EXTERNAL FLOW AROUND A 3-D RECTANGULAR CYLINDER

Compiled by Ratish Dixit (OpenFOAM CFD club)

There are 4 basic steps followed while analyzing fluid flow around bodies.

I. Construction of geometry
II. Meshing of the geometry
III. Simulation
IV. Post-processing

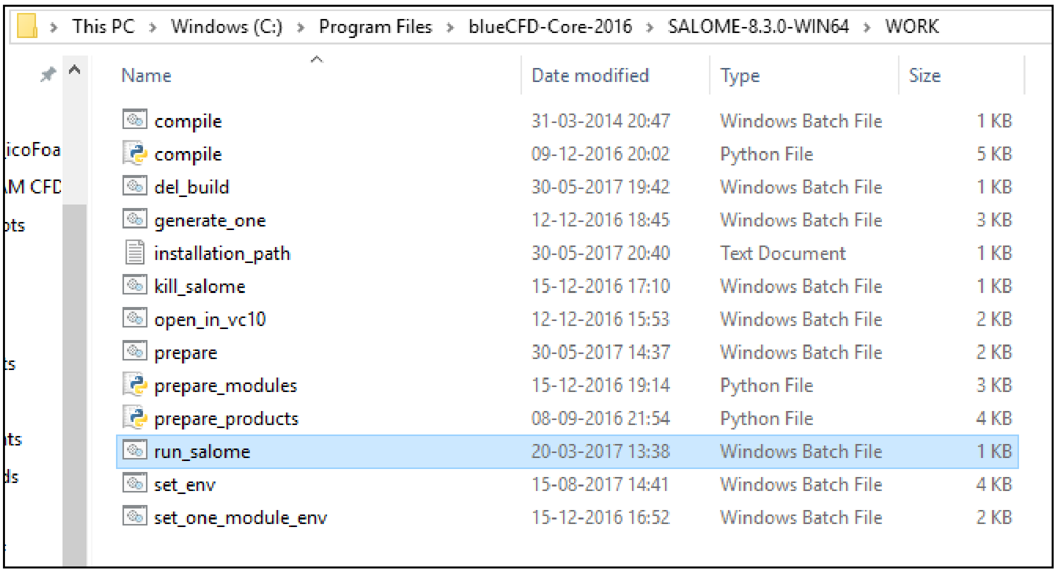
Before starting with our process, let’s know some basics.

(a) Create a folder, ‘EXT_FLOW’ on desktop. We will save all our files in this folder only.
(b) Always copy the files from the main folder to this new folder.
(c) Always dump study in .py format.
(d) icoFoam is used for unsteady state, incompressible and laminar flow.
(e) simpleFoam is used for steady state, incompressible and turbulent flows.
(f) pisoFoam is used for unsteady state, incompressible and few turbulent models.
(g) pimpleFoam is a combination of pisoFoam and simpleFoam and is more robust and efficient.
(h) In case of external flows, a larger box is made which acts as a domain and the smaller box inside the big box which acts as our body. The smaller box has to be cut from the domain to study flow around it.
(i) The walls of the smaller box are also included in the walls of the domain since we want the same conditions on the body as on the walls of the domain.

I. CONSTRUCTION OF GEOMETRY

(i) Opening salome

a. Extracted folder → blueCFD-Core-2016 → SALOME-8.3.0-WIN64 → WORK → run_salome.bat → Double click

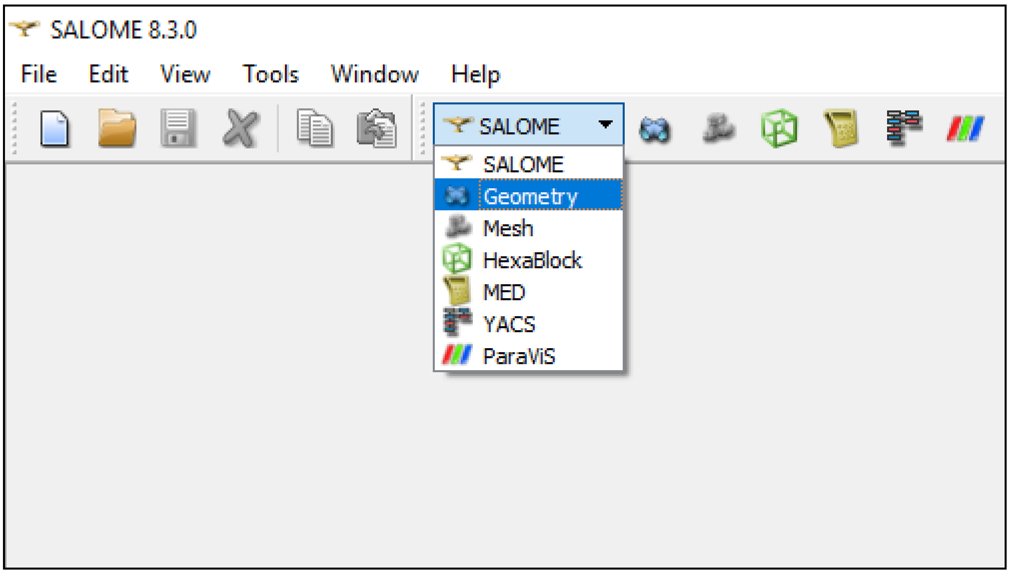


Name	Date modified	Type	Size
compile	31-03-2014 20:47	Windows Batch File	1 KB
compile	09-12-2016 20:02	Python File	5 KB
del_build	30-05-2017 19:42	Windows Batch File	1 KB
generate_one	12-12-2016 18:45	Windows Batch File	3 KB
installation_path	30-05-2017 20:40	Text Document	1 KB
kill_salome	15-12-2016 17:10	Windows Batch File	1 KB
open_in_vc10	12-12-2016 15:53	Windows Batch File	2 KB
prepare	30-05-2017 14:37	Windows Batch File	2 KB
prepare_modules	15-12-2016 19:14	Python File	3 KB
prepare_products	08-09-2016 21:54	Python File	4 KB
run_salome	20-03-2017 13:38	Windows Batch File	1 KB
set_env	15-08-2017 14:41	Windows Batch File	4 KB
set_one_module_env	15-12-2016 16:52	Windows Batch File	2 KB

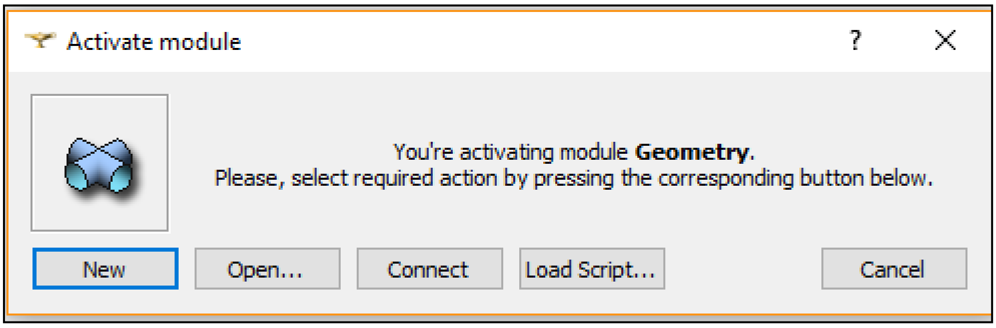
b. Never cut the terminal that opens with it.

