Spray Modeling Tutorial using ANSYS-CFX

By Ahmed Al Makky

@Ahmed Al Makky 2012
All rights reserved. No part of this publication may be reproduced, stored in a retrieval system or transmitted in any form or by any means, electronic, mechanical or photo-copying, recording, or otherwise without the prior permission of the publisher.

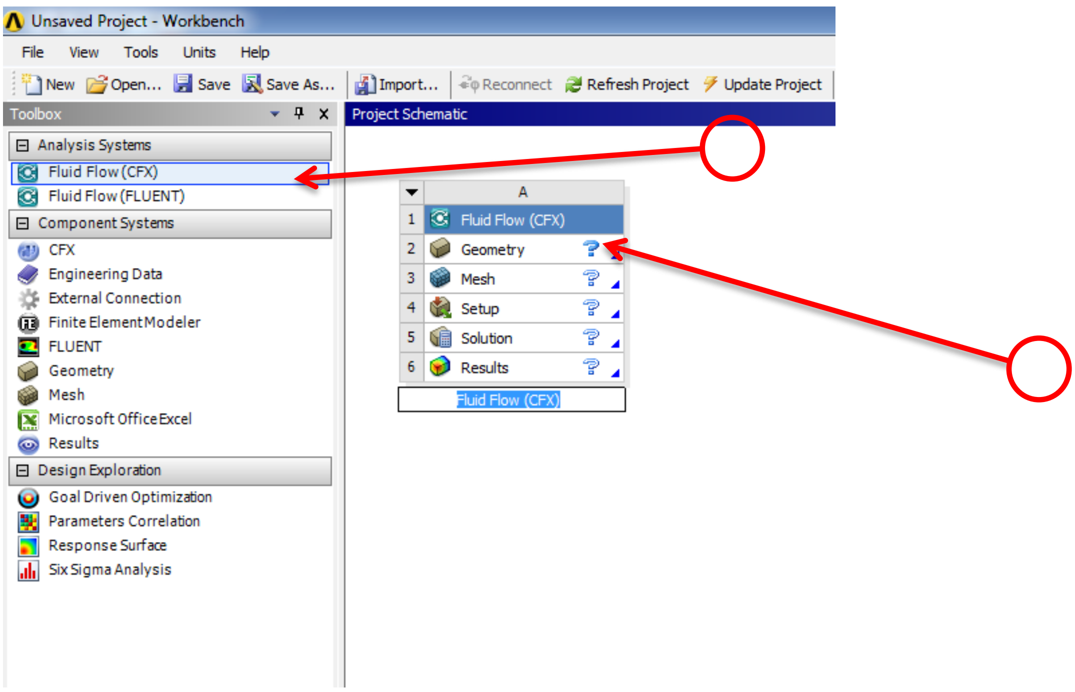
Spray Modelling using ANSYS-CFX

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 combustion simulation. Sprays are encountered in lots of engineering applications from internal combustion engines to gas turbines to air-conditioning systems to …. etc. This tutorial will show the power of multiphase flow simulation provided in ANSYS CFX.

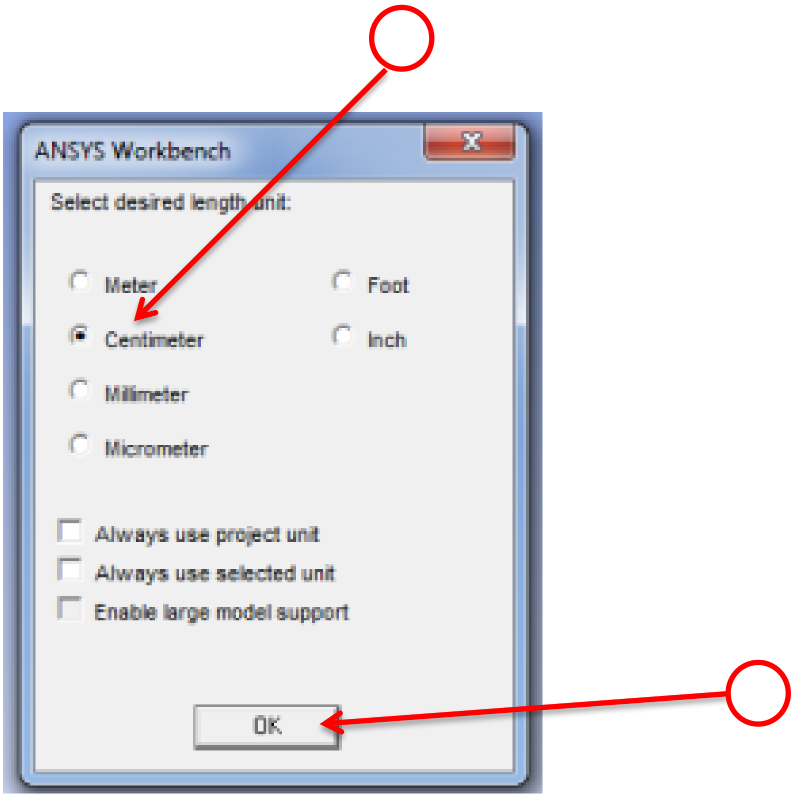
Step 1:

Under the Analysis tools double click Fluid Flow(CFX), then double click on the geometry icon.



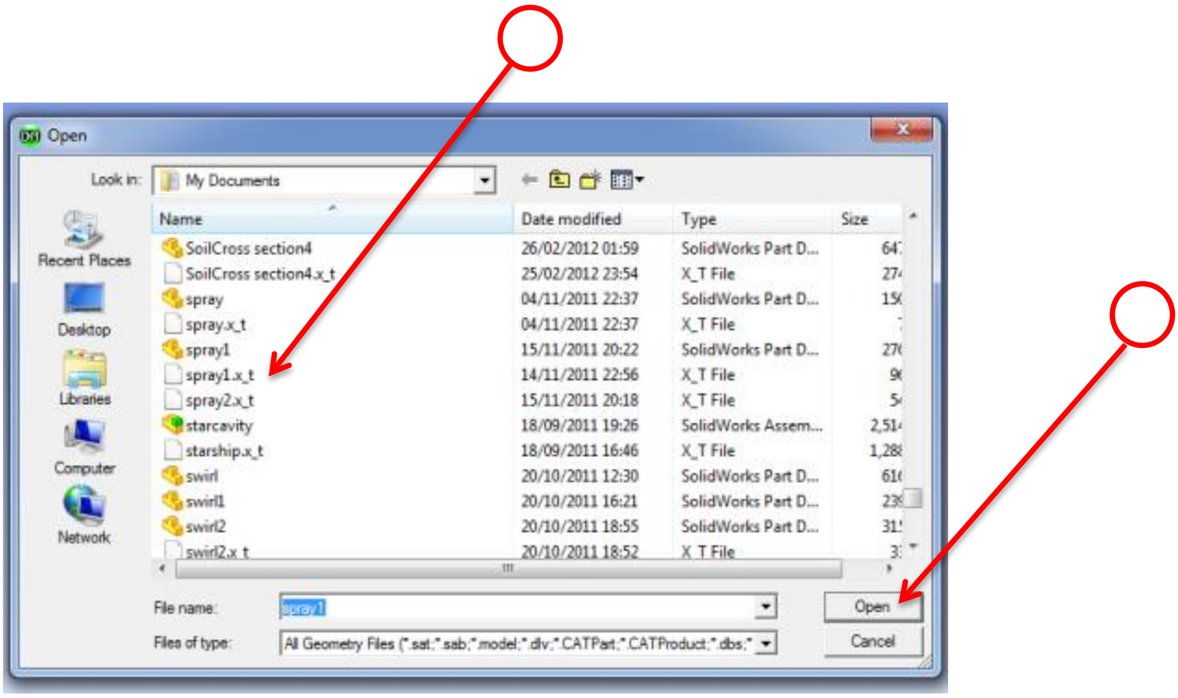
Step 2:

The next step is to select the scale of study the simulation is run at which is in centimetres and then press Ok.



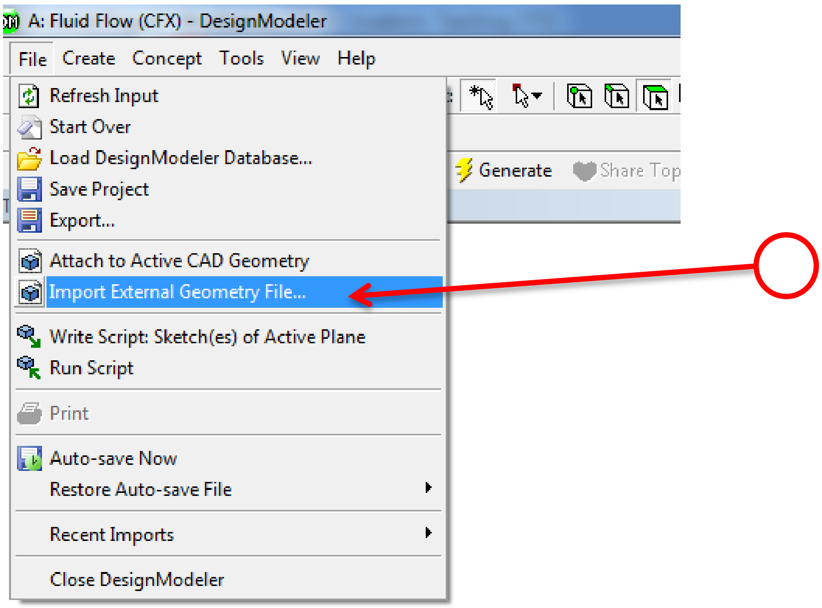
Step 3:

The next step is to select the ready modelled mesh that represnts a fuel injector used in two stroke engines. The geomtery is provided with this tutorial which has the name spray1.x_t and is in parasolid format.



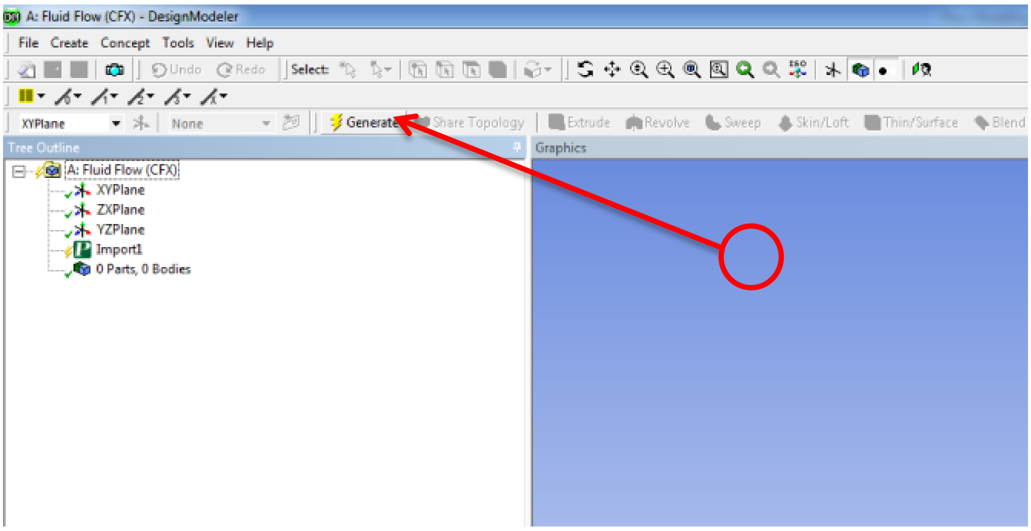
Step 4:

This step is to locate the required mesh to read into the designmodeller. This is done by going to file then from the drop down list selecting Import External Geometry file.



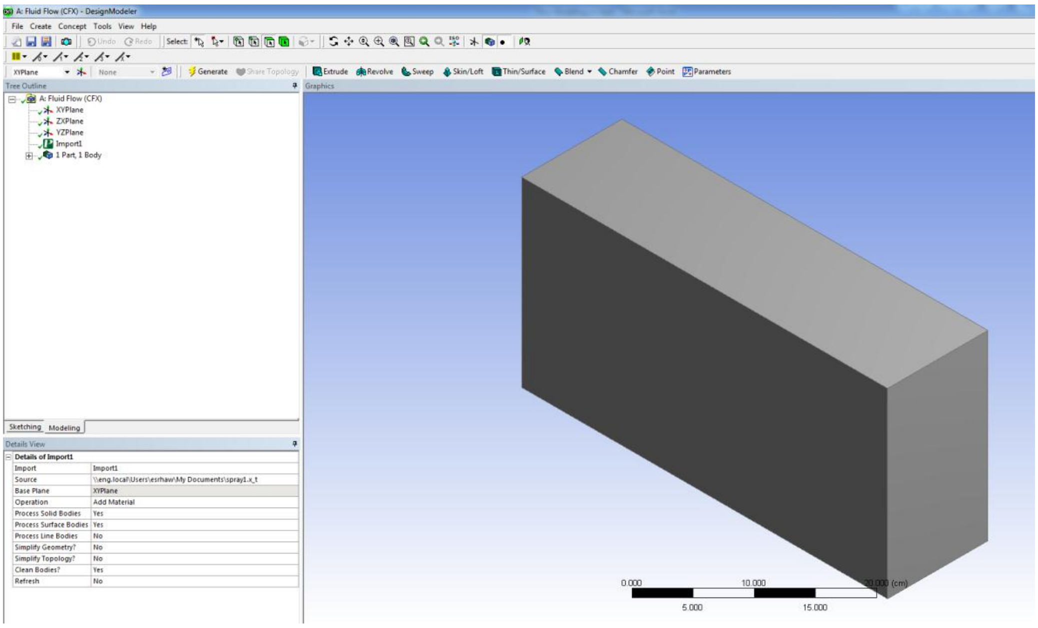
Step 5:

After the file has been selected press generate. This will read the previsouly modled geometry into Design Modeler.



