Flow Modelling in a Porous-Fluid Domain using ANSYS CFX

First Edition

Ahmed Al Makky

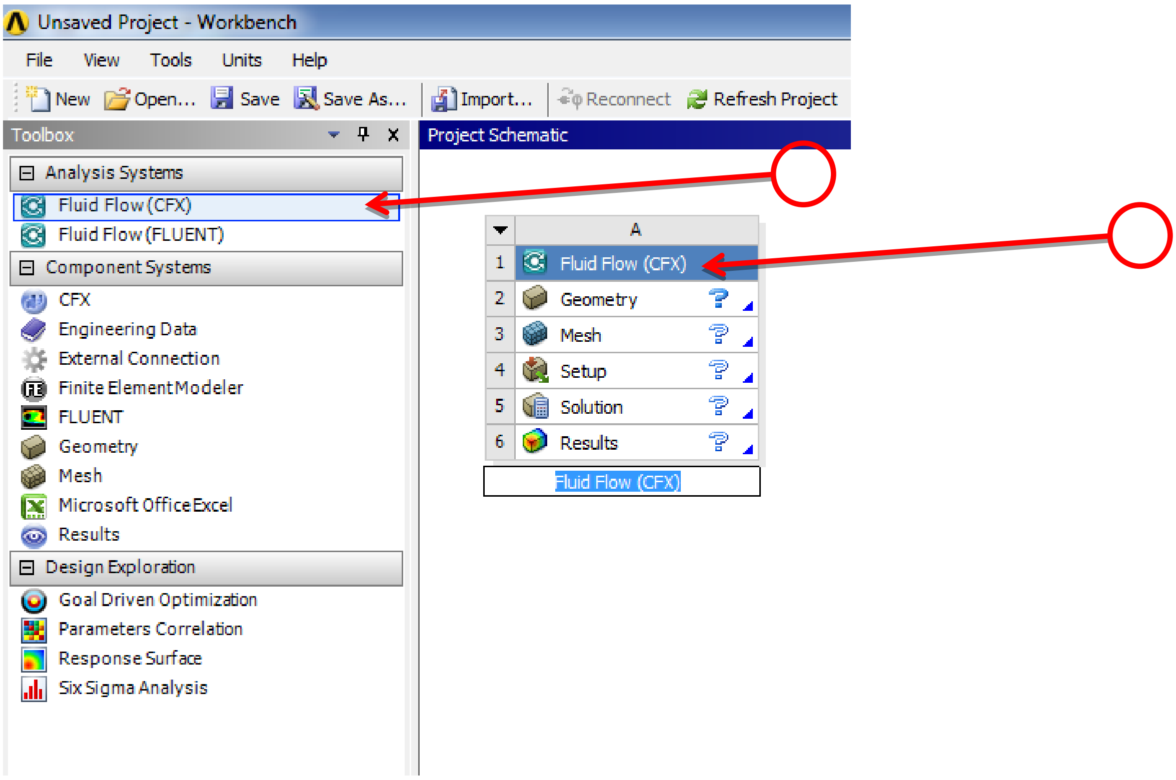
Flow Modelling in a Porous-Fluid Domain using ANSYS CFX

Introduction

The tutorial was written in a rush so it has spelling mistakes never got 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. Porous fluid problems are encountered in lots of engineering applications, in soils , in food , in filters, ….etc.

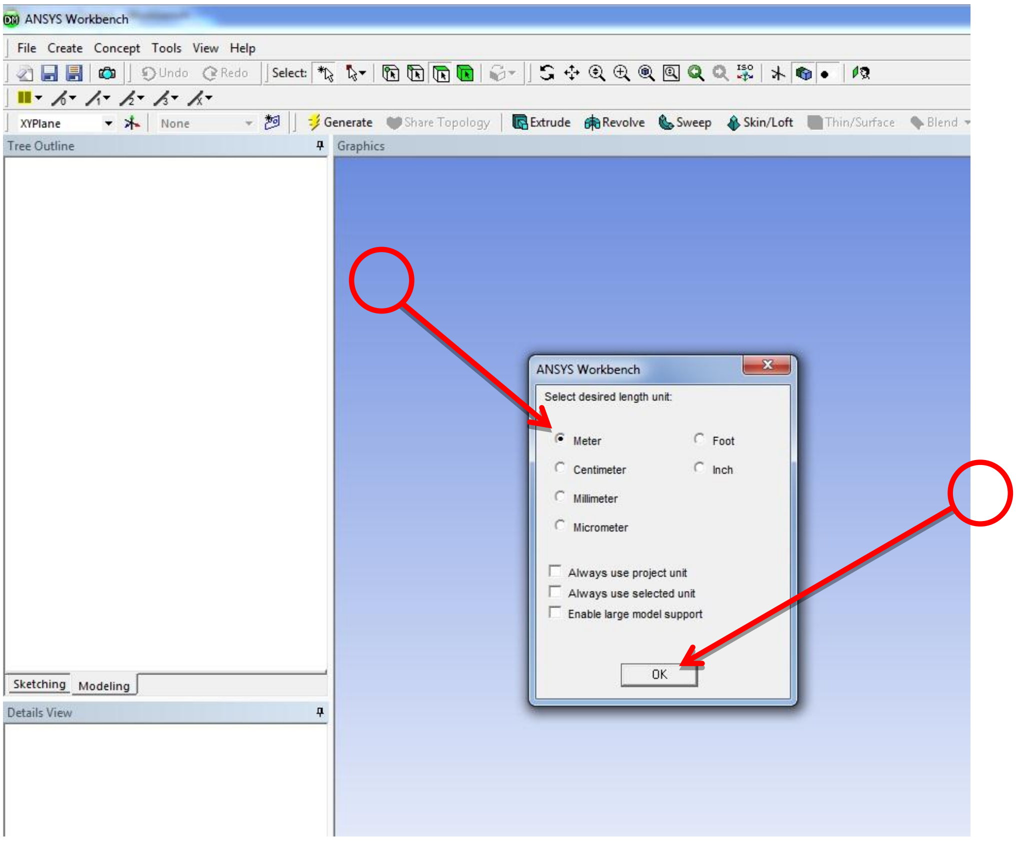
Step 1:

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



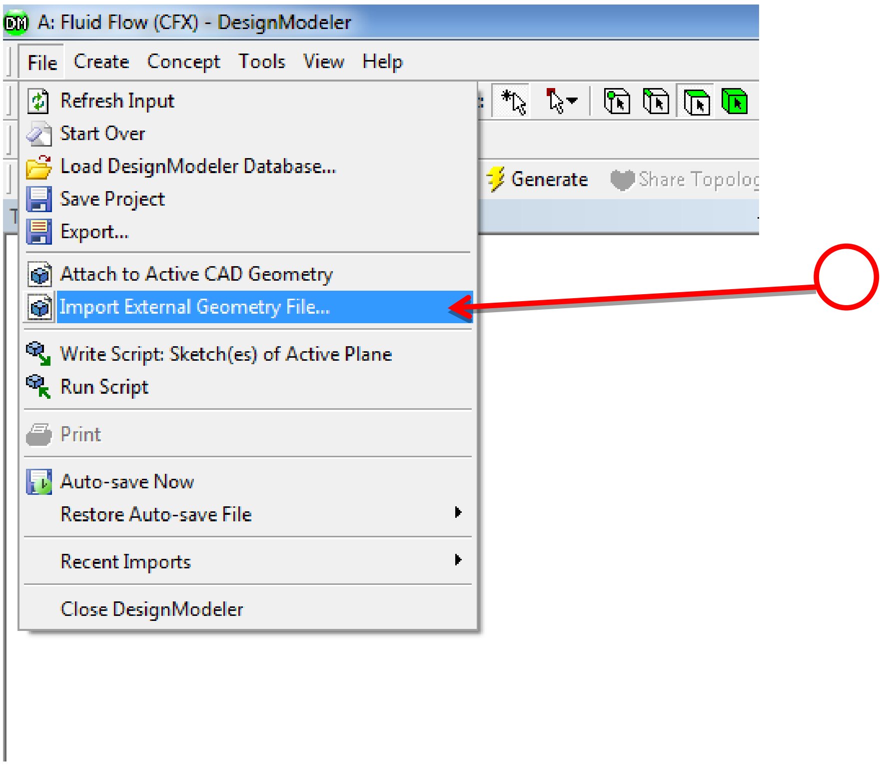
Step 2:

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.



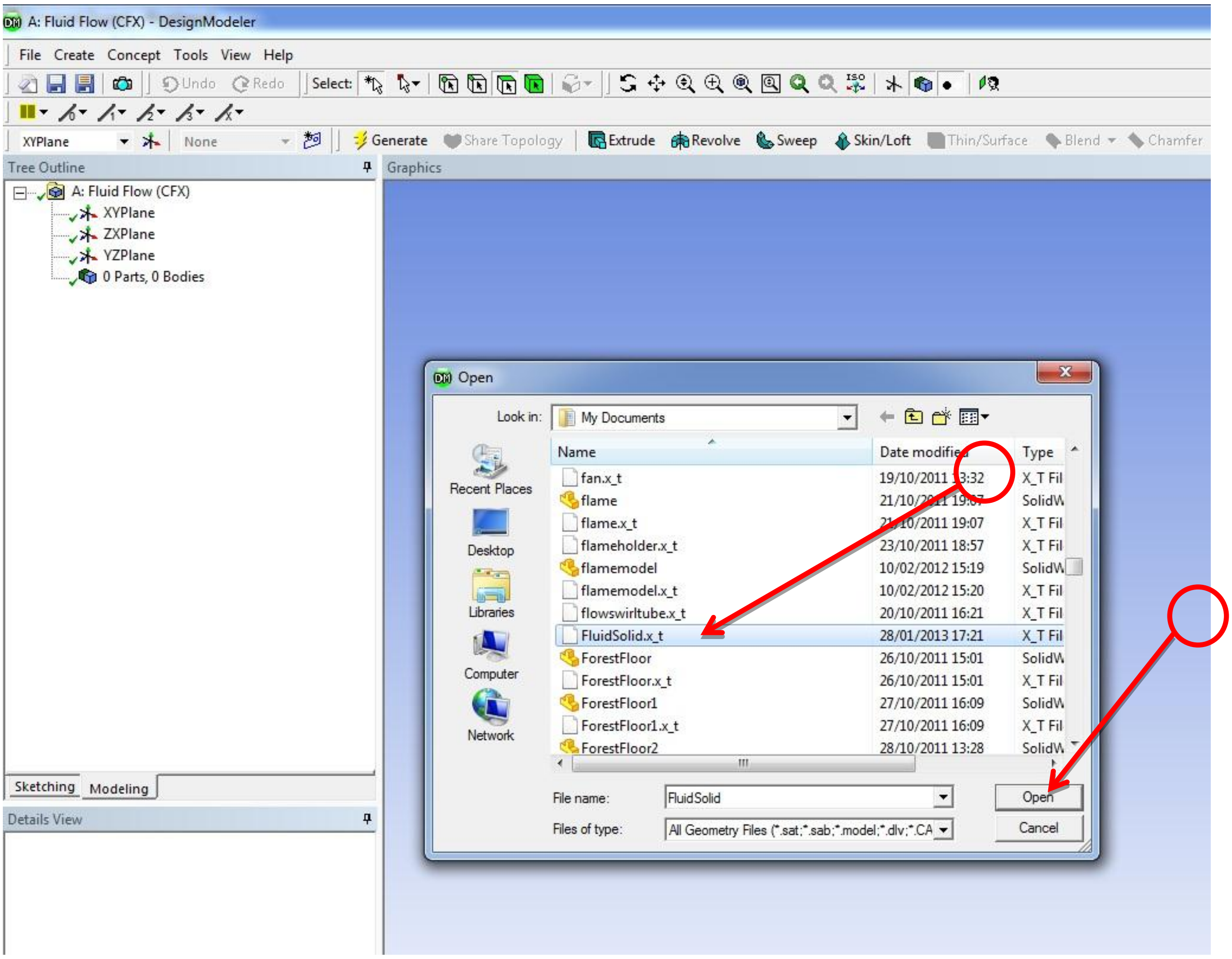
Step 3:

Go to File and select Import External Geometry file.



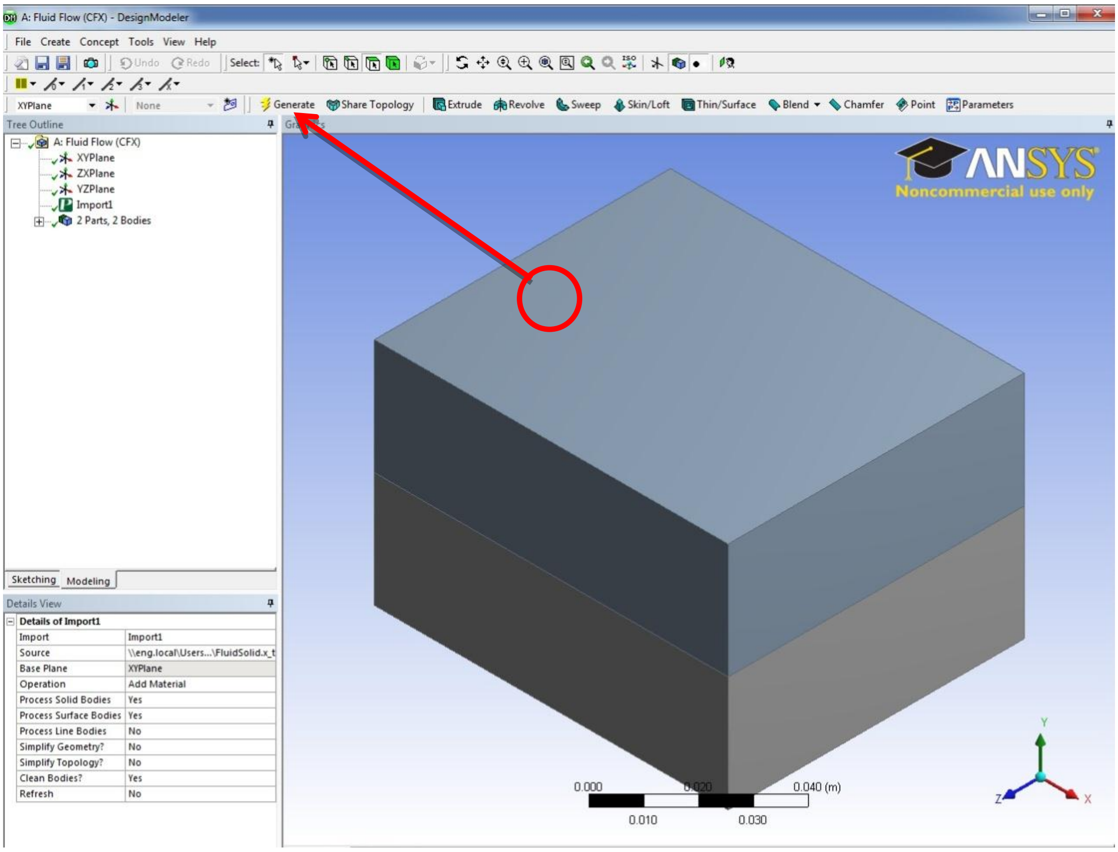
Step 4:

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



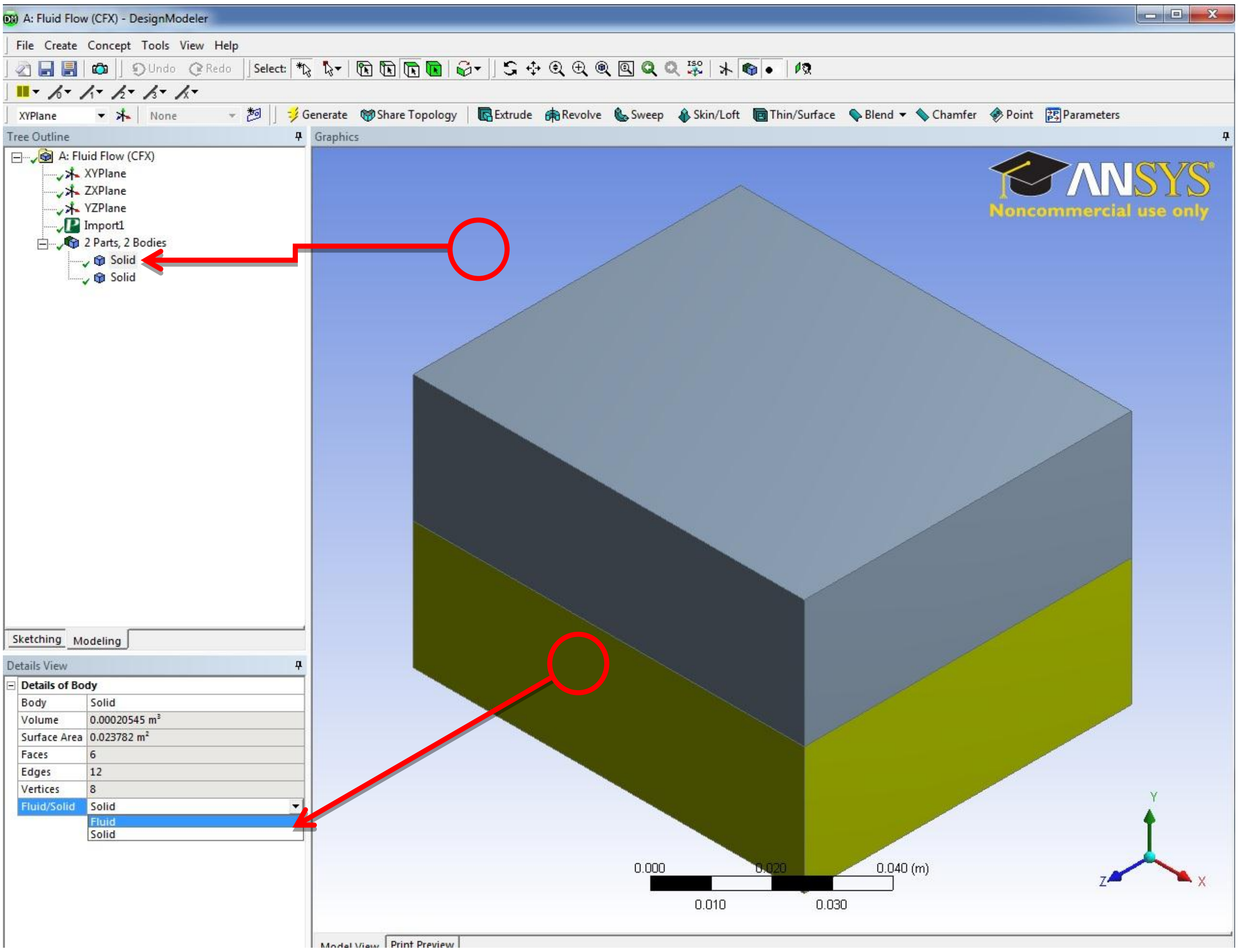
Step 5:

Press the genrate button and what you will see is the read in geometry into design modeller, for our studied case we have two domains one for the porous domain and the other is for the gas.



Step 6:

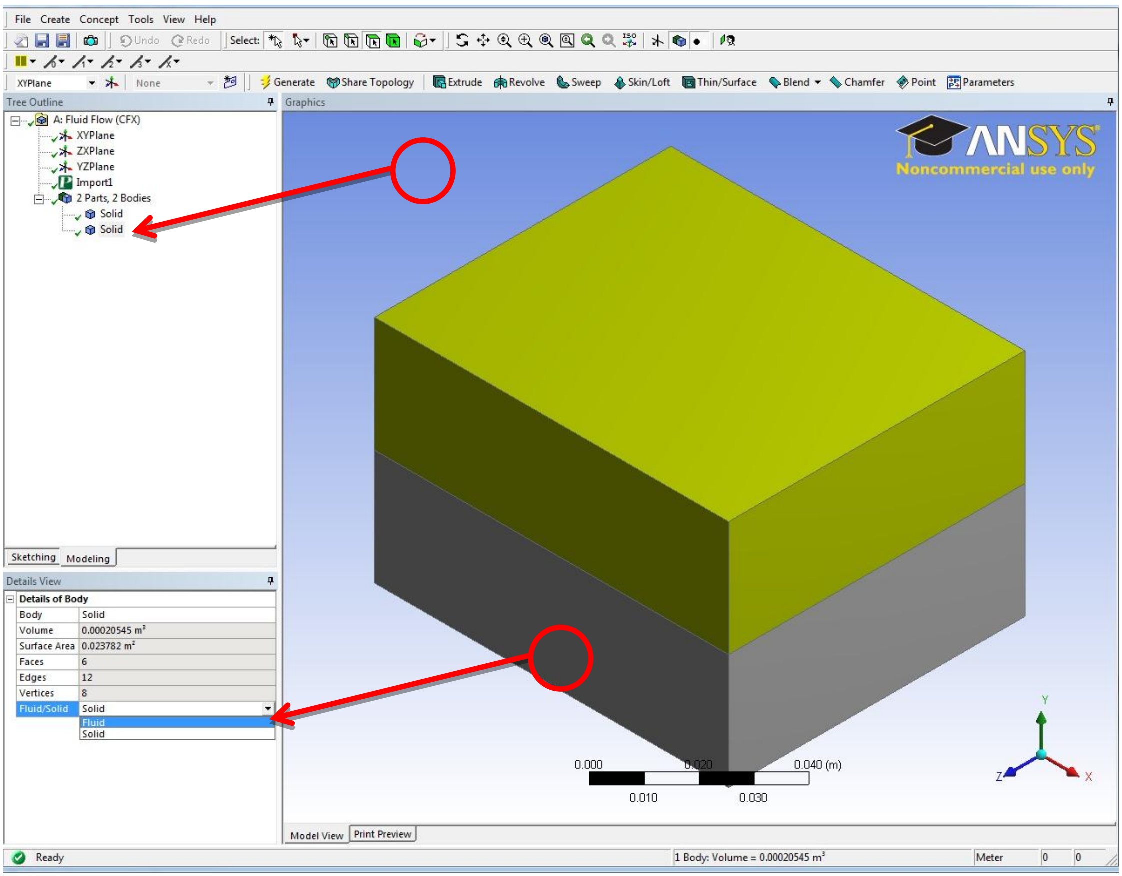
Postion the cursour on the icon (2 Parts, 2 Bodies) and then press the left button on it a subtree of three directoreis would occur. Position the cursour on the the last two solids (doing each solid seperatly) and then click the left mouse button and select one of the solids. Then go to the Details of boday and select Solid, as you can see the slelected body is colored yellow.



Step 7:

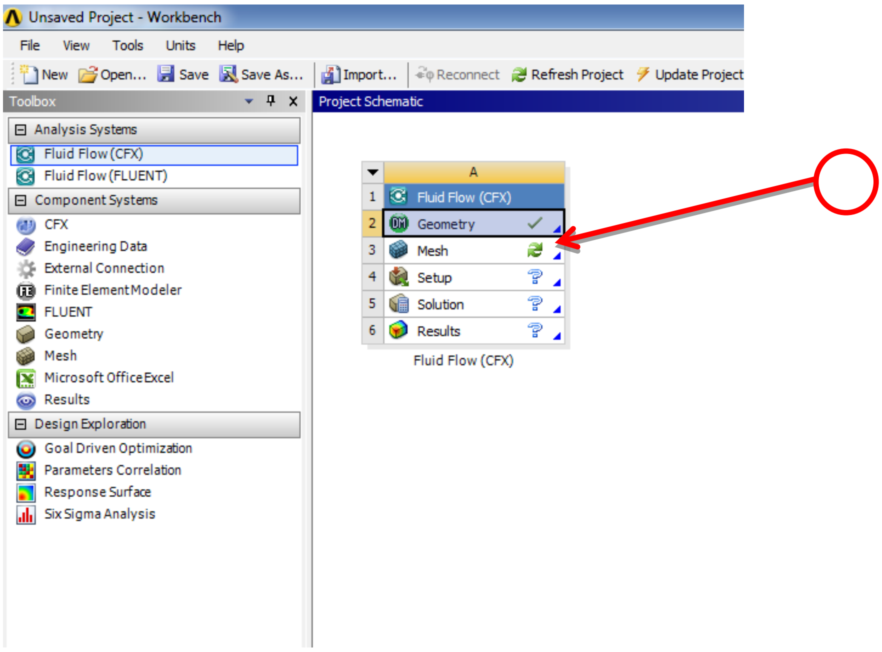
Select the second solid which turns into a yellow color go to Details of body and select Fluid.

You have now finished with design modeller our next step is to generate the mesh.



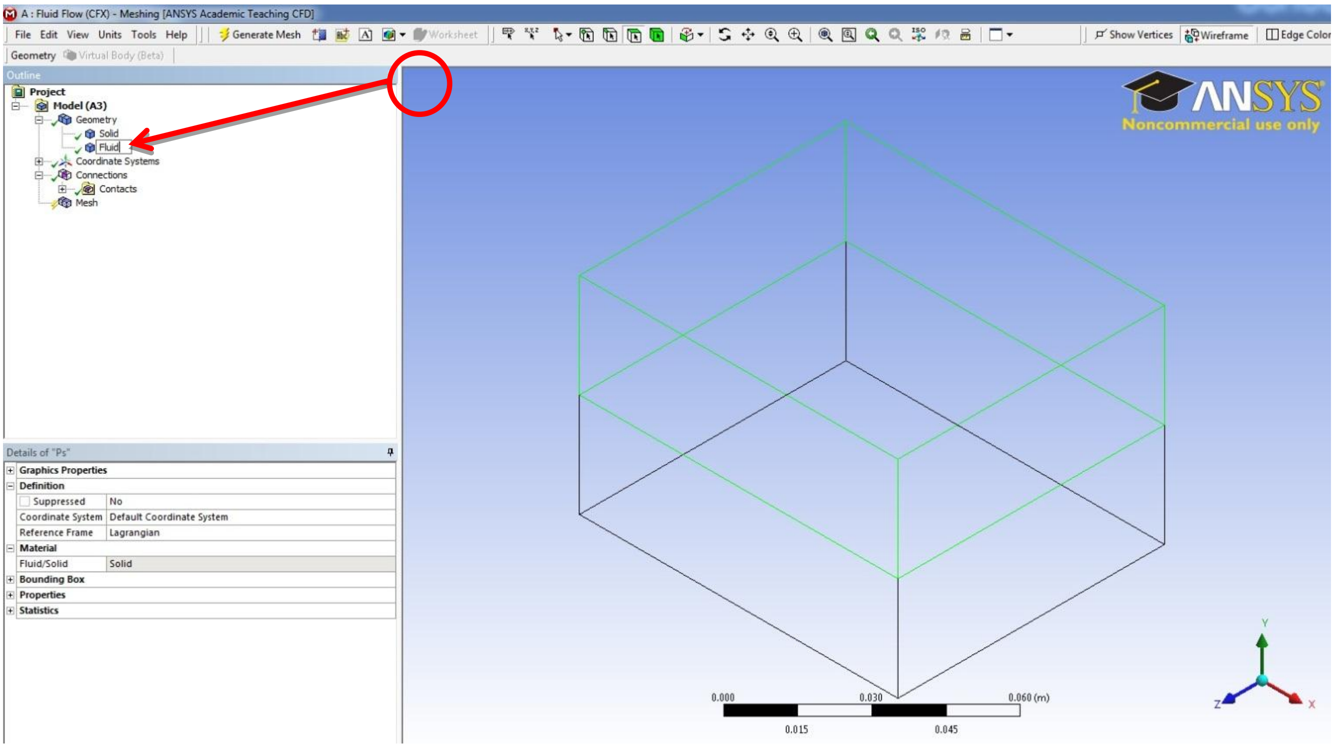
Step 8:

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



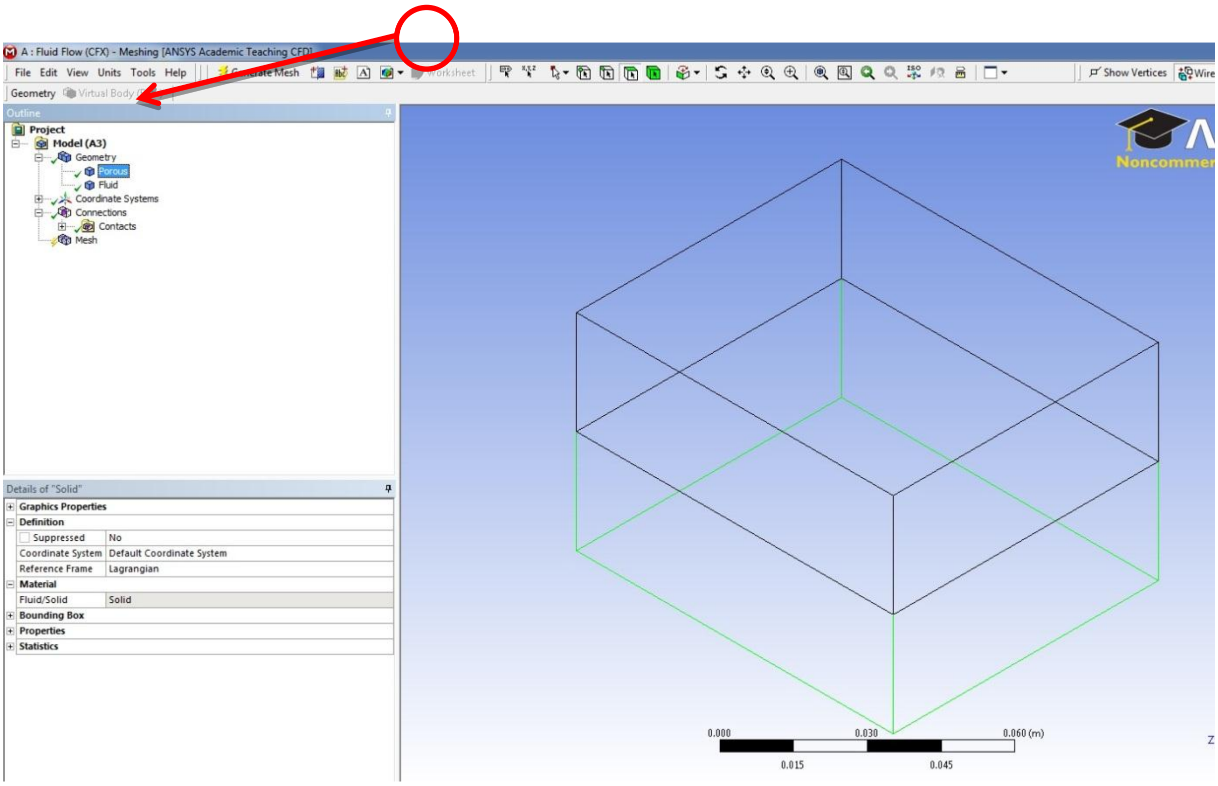
Step 9:

Click on the geometry icon, you will get to solids in the sub tree, click on the solid and right click and rename the solid to Fluid, to check that you have selected the right domain look at the highlighted box.