(ii) After salome opens

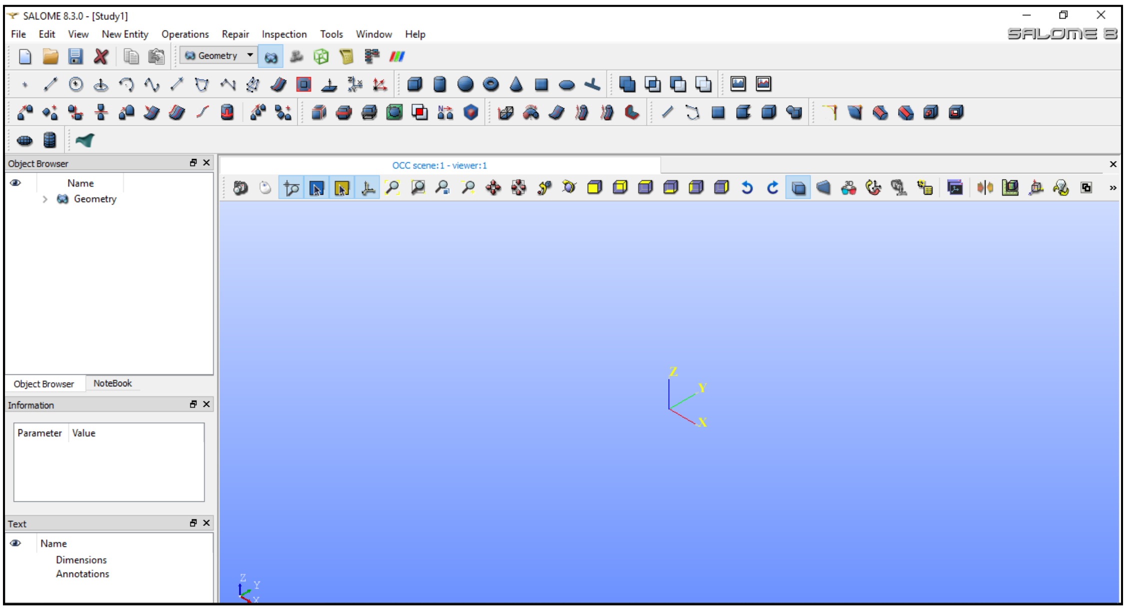
a. In the salome window, choose ‘Geometry’ from the drop down menu.



b. Opening new geometry, click on 'New'.

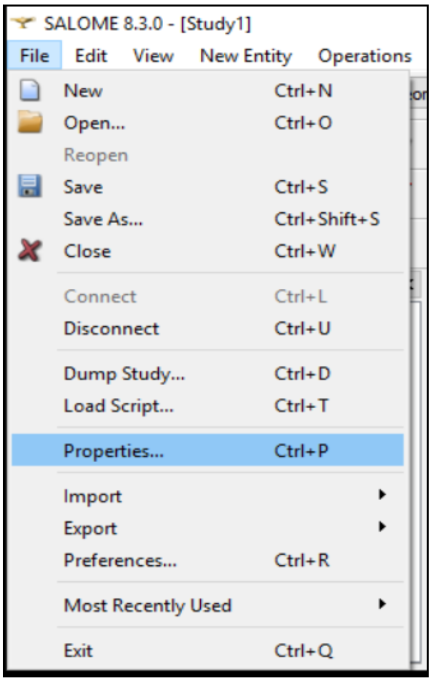


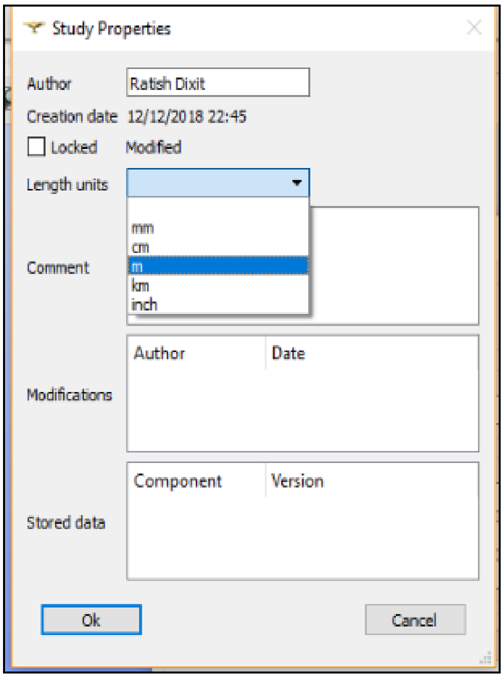
c. Salome window



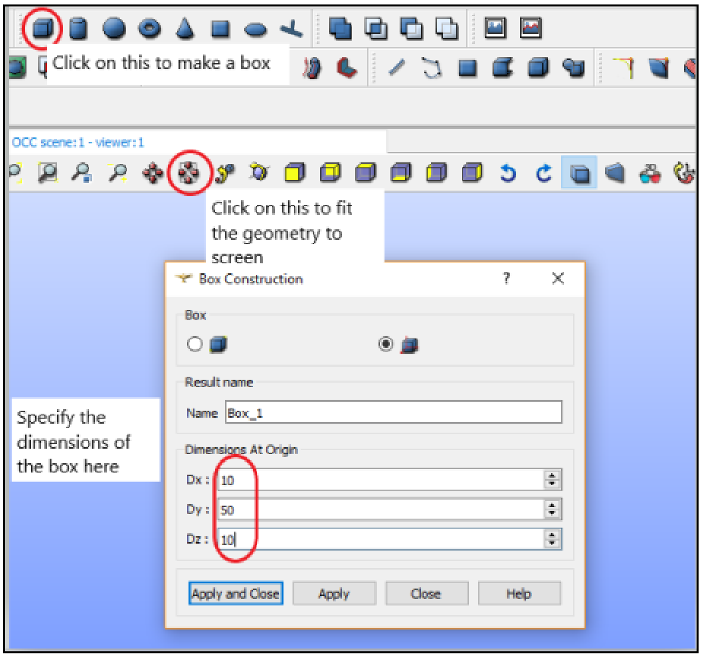
(iii) Geometry

a. Specify the units in meter and click 'Ok'.