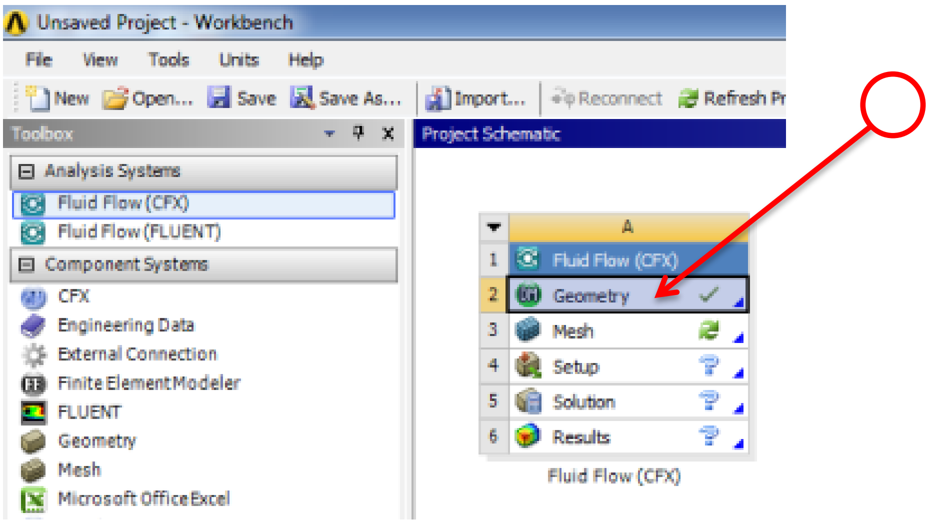
Step 6:

The imported geometery box should look something like this.



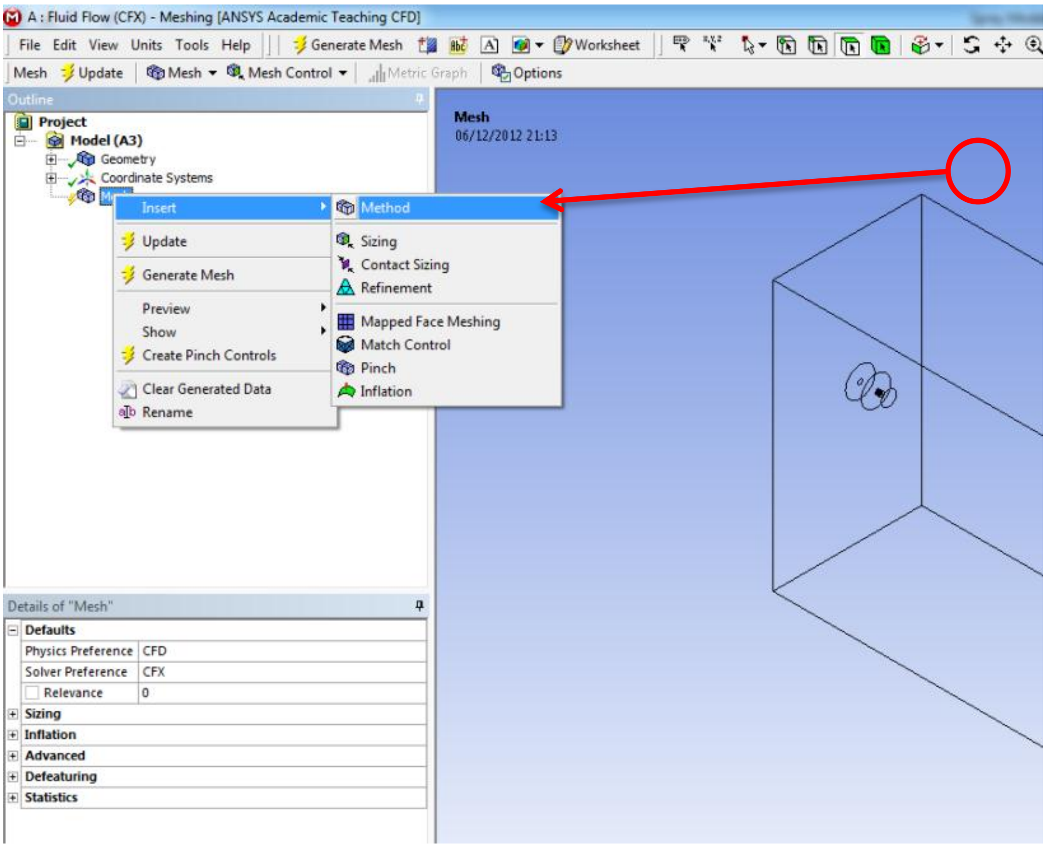
Step 6:

Check that there is a green tick beside the Geometry icon. Then double click the mesh icon.



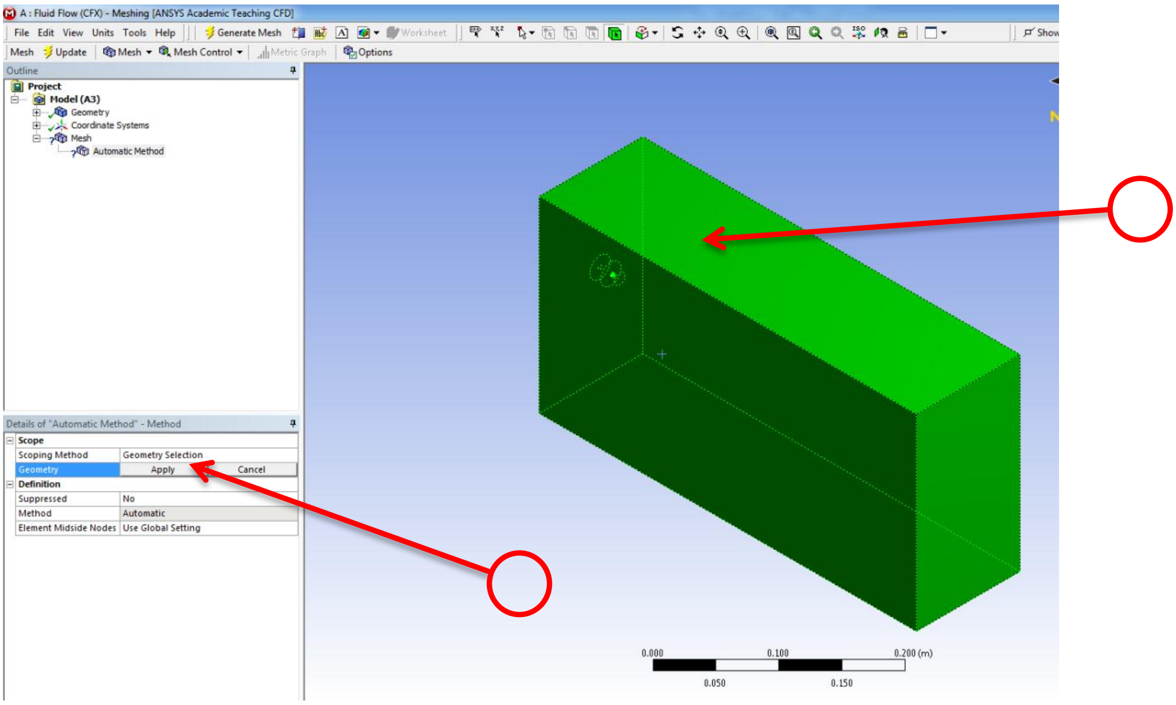
Step 7:

Postion the cursor on the mesh icon and then press the left mouse button and then press the right mouse button first select insert and then select Method.



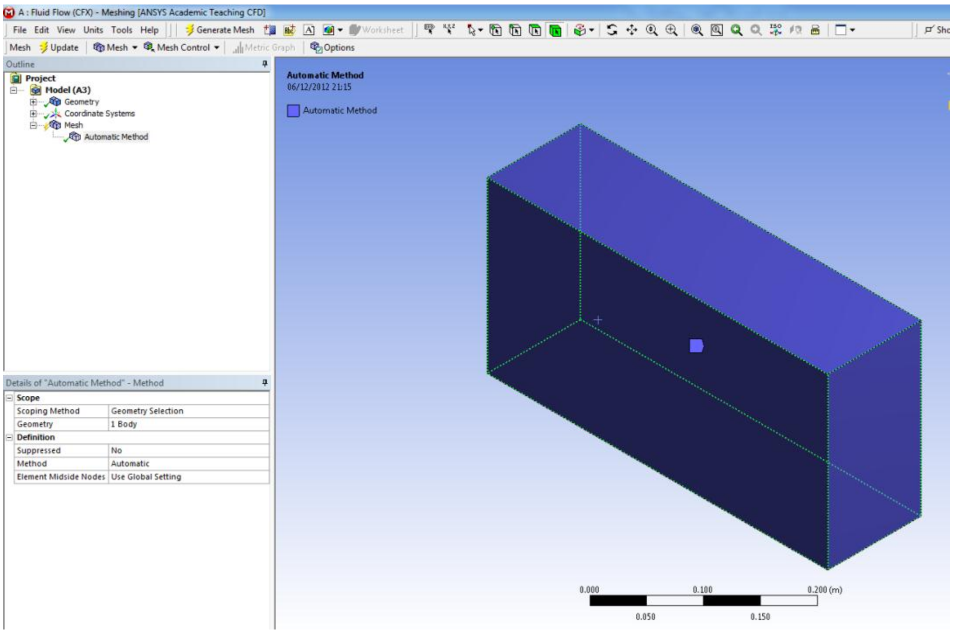
Step 8:

When you click on the geometry box it will turn green the next step is to press the apply box.



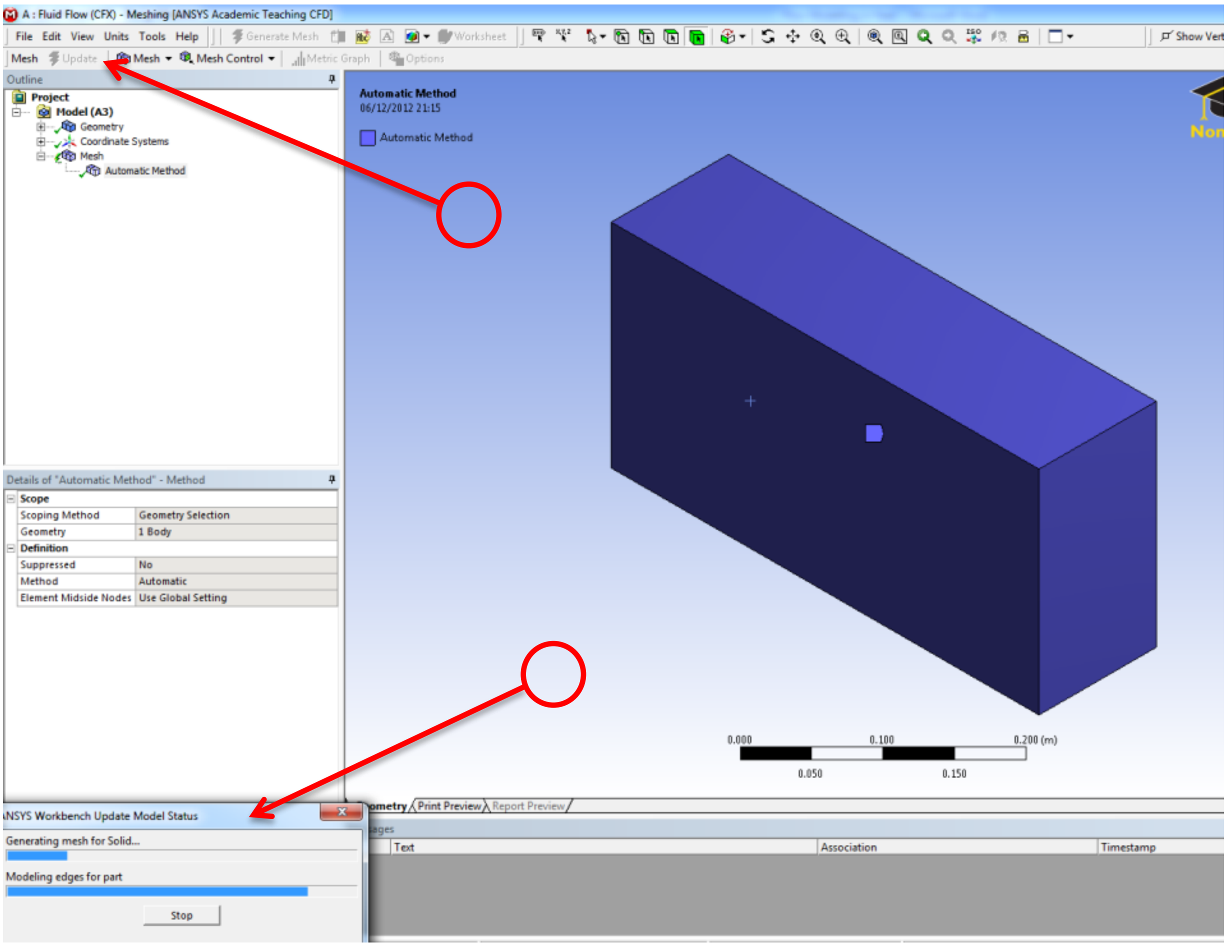
Step 9:

After the apply button had been pressed the box should turn into a blueish purpule colour.