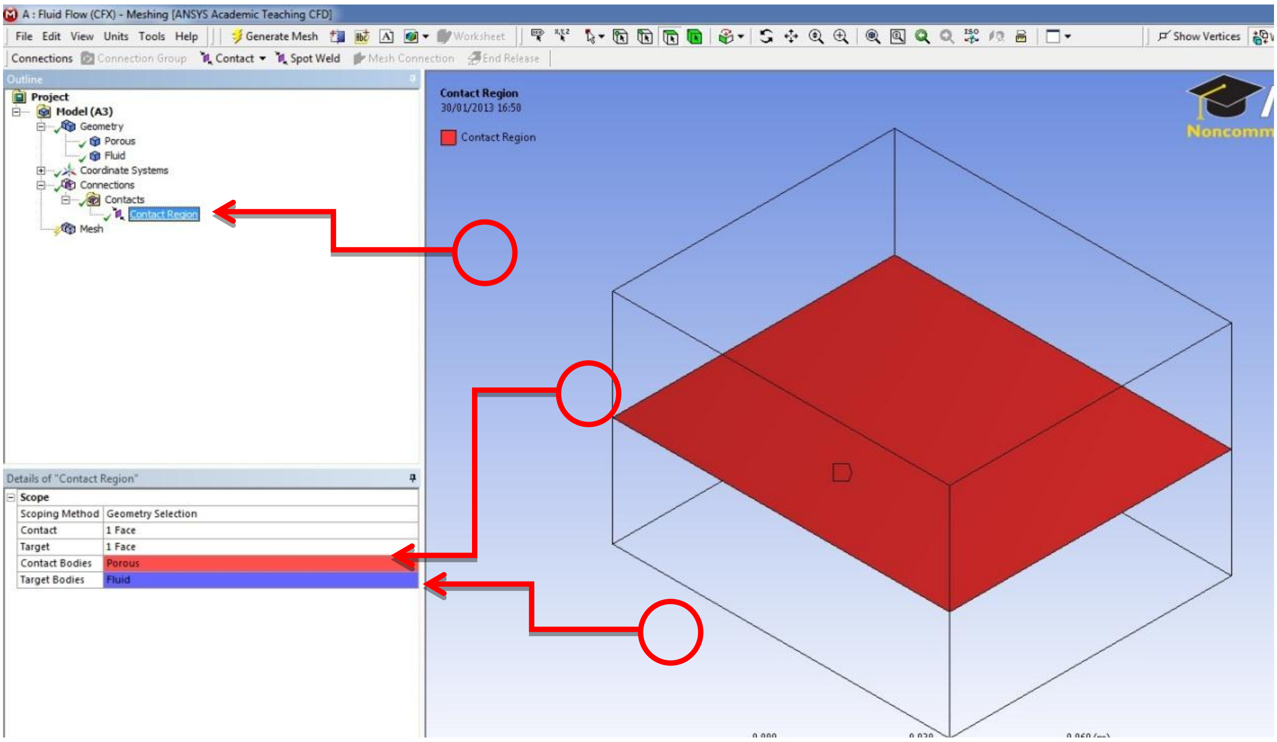
Step 10:

Click on the geometry icon, select the other solid and rename the solid to Porous (representing the porous domain), to check that you have selected the right domain look at the highlighted box.



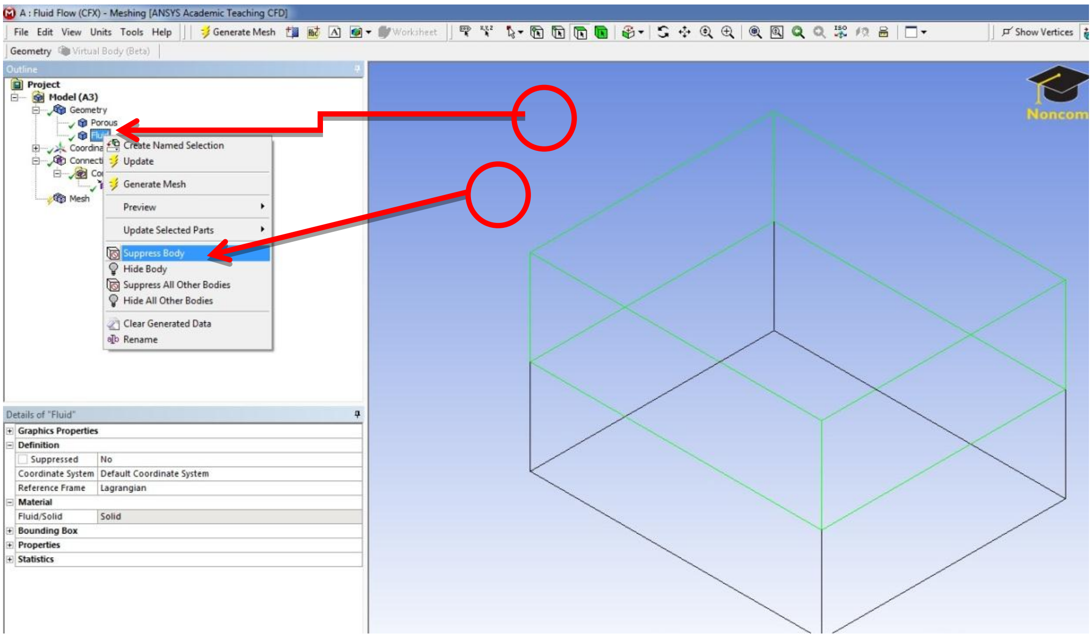
Step 11:

Click on the connections icon, then select the contact region, the contact region is in red, you can see under the details of 'Contact Region' that the contact Bodies for Porous is highlighted in pinkish red and the target Bodies is highlighted in purple .



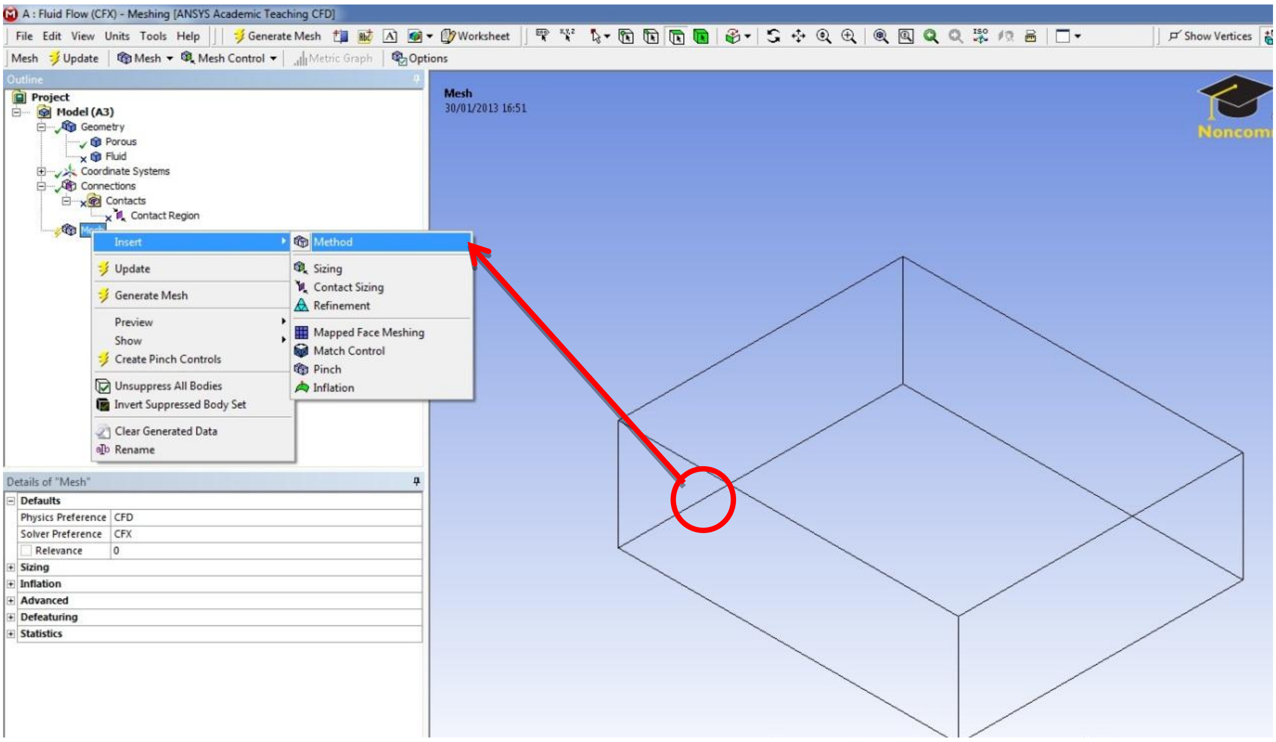
Step 12:

Under geometry click on the fluid icon and select suppress body from the drop down list.



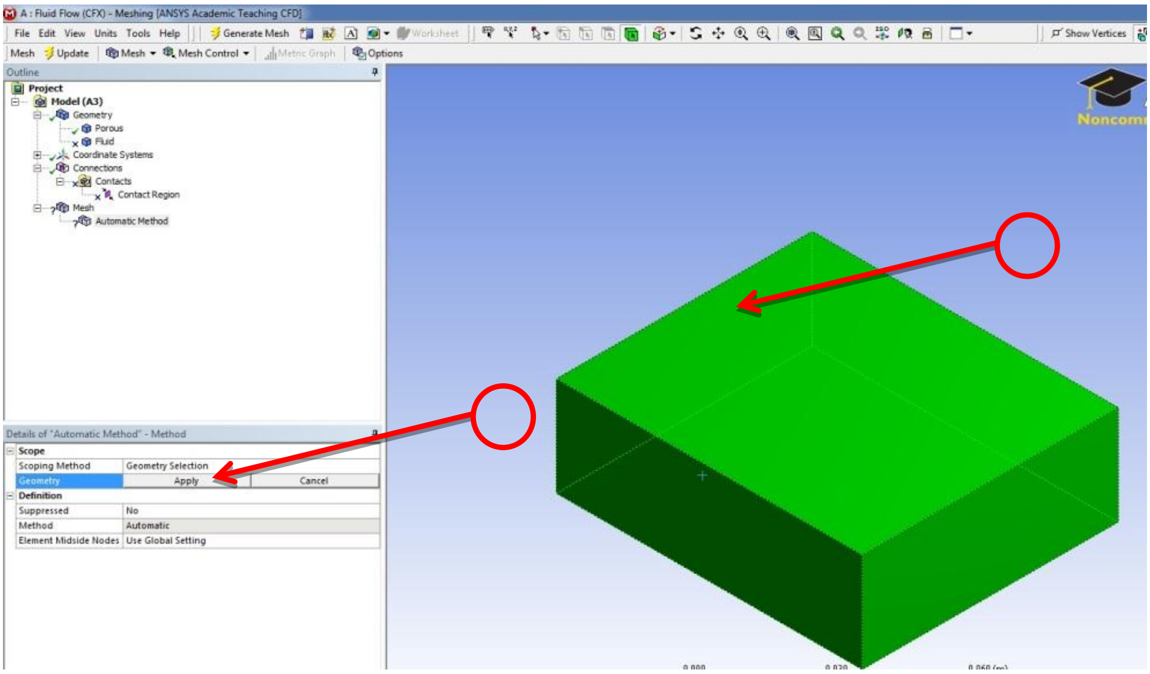
Step 13:

Click on the mesh icon, then select insert, then select method.



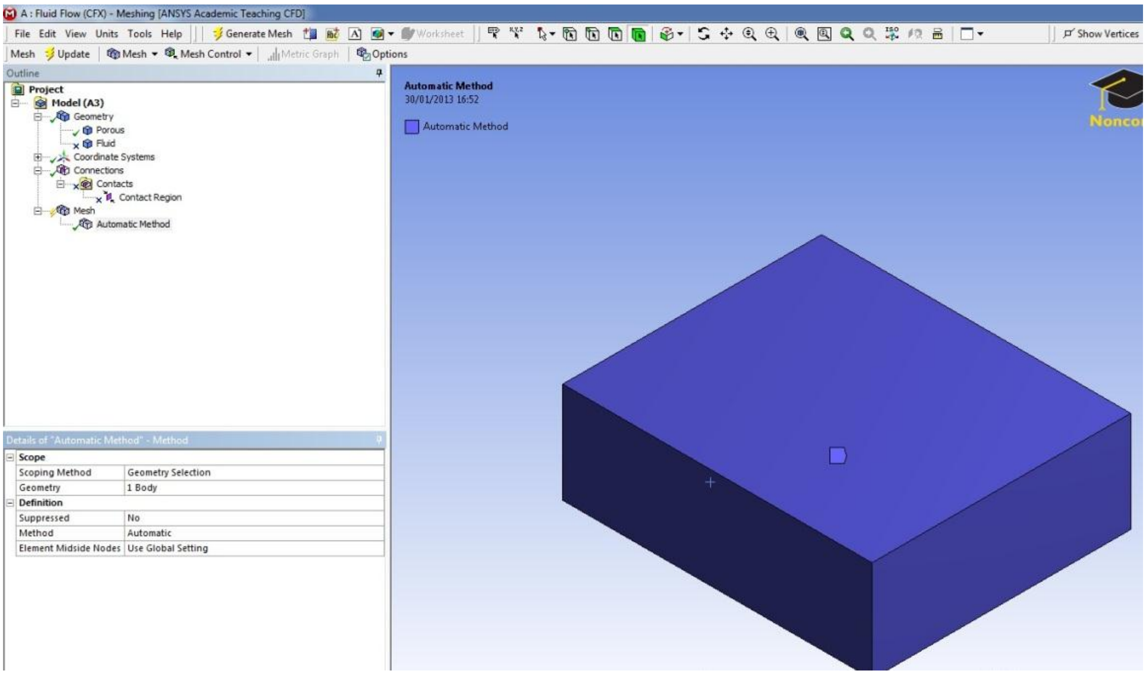
Step 14:

Select the box which should turn into a green colour, then click on the apply button.



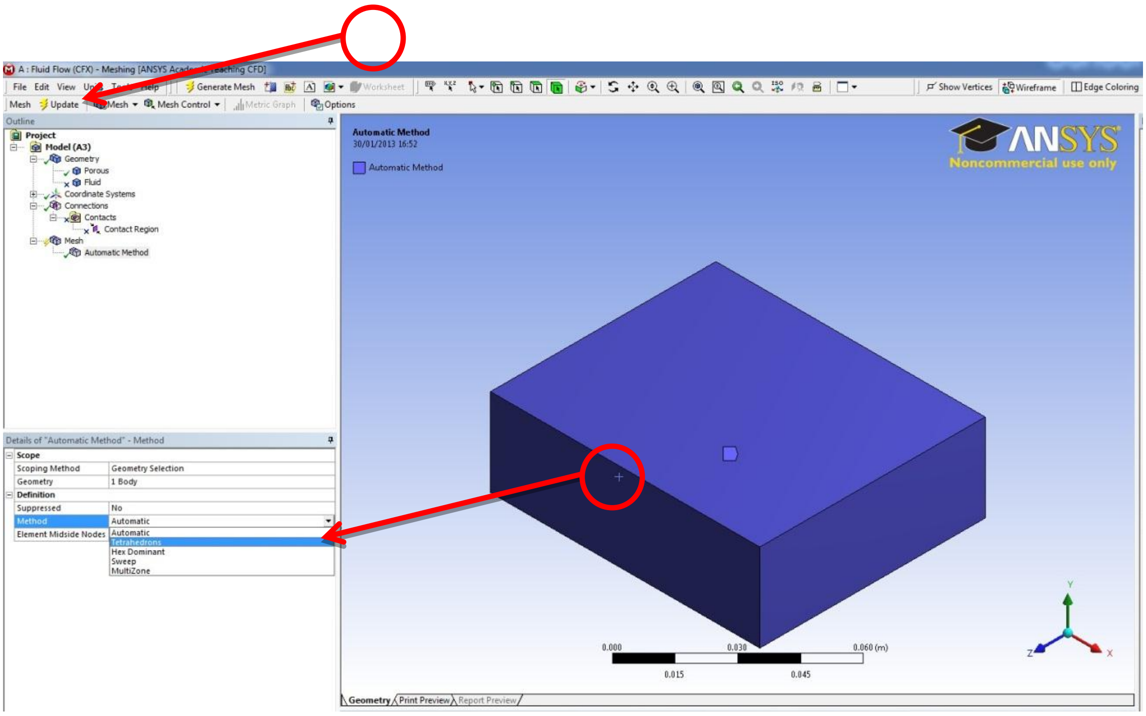
Step 15:

To check that the geometry has been correctly selected the domain should turn into a purple colour.



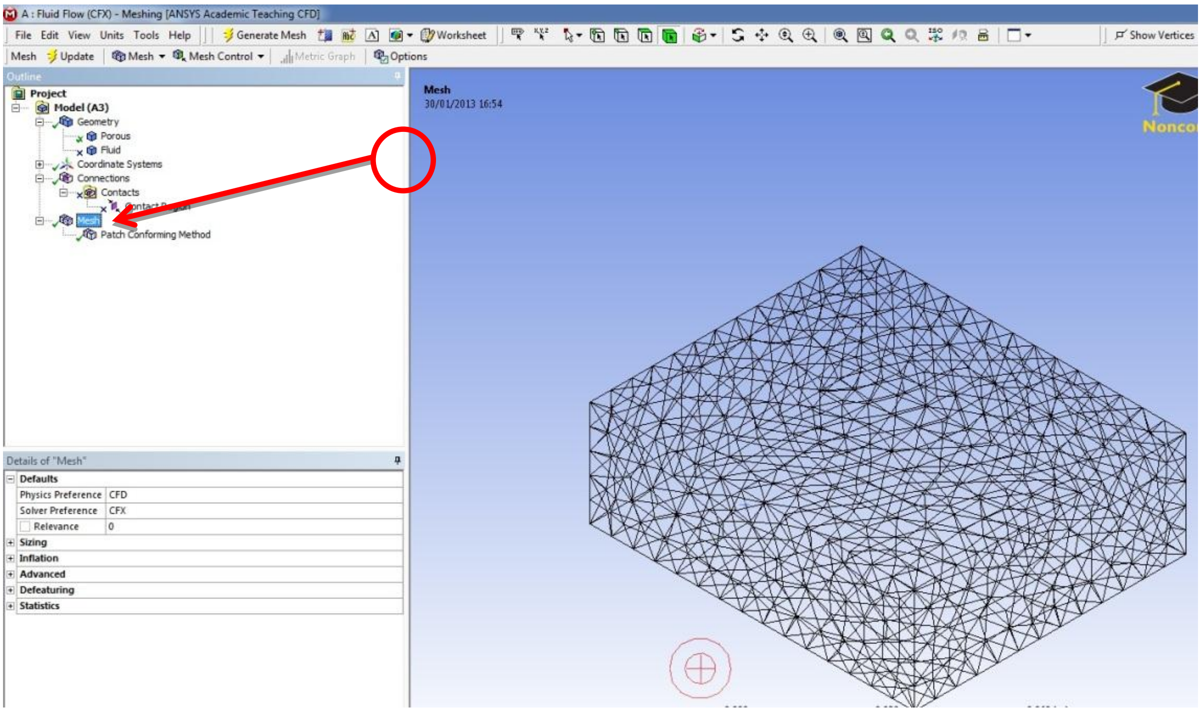
Step 16:

Select Tetrahedrons as a meshing method, then press update.



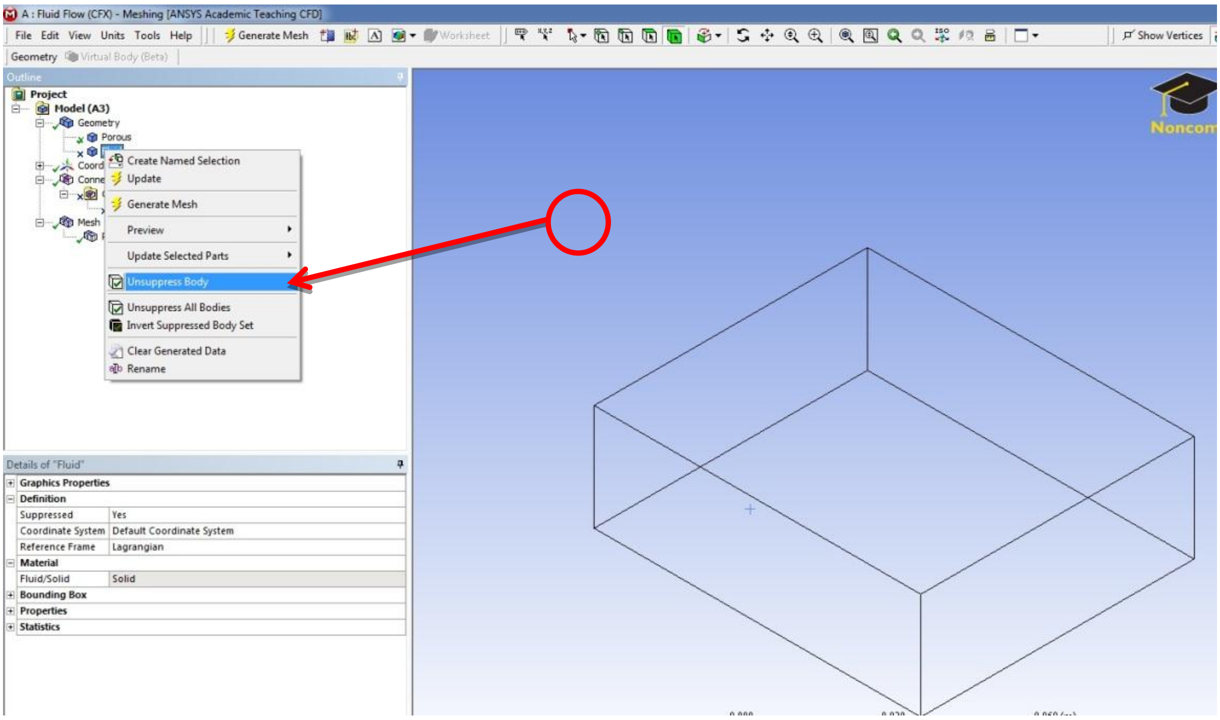
Step 17:

Click on the mesh icon and you can see the generated mesh. We are done now in generating the porous domain mesh.



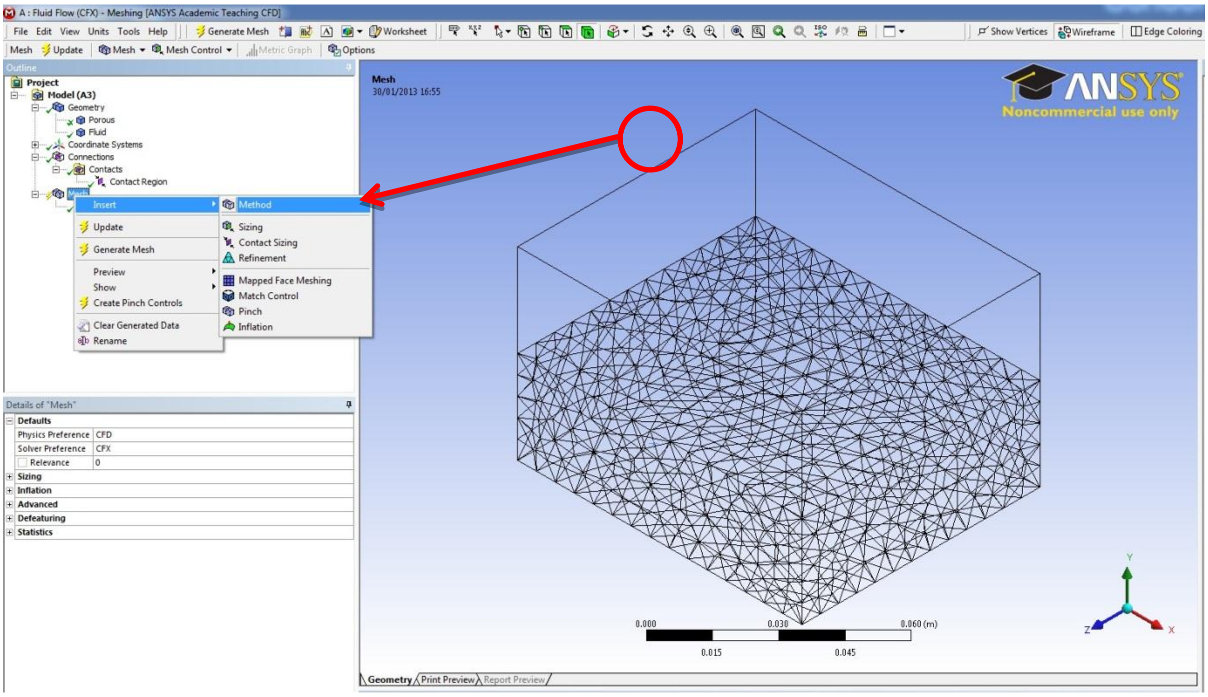
Step 18:

We now want to generate the fluid mesh domain. Select the fluid icon under geometry and right click the mouse button and select Unsuppress Body.