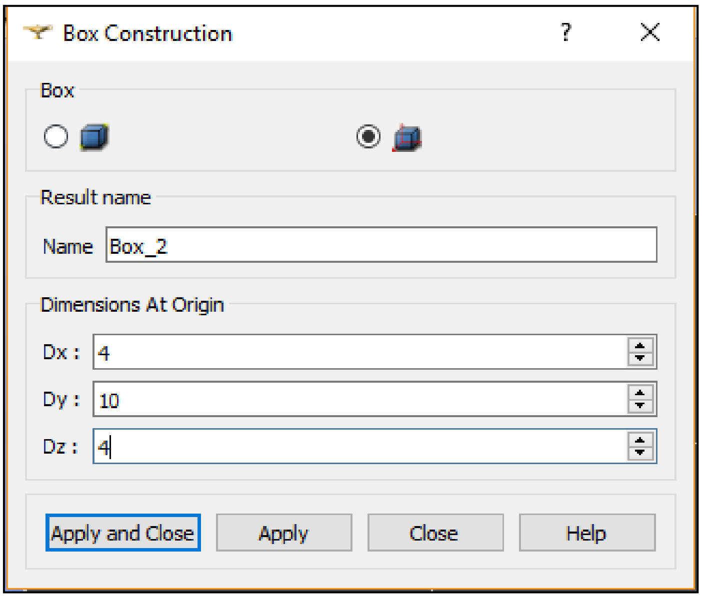




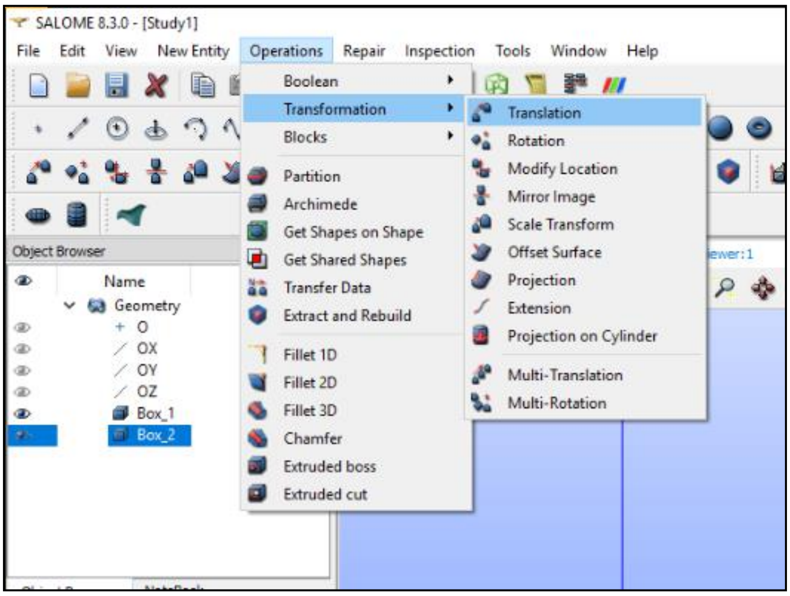
b. Creating a box, dimensions and fitting it to the window. Then click 'Apply and close'.



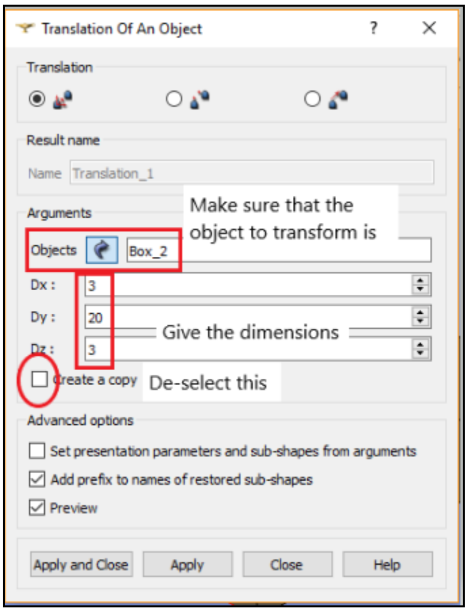
c. Now make another box. Then click 'Apply and close'.



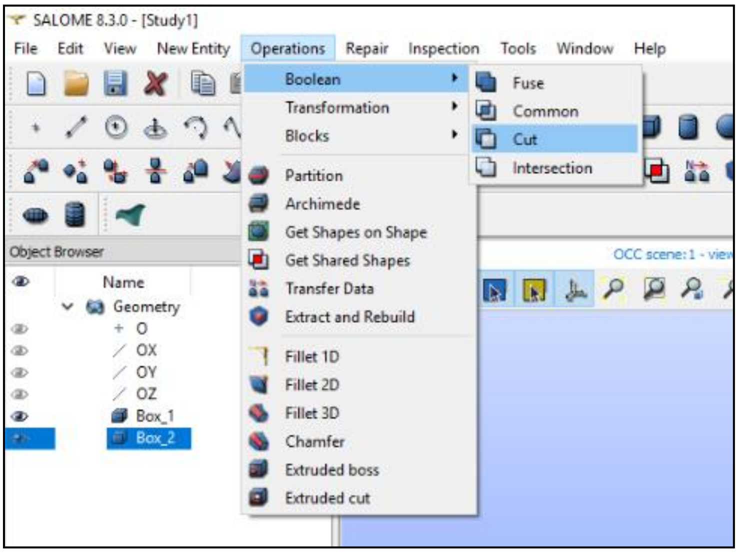
d. Translation of Box_2 (body) to its location in the domain (Box_1).



e. In the dialogue box that opens, do as below and click 'Apply and close'.

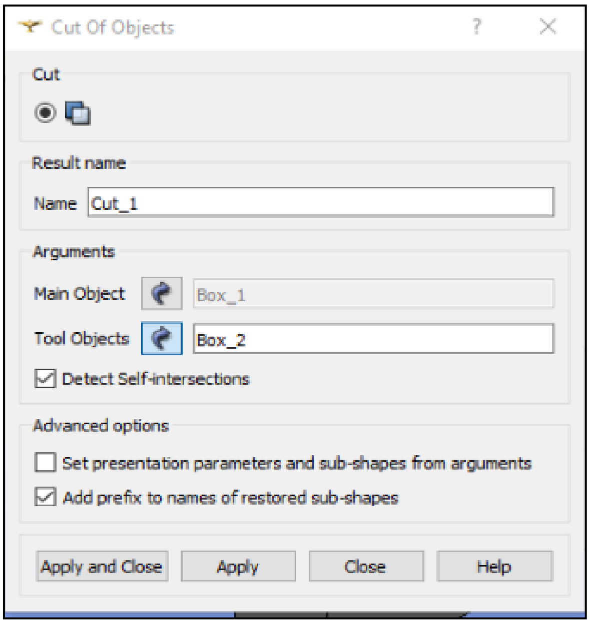


f. Now, Box_2 has to be cut-off from the domain (Box_1).