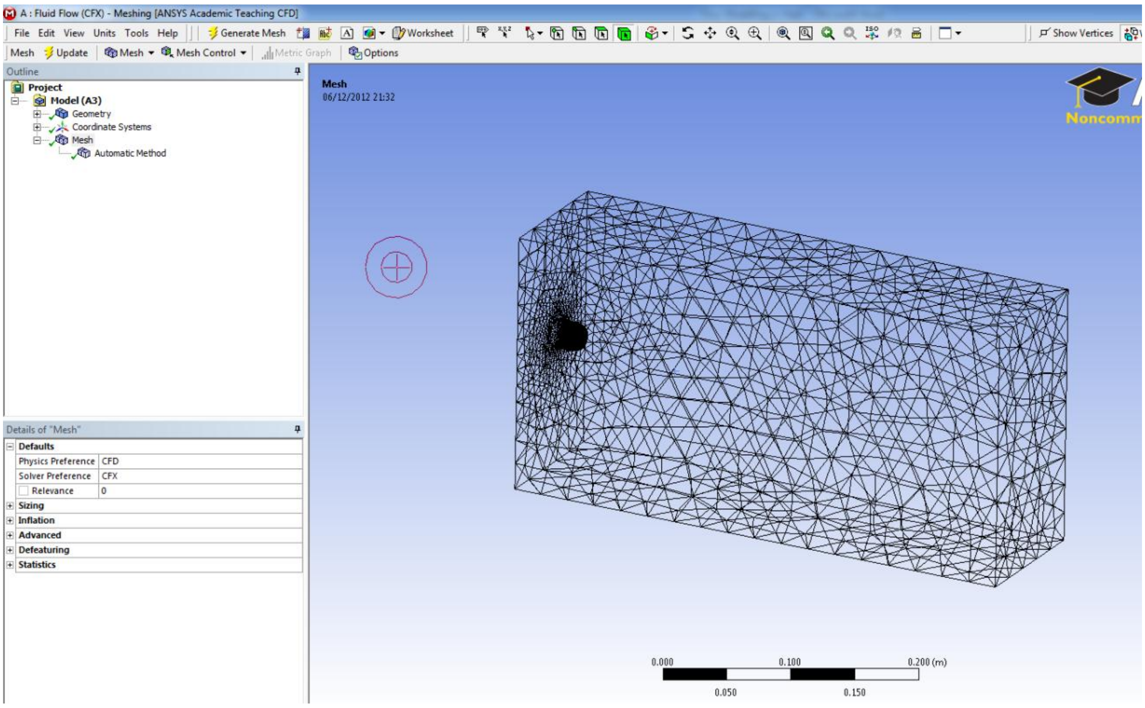
Step 10:

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.



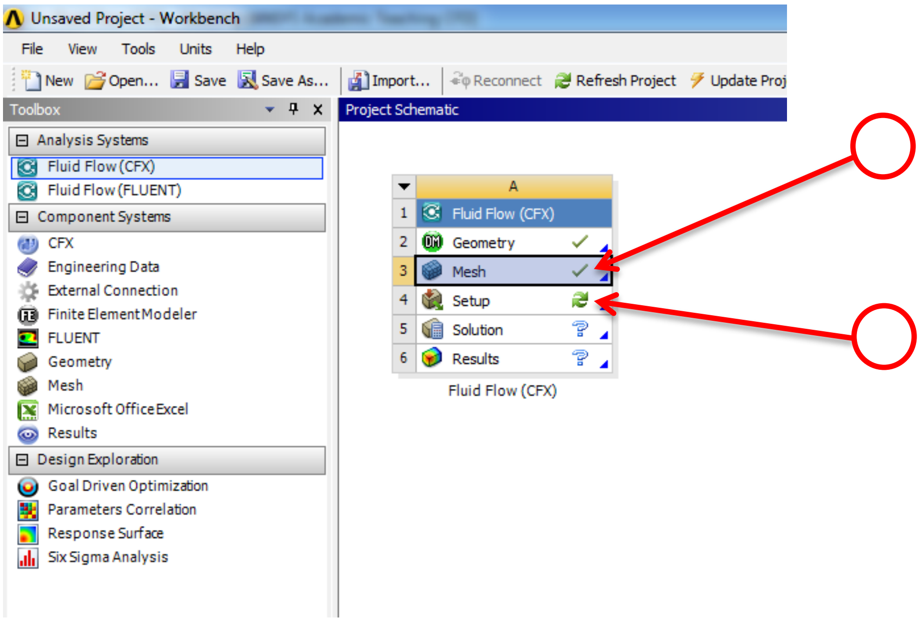
Step 11:

The generated mesh should look something like this. This method is the automatic method.



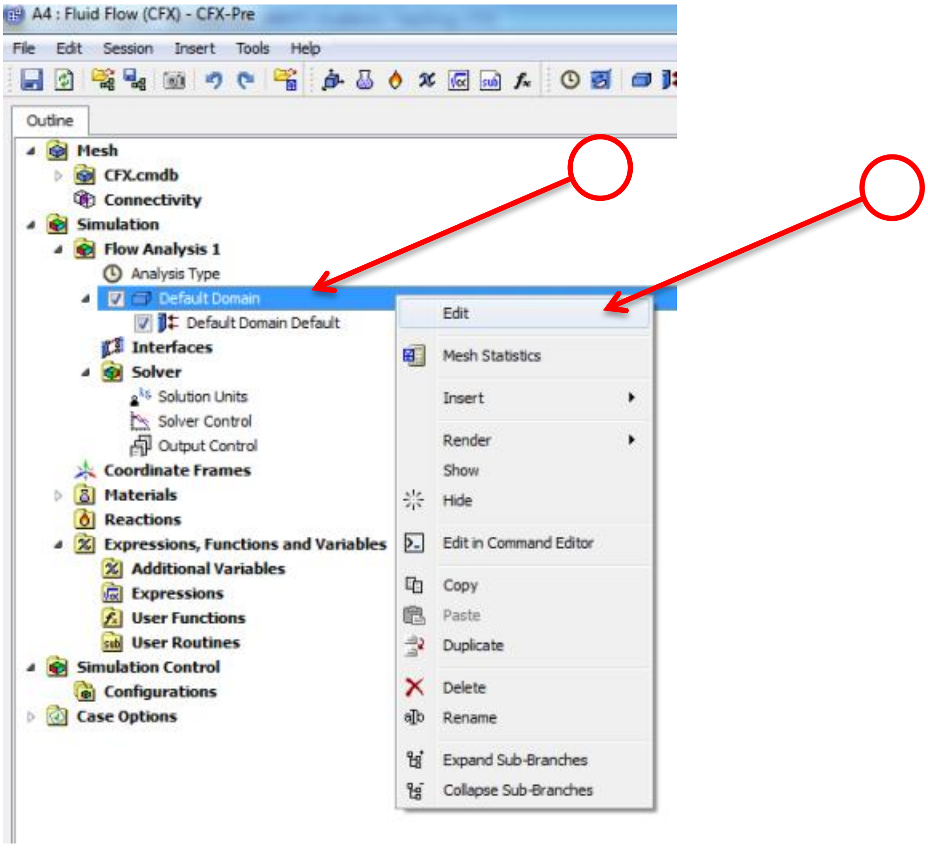
Step 12:

Check that the a green tick is shown beside the mesh icon. The next step is to double click on the setup icon.



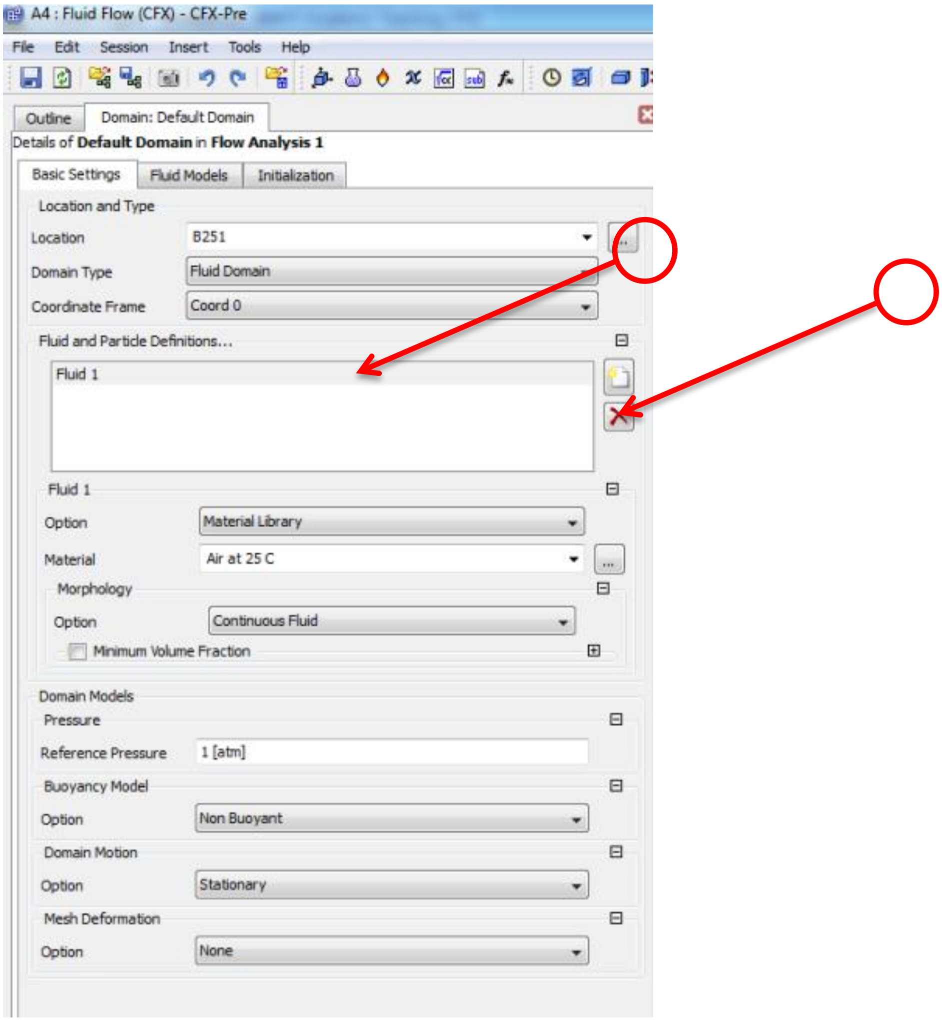
Step 13:

Position the mouse curser on the default domain icon then press the mouse left handside button then press the mouse righthand side button and select edit.



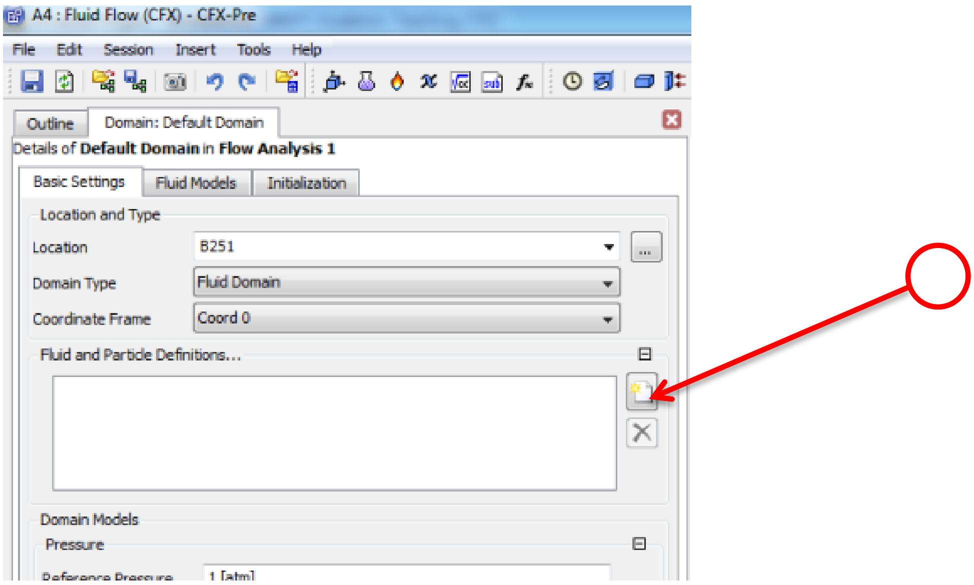
Step 14:

You would notice that by default under the Fluid and Particle Definitions the name Fluid 1, just select it and press on the red X box, this will remove the default material name.



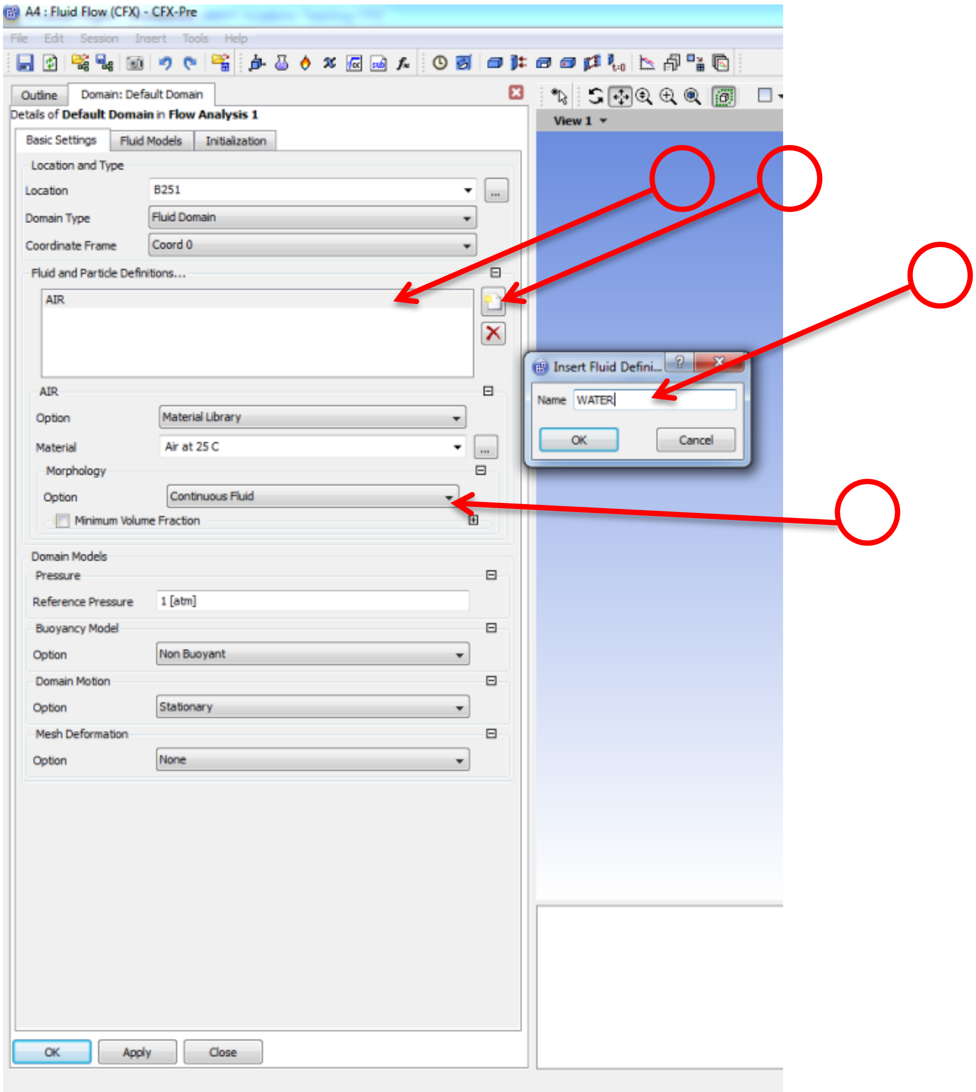
Step 15:

Press on the file star icon pointed at by the arrow. This will enable us to add a new material name into the list. A window will open asking you to enter a name of a material, write AIR and press Ok.