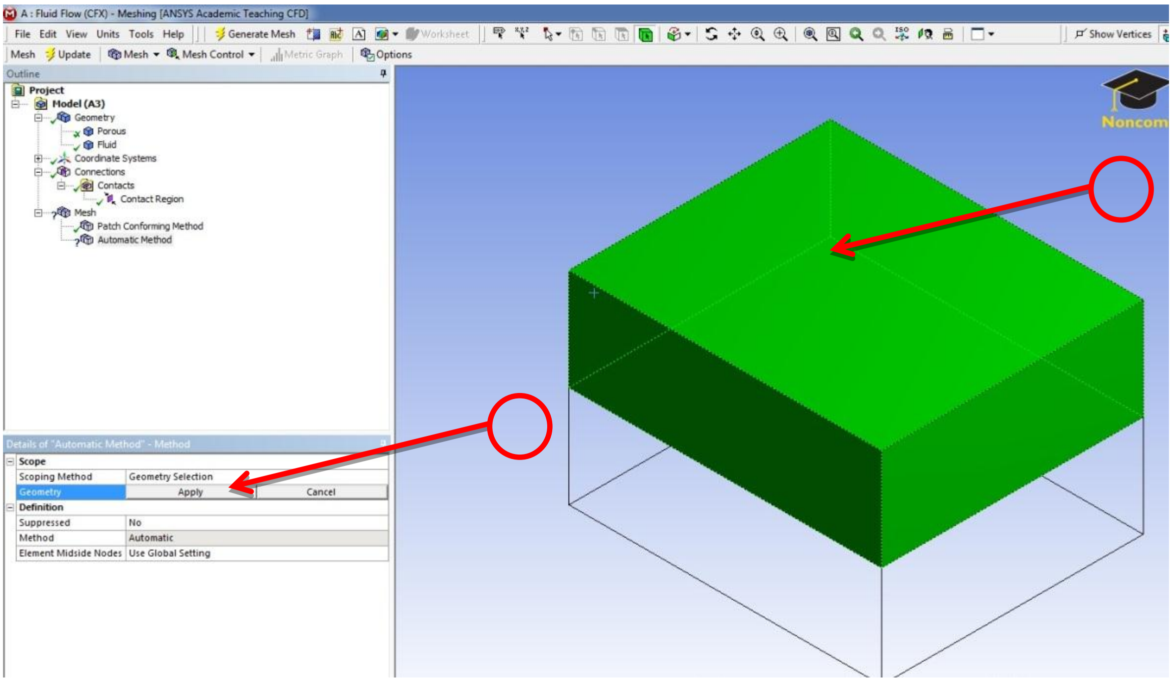
Step 19:

Select Mesh then right click mouse button then select insert then Method.



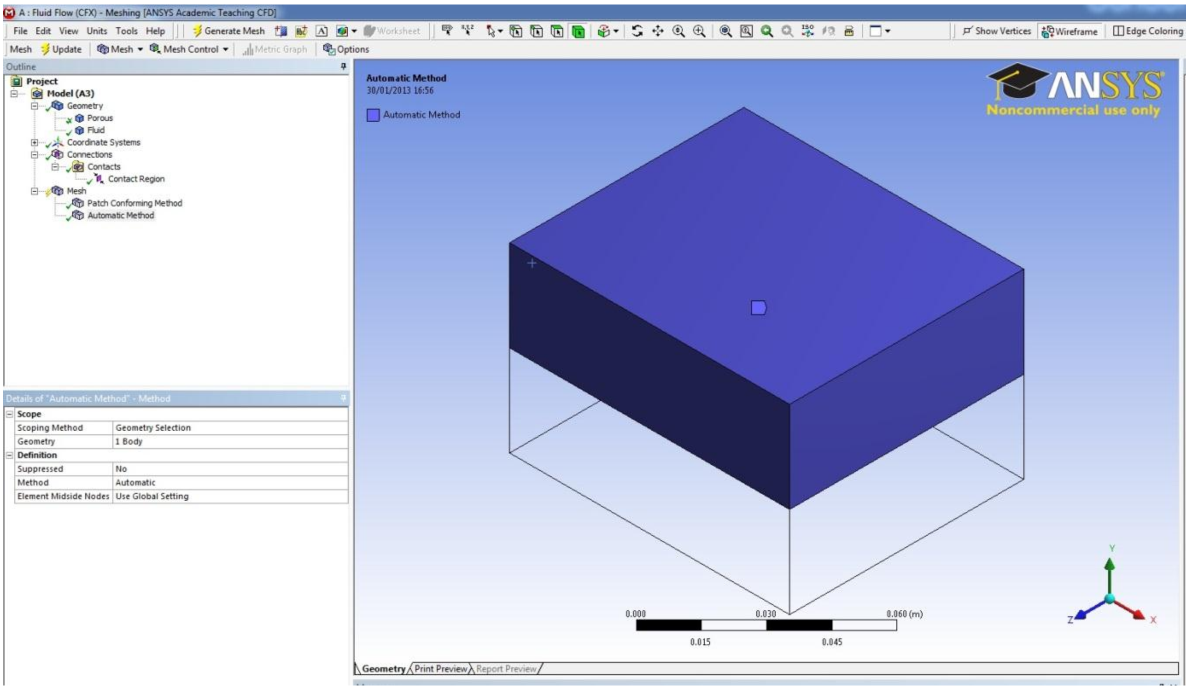
Step 20:

Select the Fluid box which should turn into a green colour, then click on the apply button.



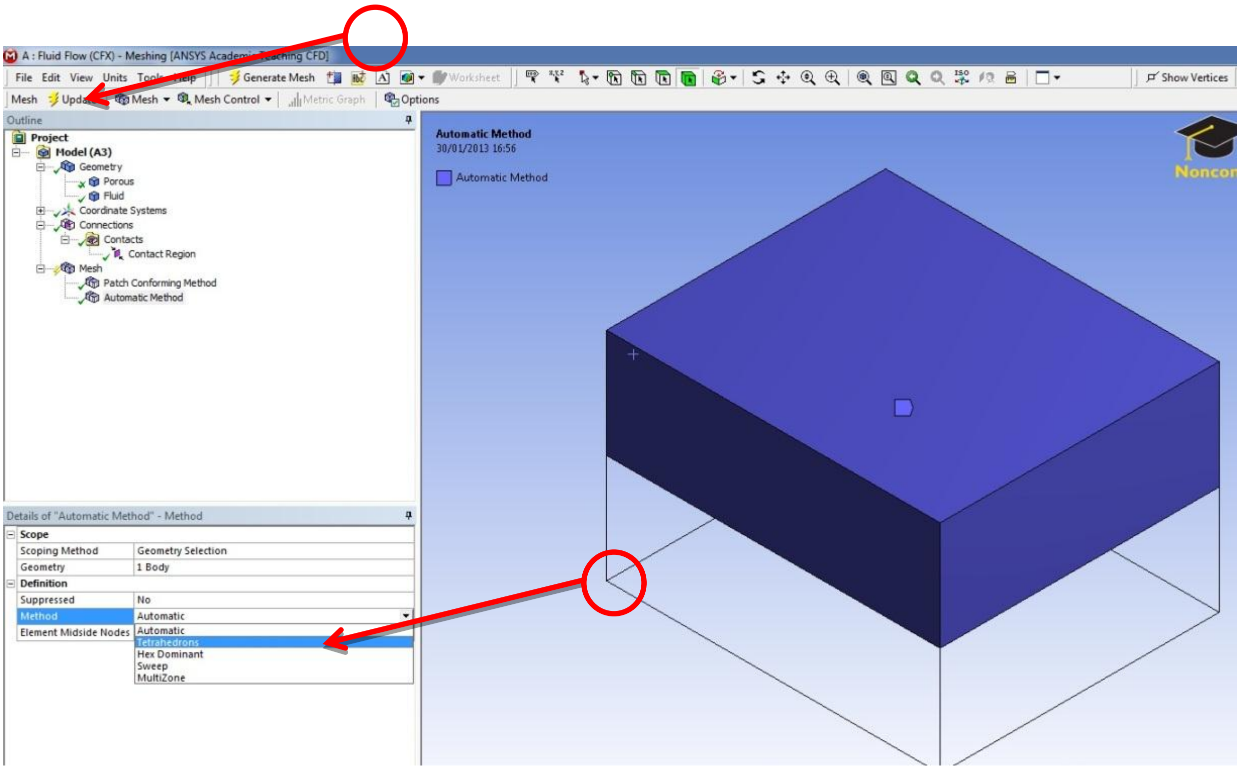
Step 21:

To check that the geometry has been correctly selected the domain should turn into a purple colour.



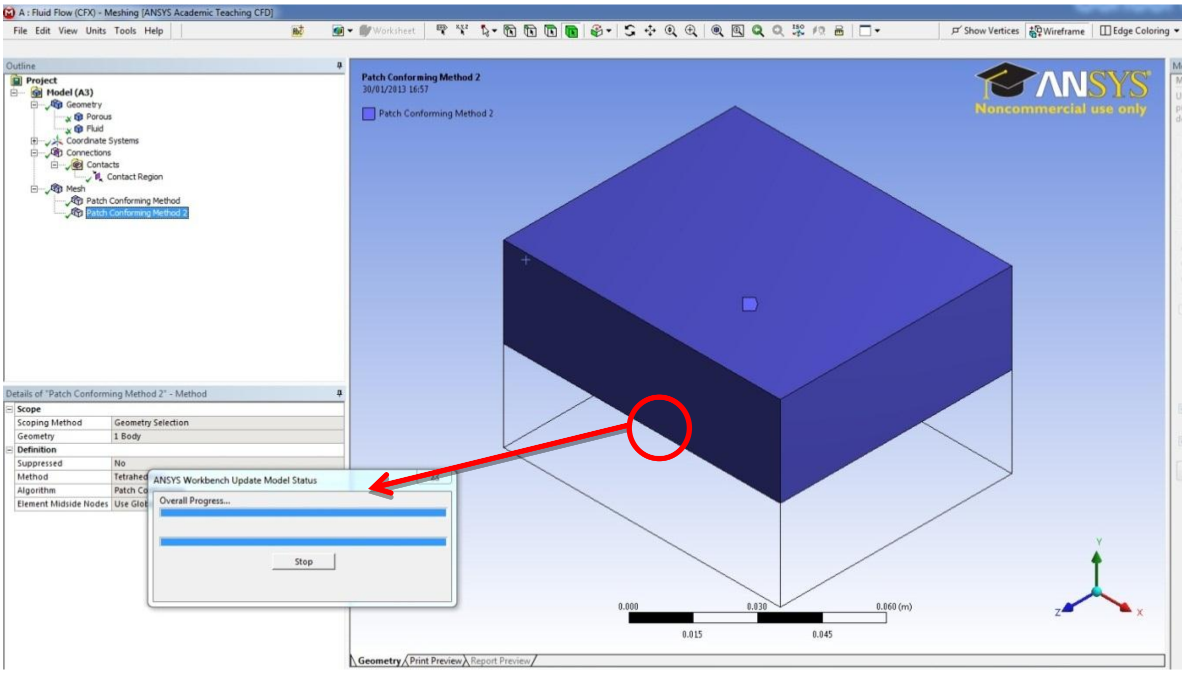
Step 22:

Select Tetrahedrons as a meshing method, then press update.



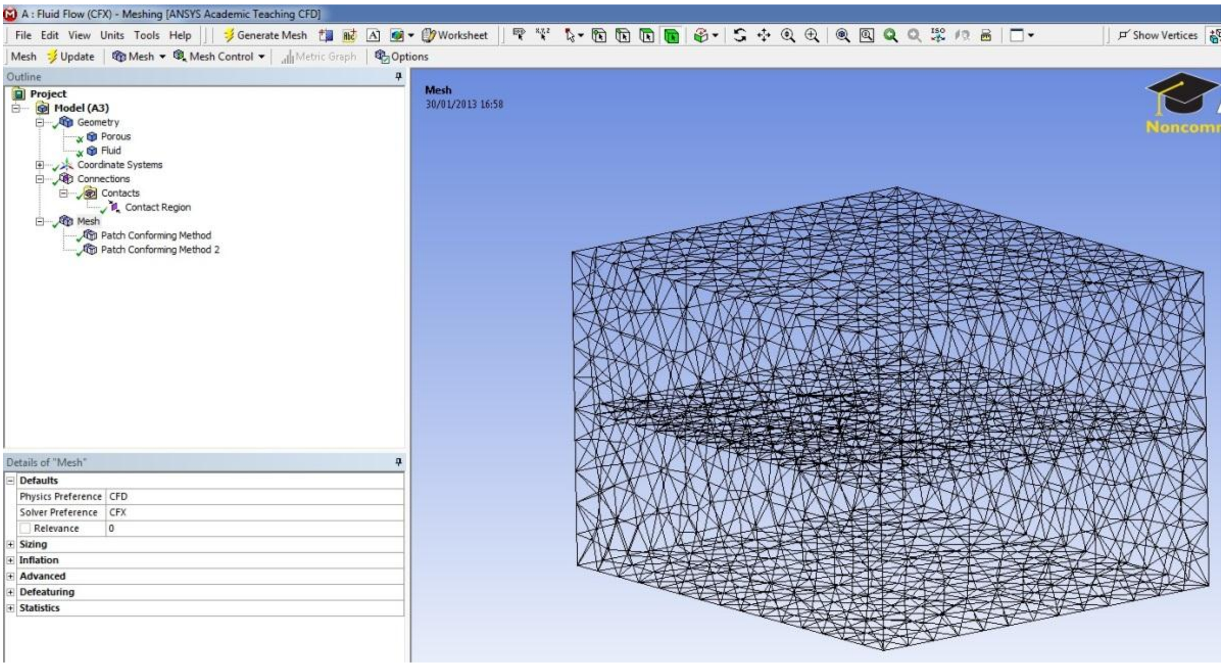
Step 23:

After pressing Update button you will see a window indicating the meshing process.



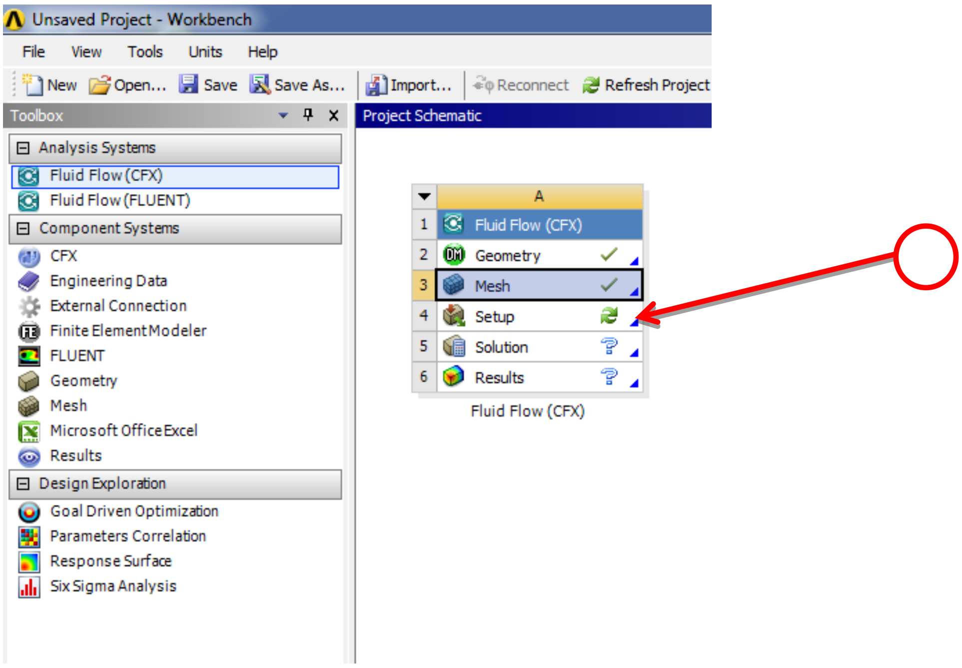
Step 24:

The genrated mesh for both domains should look somthing similair to what is shown below.



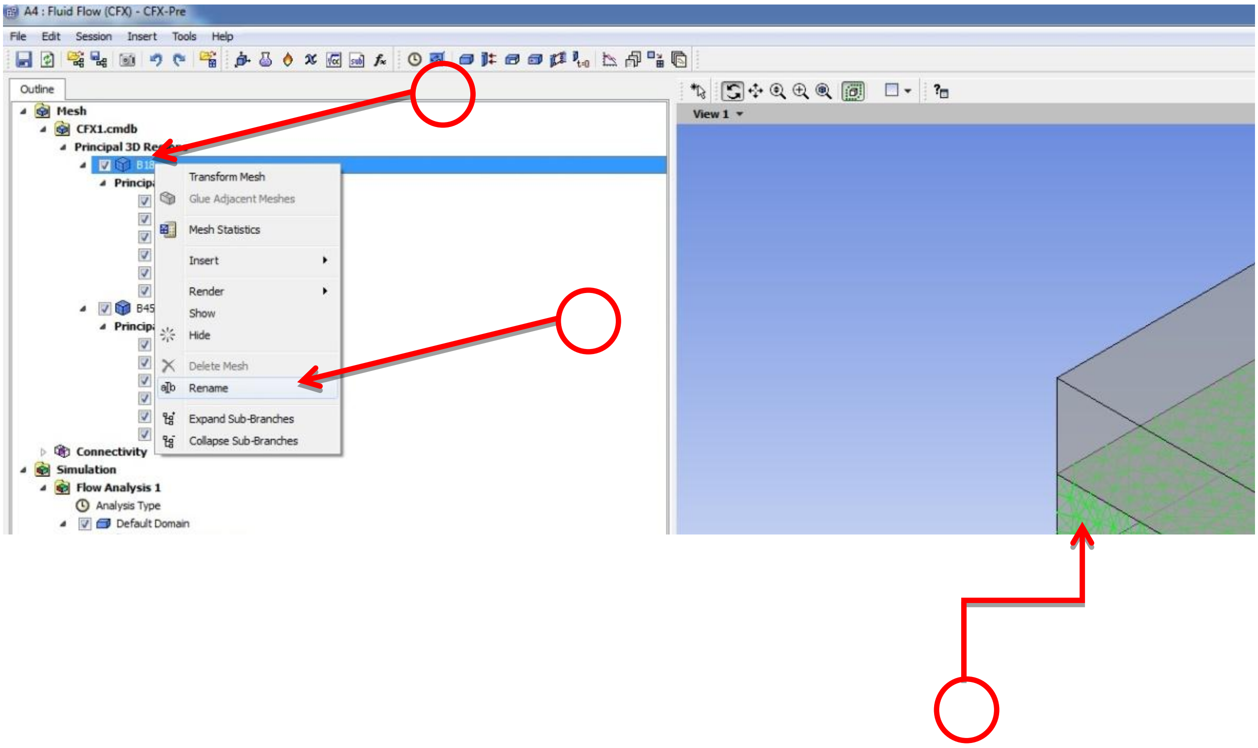
Step 25:

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



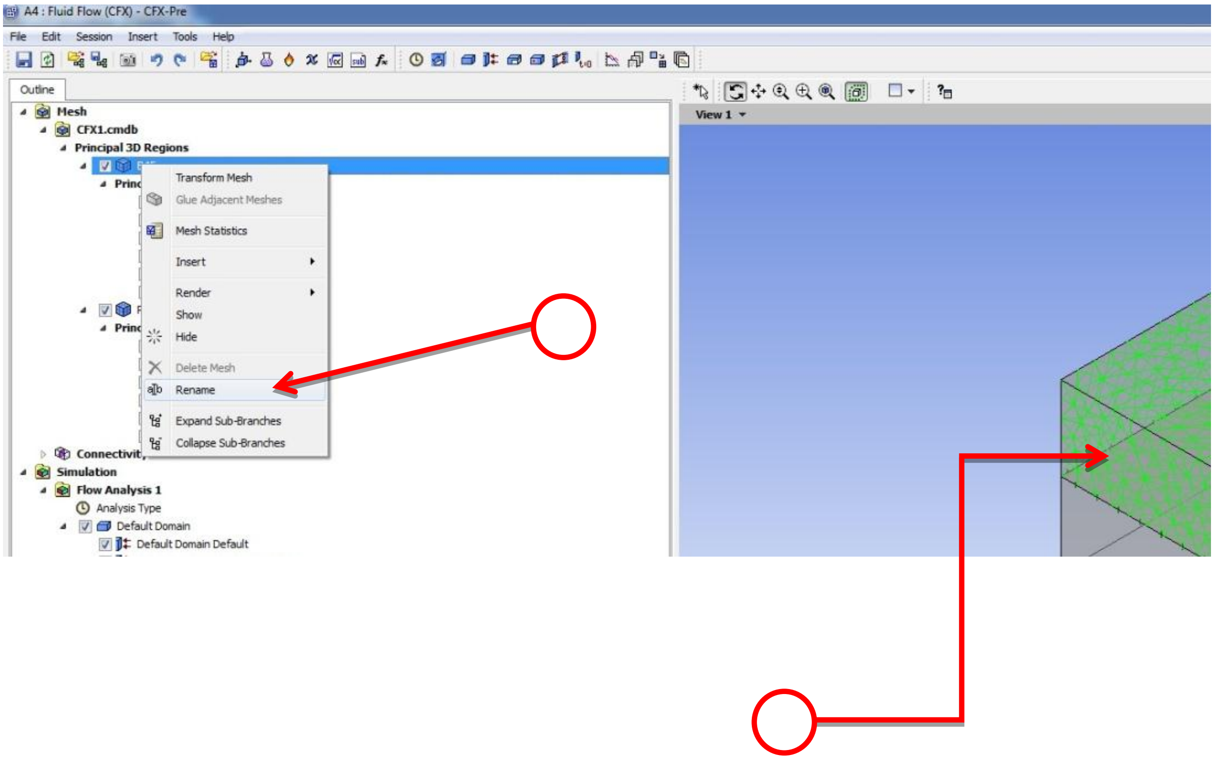
Step 26:

Go to principal 3D Region and select the porous domain region (The mesh highlighted in green) right click mouse button and select rename, assign it a name of Porous .



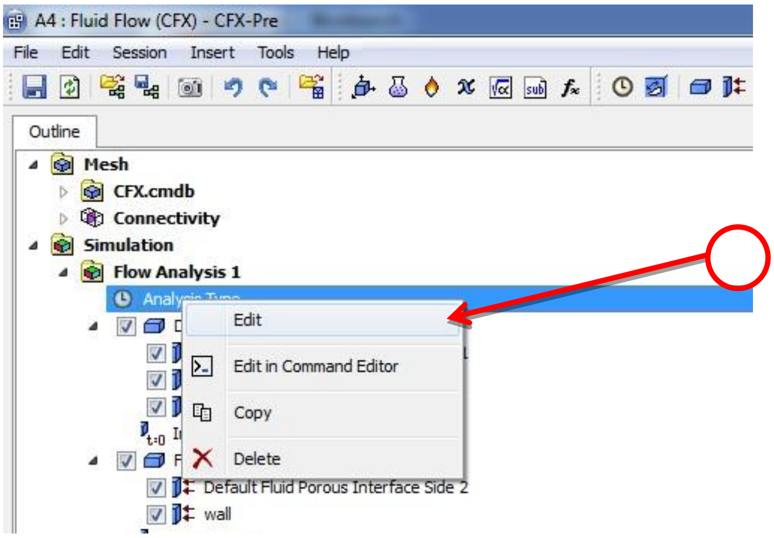
Step 27:

Go to principal 3D Region and select the Fluid domain region (The mesh highlighted in green) right click mouse button and select rename, assign it a name of Fluid .



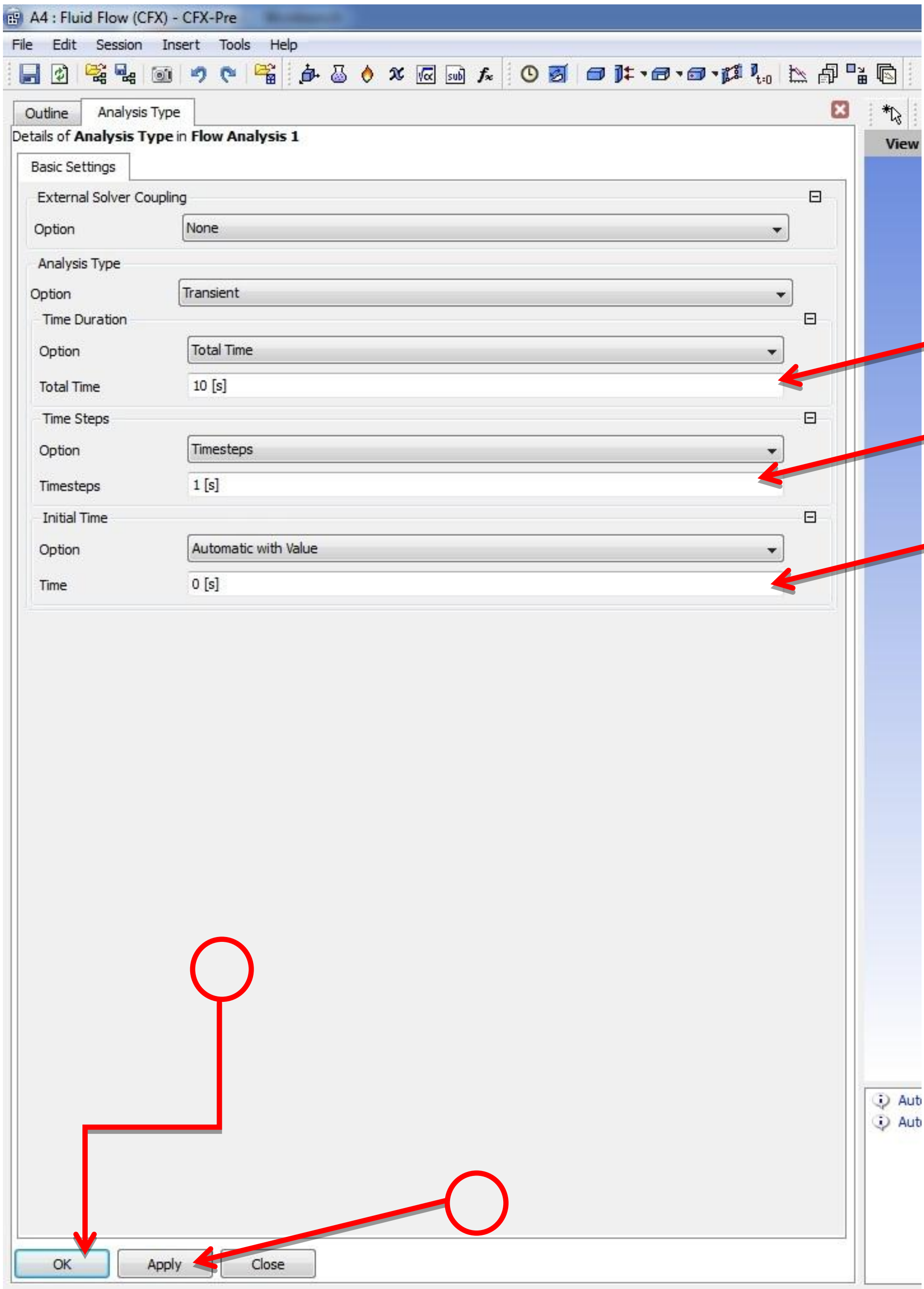
Step 28:

We need now to add the time stepping option.



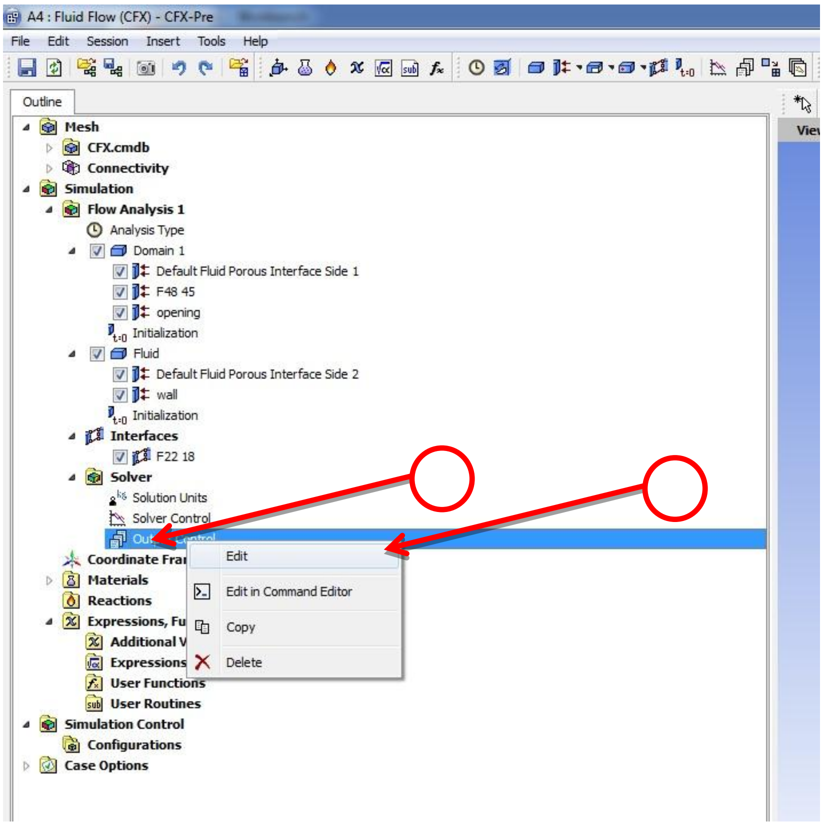
Step 29:

Enter 10 seconds for the total time, then enter 1 second for the time step and finally enter 0 second as initial time. Press the apply button and press ok.