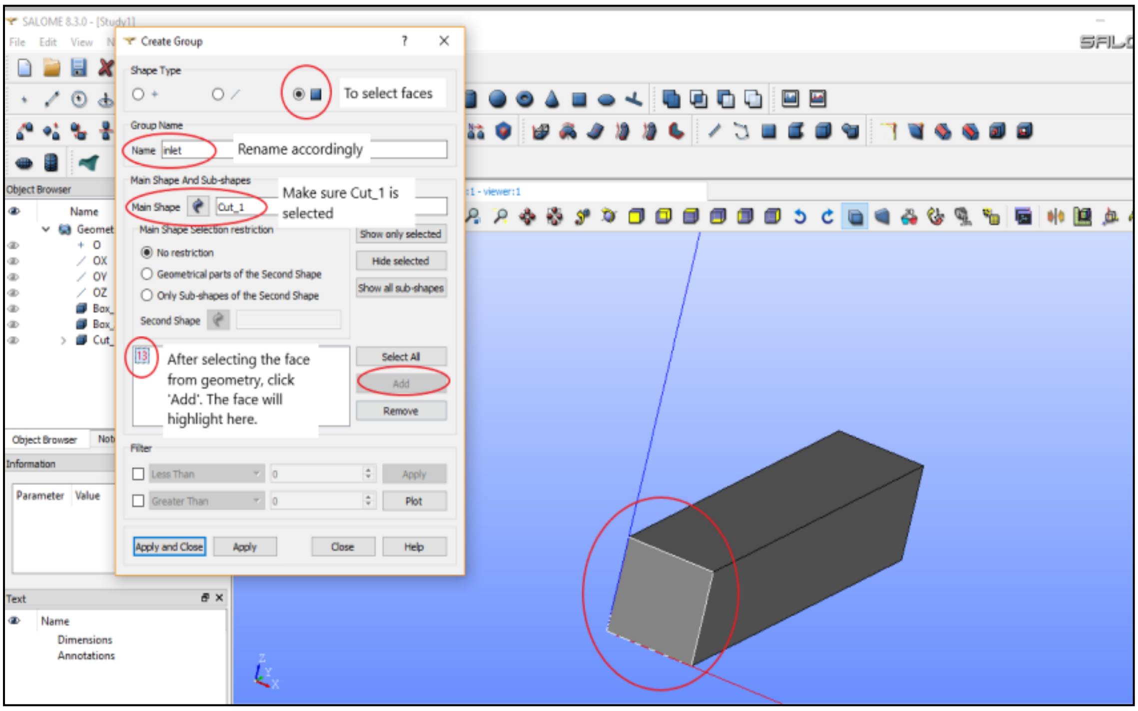


g. In the dialogue box that opens, do as below and click 'Apply and close'.

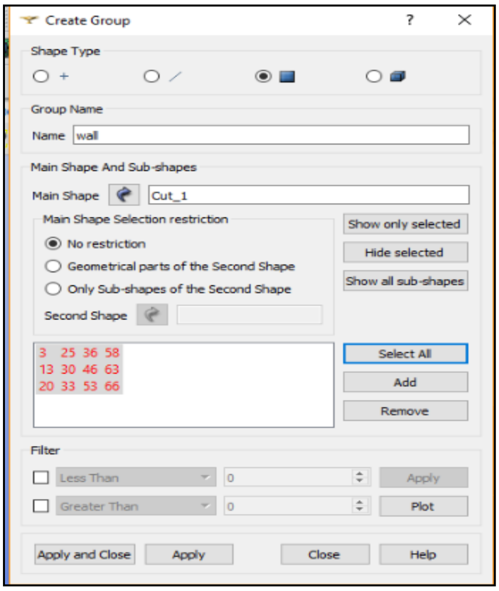
- Main object – From which the body has to be cut
- Tool object – The object that has to be cut



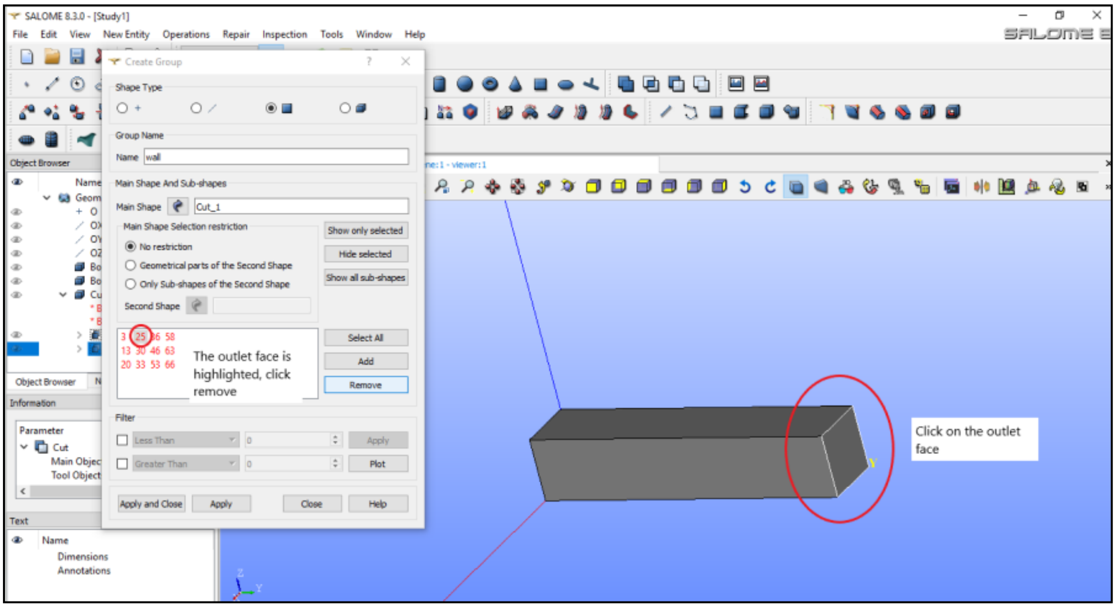
h. Creating groups. Select 'Cut_1' from the object browser window on the left side of the screen. Right click and select 'Create Group'. Then do as follows.



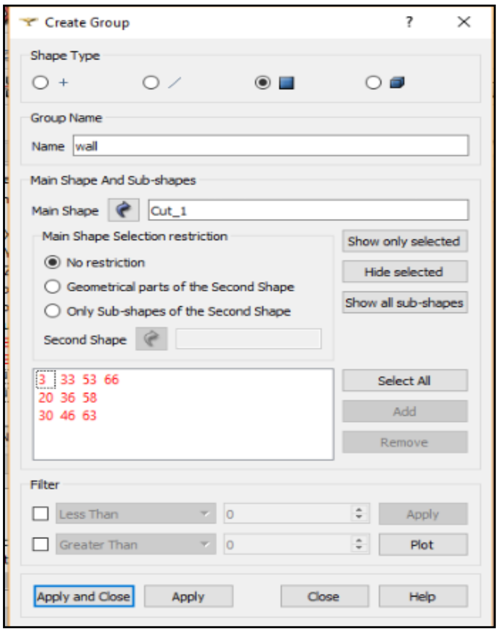
Repeat the procedure for outlet and select the face opposite to the inlet face.
For creating walls, do as follows.
Click 'Select all'. All the faces will be highlighted.



Now, when you click on ‘inlet’ and ‘outlet’ faces in the geometry and click ‘Remove’.



Now, after removing inlet and outlet faces, you will be left with 10 faces. Click ‘Apply and close’.