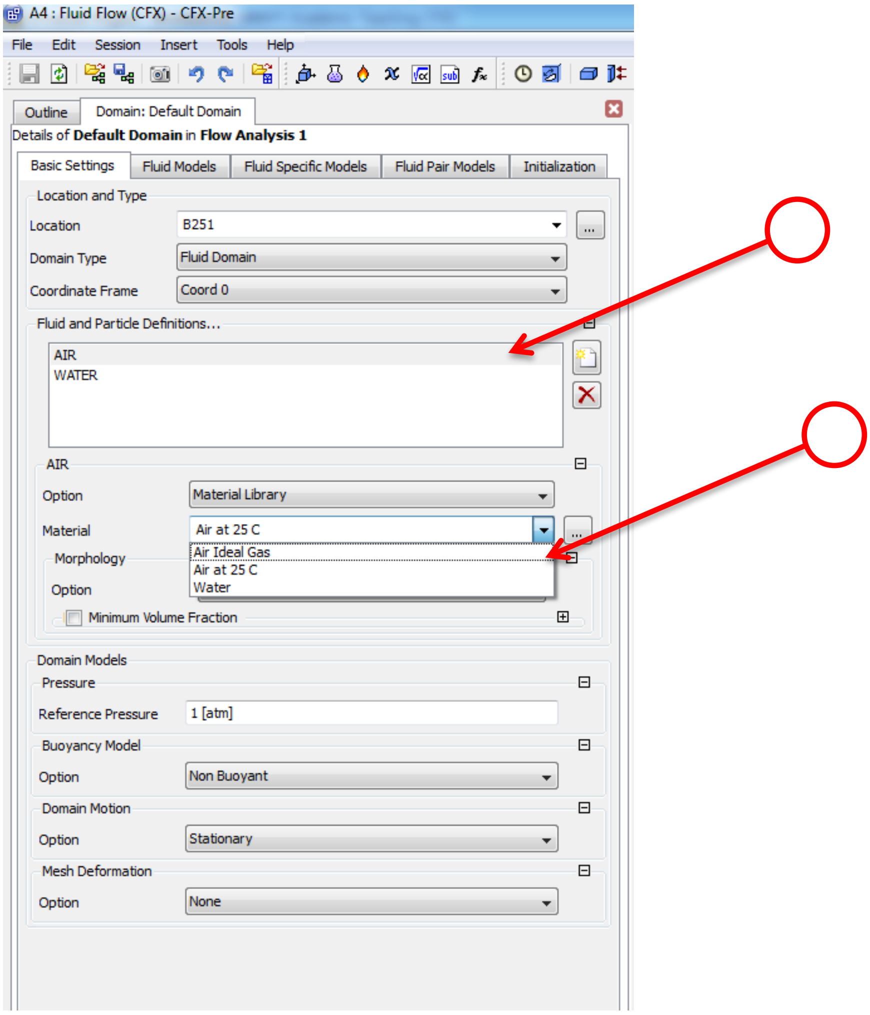
Step 16:

Then again by pressing on the star file icon add another material called WATER and press Ok. Make sure under Morphology that the continous Fluid is selected this is for the case of AIR.



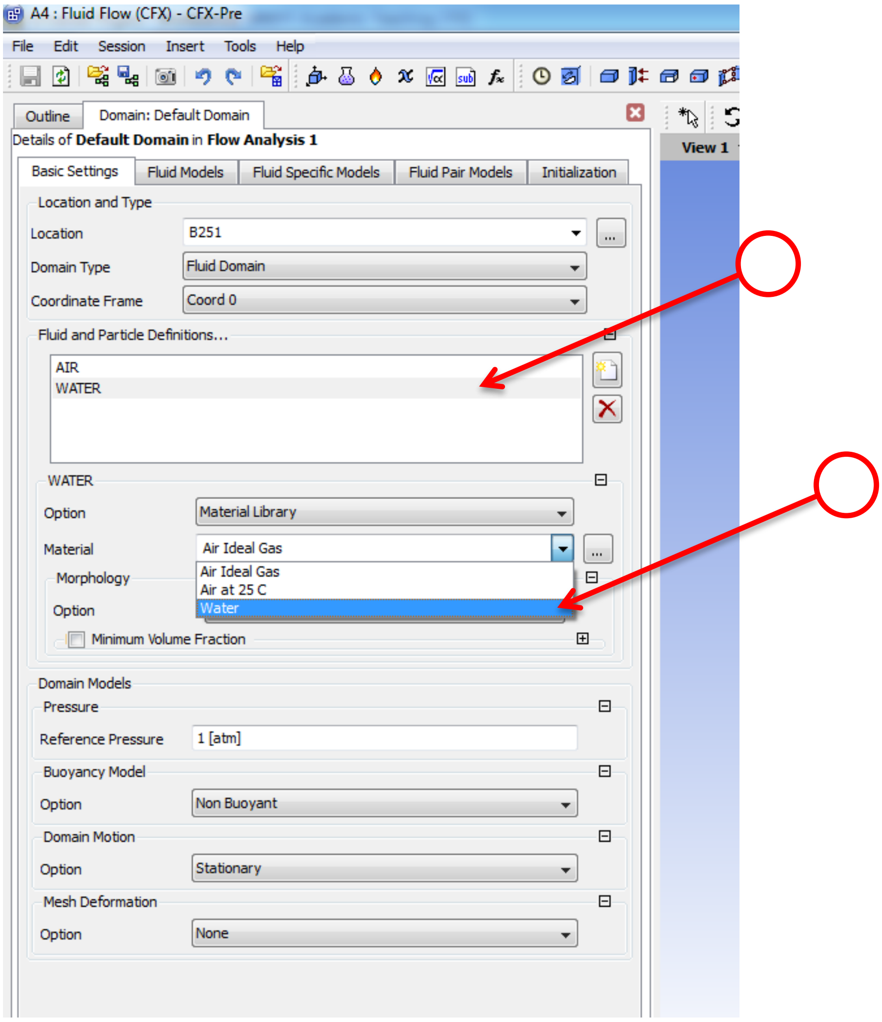
Step 17:

In the fluid and partical definitions select AIR then select from the drop down list in the material section Air at 25C. This is important becouse this step links the input name by the user to the thermodyanmics parameters data base in the software library.



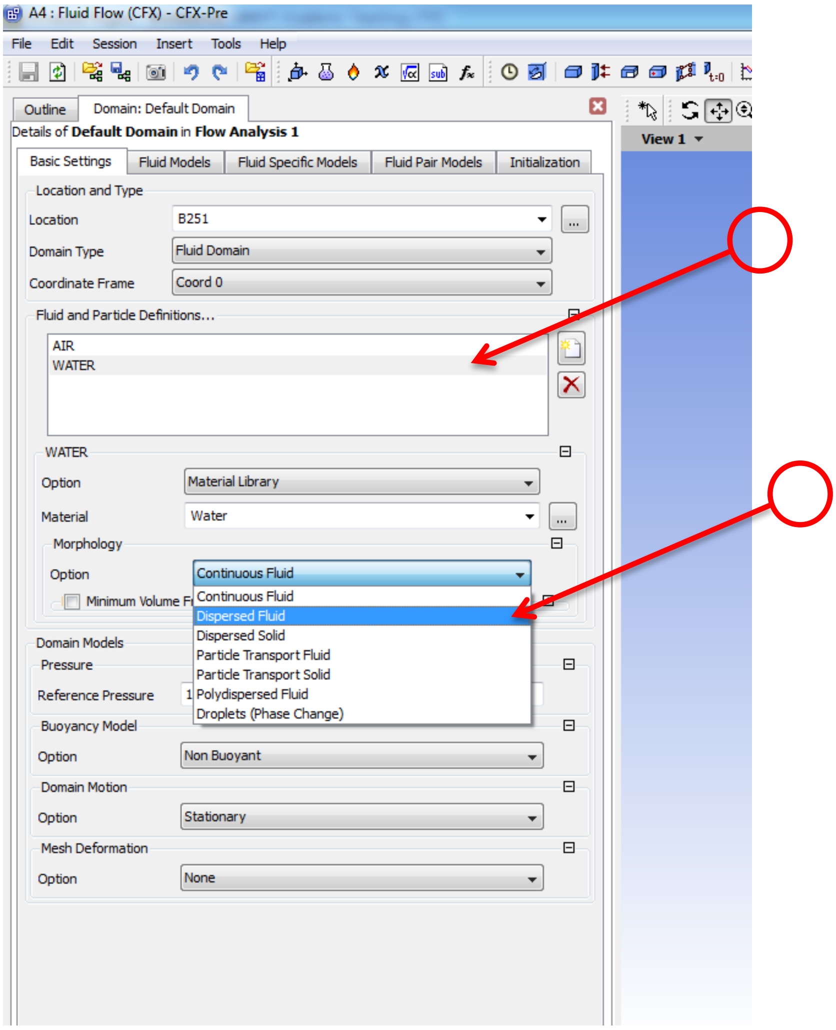
Step 18:

The next step is to select in the fluid and particle definition WATER then from the material drop down list select Water again this is important becouse this step links the input name by the user to the thermodyanmics parameters data base in the software library.



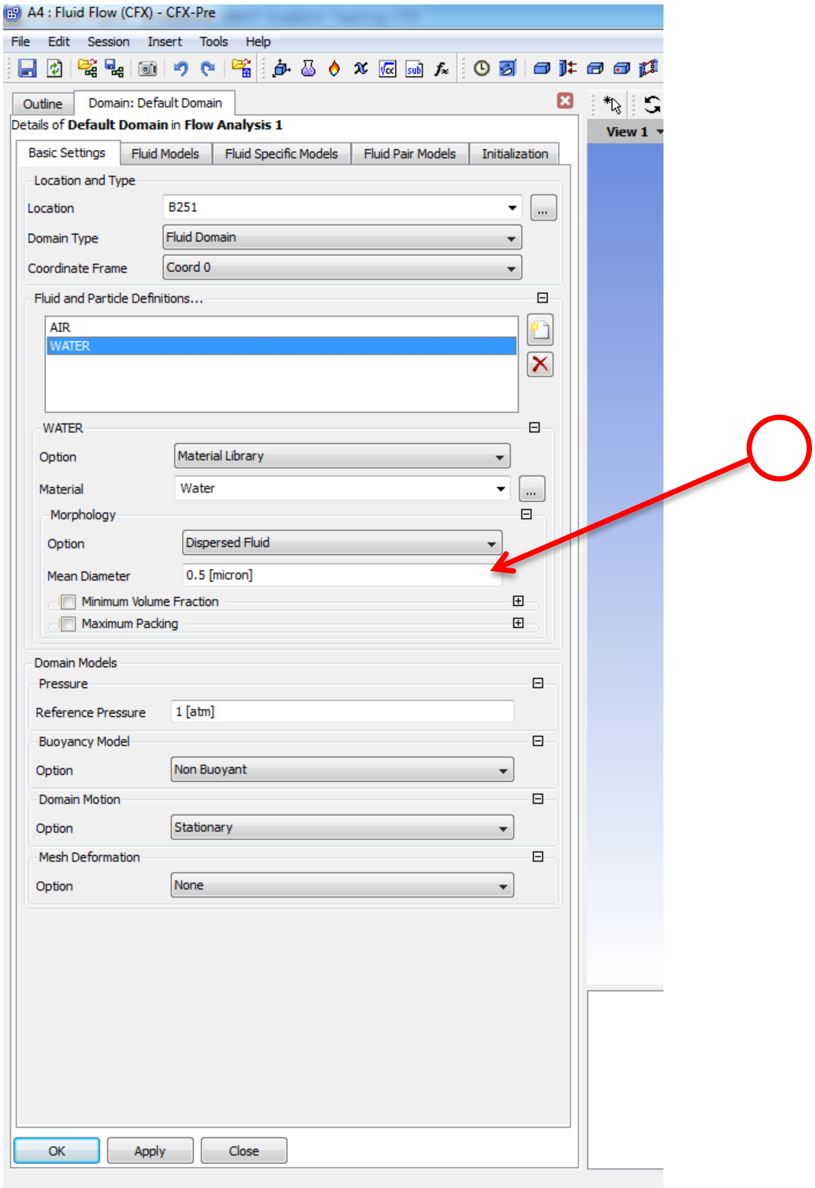
Step 19:

Having selected WATER under the Fluid and Partical Definition go to Morphology and select from the drop down list a dispersed fluid.