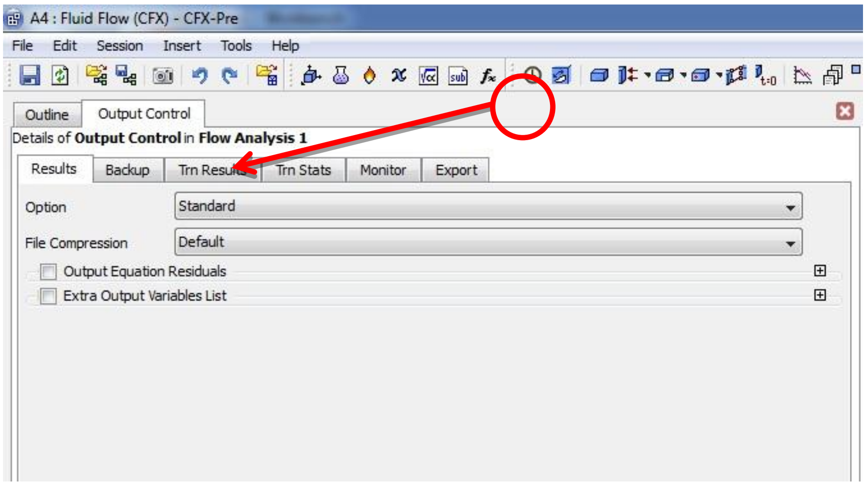
Step 30:

Right click the mouse button and select edit.



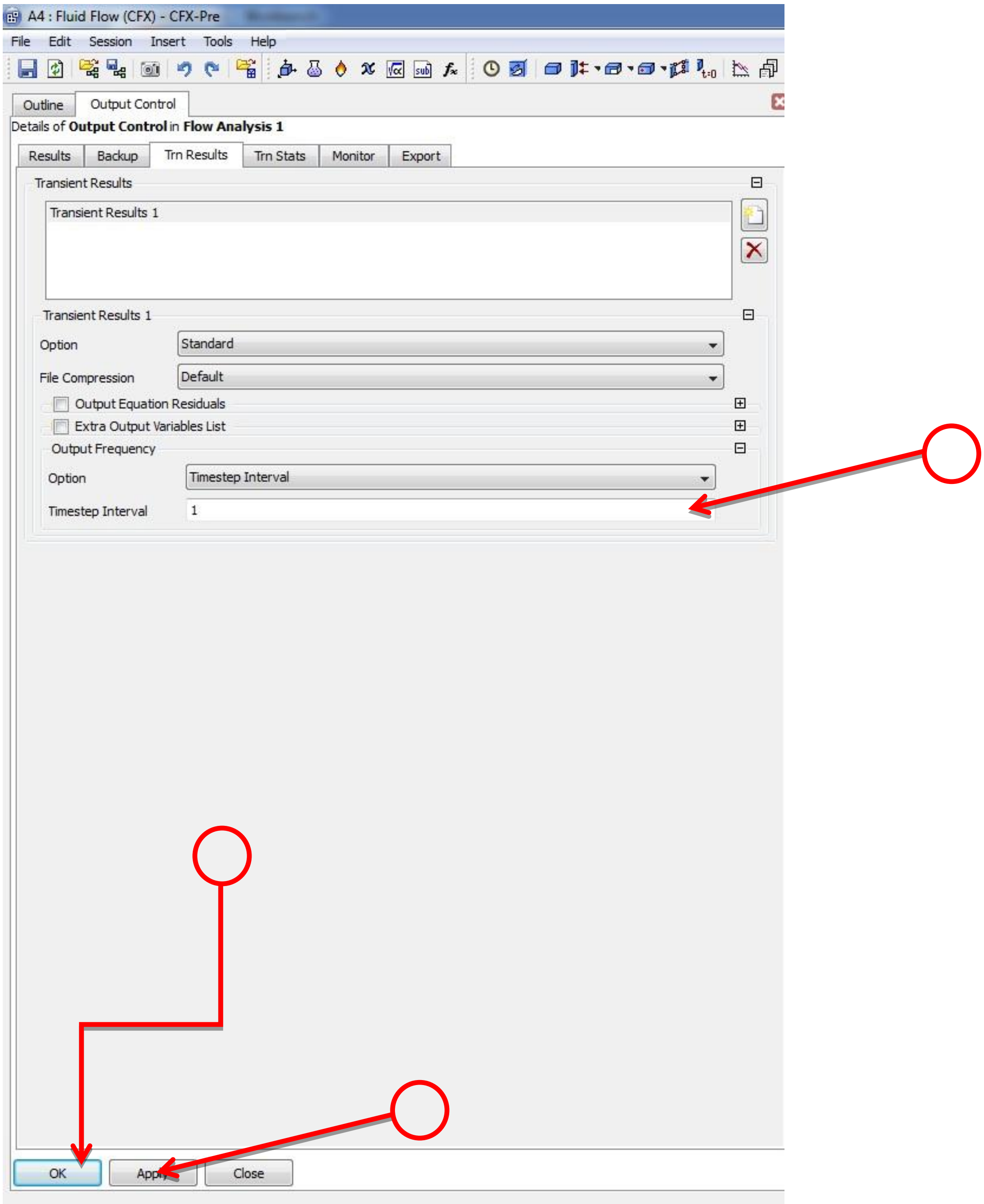
Step 31:

Right click the mouse button and select edit.



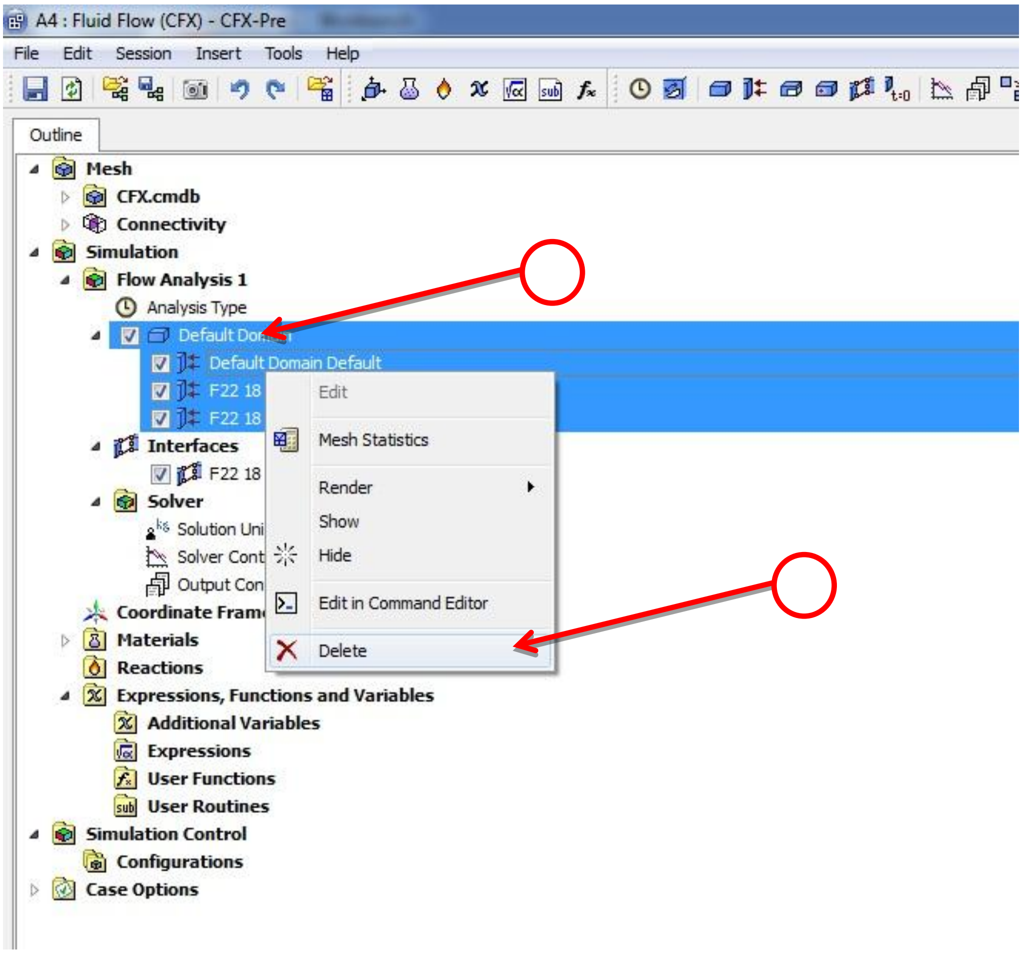
Step 32:

Under the Output Frequency enter 1 in the Timestep Interval. Press the Apply button and then press Ok.



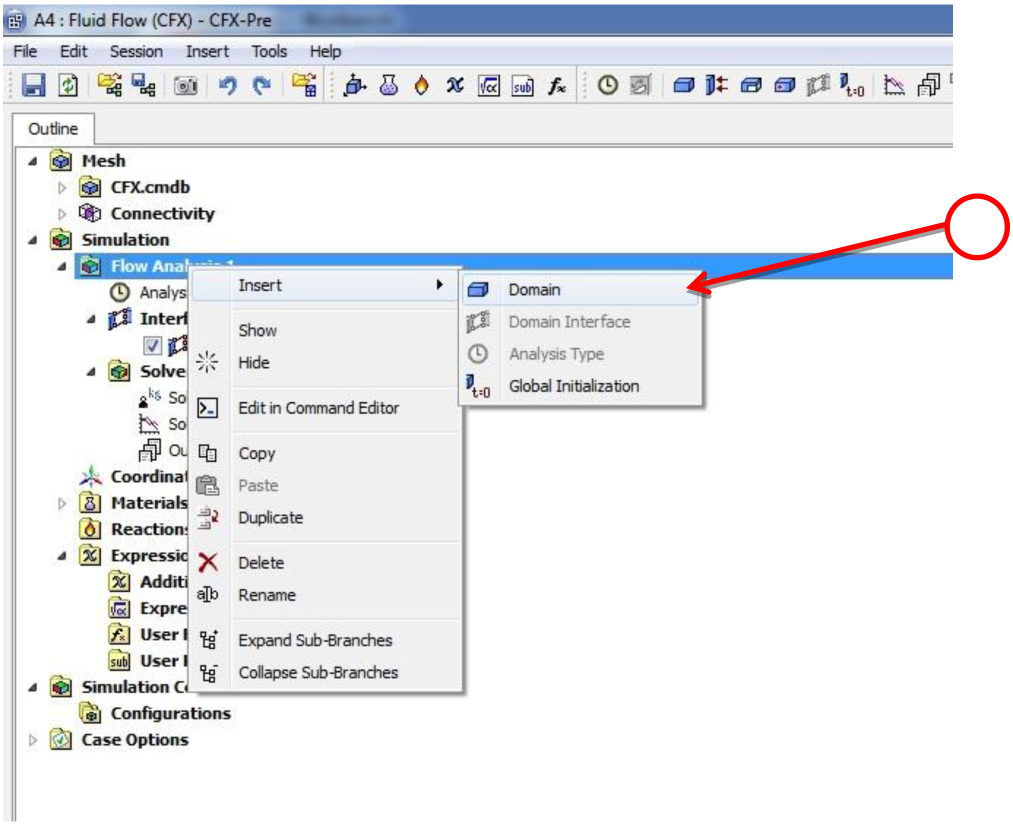
Step 33:

Select the four highlighted domains. Then press delete. Because we need to add new domains for the simulation.



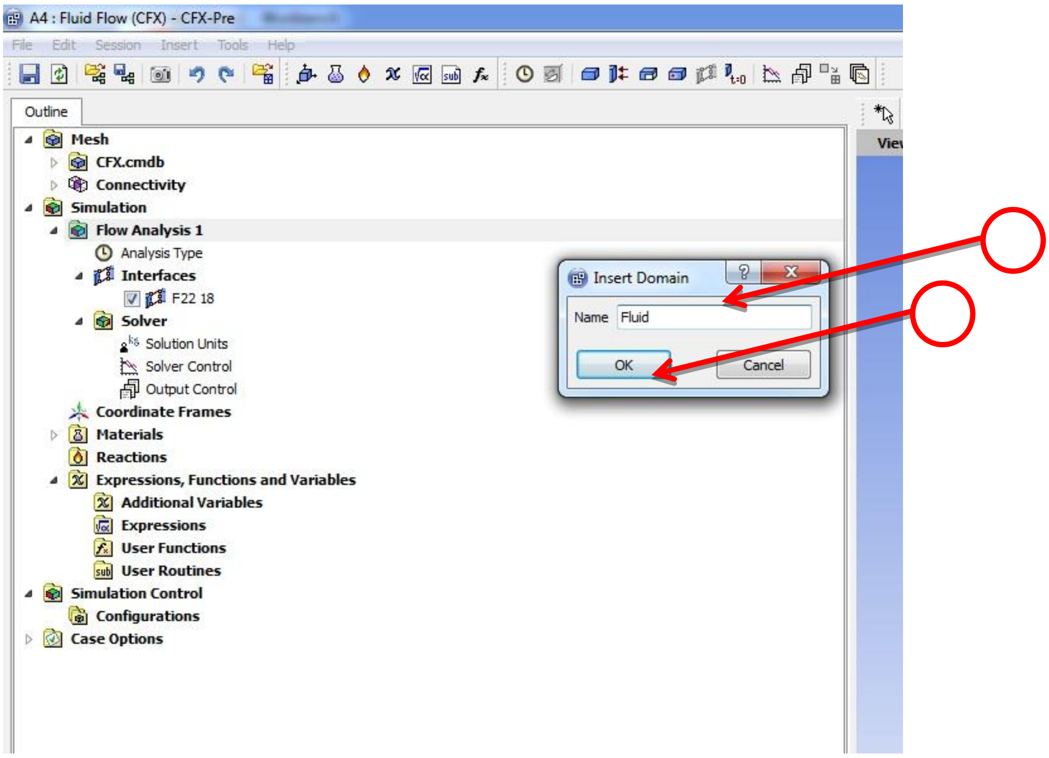
Step 34:

Call the new domain Porous and press Ok.



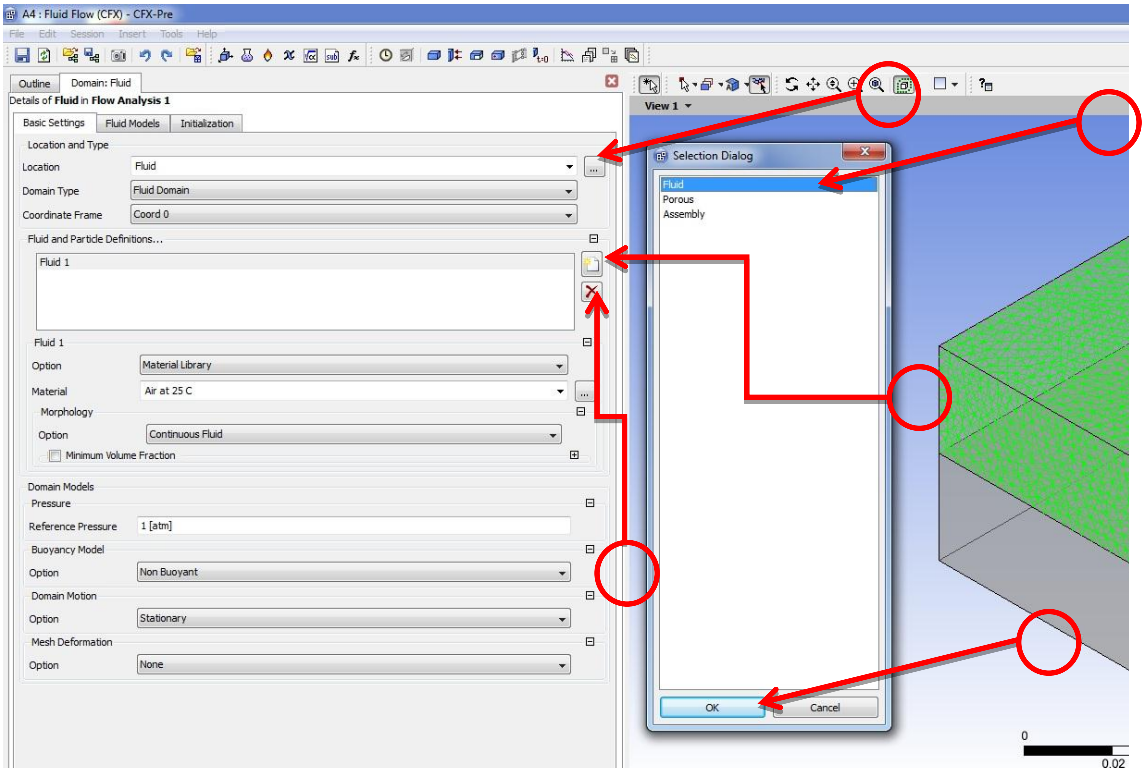
Step 35:

Call the new domain Porous and press Ok.



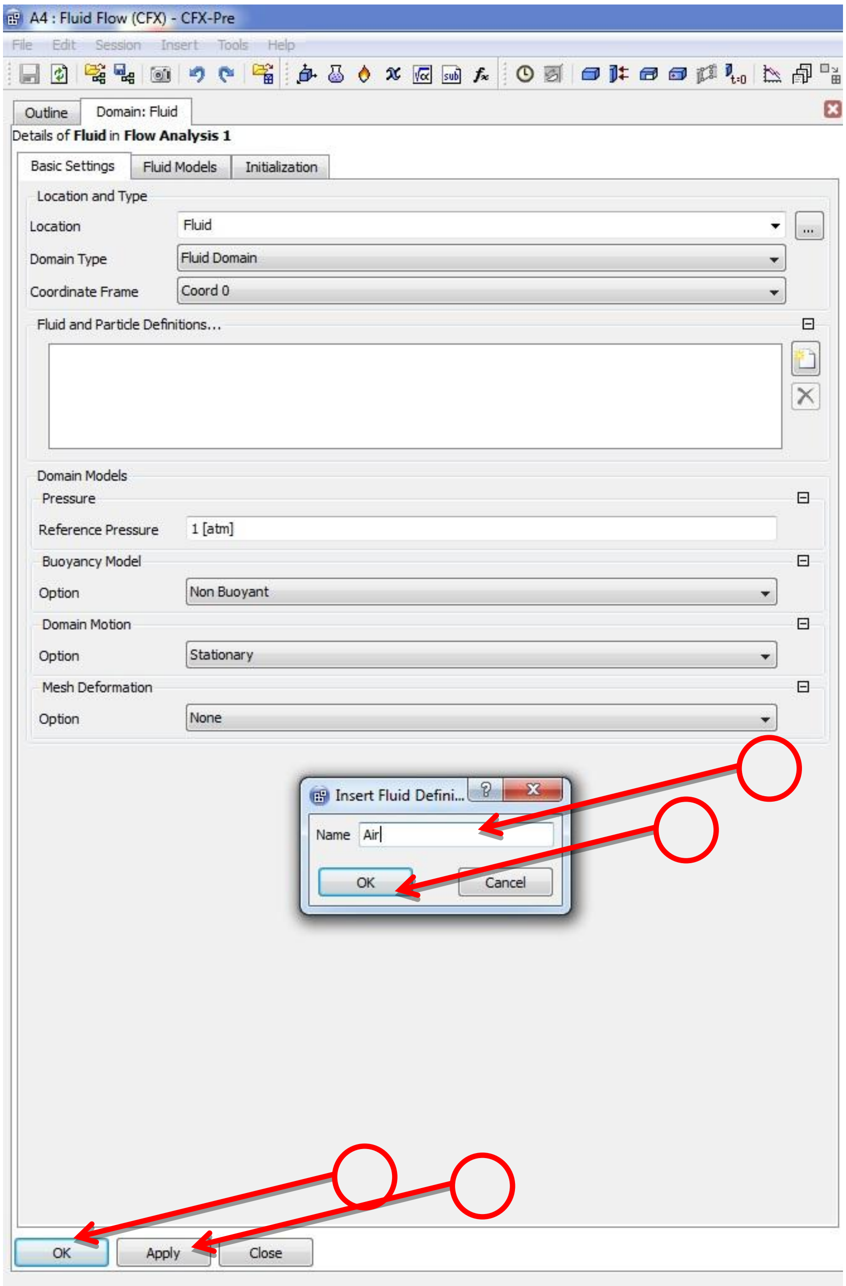
Step 36:

Press on the button beside the location selection, a window will open up select Fluid and press Ok. Finally press the red x sign to rename the Fluid 1 to Air. Then press the icon that has a document with a yellow star.



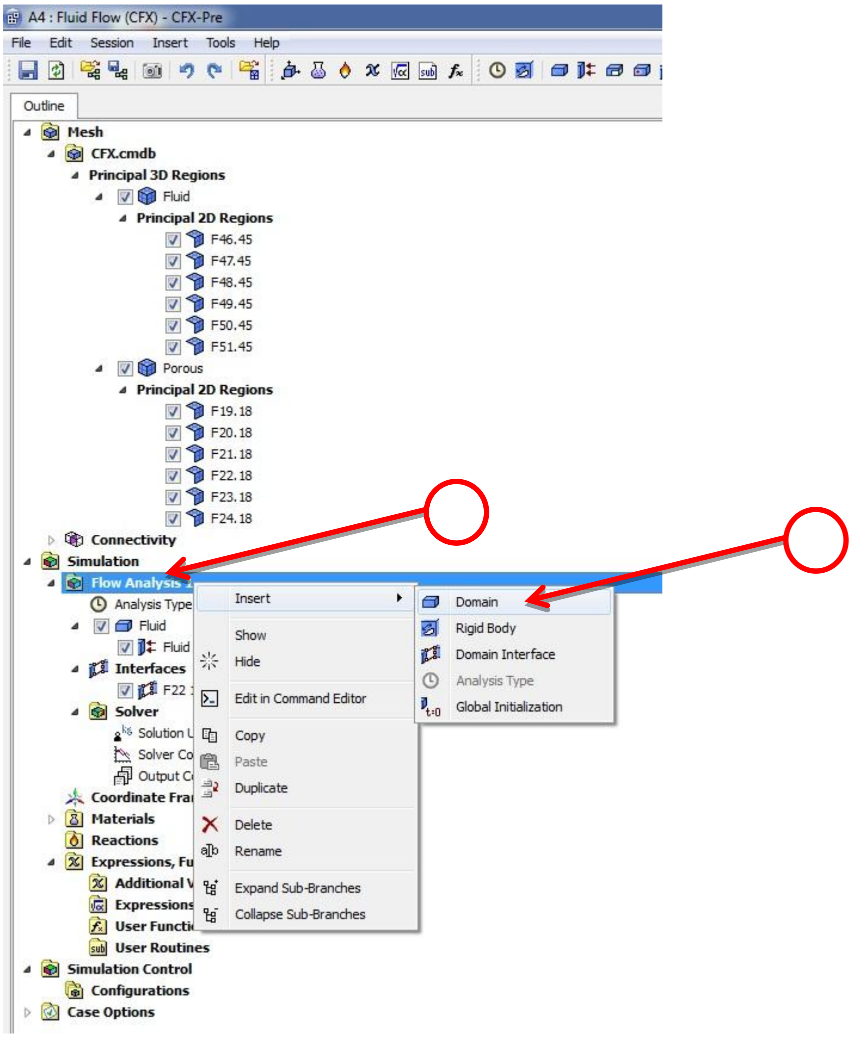
Step 37:

The next step is to enter in the Name section Air and then press OK, then press apply and then press OK.



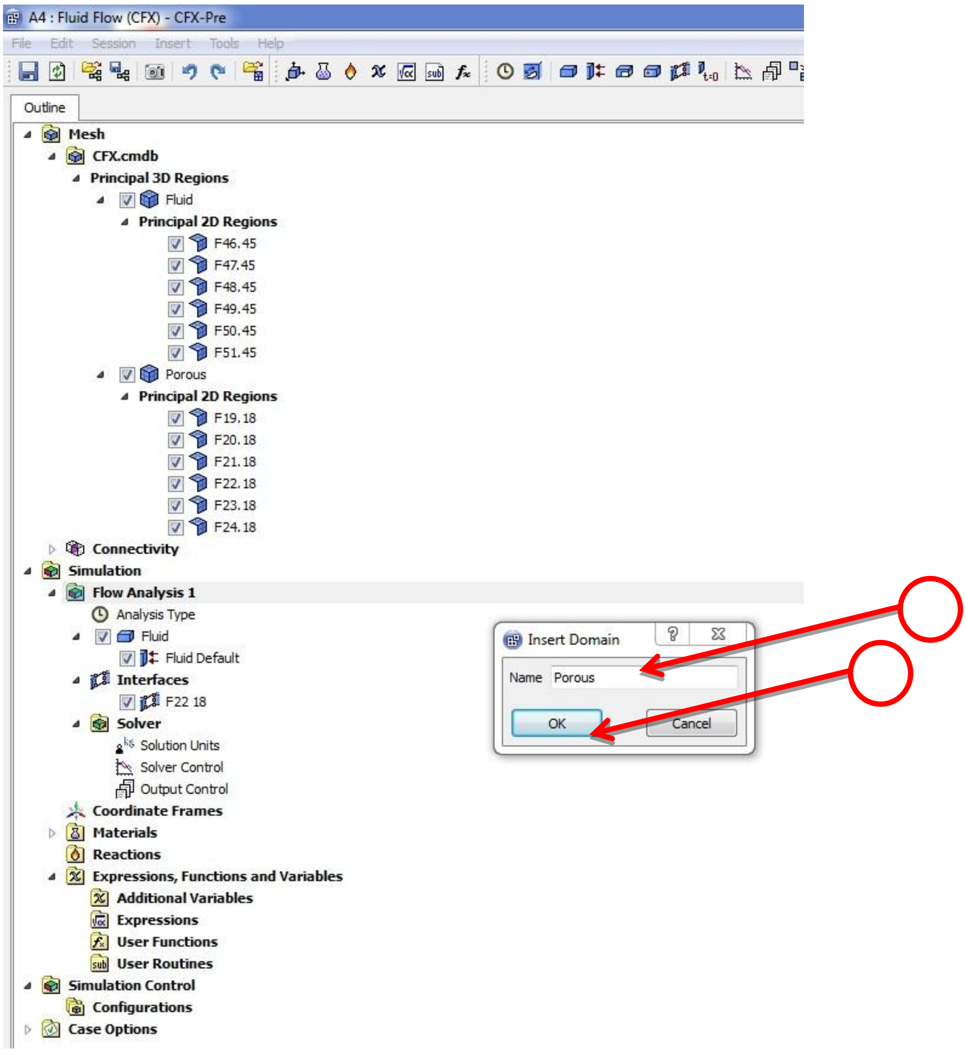
Step 38:

Select Flow Analysis and then right click the mouse button go to insert and select Domain.



