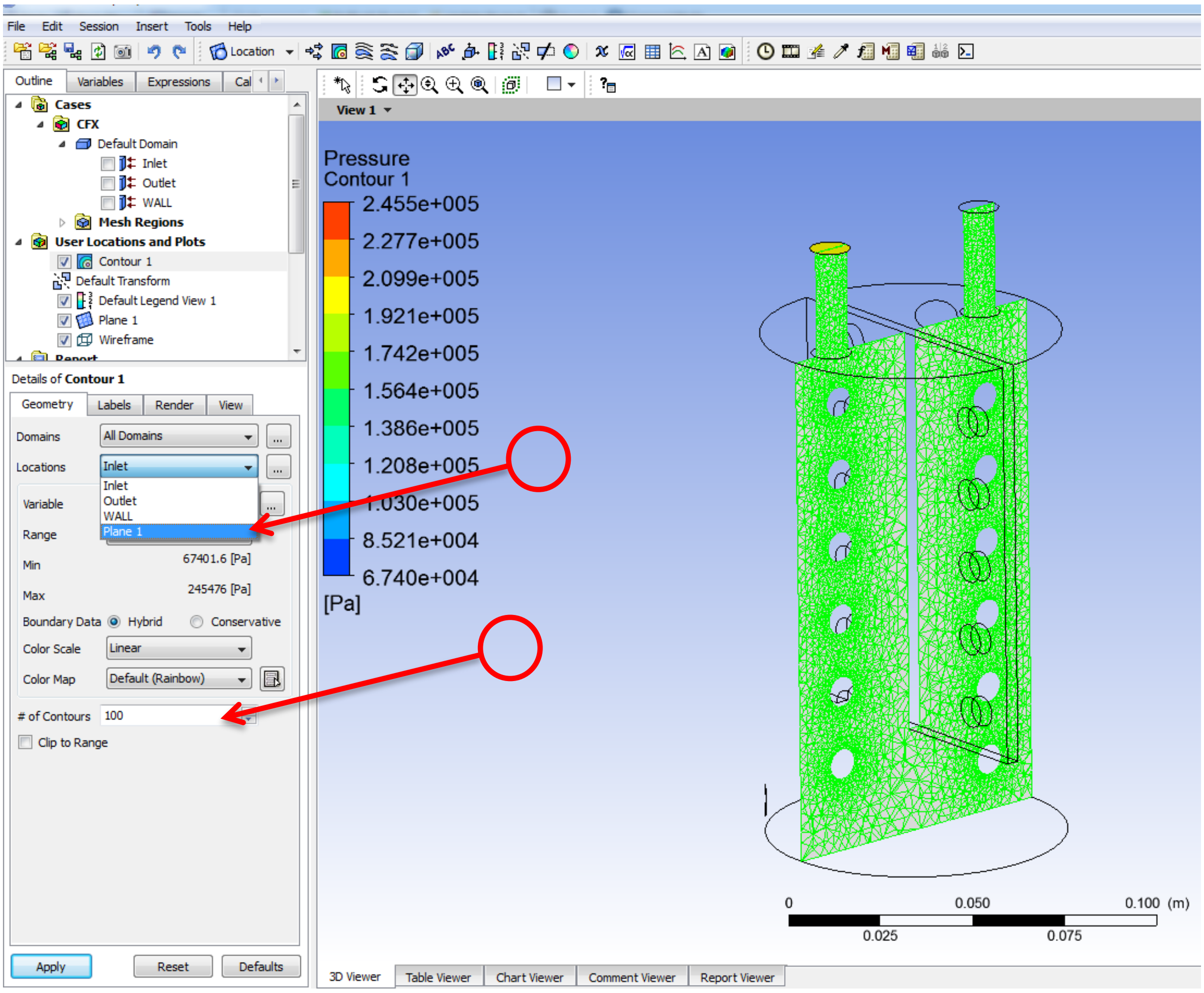
**Step 34:**

Left click the cursor on the User Locations and Plots icon, then go to insert then locations and select contour,



**Step 35:**

A new side window will open which has the details of the created plane, go to locations and chose Plane1 from the drop down list. Then go to (# of Contours) and enter into the input cell 100.



**Step 36:**

The next step is to select the Velocity Variable from the drop down list, then go to Range and select from the drop down list Local. Finally press apply and what would be visible the velocity profile of water. The user has required knowledge now to continue on his own.