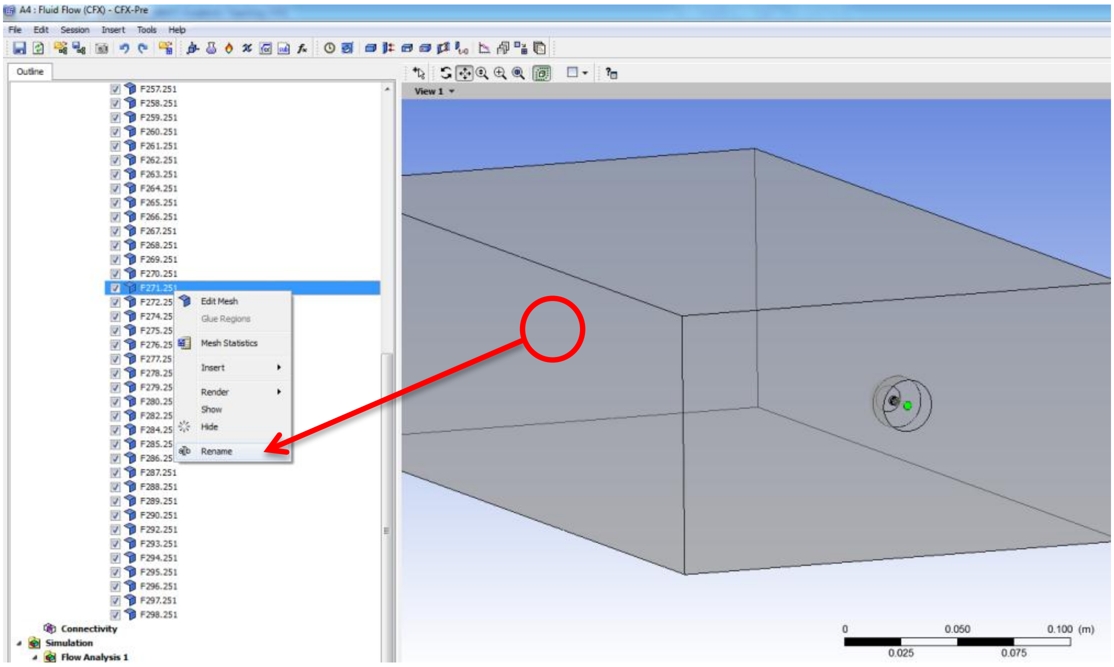
Step 20:

Once dispersed fluid is selected the user is asked input a fluid partical mean diameter. Apply the mean droplet diameter of 0.5 micron. Then press Ok.



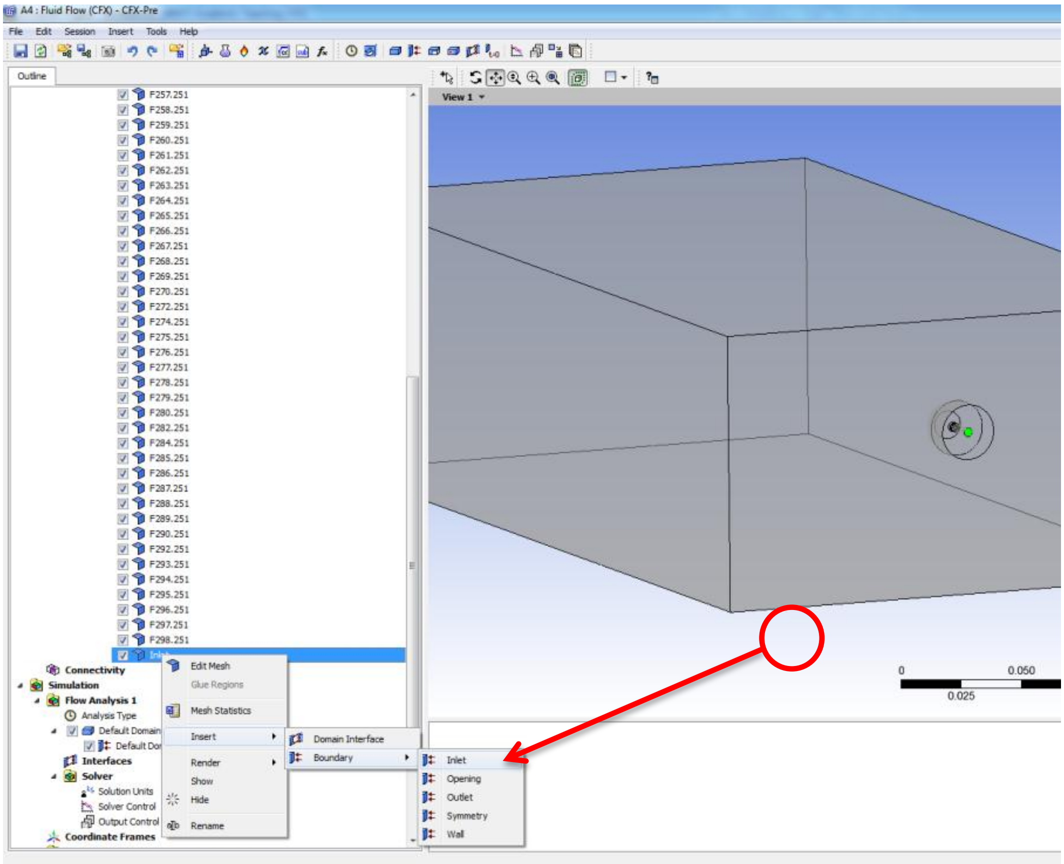
Step 21:

Go to the CFX.cmdb and from the drop down list for the Principial 2D Regions select the region numbered F211.251. Then right click and select the Rename option, this action will help us in applying a name to the inflow face.



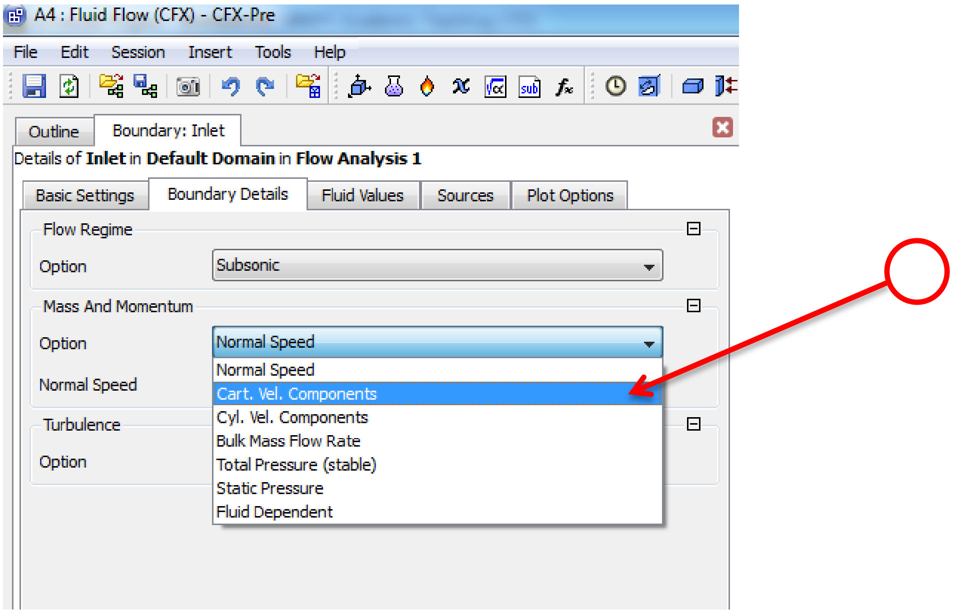
Step 22:

After renaming the surface right click on the renamed region, go to Insert , then go to boundary and then under boundary select inlet.



Step 23:

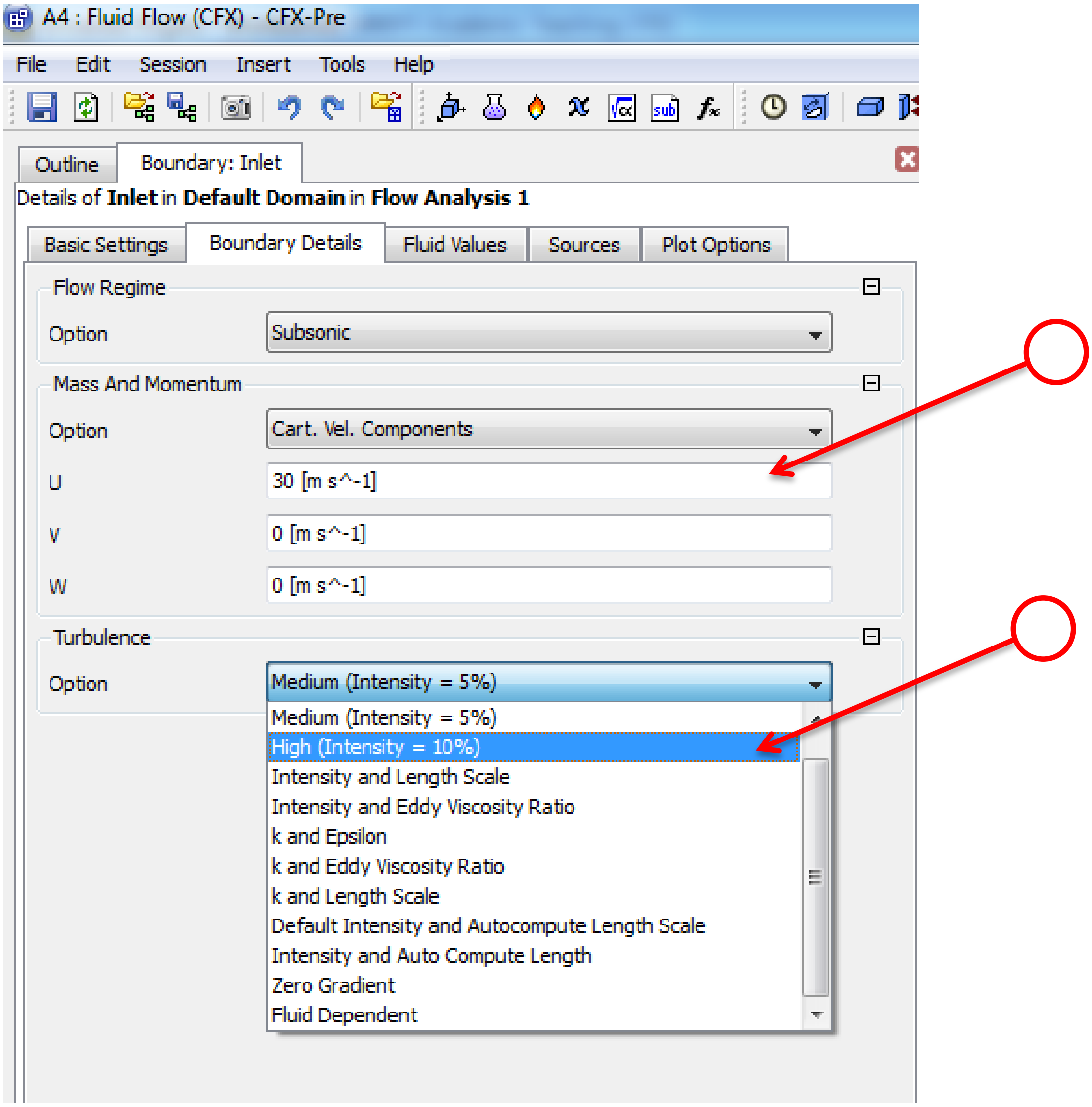
Once the inlet option is selected go to boundary details and under Mass and Momentum select the Cart. Vel. Components. In the main view look at the coordiante system shown on the bottom right hand side this will clarfiy that the directions of the selected velocities is correct.



Step 24:

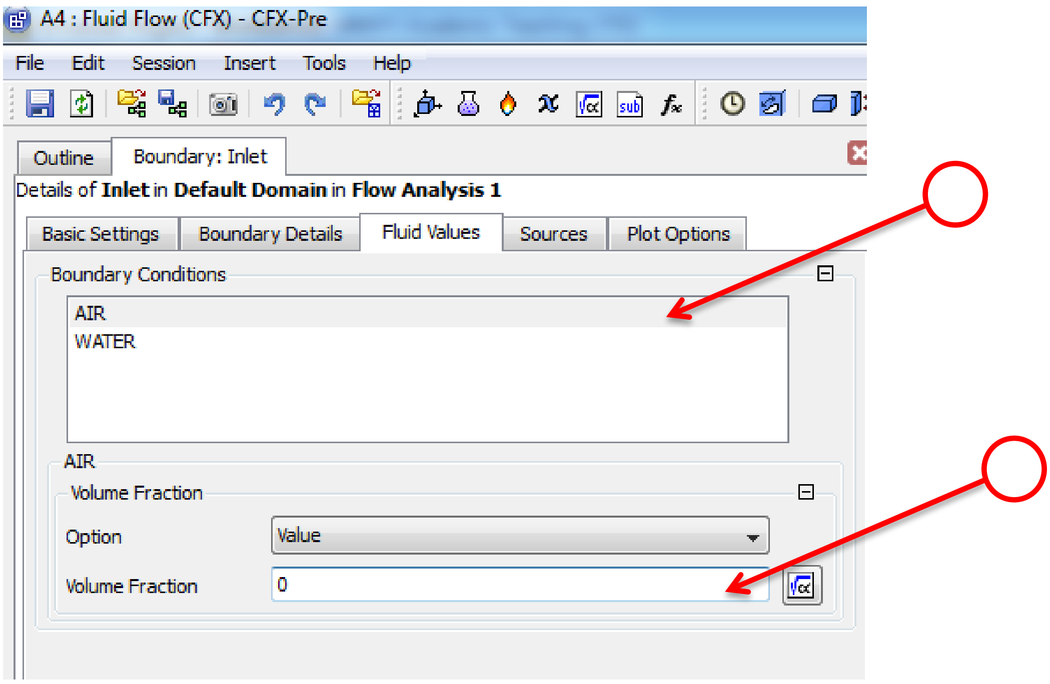
Type the value of 30 into the U velocity input box, while for the V and W cell type in zero.

Under the Turbulnce option select High (Intensity=10%).