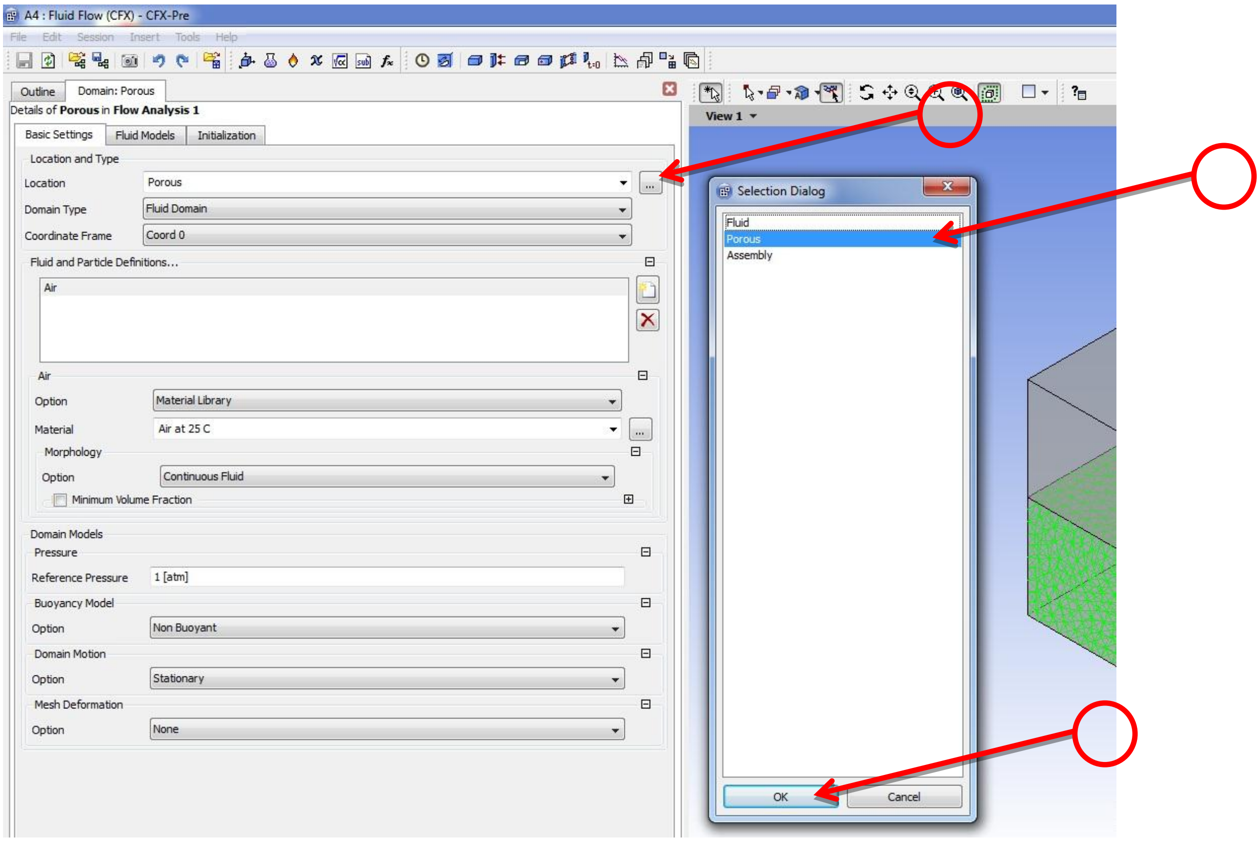
Step 39:

Type in the Name Porous and press OK.



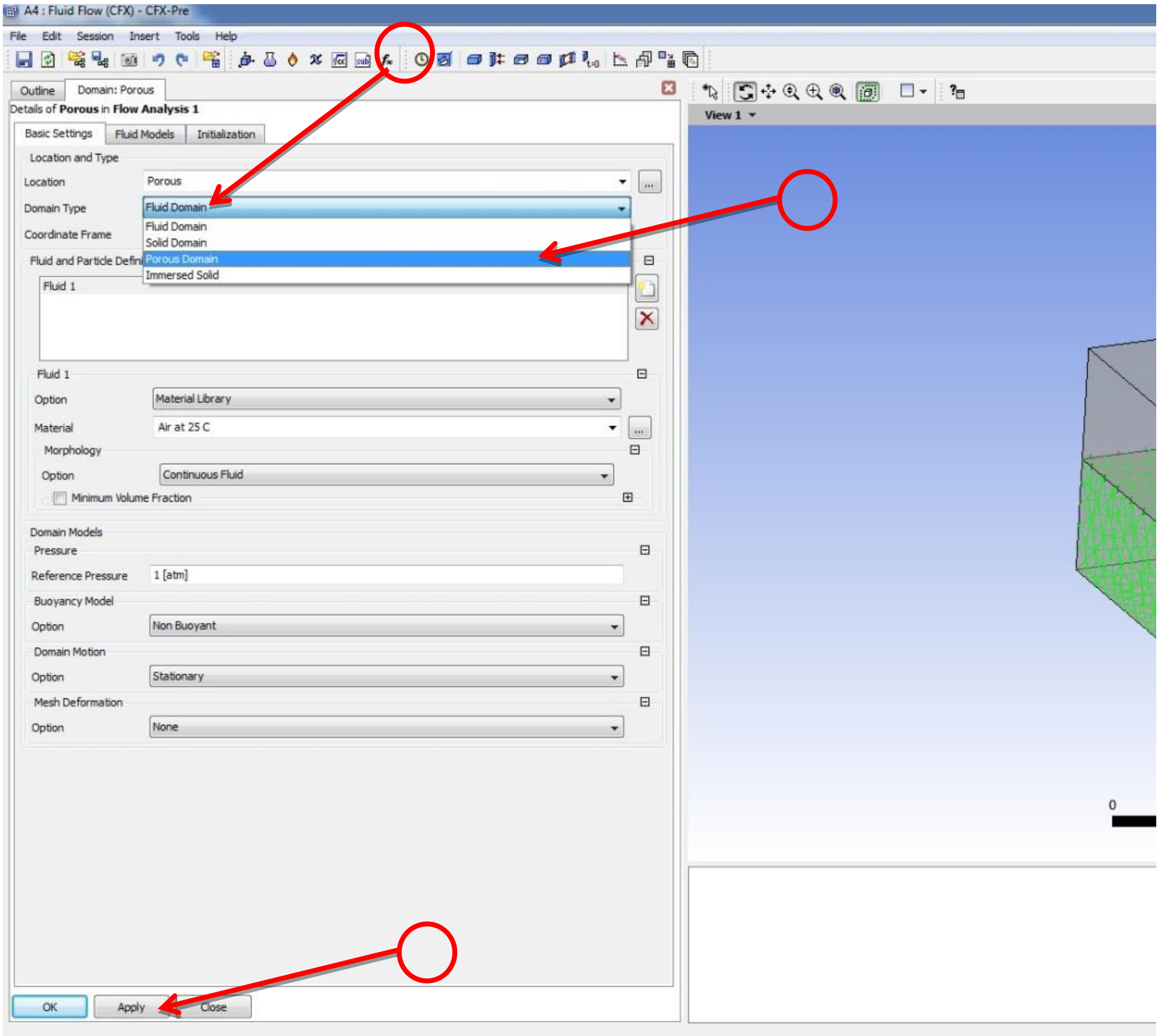
Step 40:

Press on the button beside the location selection, a window will open up select Porous and press Ok.



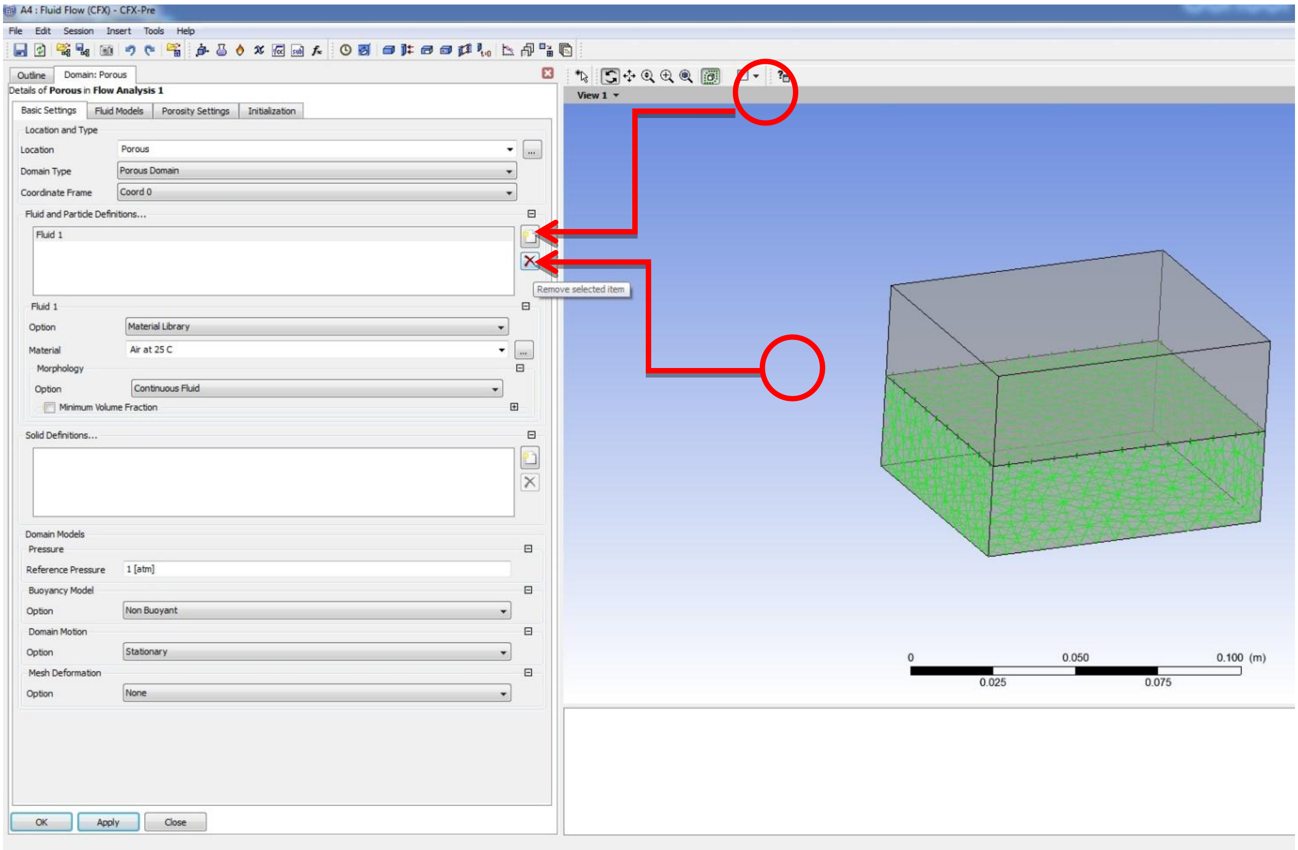
Step 41:

Select the porous domain right click the mouse button and select edit, then go to Basic Settings and select Porous Domain.



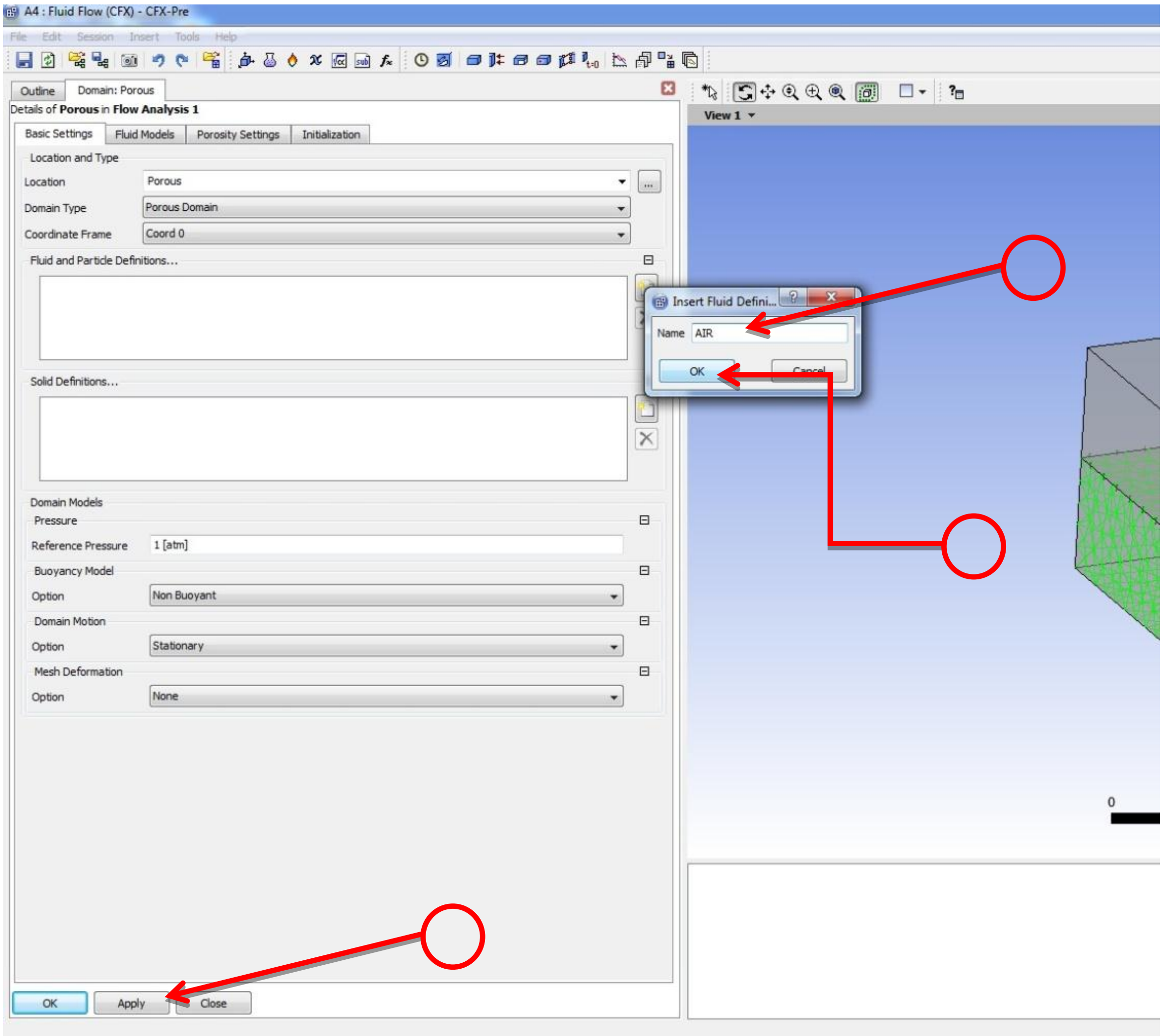
Step 42:

The next step is to go to Basic Settings and click on the remove selected item, next comes clicking on the new item icon.



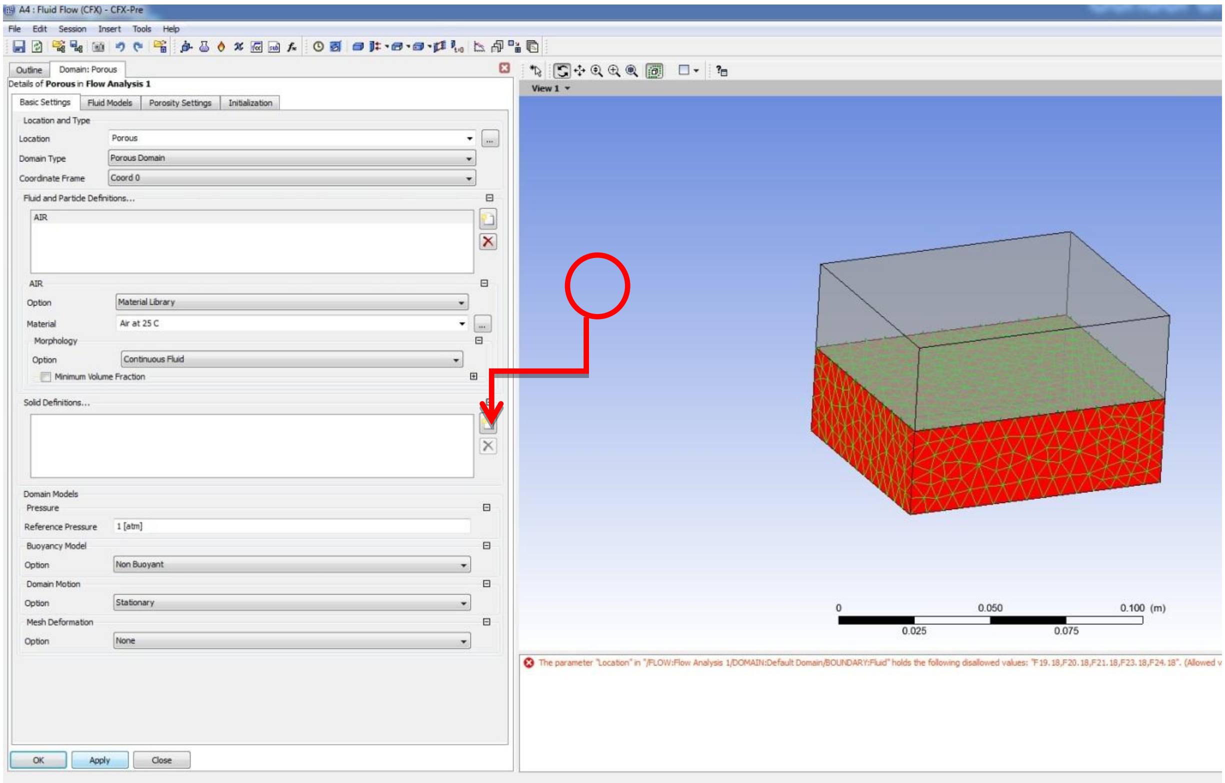
Step 43:

Enter in the name cell: Air and press Ok. Then press the apply button.



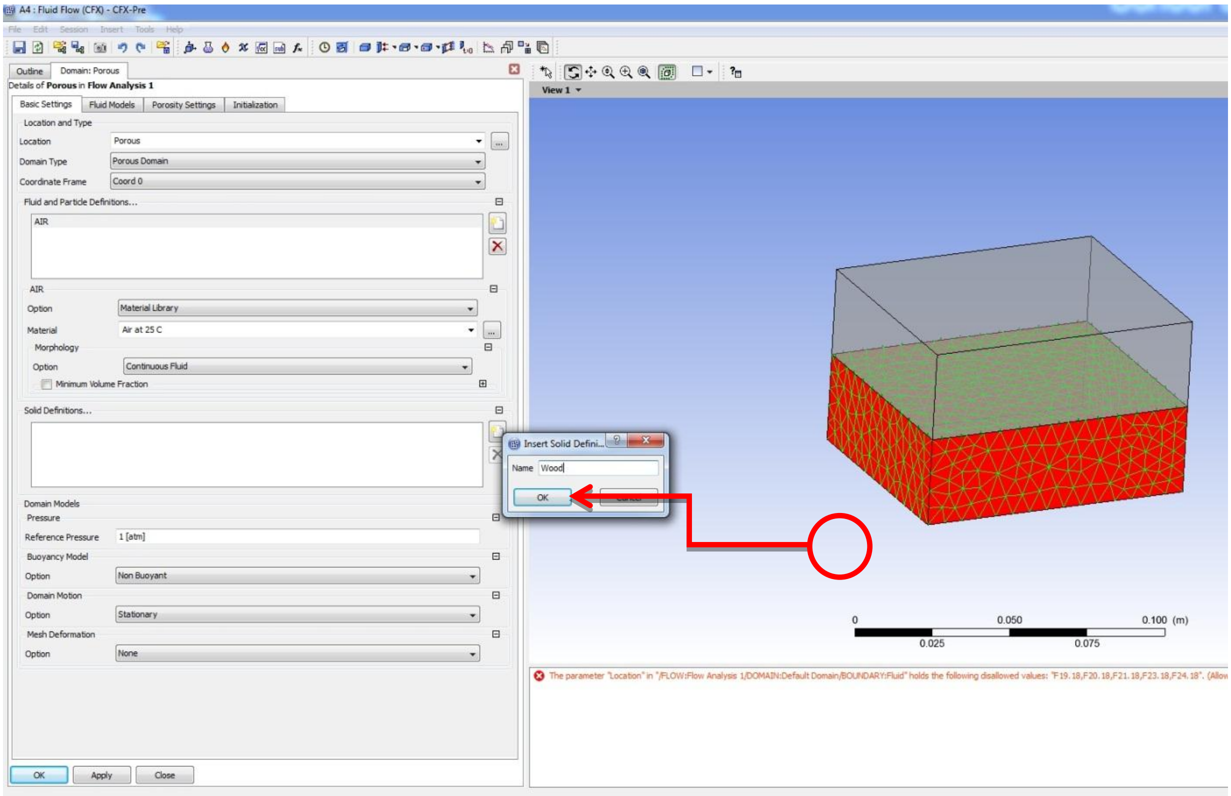
Step 44:

Ignore for now the caution sign shown in red, go to Solid Definition and press the material assign icon.



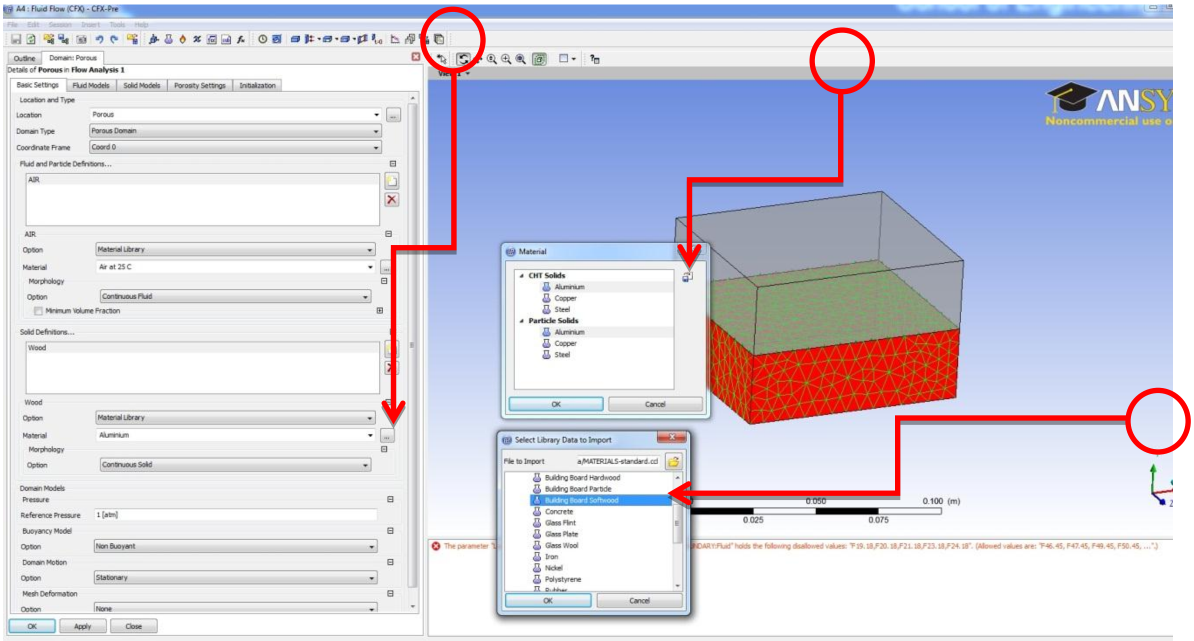
Step 45:

Enter the name wood and press Ok. The material names will be added to the list.



Step 46:

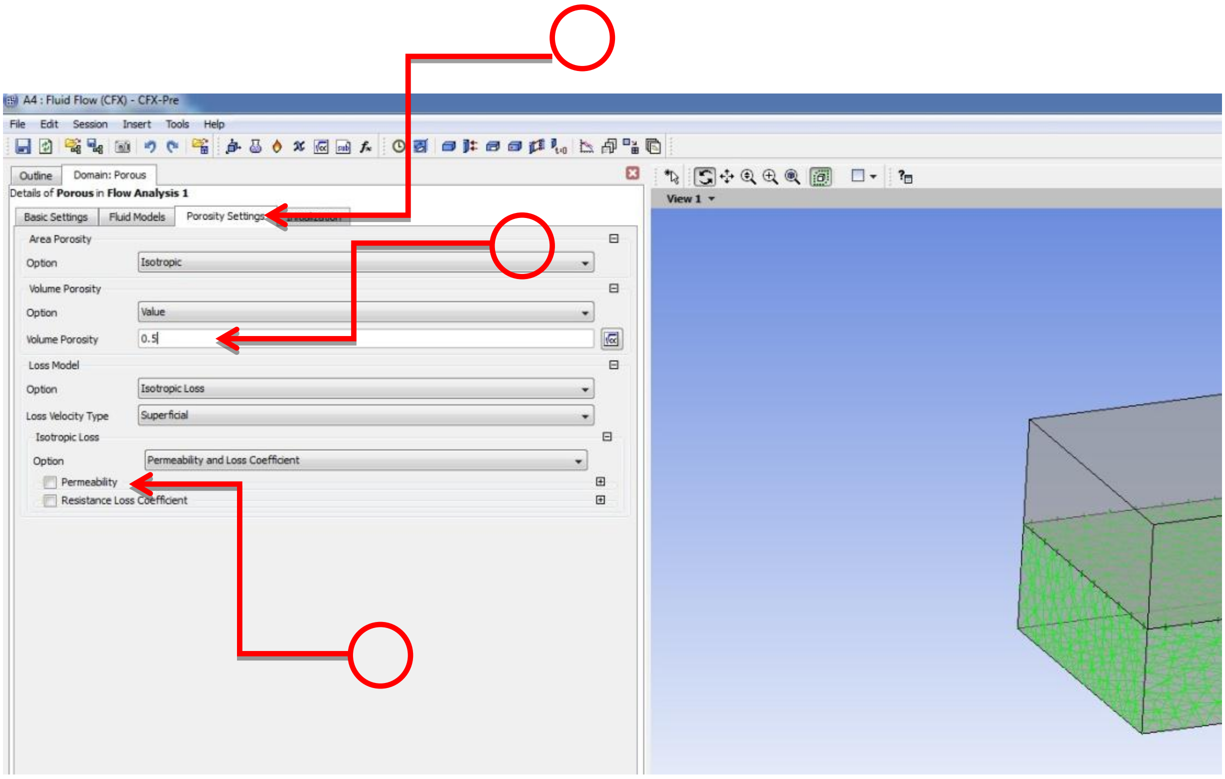
Press on the material library button, then access the library by pressing on the icon that has a floppy disc sign, a new window will open select the material Building Board Softwood (You can choose a material that mimics your studied case). If the material is not available in the library then the user can manually input the properties of the material.