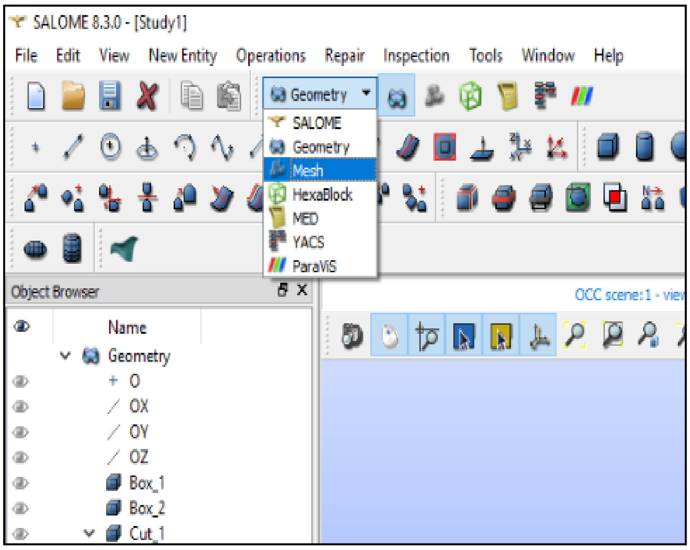


(iv) Dump study

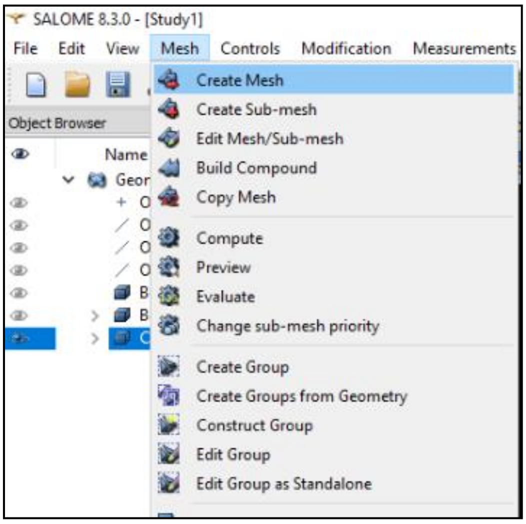
File → Dump Study → Rename → Save in .py format in the 'EXT_FLOW' folder.

II. MESHING

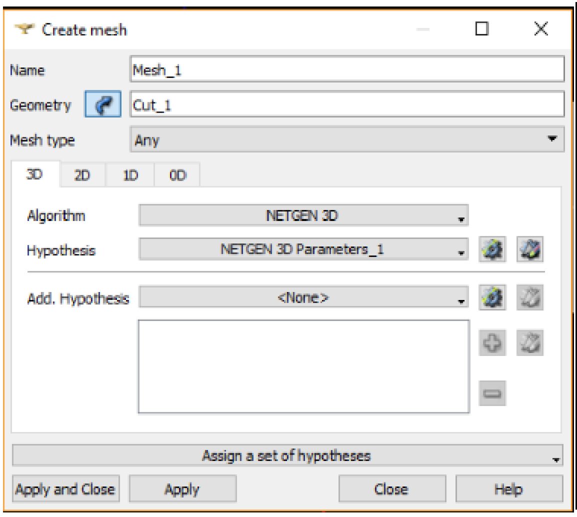
(i) Start meshing. Click on 'Geometry' and select 'Mesh' from the drop-down menu.

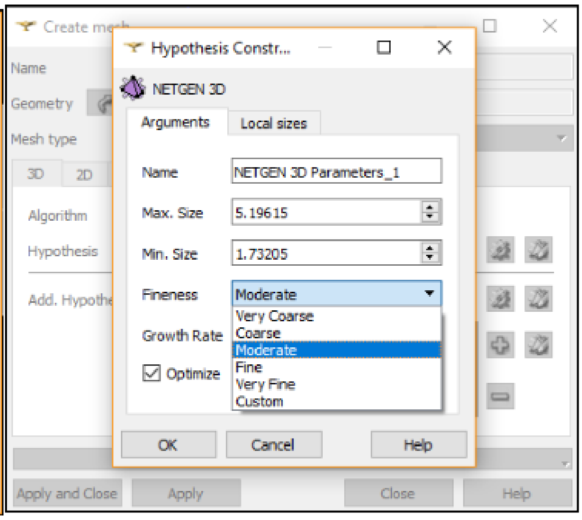


(ii) Go to 'mesh' in the tool bar, and click 'create mesh'.

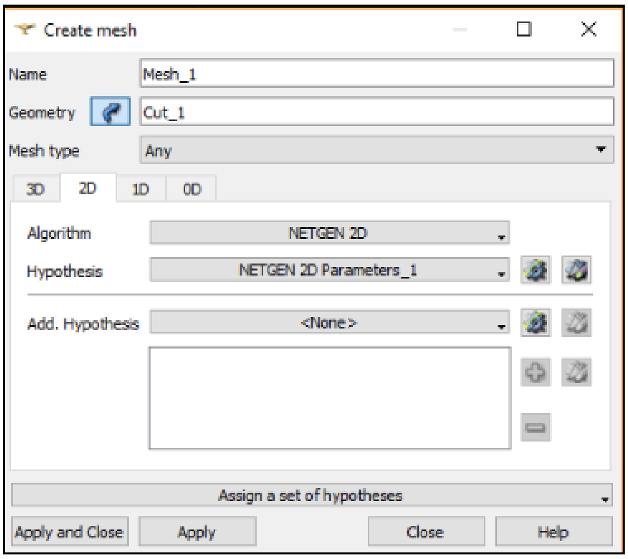


(iii) In the dialogue box that opens, select the following parameters. For 3D, algorithm → NETGEN 3D, hypothesis → NETGEN 3D Parameters. Again a dialogue box will open. Do as done in the second image. Click 'Ok'.

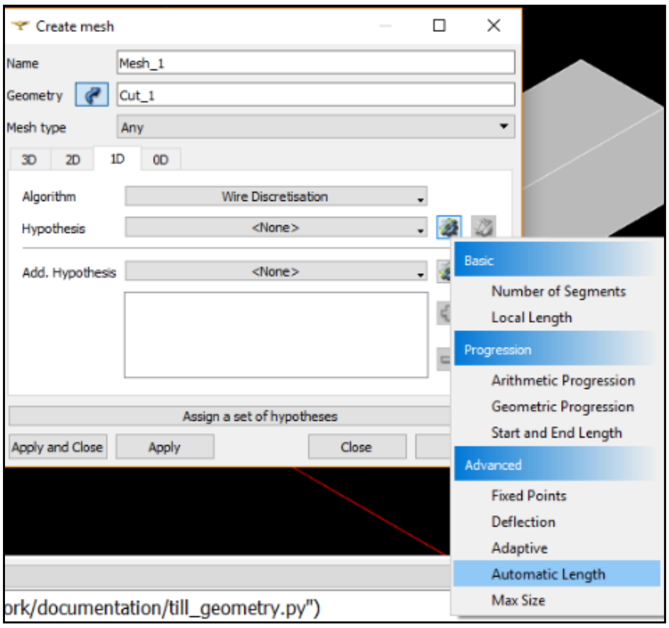


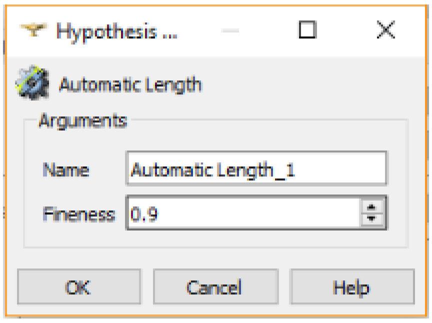


Similarly for 2D, algorithm → NETGEN 2D, hypothesis → NETGEN 2D Parameters. Select the same parameters in the second dialogue box as did for 3D.