Step 25:

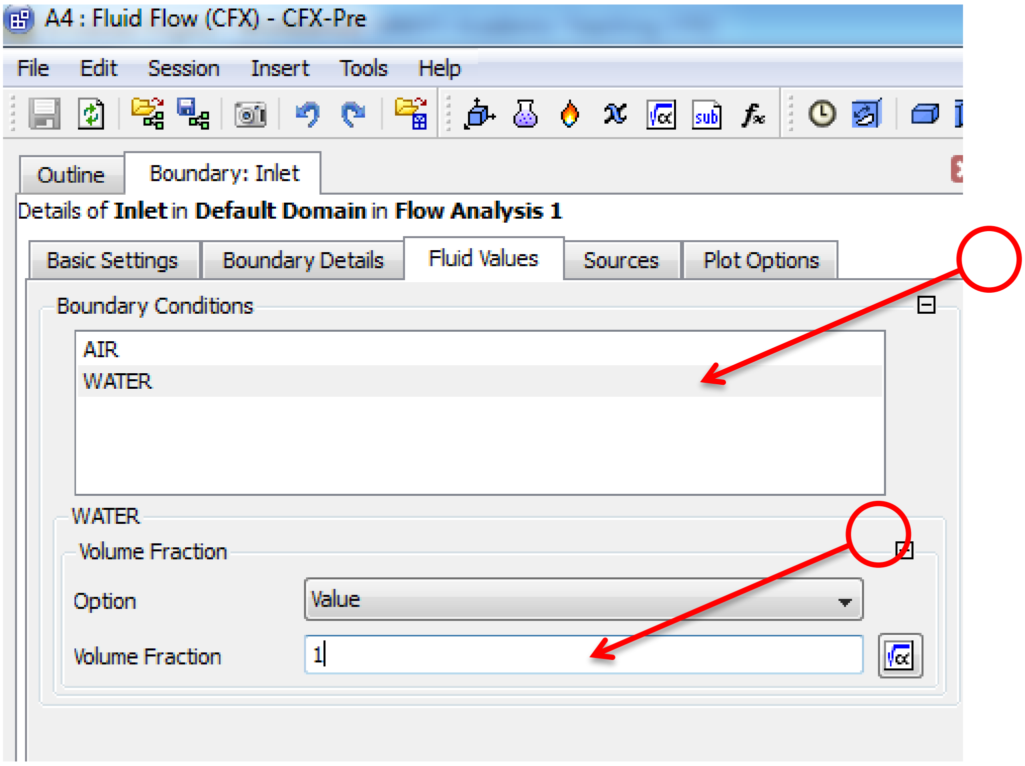
In the boundary conditions section go to Fluid Values and select AIR then in the volume fraction section input zero as.



Step 26:

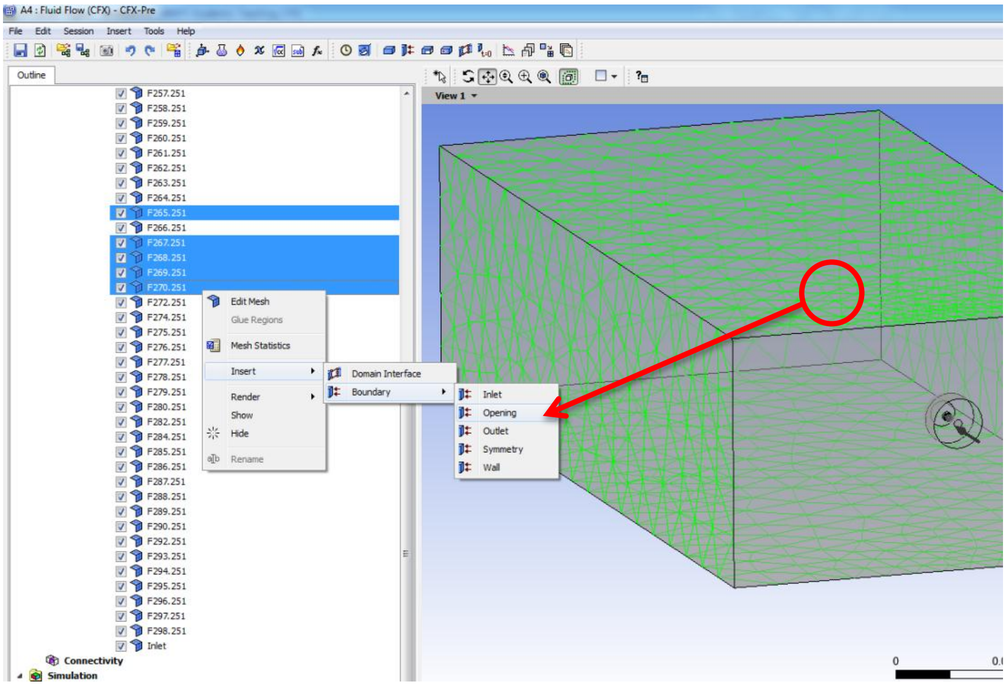
The next step is to go again to Fluid Values and select WATER under boundary conditions.

Then apply 1 as the volume fraction for WATER.



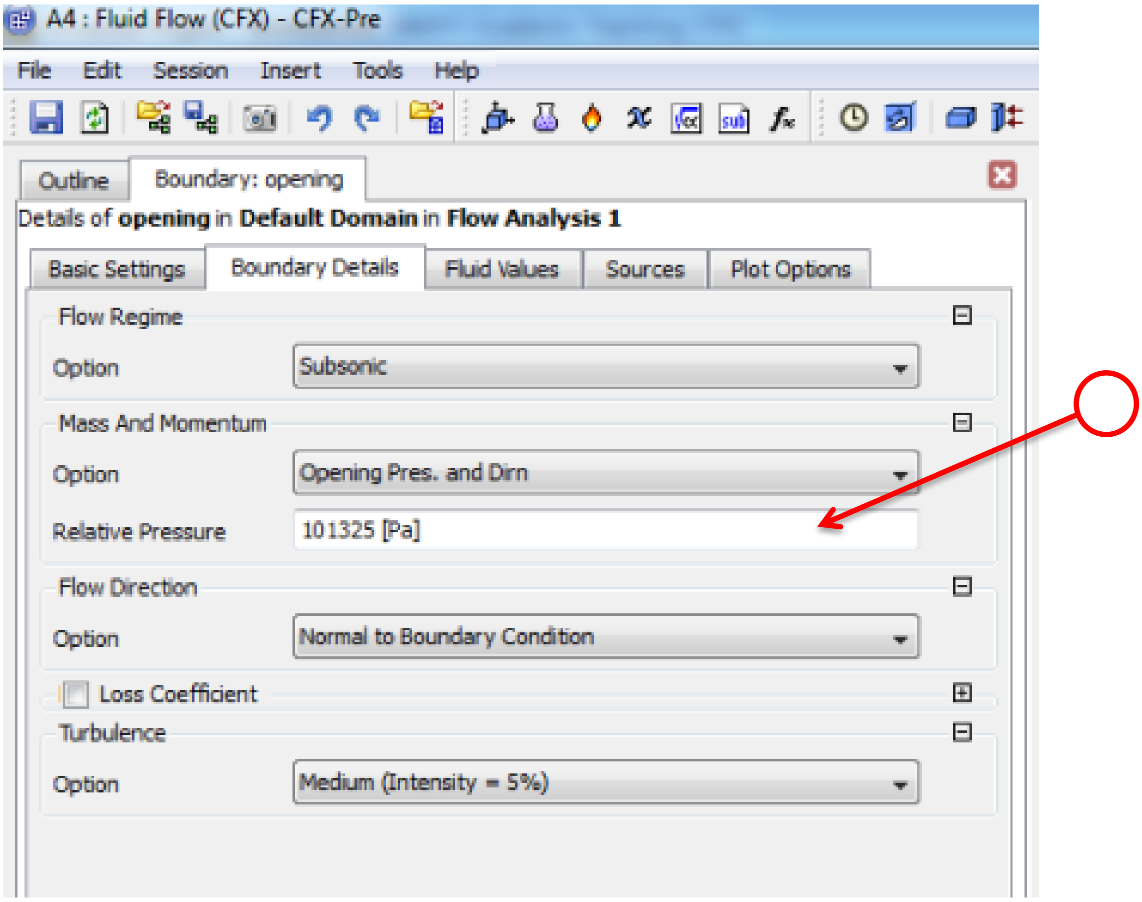
Step 27:

Go to the CFX.cmdb and from the drop down list for the Principial 2D Regions select the five surfaces having the names of F265.251, F267.251, F268.251, F269.251 and 270.251, the go to Insert , then go to boundary and then under boundary select Opening.



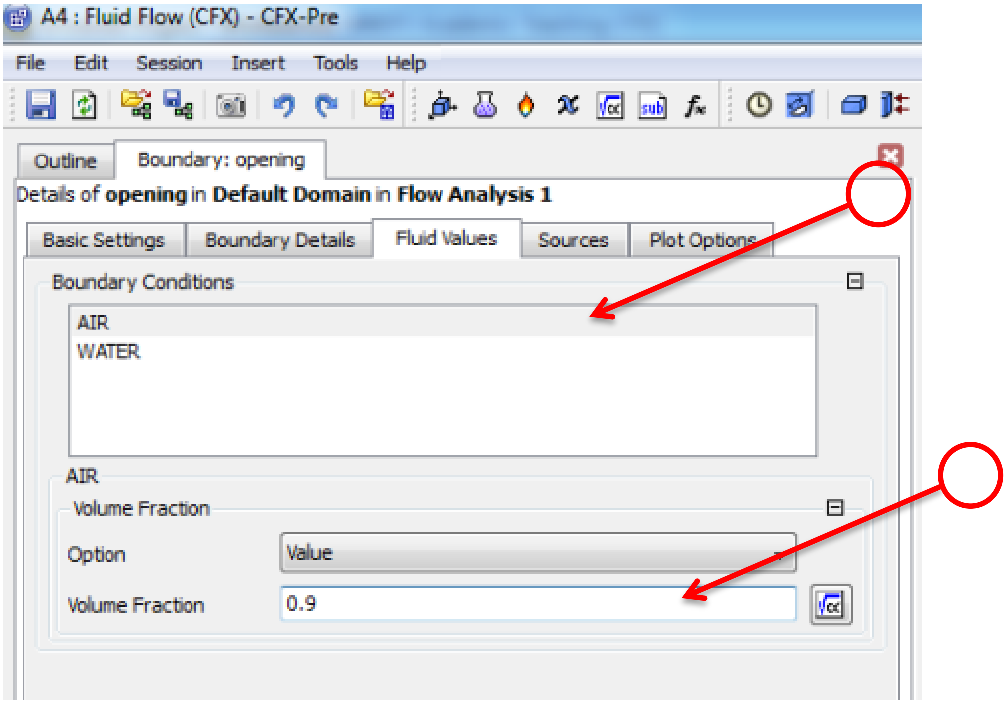
Step 28:

Type into the relative pressure cell a pressure value of 101325, this would apply atmosphric conditions to the spray domain.



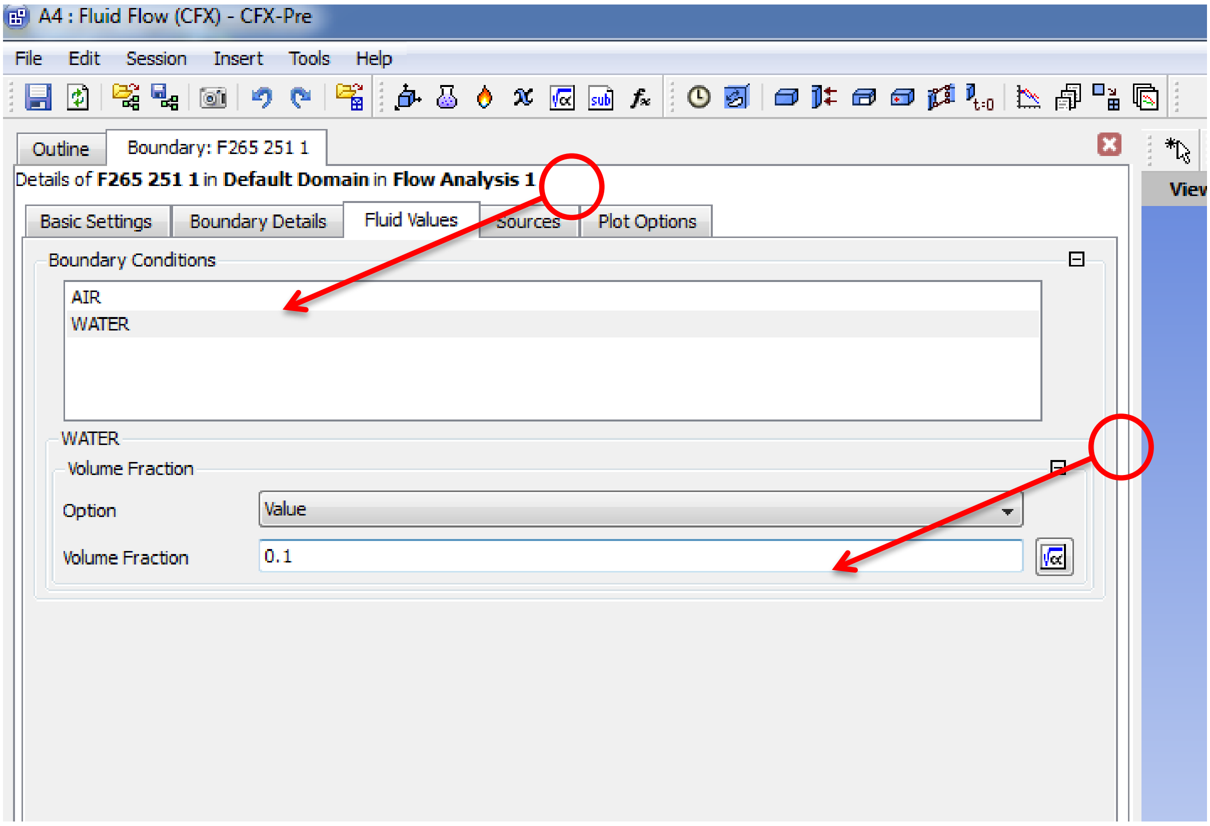
Step 29:

In the boundary conditions section go to Fluid Values and select AIR then in the volume fraction section input a value of 0.9.