Step 47:

The next step comes is through going to the porosity Settings section and assigning a value of 0.45 to the volume porosity input cell. To apply permeability click in the tick box. A note to the user that he can assign the value of porosity according to the material he is studying meaning that different soils have different values of porosity the same applies for fruits, woods ….etc. The values of porosity for different material are found in handbooks or specialized books such as :

Principles of Heat Transfer in Porous Media (Mechanical Engineering Series) byMaasoud Kaviany

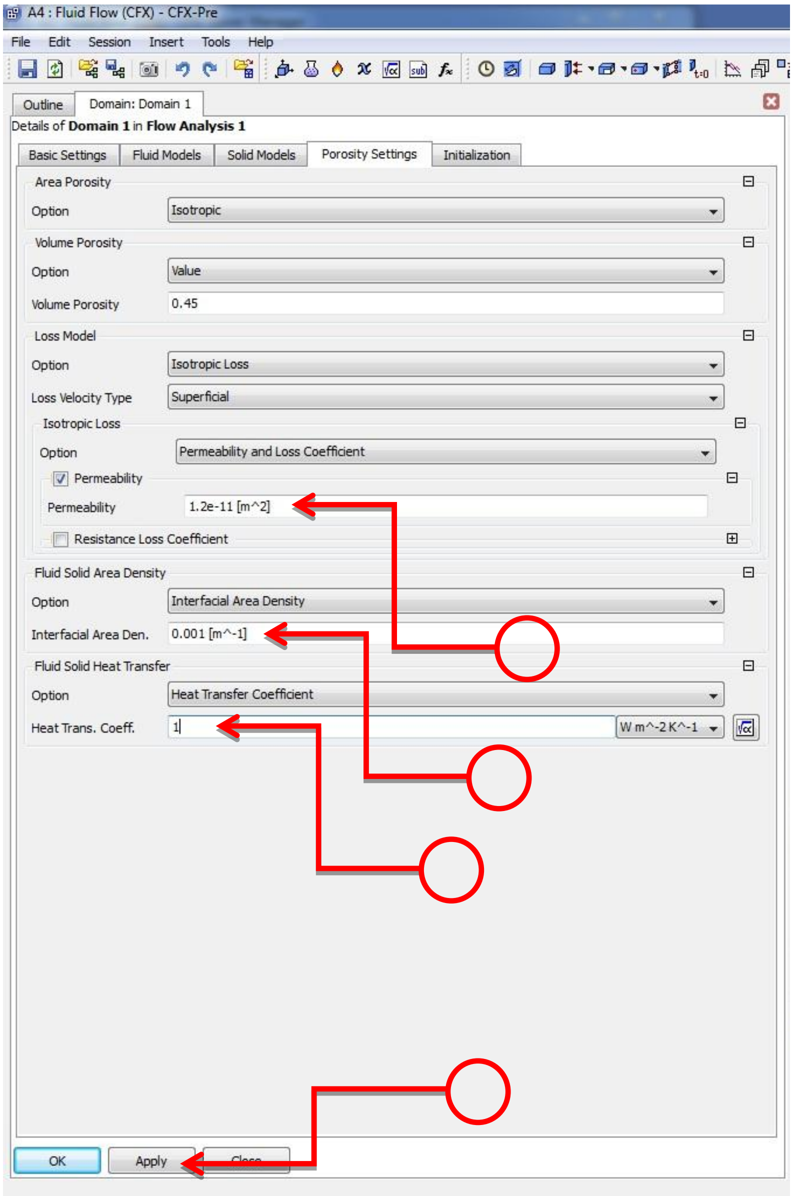


Step 48:

The next step comes here in applying the permeablity values which are the following

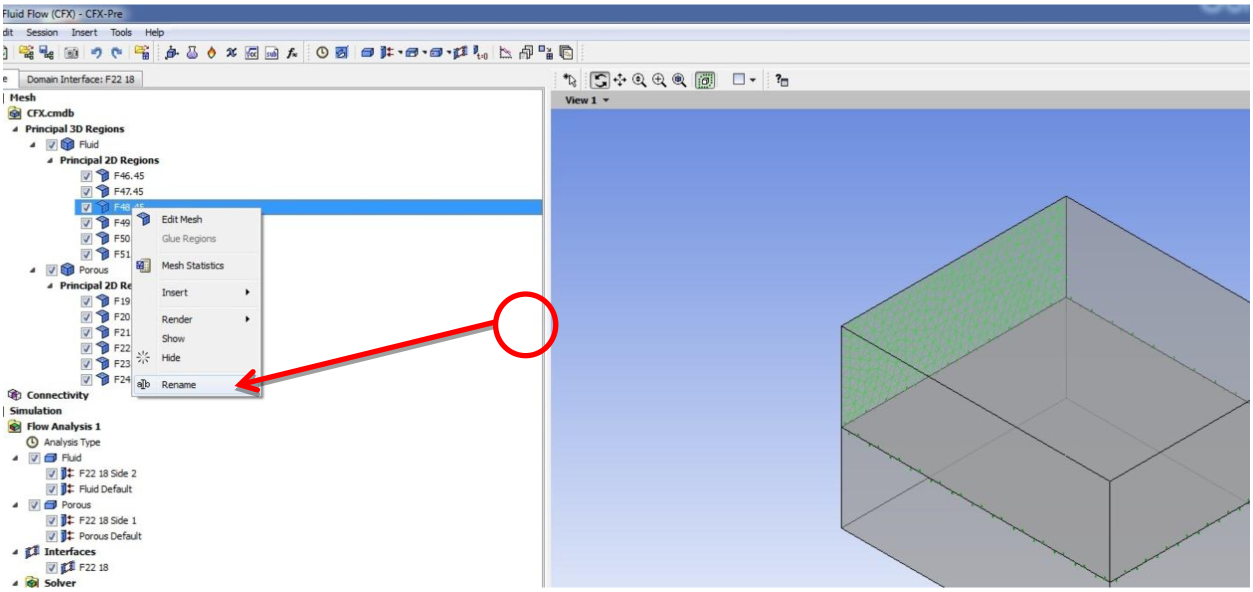
$$K = 1.2 \times 10^{-11}\ m^2$$

Again the value of permiability varies depending on the type of studied material. Assign a value of 0.001m for the Interfacial Area Den. Finaly assign the Heat TRANSFER. Coefficient a value of 1 again you choose its value depending on your problem.



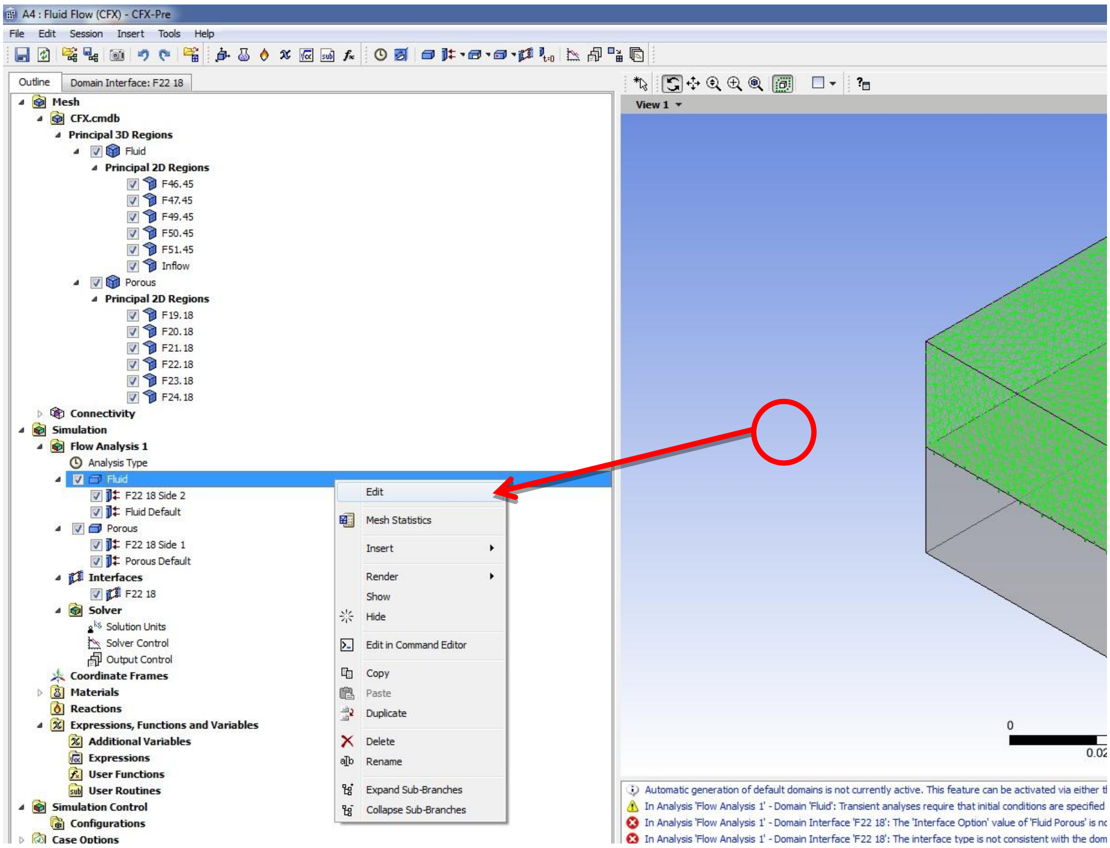
Step 49:

Select the highlighted region and rename it to inflow.



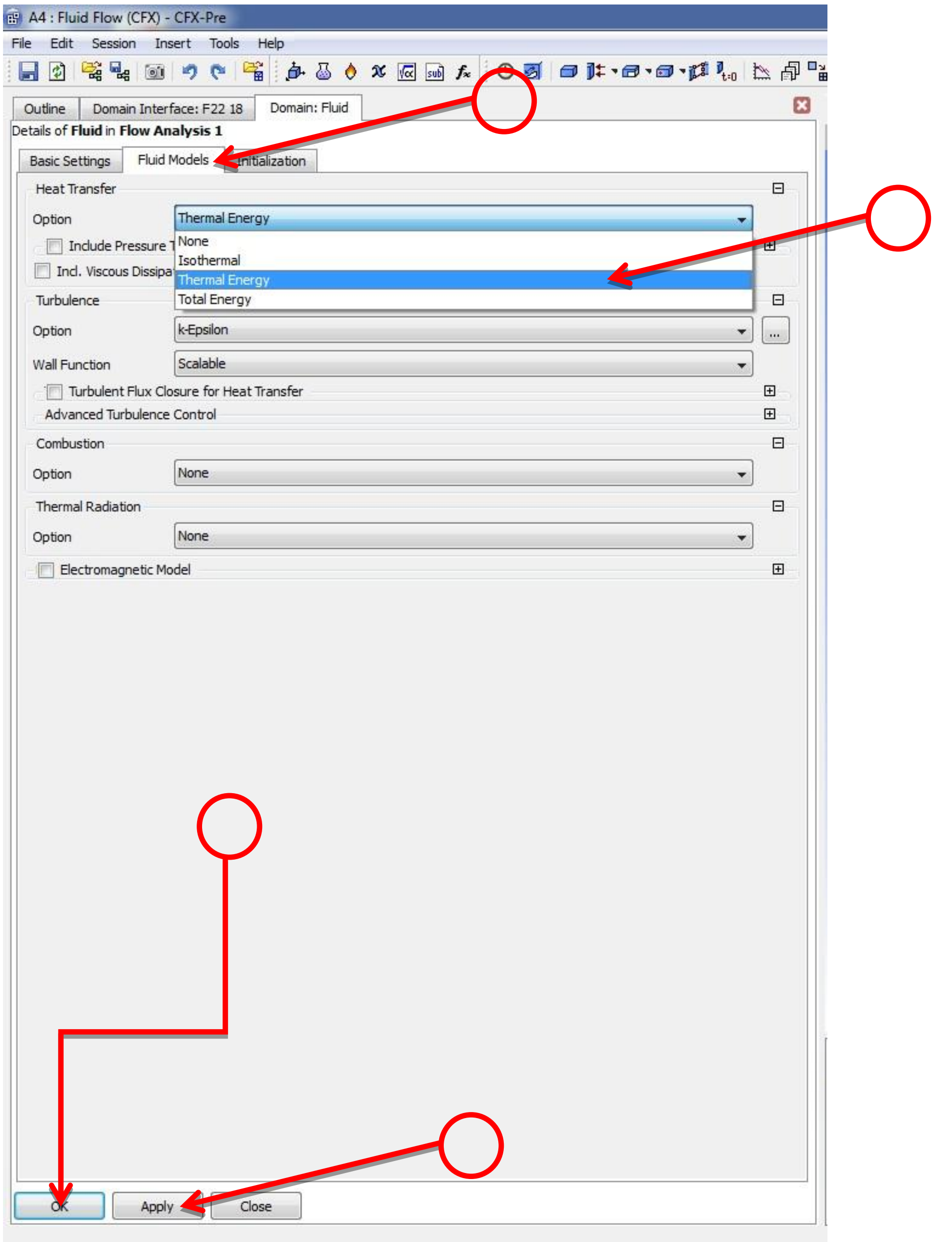
Step 50:

We need to solve the heat equation therefore we need to specify that. This is done through selecting the Fluid domain and then clicking the right mouse button and selecting edit.



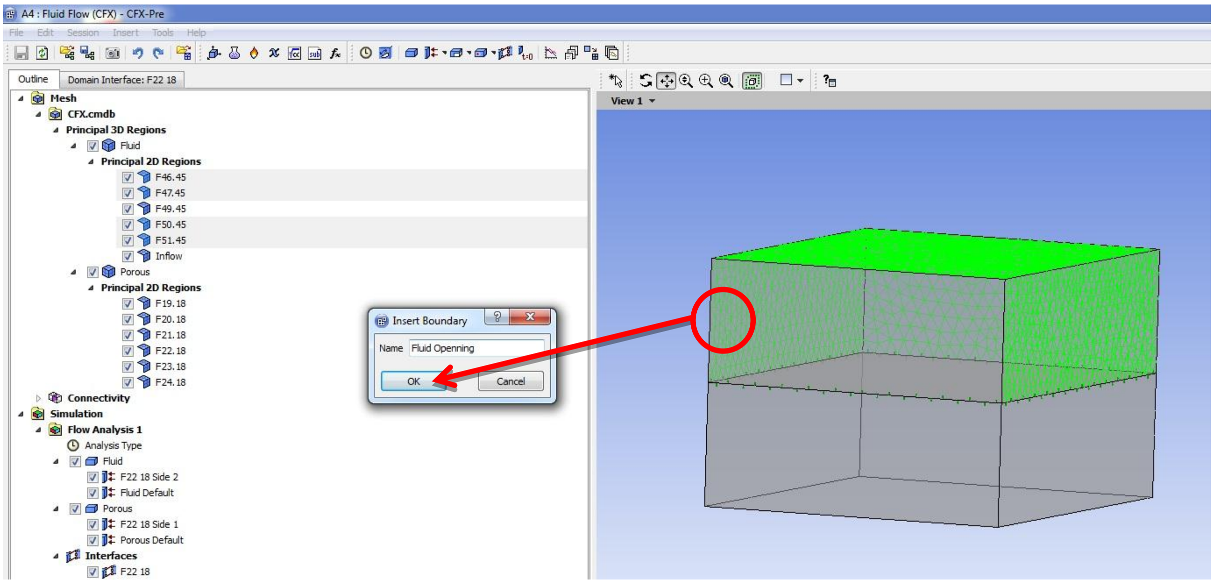
Step 51:

Go to Fluid Models and select Thermal Energy. Then press apply and OK.



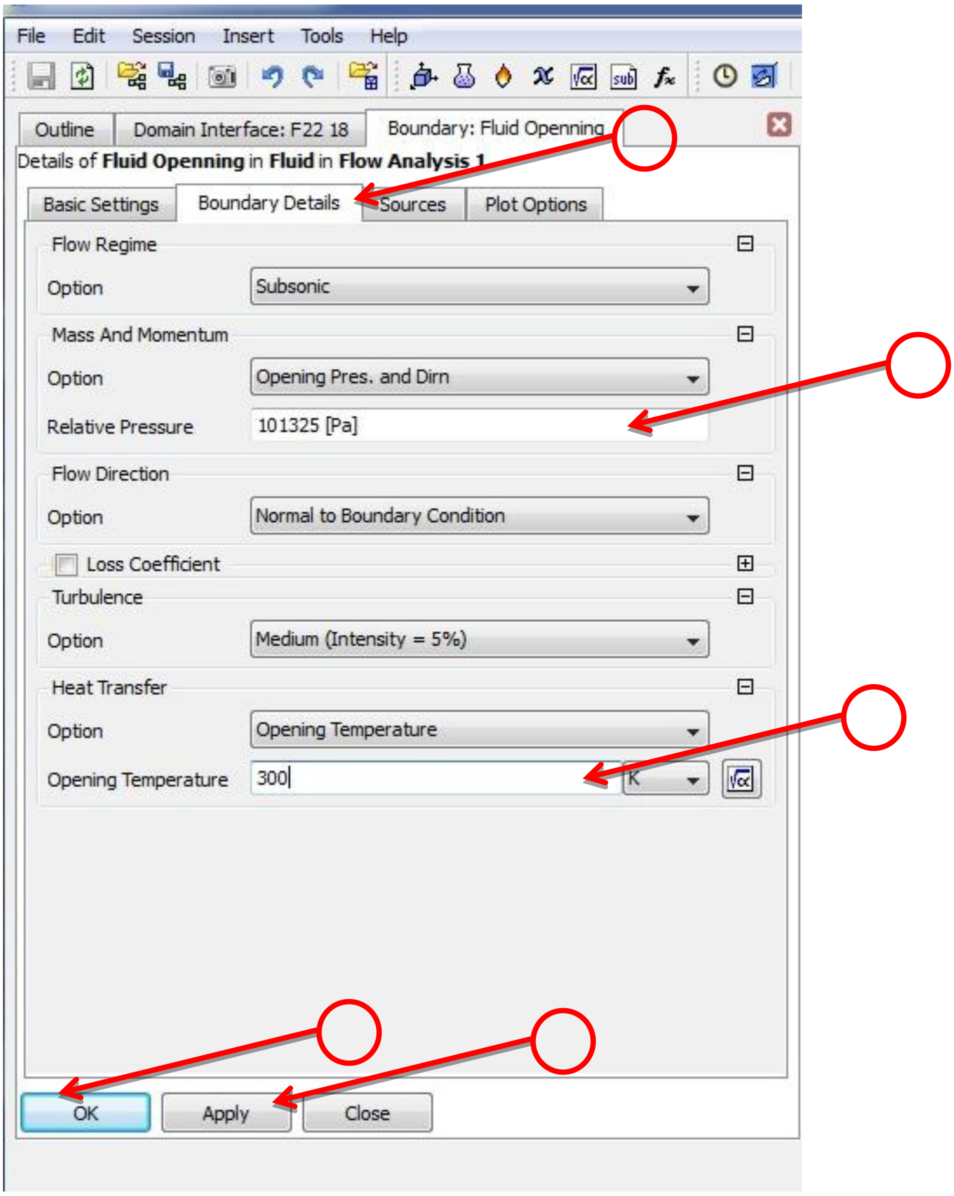
Step 52:

Select the green regions right click on the mouse button and select Insert boundary rename it as Fluid Opening and then press OK.



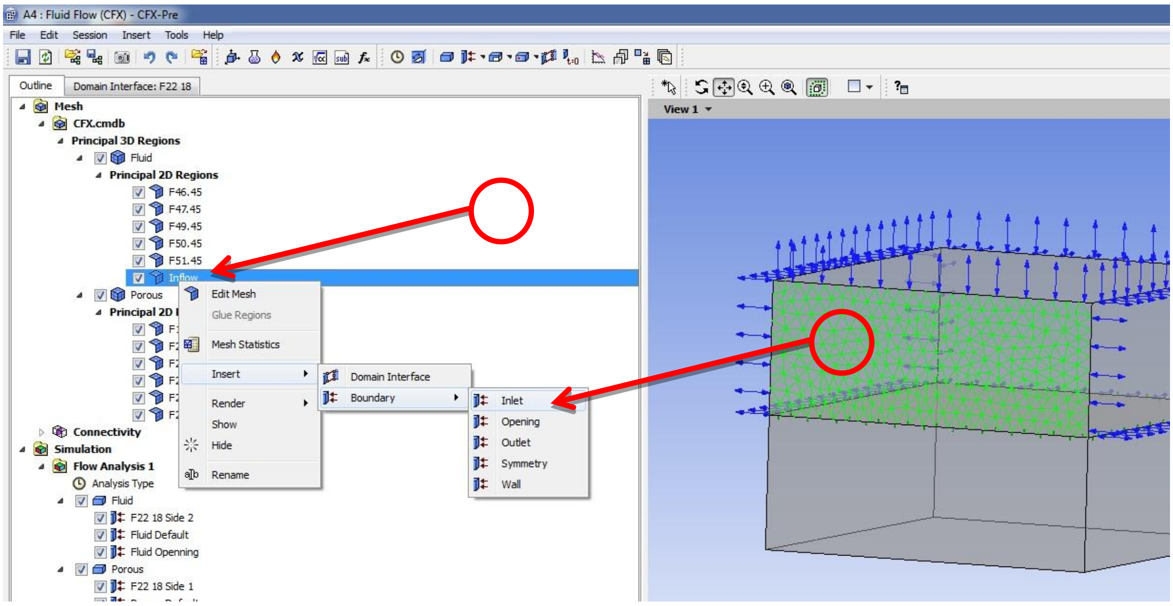
Step 53:

Select the boundary details, enter in the relative pressure section 101325 Pa, also enter the value of 300 K in the Opening Temperature Section. Then press apply and press OK.



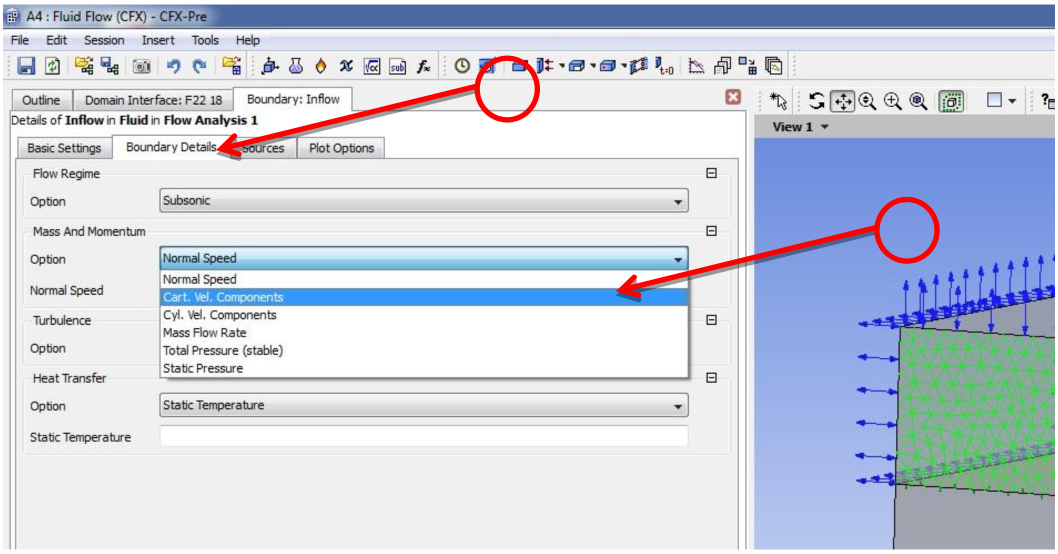
Step 54:

Select the inflow region, click on the mouse right button go to boundary, then select Inlet.



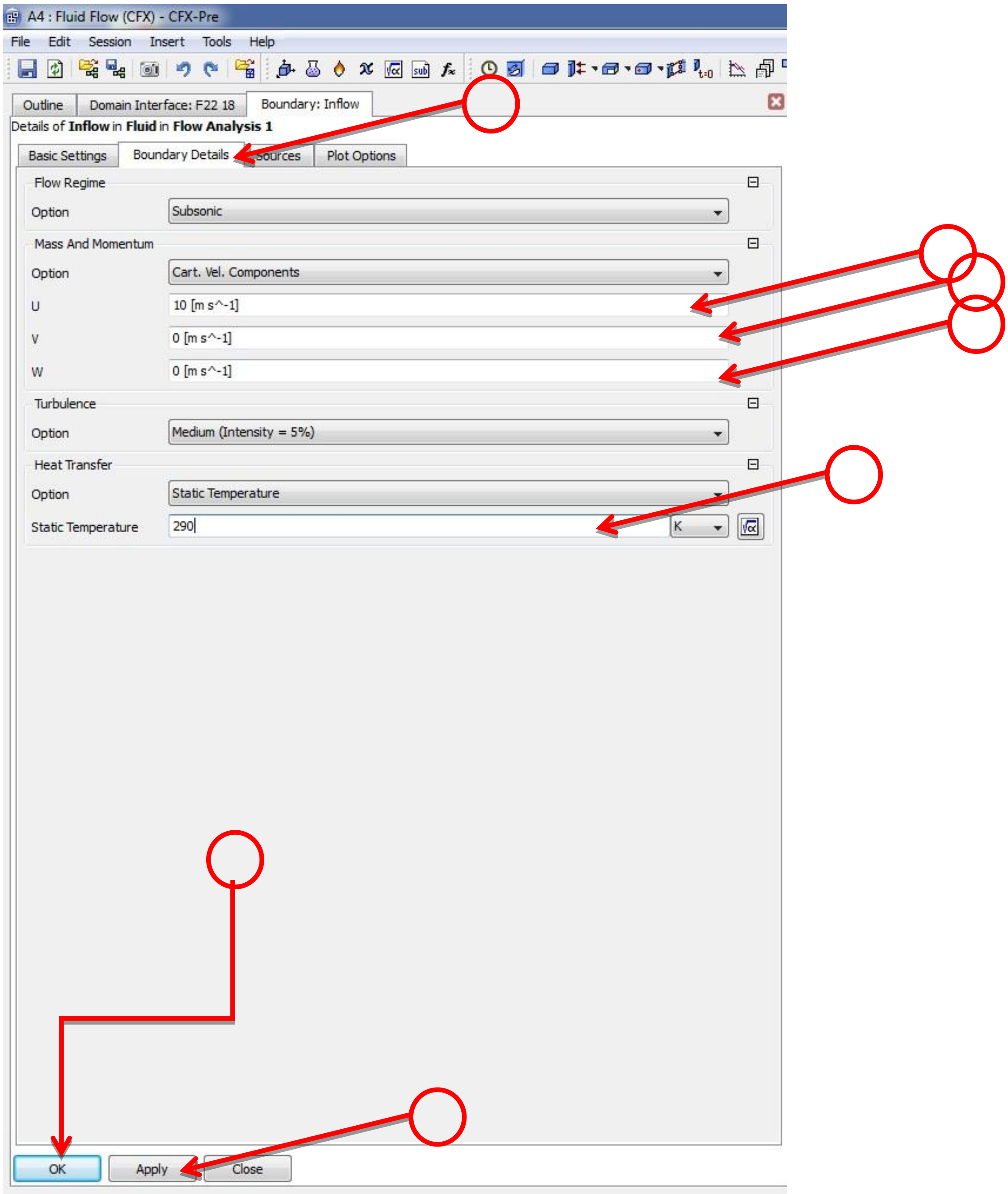
Step 55:

Click on Boundary details, then select Cart Vel. Components.



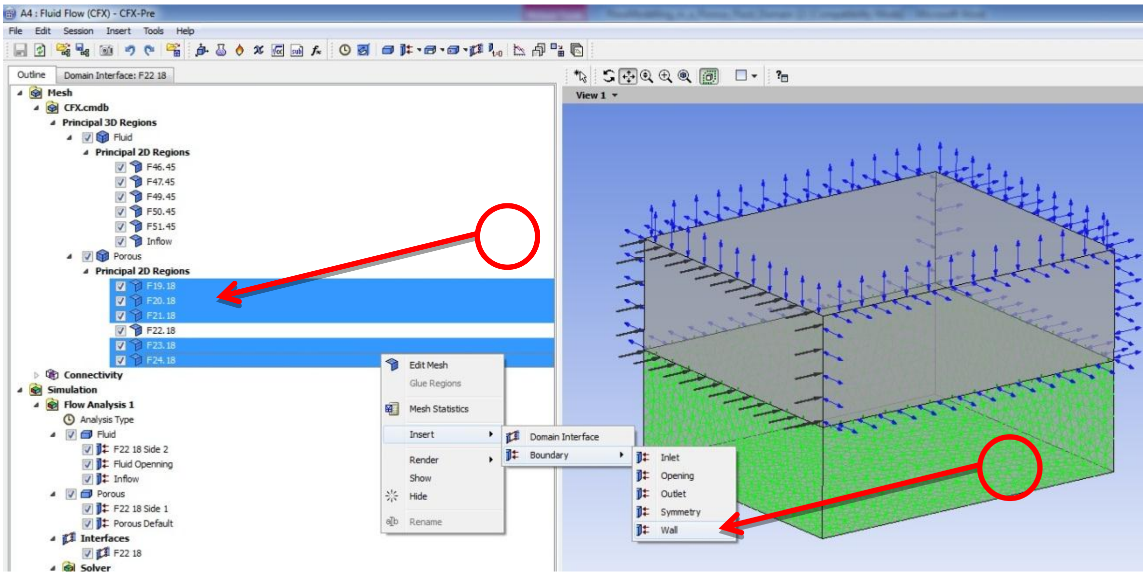
Step 56:

Enter the value of 10 m/s for the U velocity while apply zero for both the V and W component. Then assign a static temperature value of 290 K. Then press apply and OK.