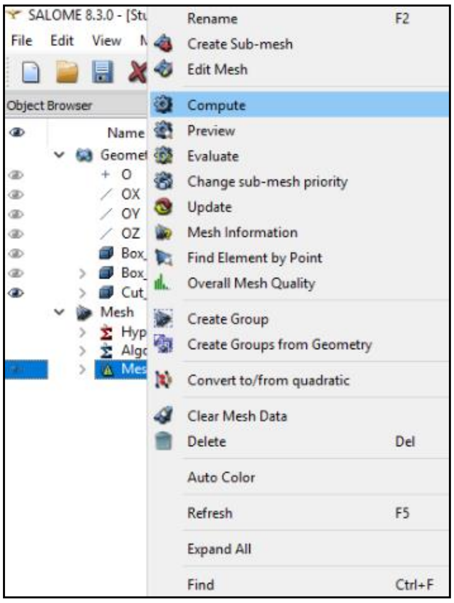
For 1D mesh, choose ‘wire discretisation’ and ‘automatic length’. Give fineness ratio of 0.1 (coarse mesh) - 0.9 (fine mesh). And click ‘Ok’ → ‘Apply and close’.



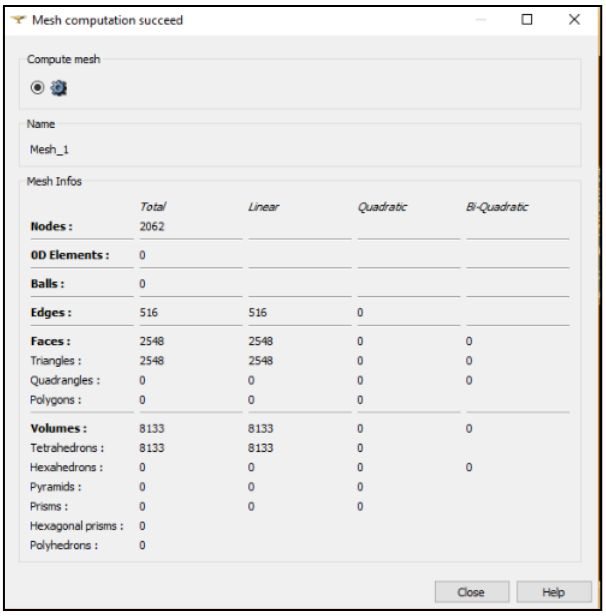


(iv) Computing mesh

Right click ‘Mesh_1’. Click ‘Compute’ to compute mesh.



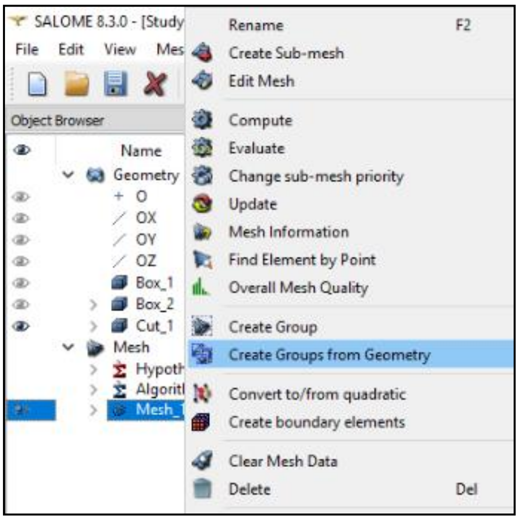
After computation, a dialogue box will open showing the numbers of different elements in the geometry.



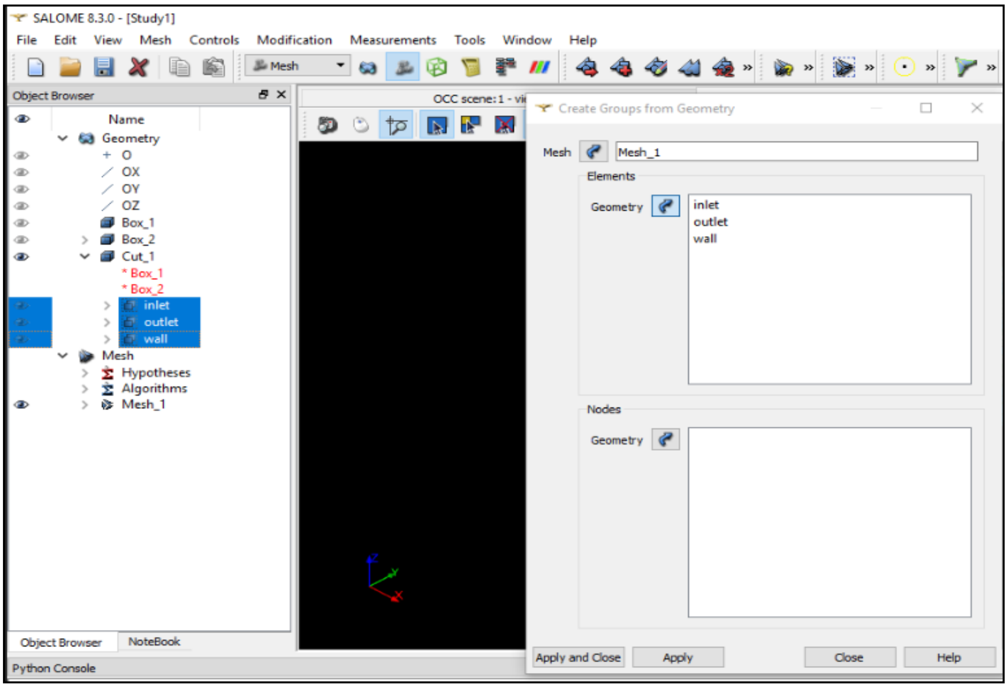
Note: Pyramids should always be zero then only your geometry and meshing is correct.