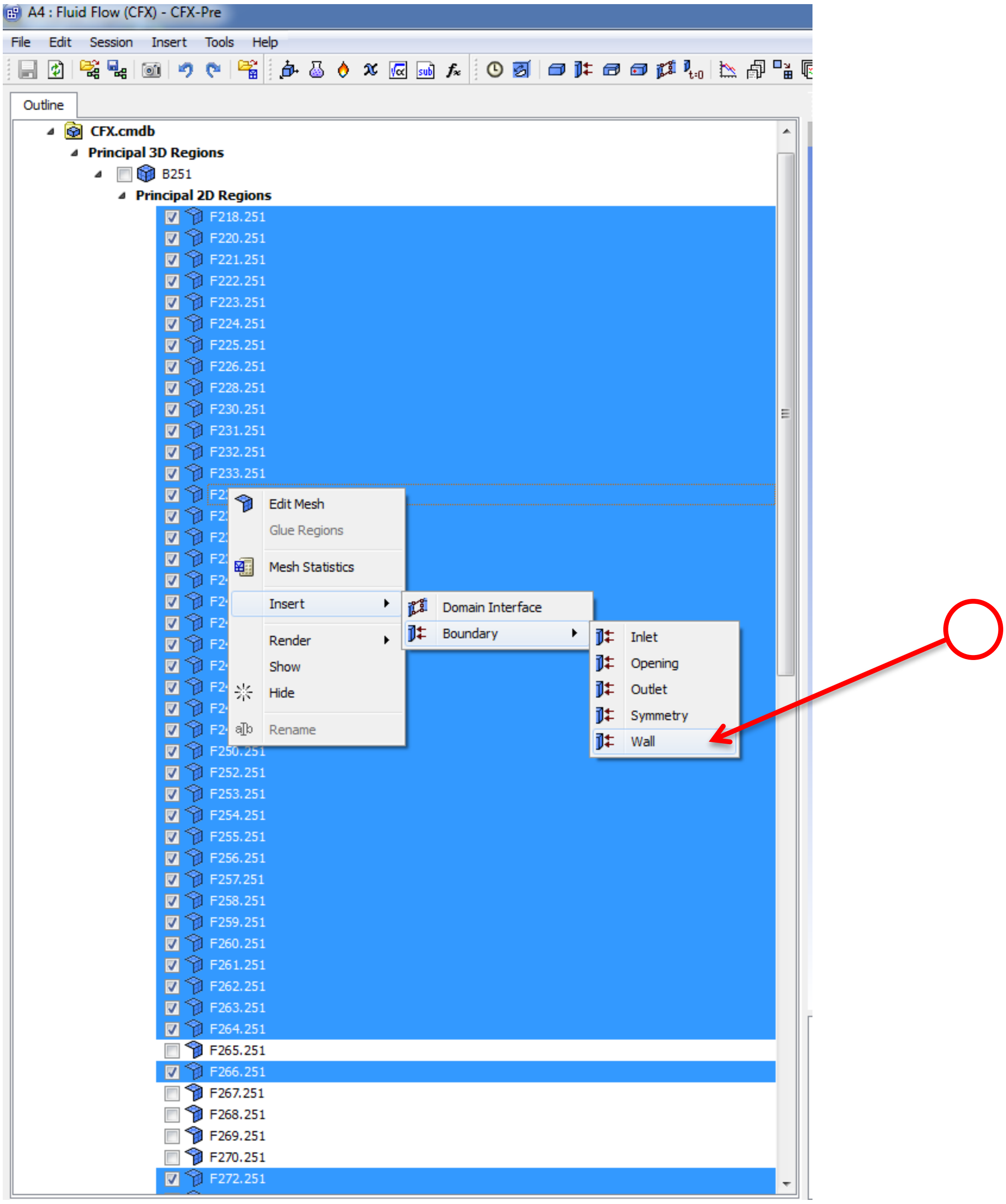
Step 30:

The next step is to go again to Fluid Values and select WATER under boundary conditions. Then apply 0. 1 as the volume fraction for WATER.



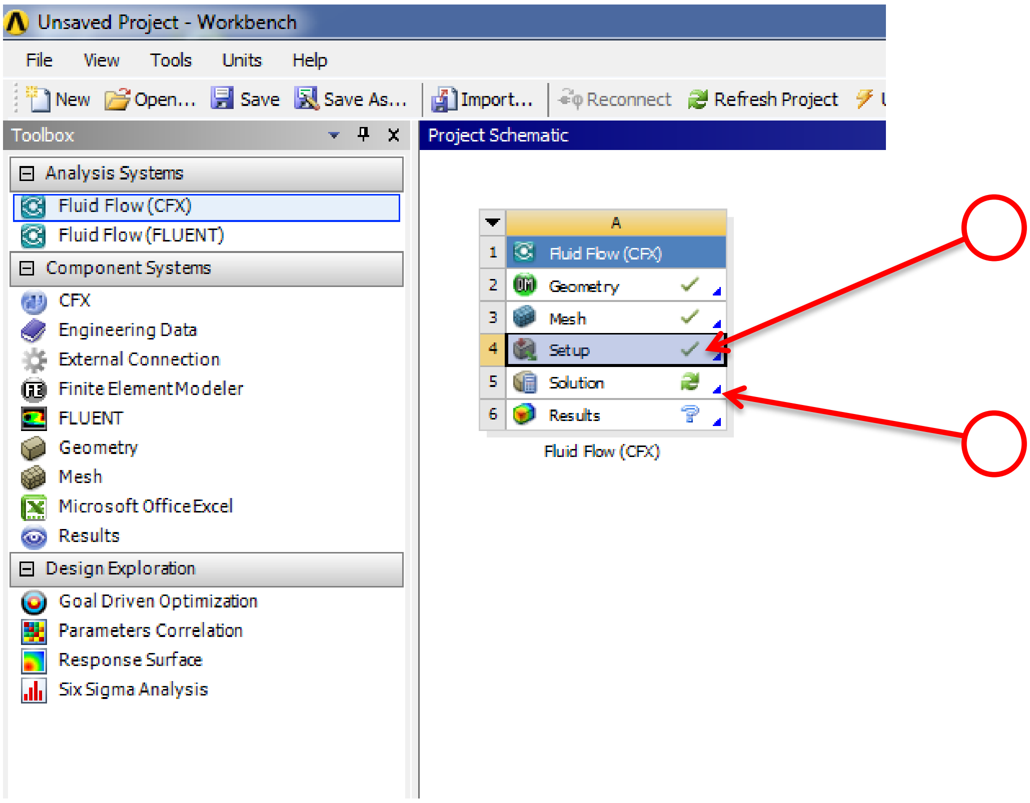
Step 31:

Go to the CFX.cmdb and from the drop down list for the Principial 2D Regions select the rest of the regions which havent been selected before, then go to Insert , then go to boundary and then under boundary select wall.



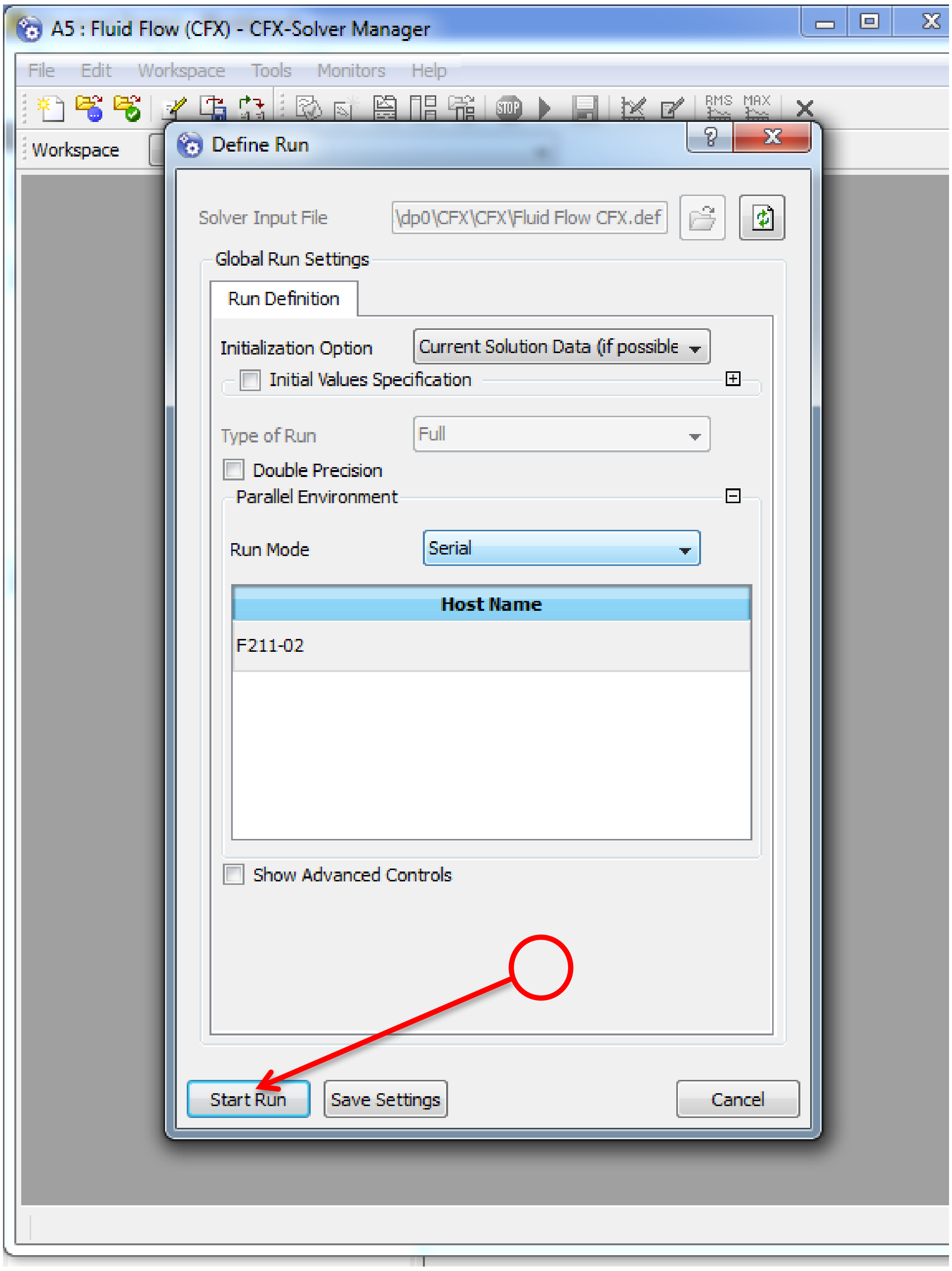
Step 32:

The next step is to check that there is a green tick beside the setup icon then double click on solution.



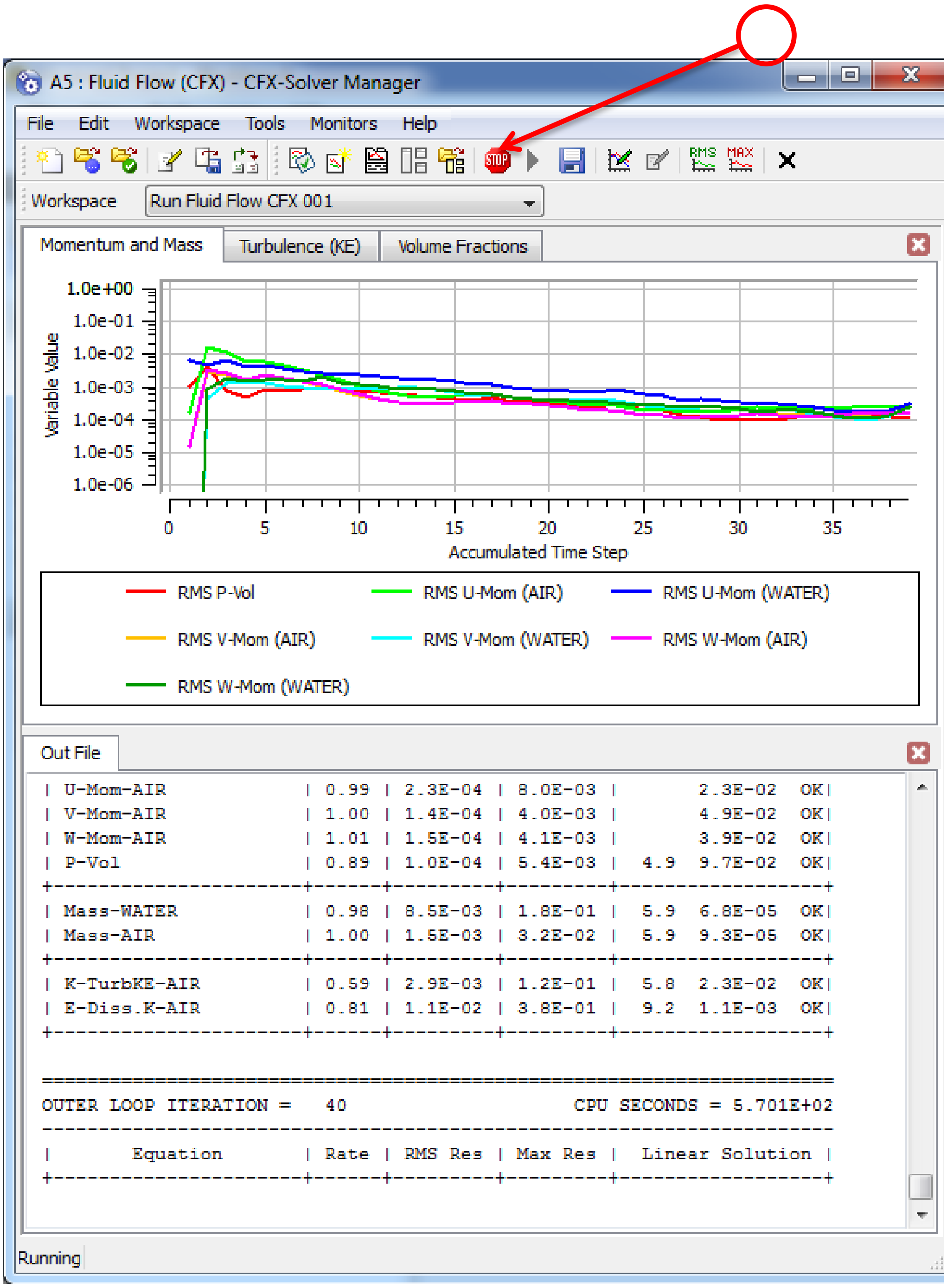
Step 33:

The final step is press on Start Run and the calculation should start.