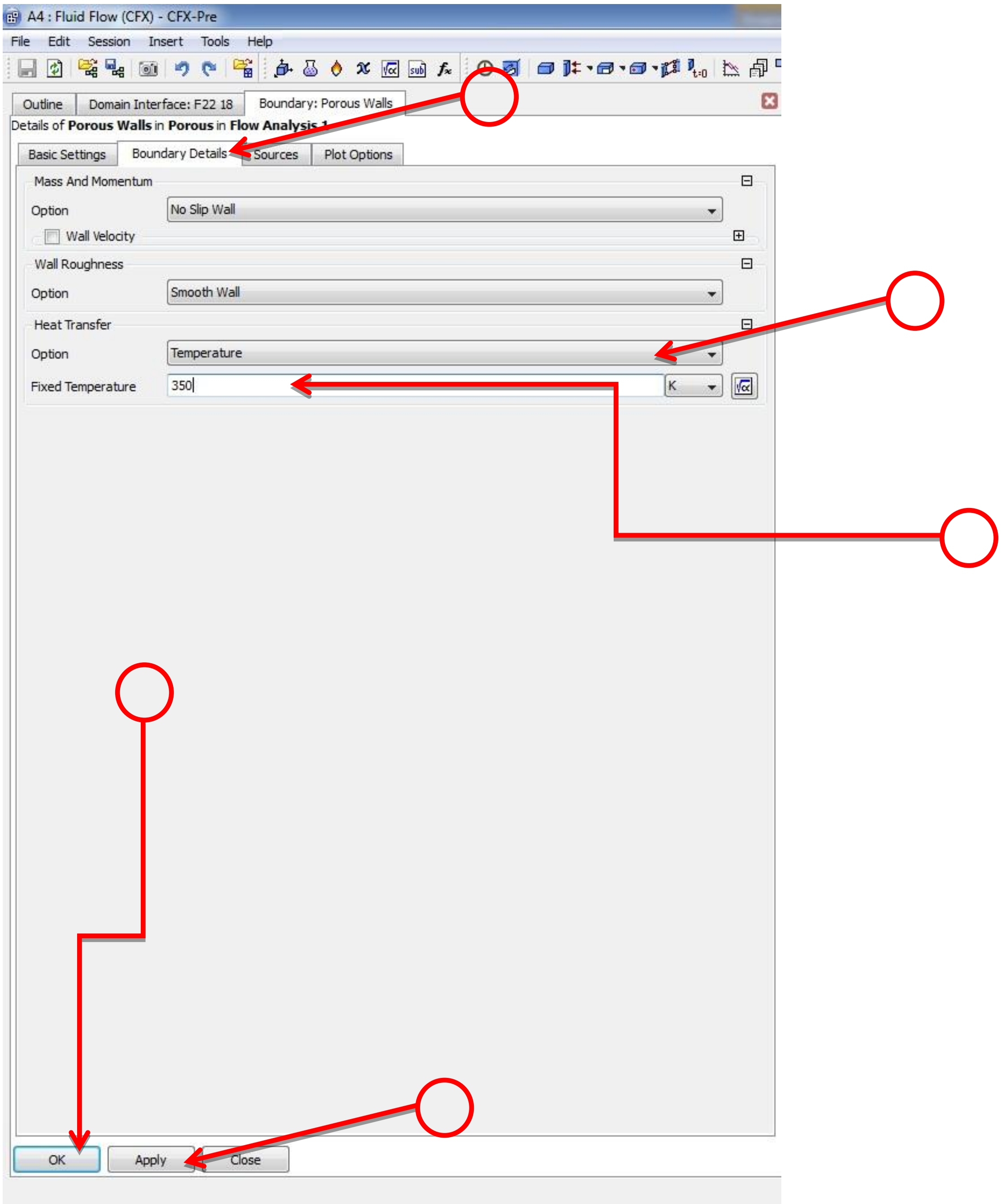
Step 57:

Select the 5 walls of the Porous domain (excluding the interface wall) then click the right mouse button, select insert, select Boundary then wall.



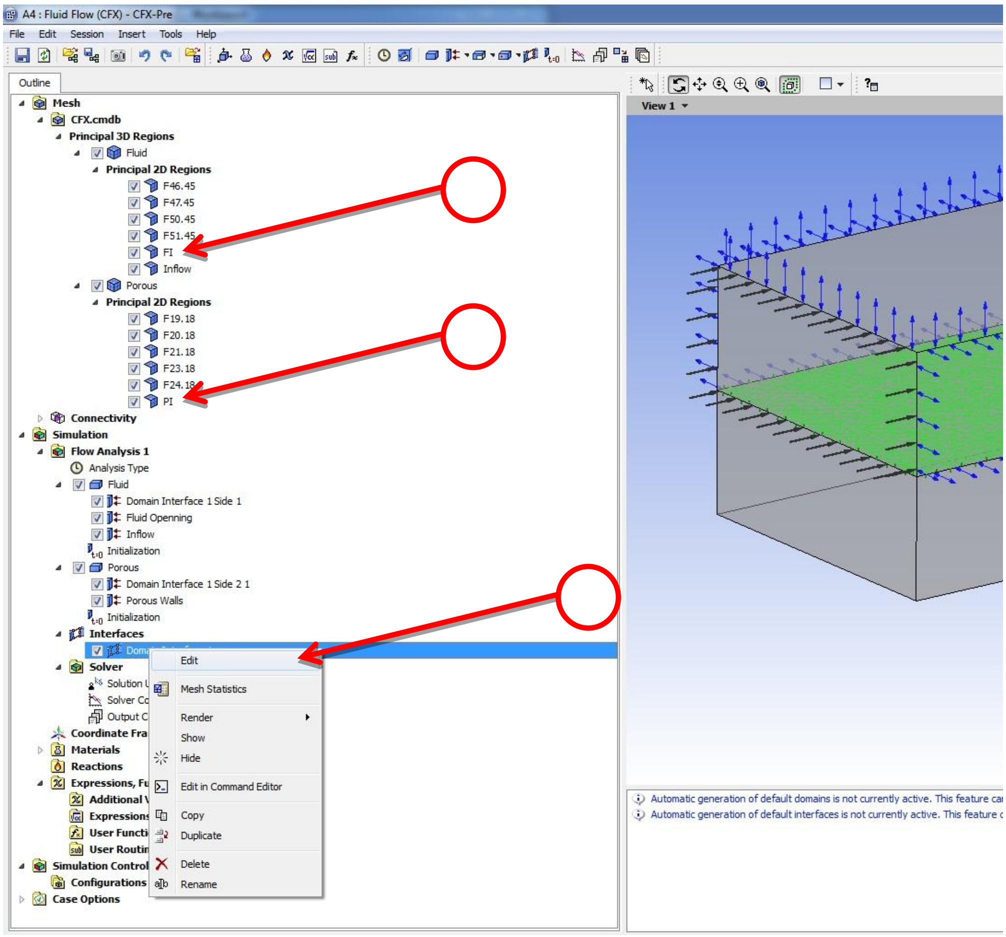
Step 58:

Click Boundary details then select the Option Temperature instead of adiabatic then enter a value of 350 K press apply and click OK.



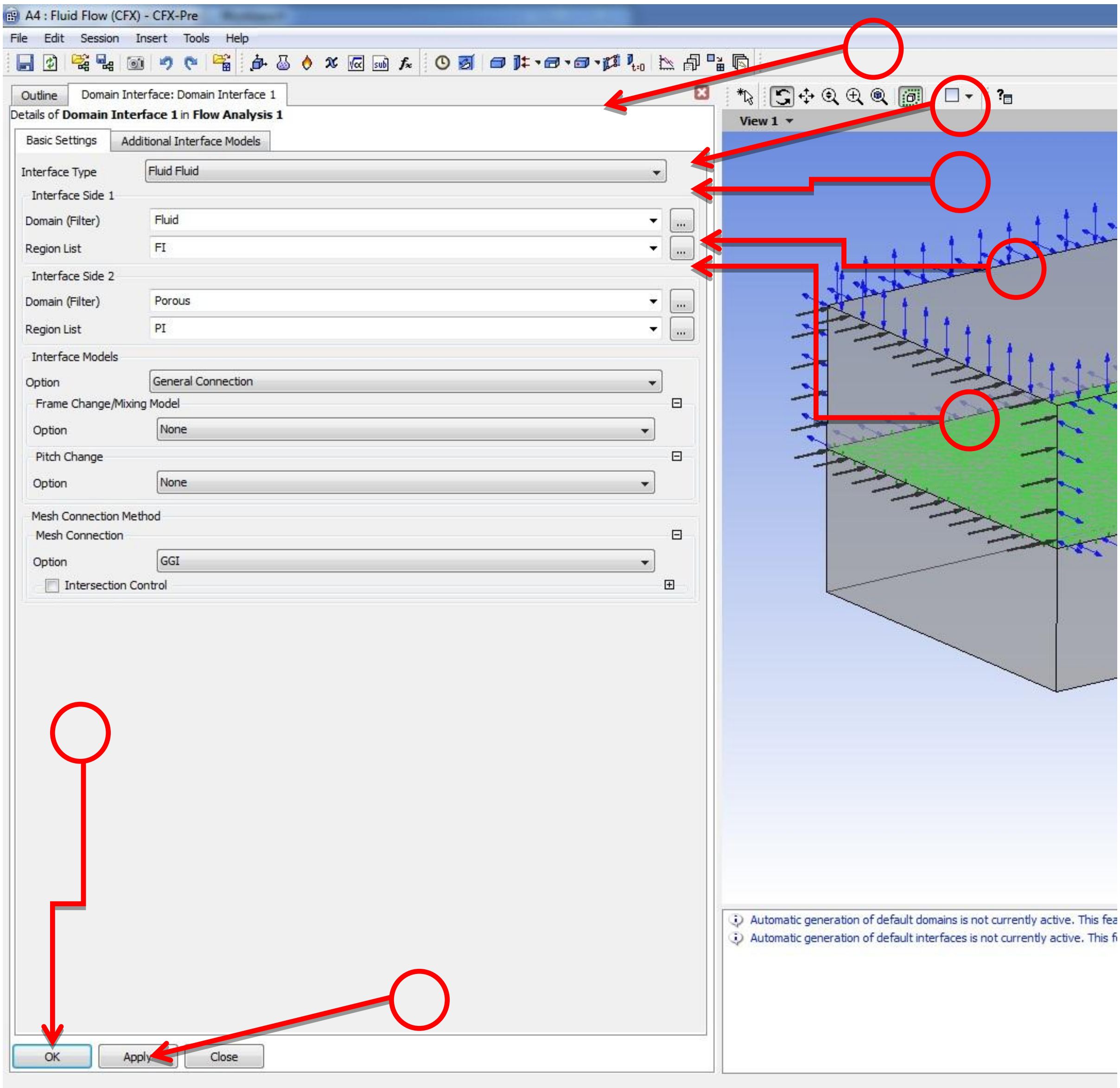
Step 59:

In the Fluid domain Principal 2D regions rename the interface surface with FI, in addition to renaming the interface surface in the Porous Principal 2D regions with PI. Select Domain interface and right click the mouse button and select Edit.



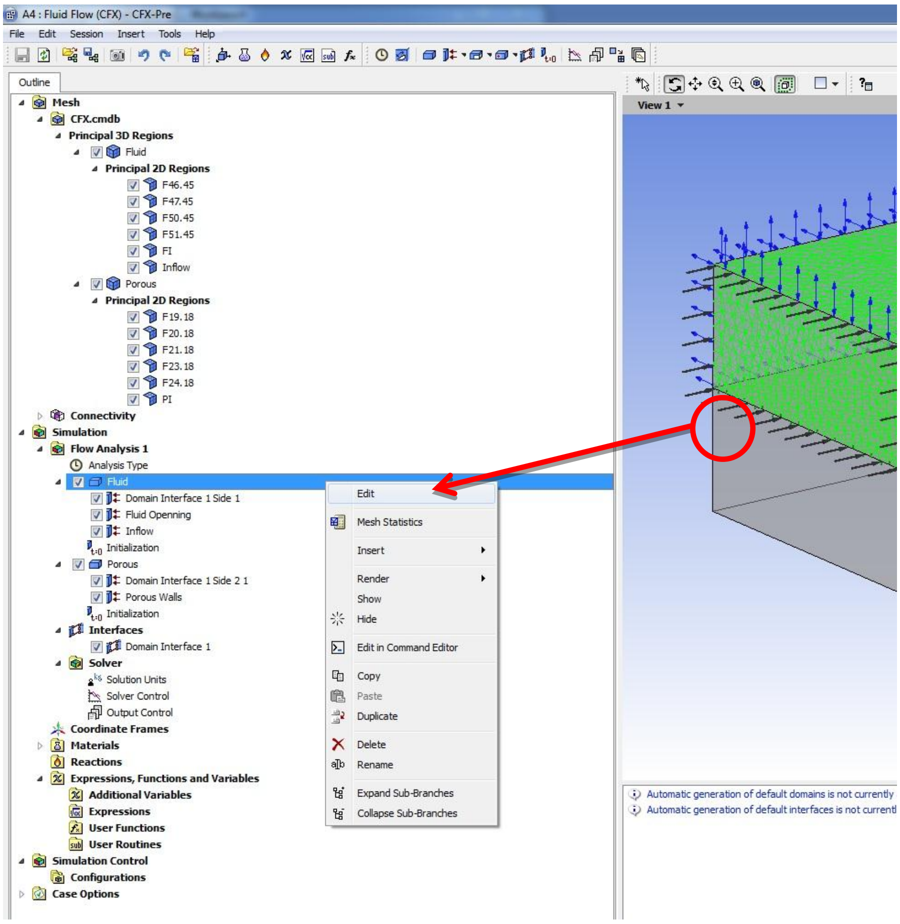
Step 60:

In the interface Type select Fluid Fluid, in the interface side 1 select in domain filter Fluid and in the regions List FI. In the interface side 2 select in domain filter Porous and in the regions List PI. The next step is to press apply and OK.



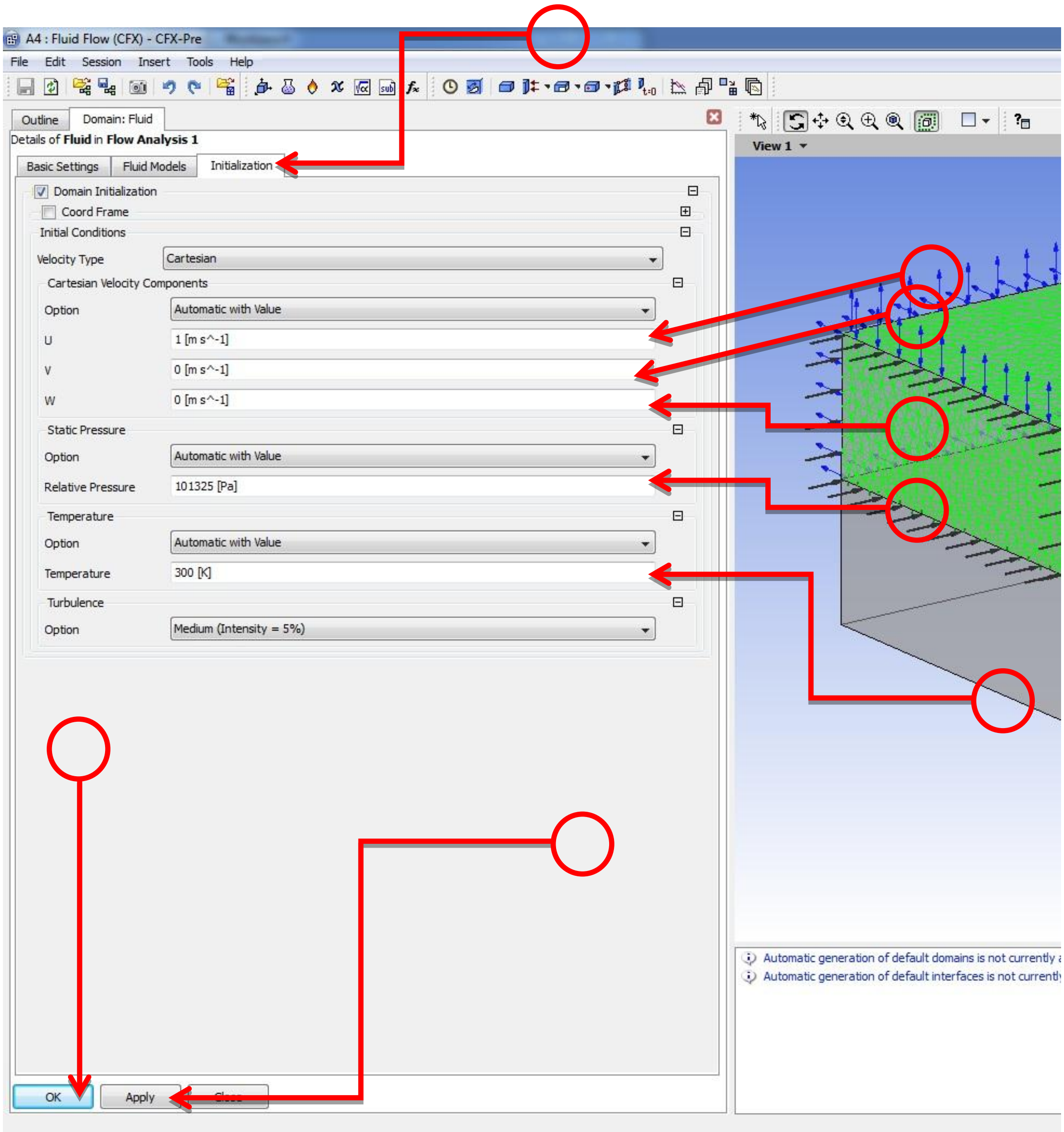
Step 61:

Edit the Fluid domain.



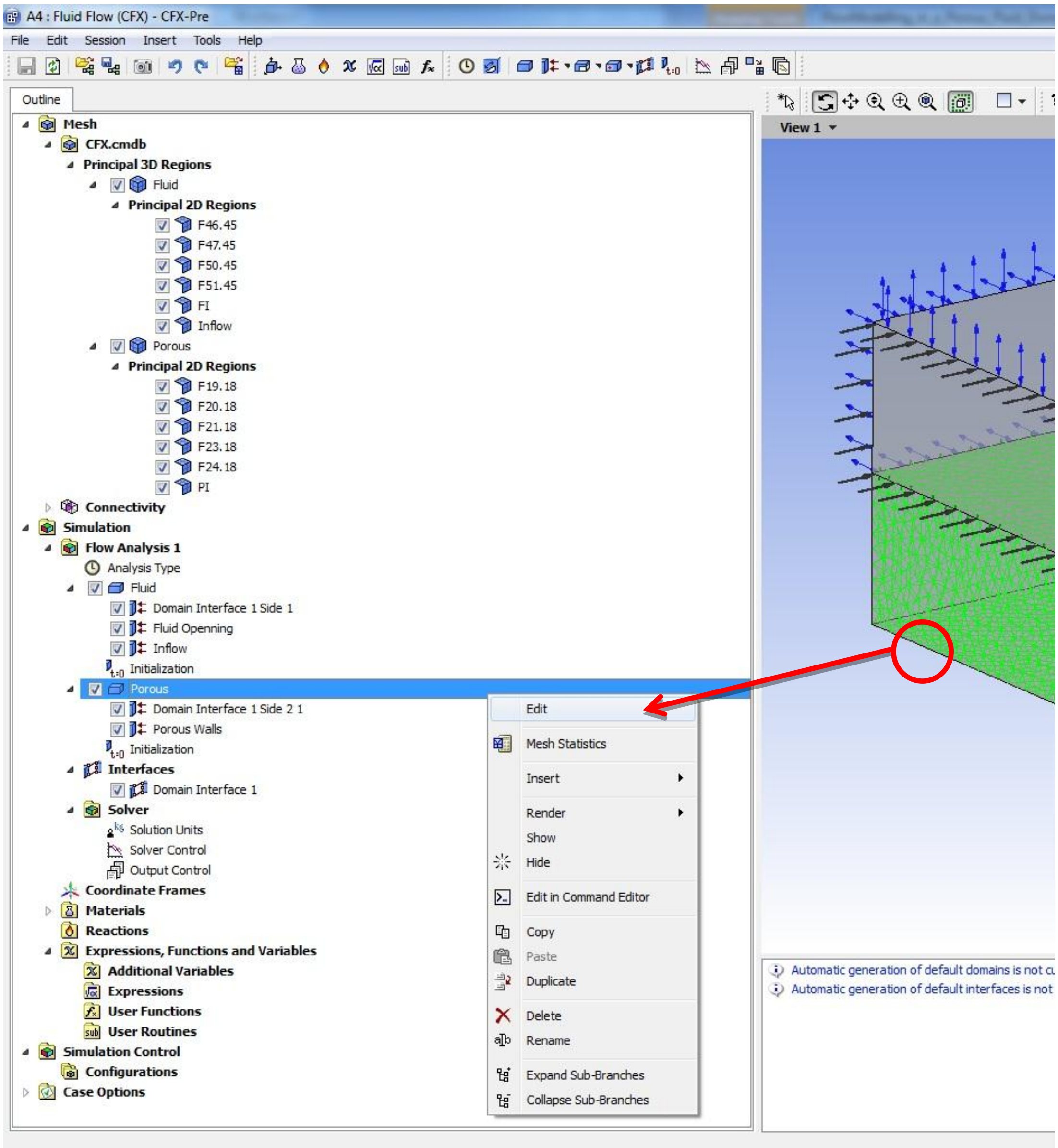
Step 62:

Click on Initialization, then assign the U component a velocity of 1 m/s, type in a relative pressure of 101325 Pa, and finally type a temperature value of 300 K. Then press apply and OK.



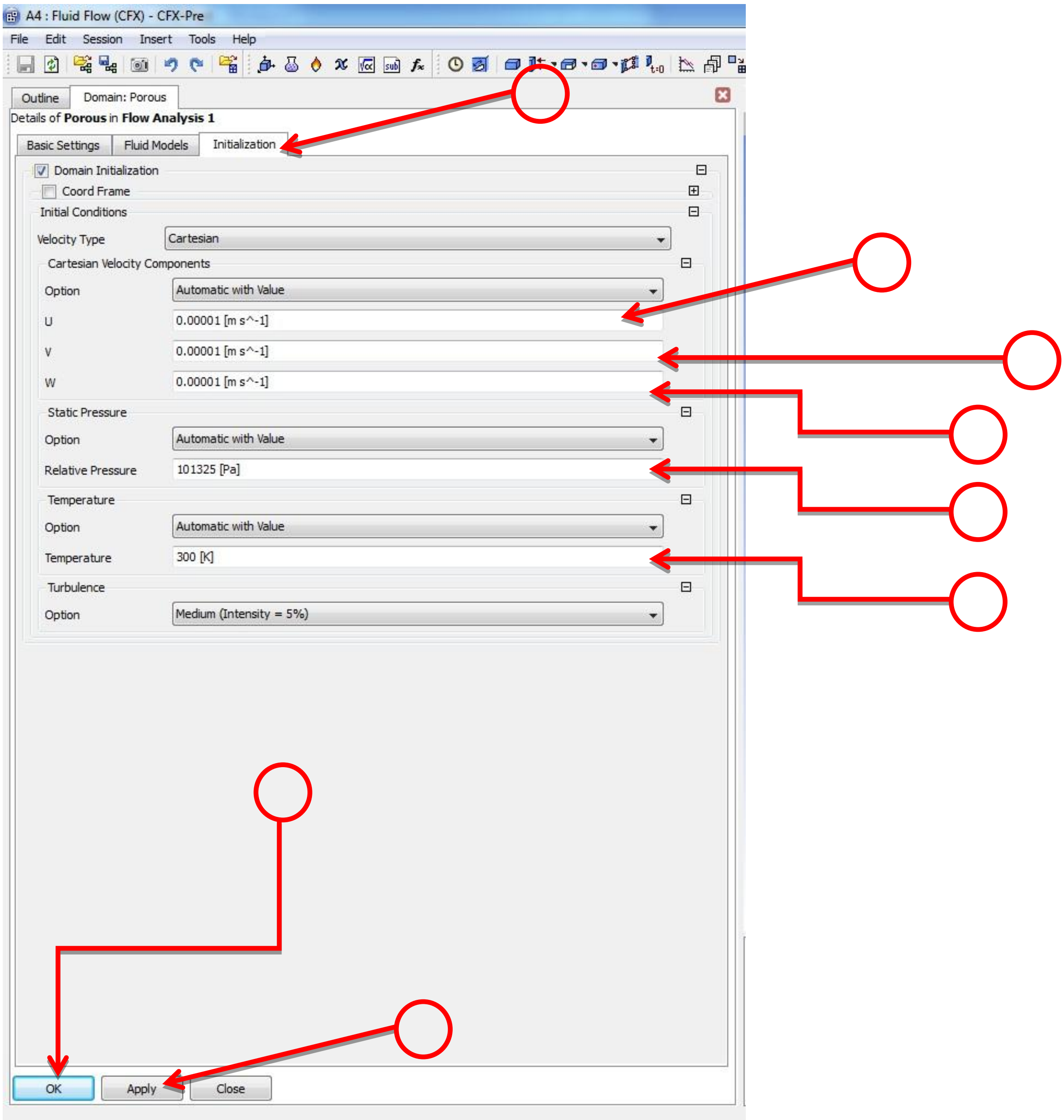
Step 63:

Edit the Porous domain.



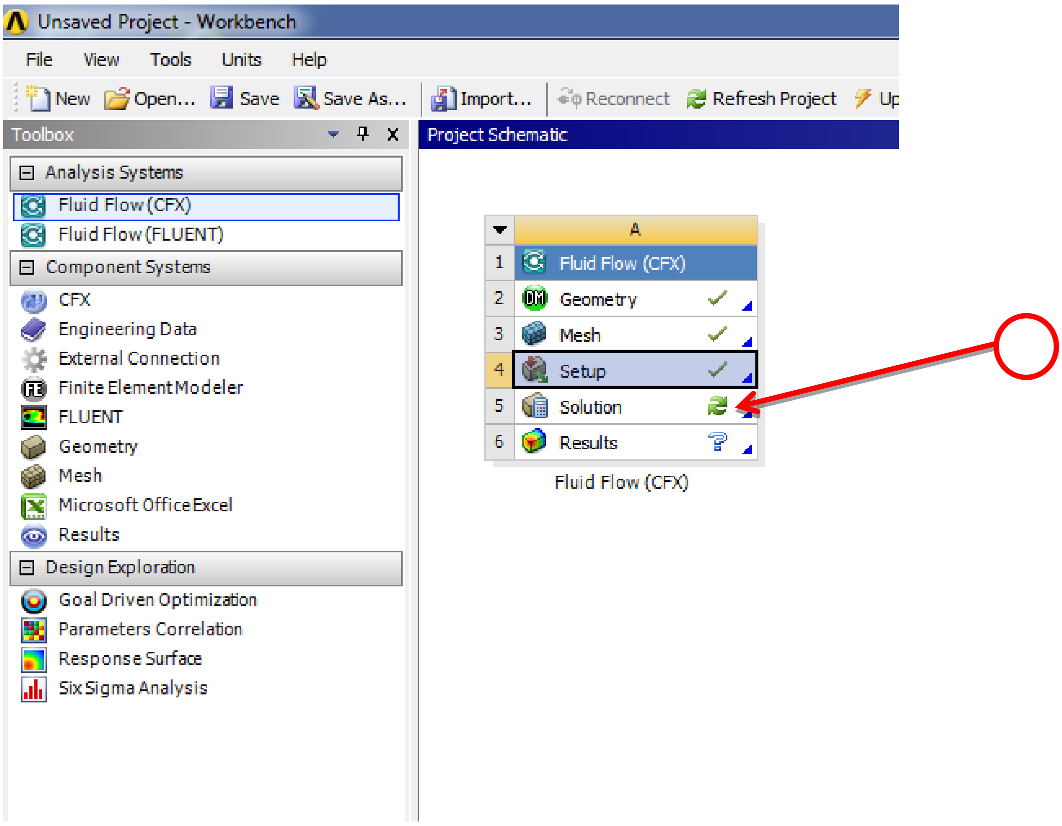
Step 64:

Click on Initialization, then assign the U,V and Wcomponents a velocity of 0.00001 m/s, type in a relative pressure of 101325 Pa, and finally type a temperature value of 300 K. Then press apply and OK.