(v) Creating groups in the mesh

Right click ‘Mesh_1’ → ‘Create Groups from Geometry’

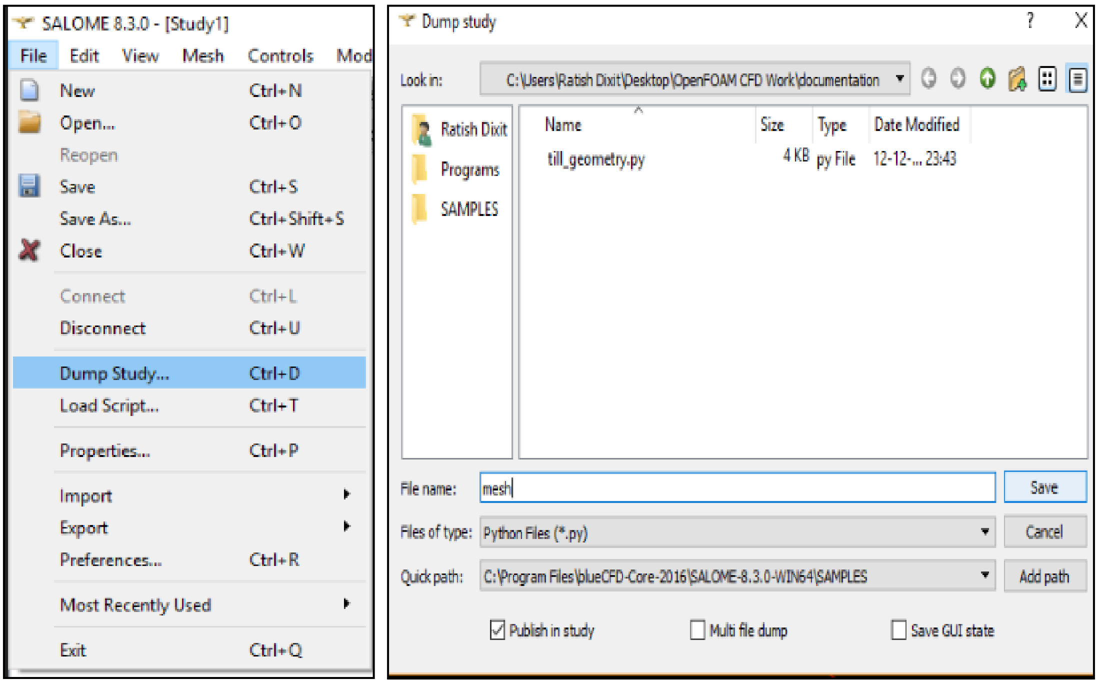


In the dialogue box that opens, select ‘inlet’, ‘outlet’ and ‘wall’ from the geometry. Your selections will automatically be shown in the Geometry window. Click ‘Apply and close’.



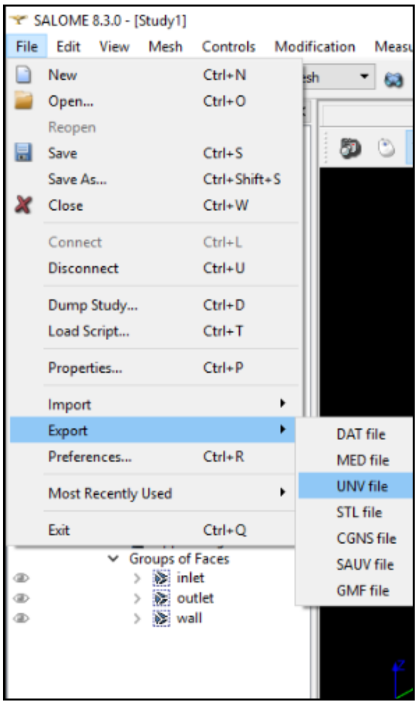
(vi) Dump study

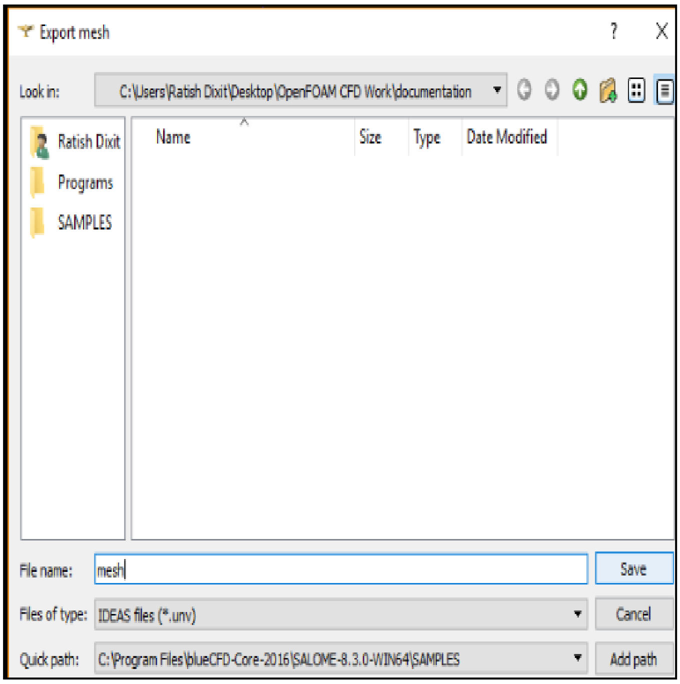
File → Dump study → Rename → Save in .py format in the 'EXT_FLOW' folder.



(vii) Exporting the mesh to .unv format

Select 'Mesh_1' →Go to 'File' →Export → UNV file → Rename →Save

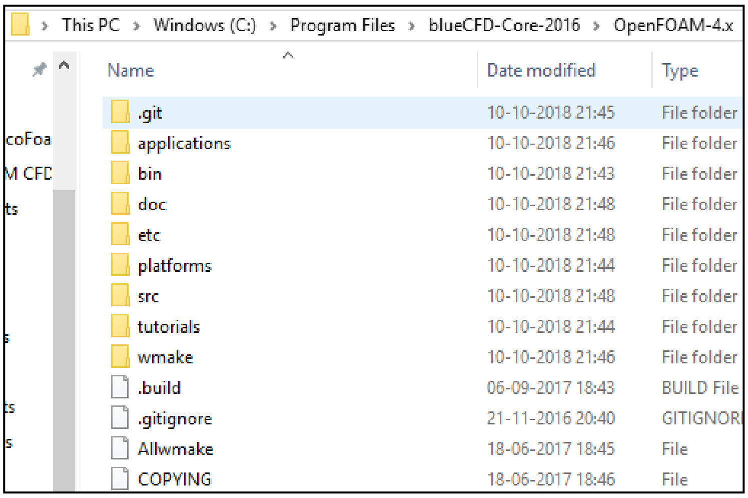




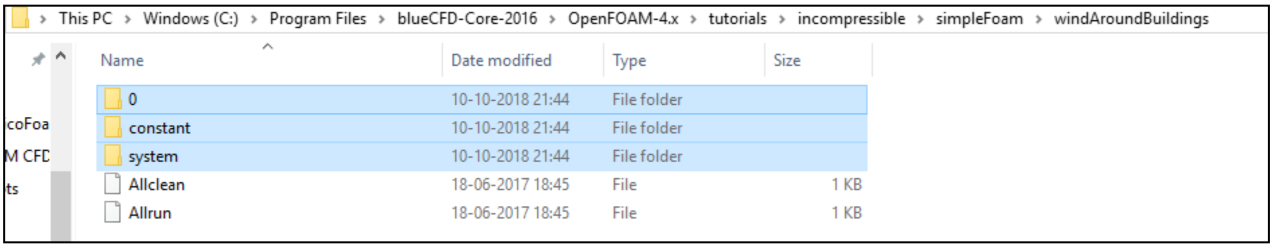
(viii) Exit salome

III. SIMULATING OUR FLOW

(i) Go to the location where you have extracted the software. For example: C:\Program Files\blueCFD-Core-2016\OpenFOAM-4.x



Click on 'tutorials' → 'incompressible' → 'simpleFoam' → 'wind around building'.