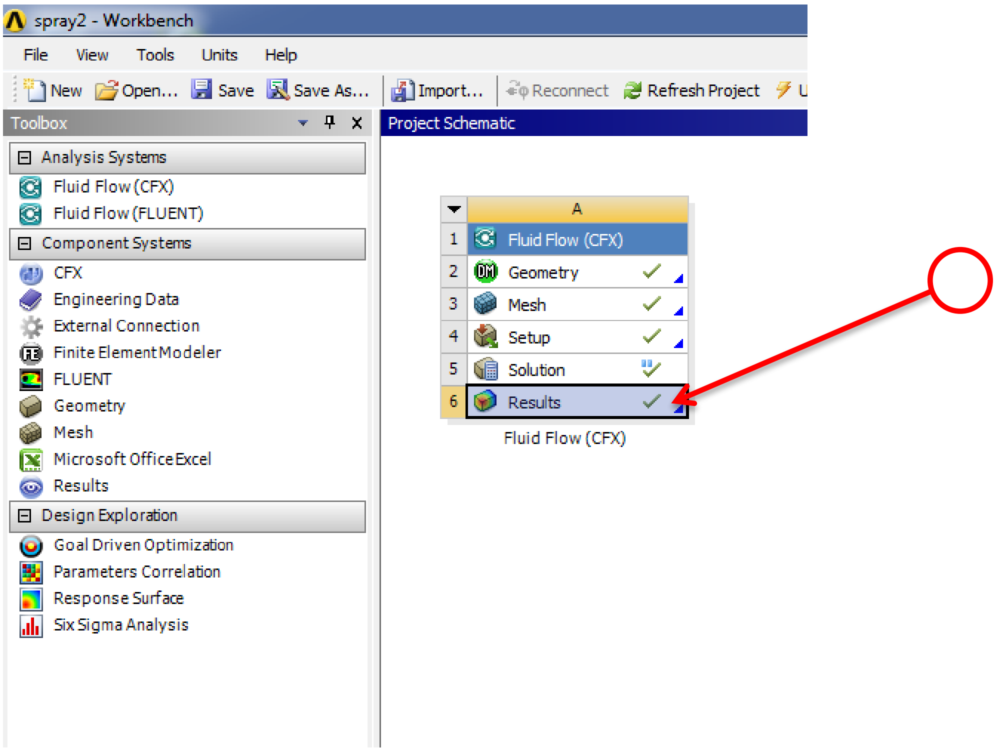
Step 34:

The solver should show you something like this. You can press stop after a number of time steps then check if the required results fall into the wanted ranges. In this case I will press stop you can go back and continue the simulation from the stoped point.



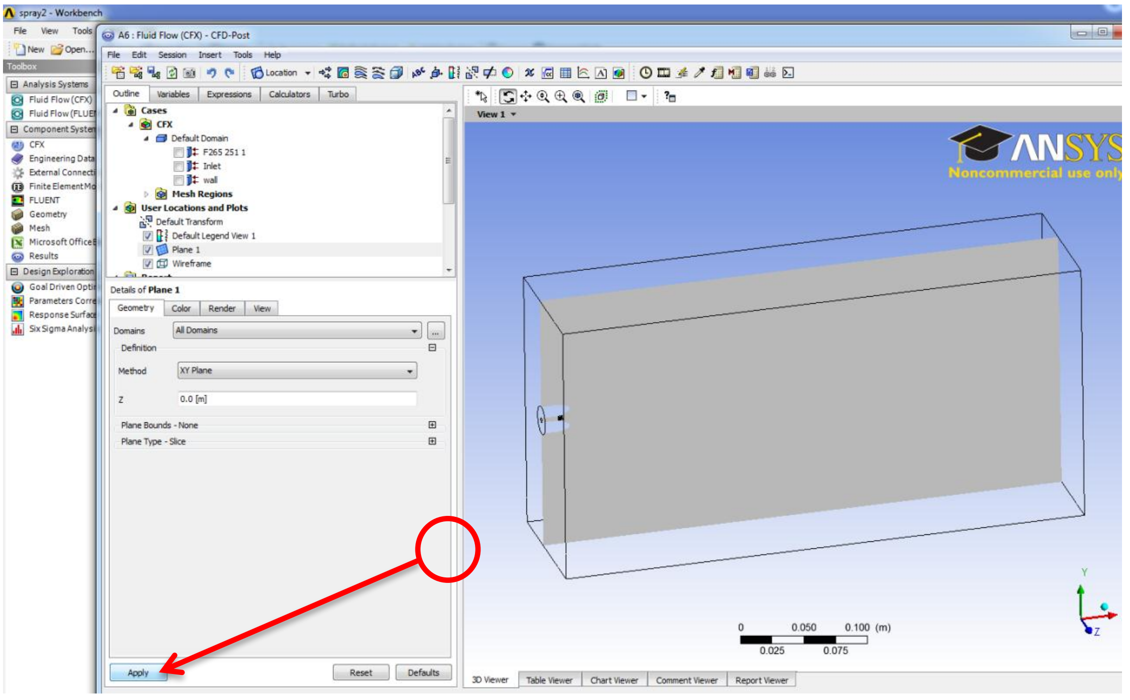
Step 35:

The user can see a pause and a green tick mark beside the solution icon by double clicking on the results icon the researcher can proceed to the data analysis part.



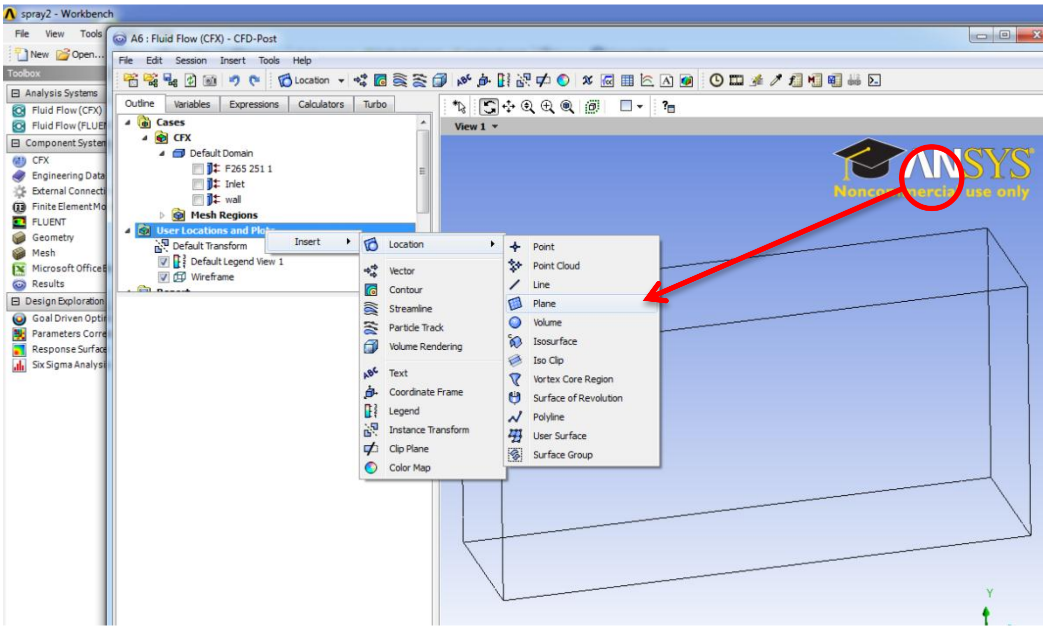
Step 37:

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



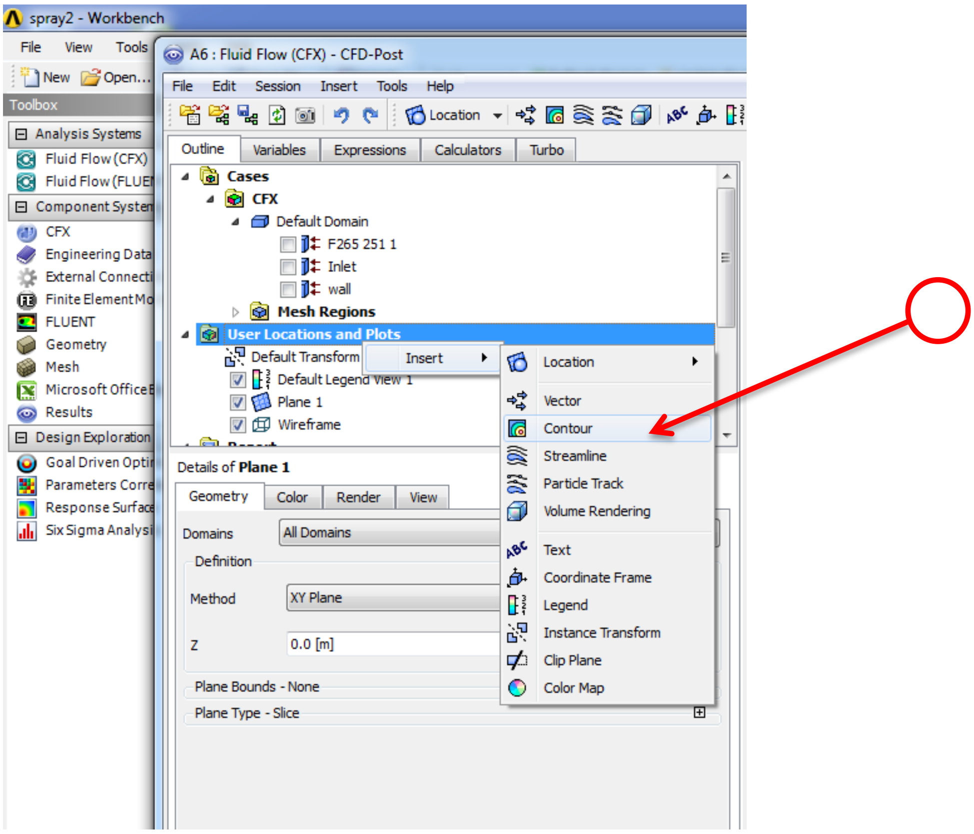
Step 36:

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



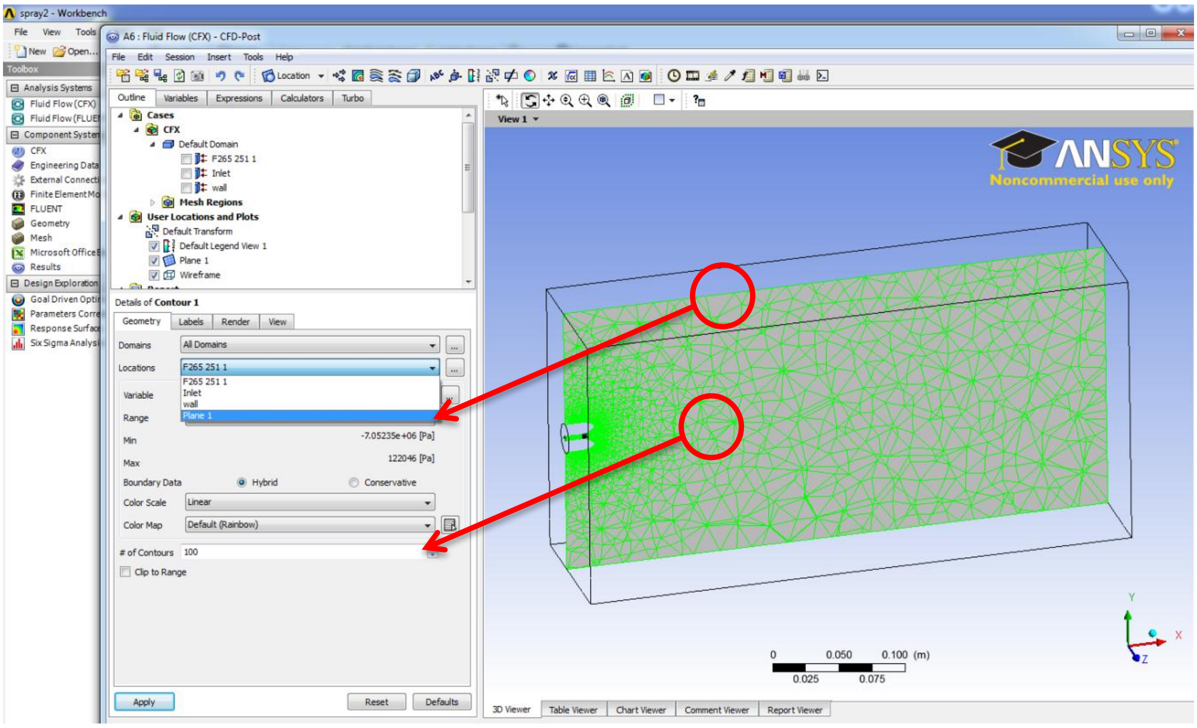
Step 37:

Left click the cursor on the User Locatins and Plots icon, then go to insert then from the drop down list select Contour, a window will open with a plane default name of contour 1 higlighted in blue changing its name is upto the user.



Step 38:

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 39:

The next step is to select the Water.Velocity Variable from the drop down list, then go to the Range and select from the drop down list User Specfied.When user specfied is selected two new options are added the minimum and the maximum enter a minimum velocity value of zero and a maximum velocity value 0.9. Finaly press apply and what would be visibile the velocity profile of water. The user has required knowlagde now to continue on his own.