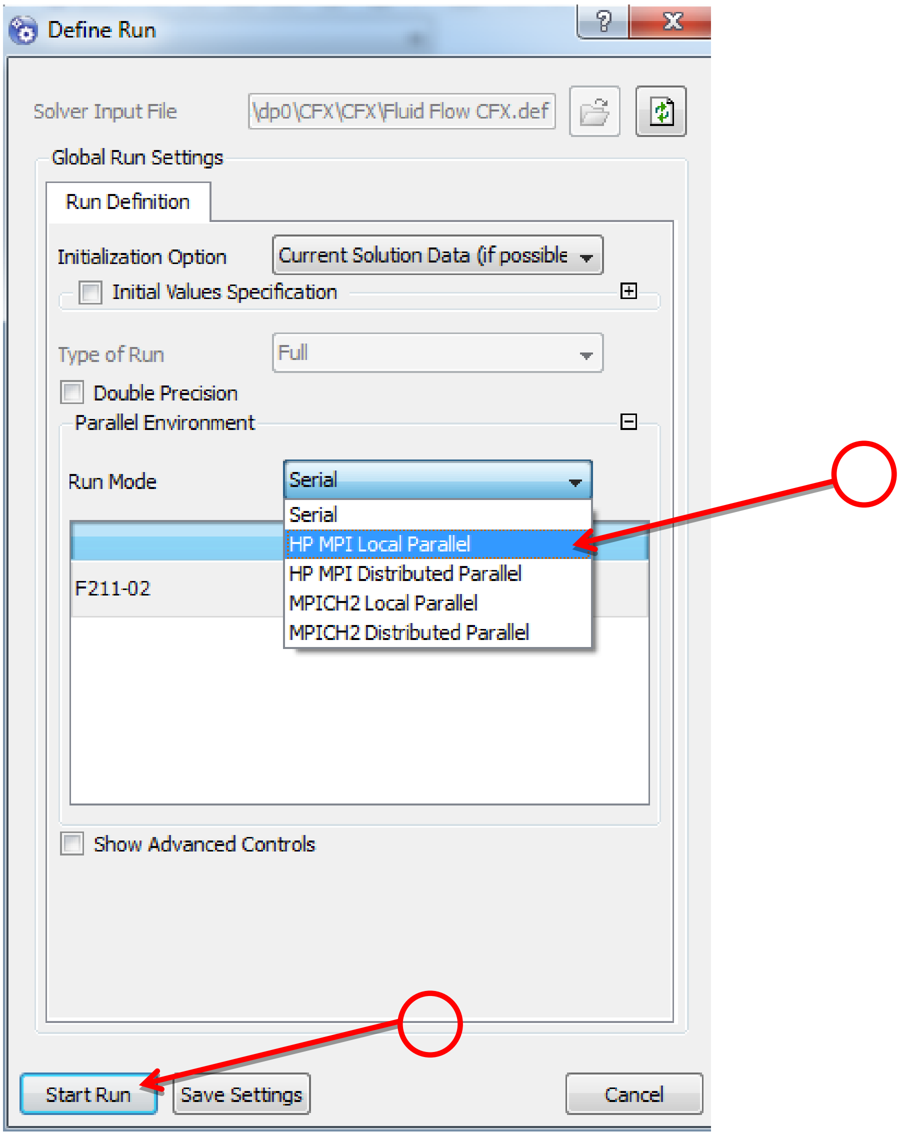
Step 65:

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



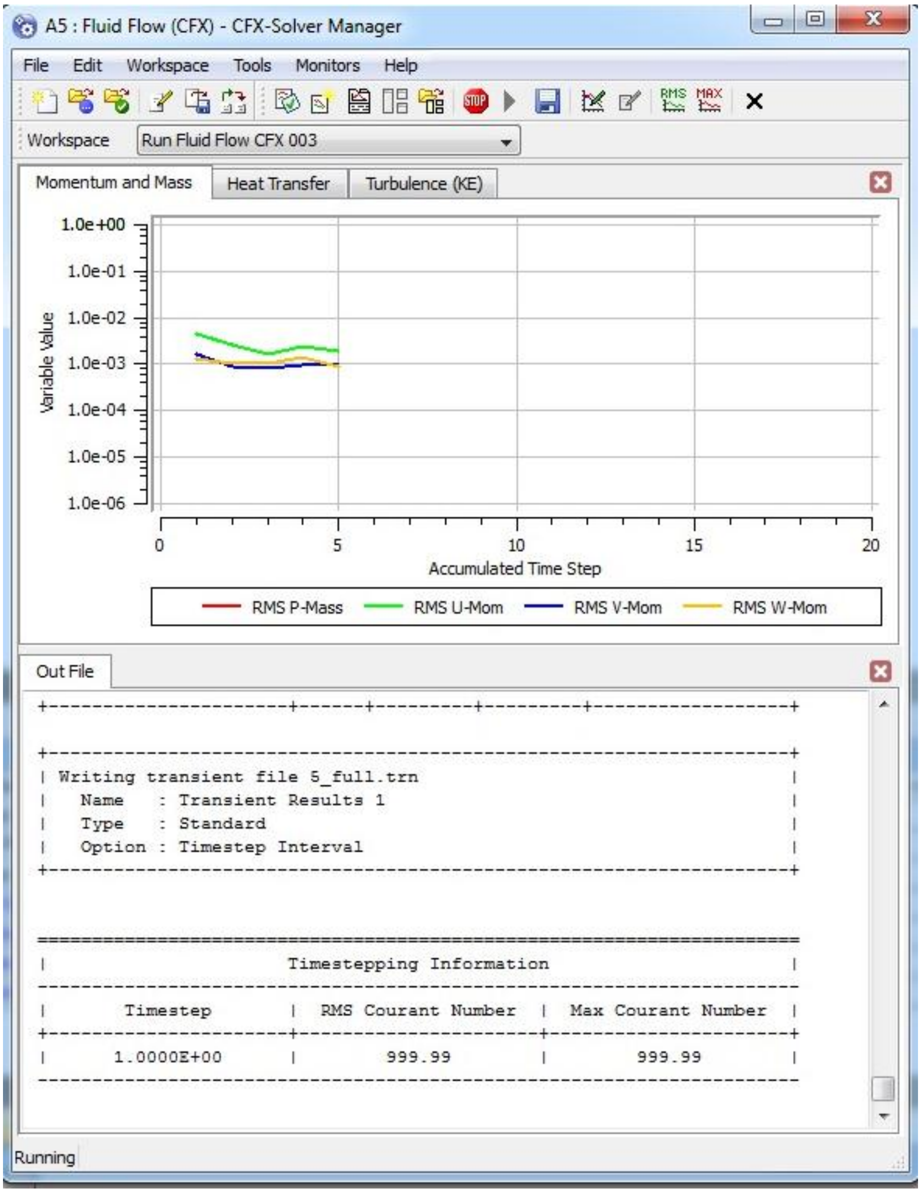
Step 66:

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.



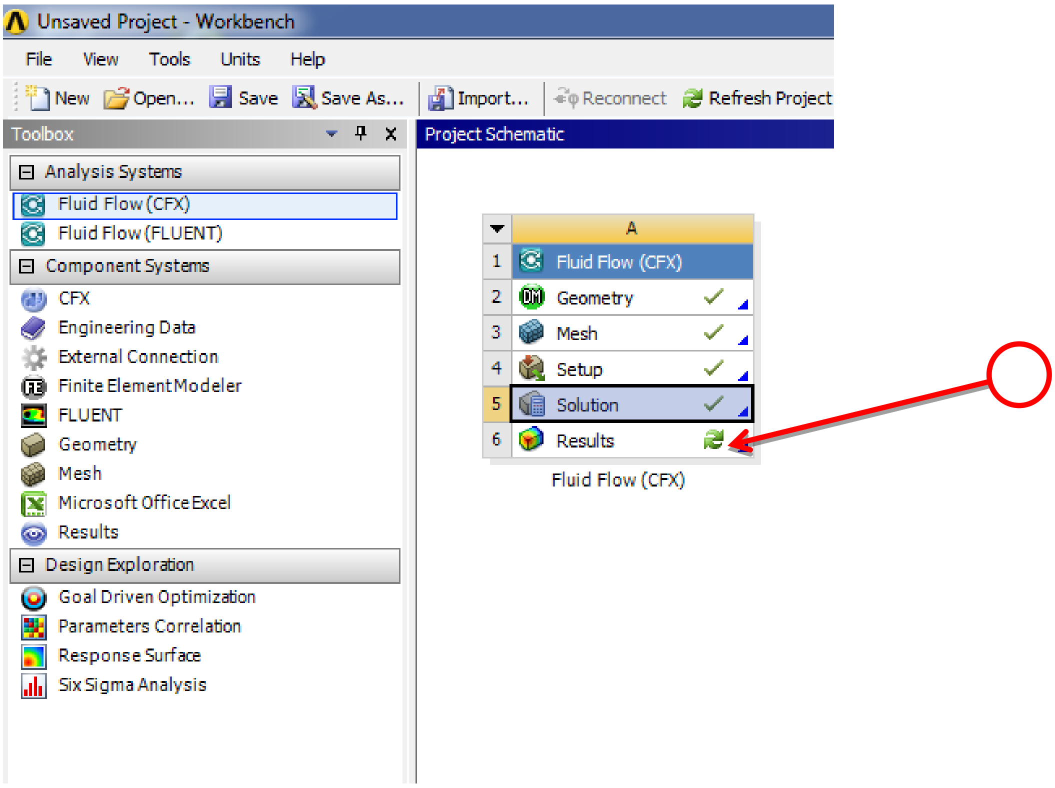
Step 67:

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.



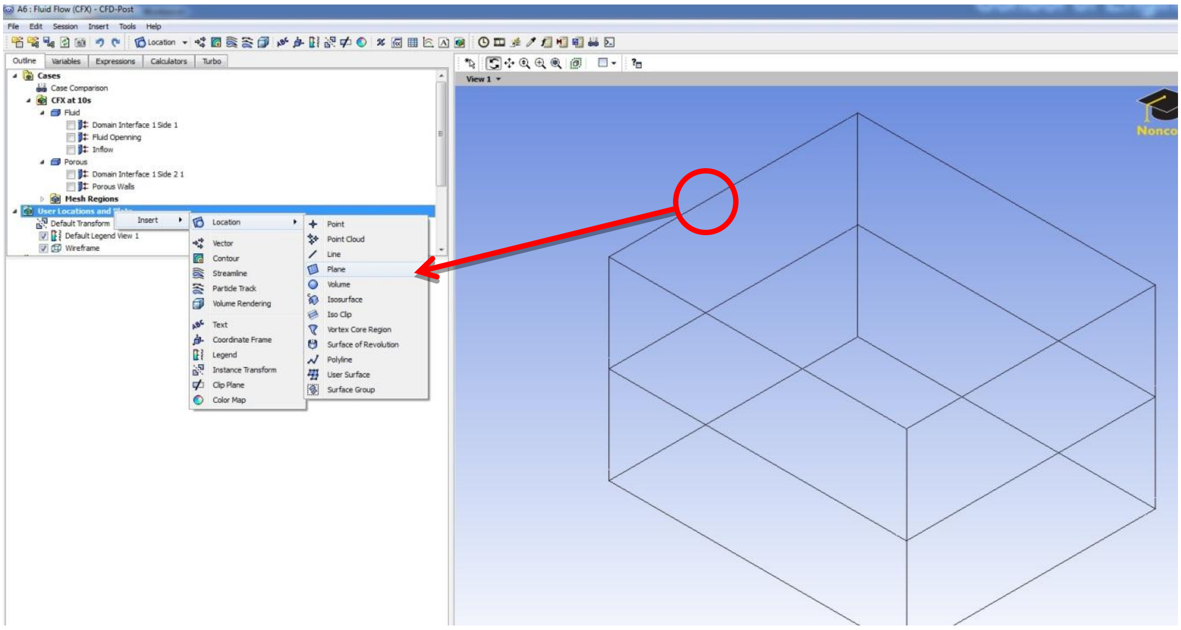
Step 68:

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



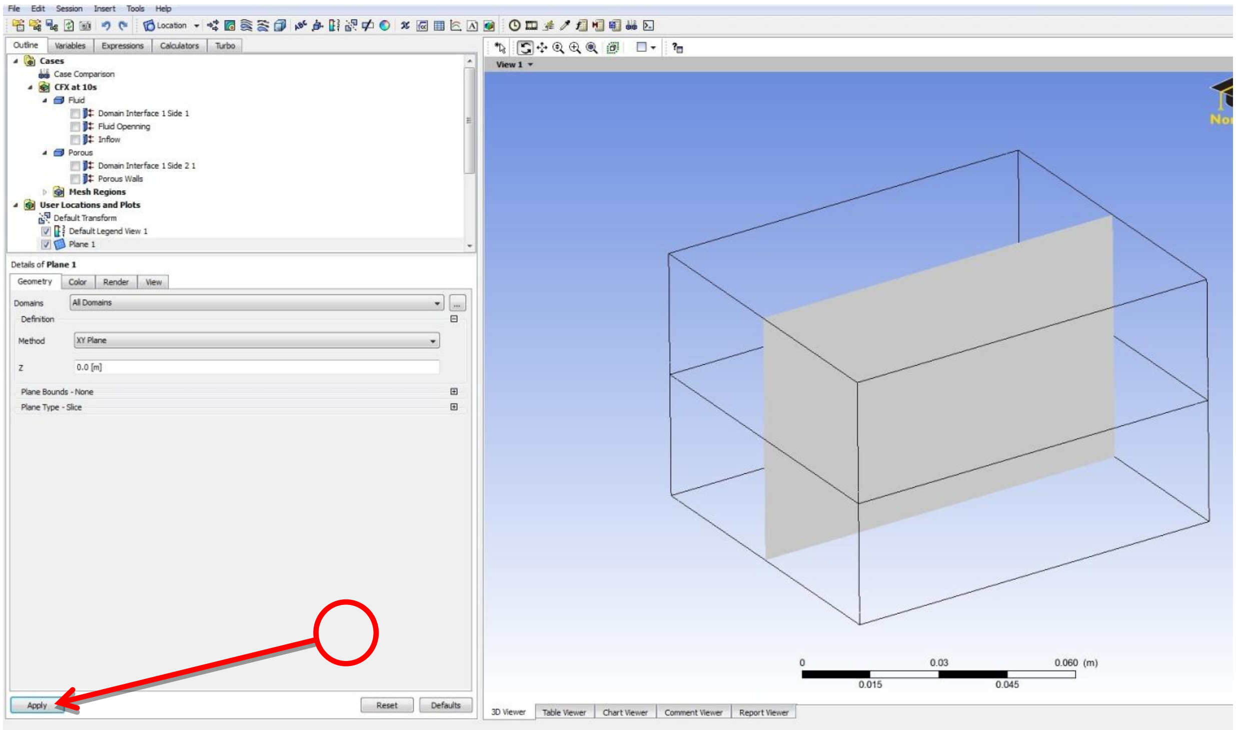
Step 69:

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.



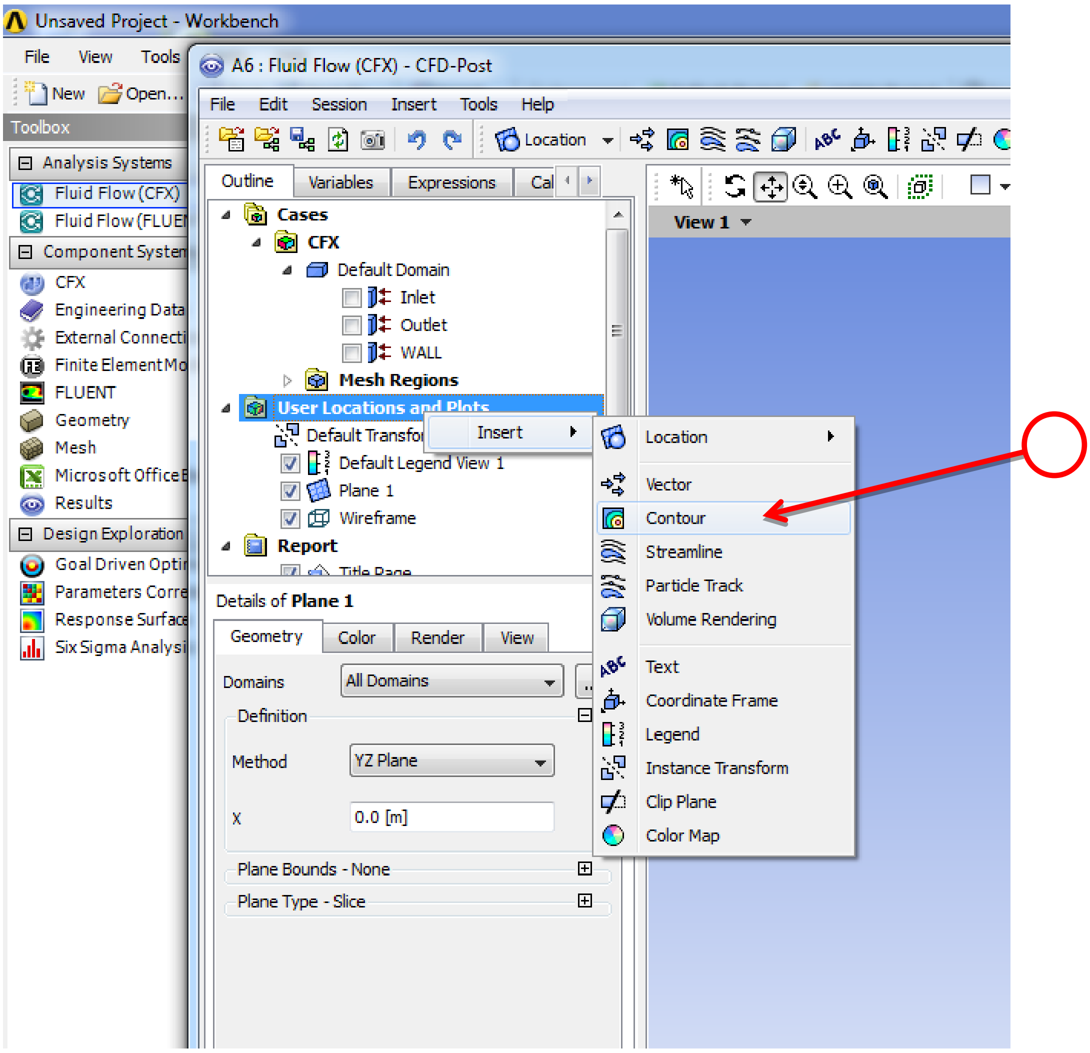
Step 70:

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



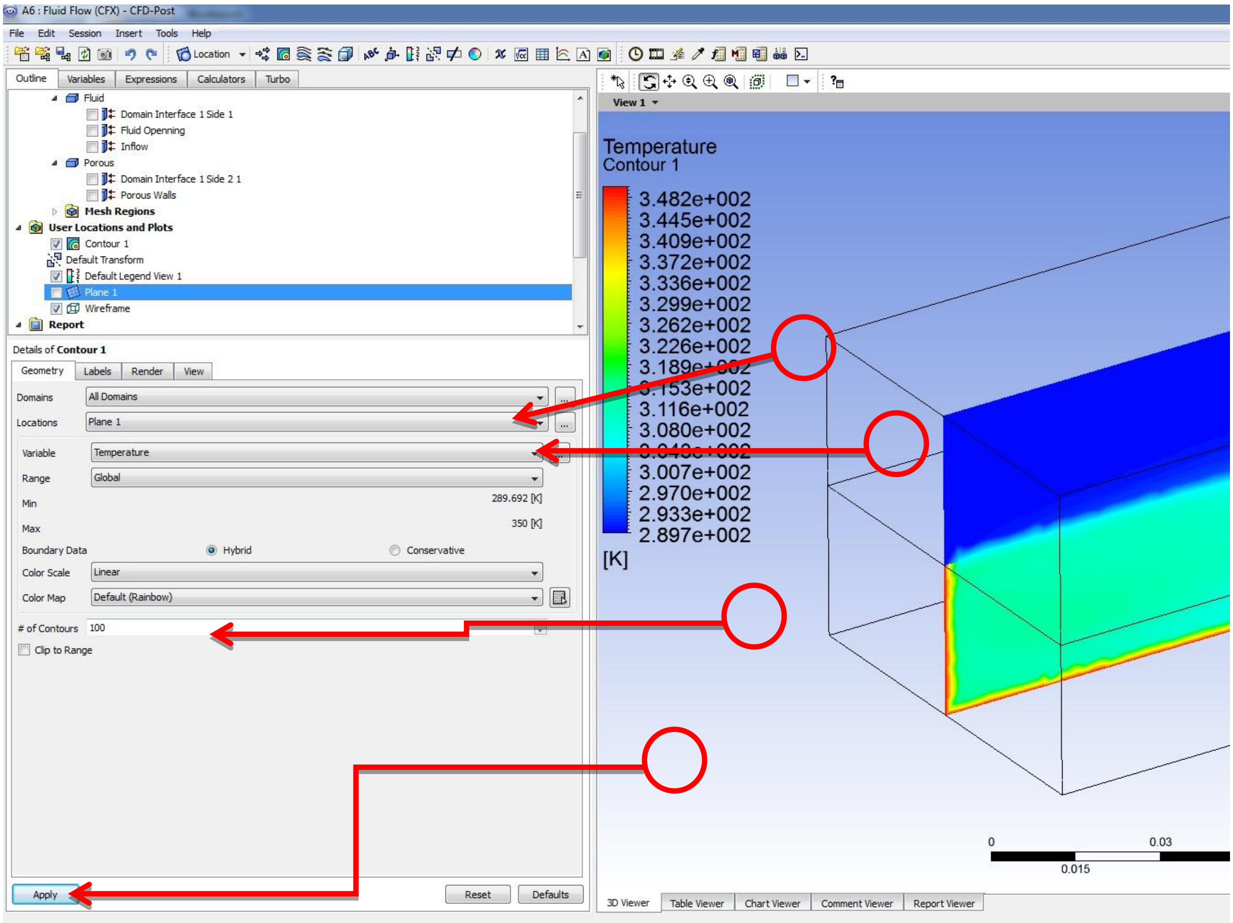
Step 71:

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



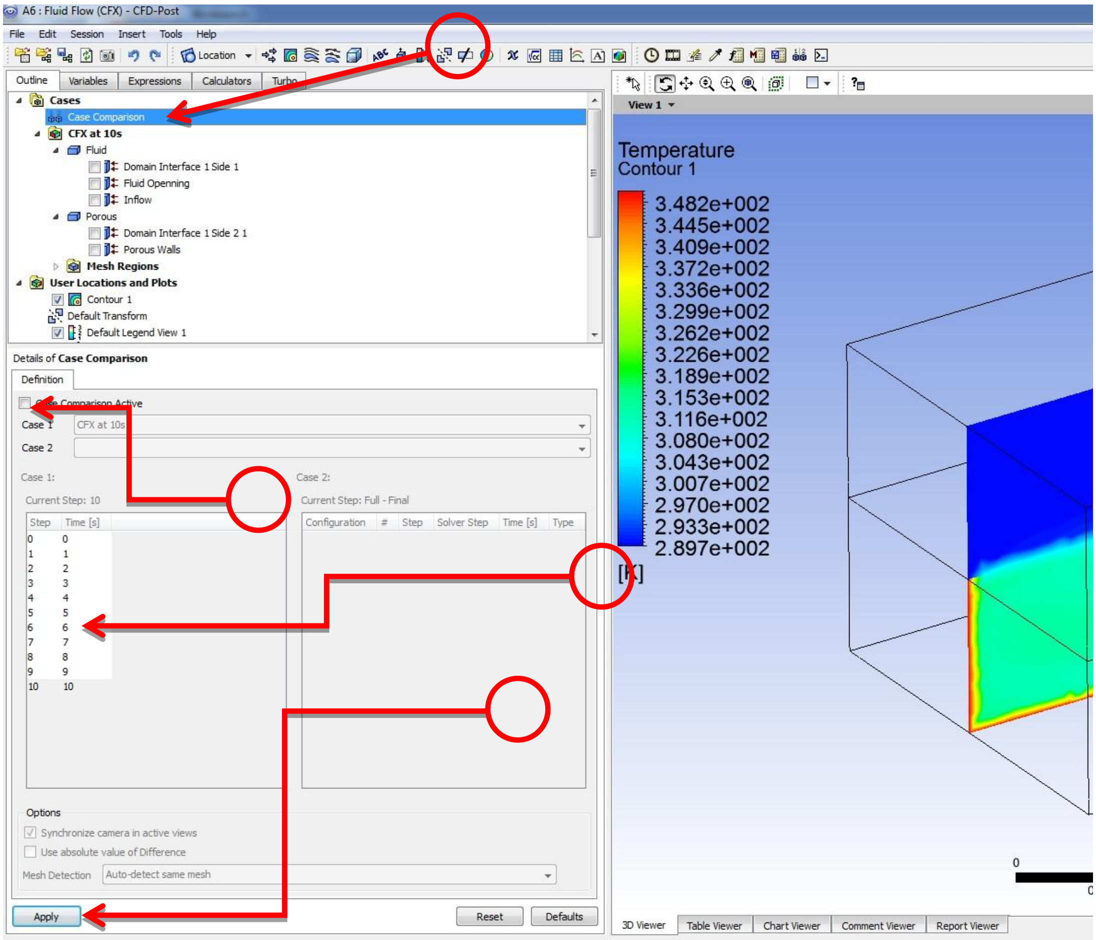
Step 72:

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. In the variable section select Temperature. Then go to (# of Contours) and enter into the input cell 100.



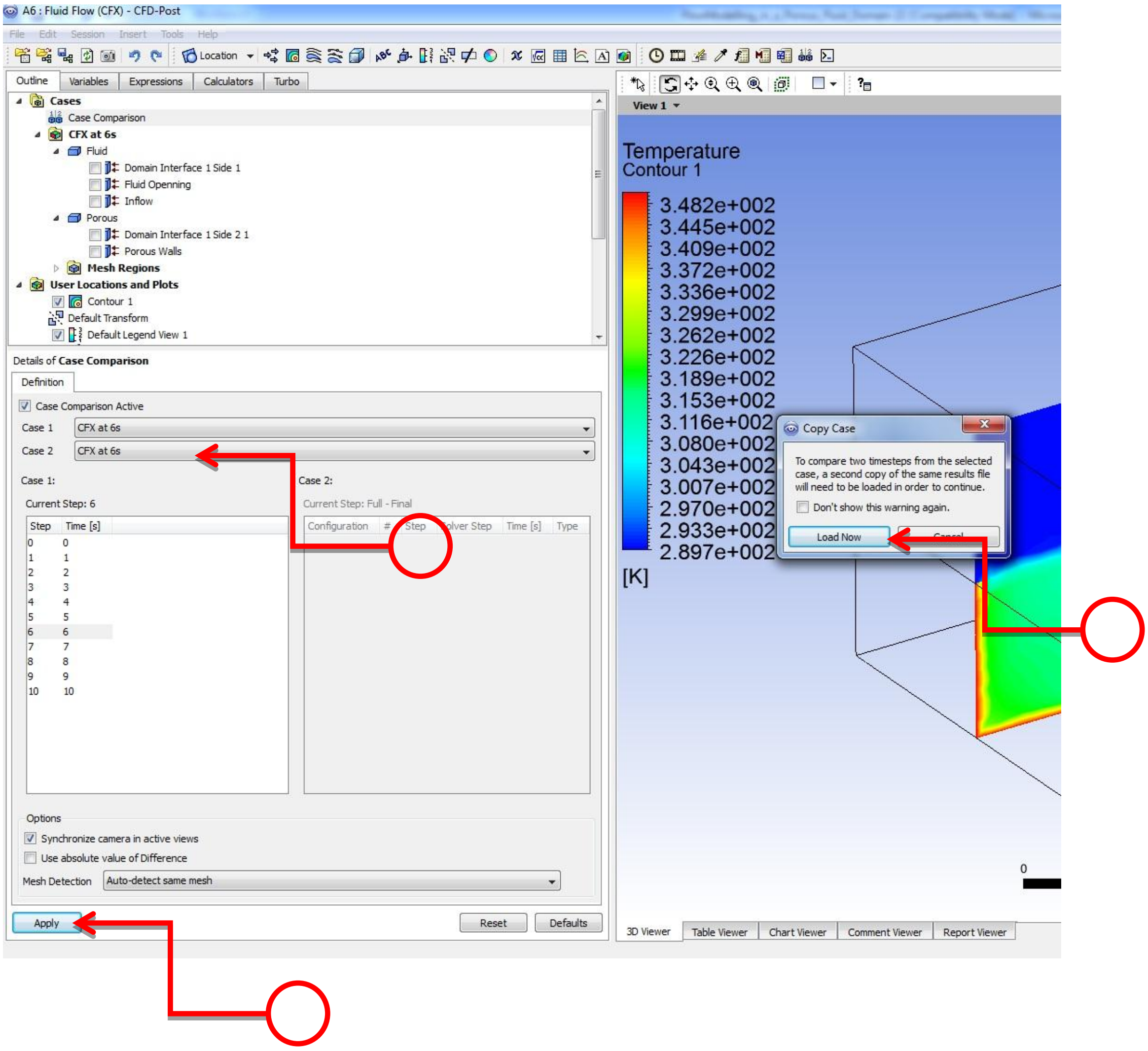
Step 73:

The next step is double click the case comparison, then click on the tick box of case comparative Active. Then double click on the time step numbered 6. You will notice that the comparative case gets deactivated.



Step 73:

Select time step 6. Then press Load now. Finally press apply.



Step 74:

The results shown represent time step number 6, time step number 10, and finally the subtraction between the two time steps which help in calculating the heat loss (gain).