Copy the 0, constant and system folders to the 'EXT_FLOW' folder. Always COPY never CUT.

(ii) Editing the files in 0, constant and system folders for our problem in the 'EXT_FLOW' folder.

a. In 0 folder, only keep the following files and delete rest.

epsilon	12-12-2018 19:54	File	2 KB
k	12-12-2018 19:54	File	2 KB
nut	12-12-2018 19:54	File	2 KB
p	12-12-2018 19:55	File	2 KB
U	12-12-2018 22:14	File	2 KB

b. Open ‘epsilon’ file. Do the following changes and save.

```
boundaryField
{
    inlet
    {
        type            fixedValue;
        value           uniform $epsilonInlet;
    }

    outlet
    {
        type            inletOutlet;
        inletValue      uniform $epsilonInlet;
        value           uniform $epsilonInlet;
    }

    wall
    {
        type            epsilonWallFunction;
        value           uniform $epsilonInlet;
    }

}
```

c. Open the ‘k’ file. Do the following changes and save.

```
boundaryField
{
    inlet
    {
        type            fixedValue;
        value           uniform $kInlet;
    }

    outlet
    {
        type            inletOutlet;
        inletValue      uniform $kInlet;
        value           uniform $kInlet;
    }

    wall
    {
        type            kqRWallFunction;
        value           uniform $kInlet;
    }

}
```

d. Open the ‘nut’ file. Do the following changes and save.

```
boundaryField
{
    inlet
    {
        type            calculated;
        value           uniform 0;
    }

    outlet
    {
        type            calculated;
        value           uniform 0;
    }

    wall
    {
        type            nutkWallFunction;
        value           uniform 0;
    }

}
```

e. Open the ‘p’ file. Do the following changes and save.

```
boundaryField
{
    inlet
    {
        type            zeroGradient;
    }

    outlet
    {
        type            totalPressure;
        p0              uniform 0;
        gamma           1.4;
        value           uniform 0;
    }

    wall
    {
        type            zeroGradient;
    }

}
```

f. Open the ‘U’ file. Do the following changes and save.

```
Uinlet          (0 20 0);

dimensions      [0 1 -1 0 0 0 0];

internalField   uniform (0 0 0);

boundaryField
{
    inlet
    {
        type            fixedValue;
        value           uniform $Uinlet;
    }

    outlet
    {
        type            pressureInletOutletVelocity;
        value           uniform (0 20 0);
    }

    wall
    {
        type            noSlip;
    }

}
```

(iii) Nothing to be changed in 'constant' folder for now.

(iv) In the 'system' folder

a. In the 'system' folder only keep the 3 files below and delete all other files.

controlDict	12-12-2018 21:40	File	2 KB
fvSchemes	21-11-2016 20:27	File	2 KB
fvSolution	21-11-2016 20:27	File	2 KB

b. Open 'controlDict' file. Do the following changes and save. Nothing to be changed in other two folders.

```
application     simpleFoam;

startFrom       latestTime;

startTime       0;

stopAt          endTime;

endTime         20;

deltaT          0.1;

writeControl    timeStep;

writeInterval   20;

purgeWrite      0;

writeFormat     ascii;

writePrecision  6;

writeCompression off;

timeFormat      general;

timePrecision   6;

runTimeModifiable true;

// adjustTimeStep  yes;
```

(v) Right click the 'EXT_FLOW' folder. Click 'Open in blueCFD core terminal'.

(vi) In the terminal

a. Type ‘ls’ to view the files in that folder. ‘ls’ is a foam run command.

```
Ratish Dixit@LAPTOP-BEOHK82O MINGW64 OpenFOAM-4.x /c/Users/RATISH~1/Desktop/O
$ ls
0  constant  mesh.py  mesh.unv  system  till_geometry.py
```

b. Type ‘ideasUnvToFoam.exe’ give a space and type the mesh file name ‘mesh.unv’ and click enter.

```
Ratish Dixit@LAPTOP-BEOHK82O MINGW64 OpenFOAM-
$ ideasUnvToFoam.exe mesh.unv
```

After the command runs, it will end with a message as below.

```
Sorting boundary faces according to group (patch)
0: inlet is patch
1: outlet is patch
2: wall is patch

Constructing mesh with non-default patches of size:
    inlet       166
    outlet      170
    wall        2212

End
```

c. Now, go to the ‘constant’ folder. Open ‘polymesh’ folder. Click and open ‘boundary’ file in the notepad.

Change the wall type to ‘wall’ which will be written as patch.

```
3   This number tells the number of groups
(   in our geometry
    inlet
    {
        type            patch;
        nFaces          166;
        startFace       14992;
    }
    outlet
    {
        type            patch;
        nFaces          170;
        startFace       15158;
    }
    wall             patch --> wall
    {
        type            wall;
        nFaces          2212;
        startFace       15328;
    }
)
```

d. Go to command window again. Type ‘checkMesh.exe’ and click enter.

```
Ratish Dixit@LAPTOP-BEOHK82(
$ checkMesh.exe
```

After the command runs, it will show a message as below.

```
Mesh OK.

End
```

e. Now, type the solver you have used. As here we will use ‘simpleFoam.exe’.

```
Ratish Dixit@LAPTOP-E
$ simpleFoam.exe |
```

It then starts performing the iterations. After it successfully runs, it will show a message as below.

```
Time = 20

smoothSolver:  Solving for Ux, Initial residual = 0.00647815, Final residual = 0.000215193, No Iterations 2
smoothSolver:  Solving for Uy, Initial residual = 0.00188096, Final residual = 5.64591e-005, No Iterations 2
smoothSolver:  Solving for Uz, Initial residual = 0.00617601, Final residual = 0.000186012, No Iterations 2
GAMG:  Solving for p, Initial residual = 0.00862202, Final residual = 0.000772304, No Iterations 1
time step continuity errors : sum local = 0.000154501, global = -9.55895e-006, cumulative = -0.00210669
smoothSolver:  Solving for epsilon, Initial residual = 0.00621438, Final residual = 0.000562419, No Iterations 1
bounding epsilon, min: -15.7909 max: 1786.49 average: 44.1148
smoothSolver:  Solving for k, Initial residual = 0.00932598, Final residual = 0.000173984, No Iterations 2
ExecutionTime = 17.016 s  ClockTime = 17 s

End
```

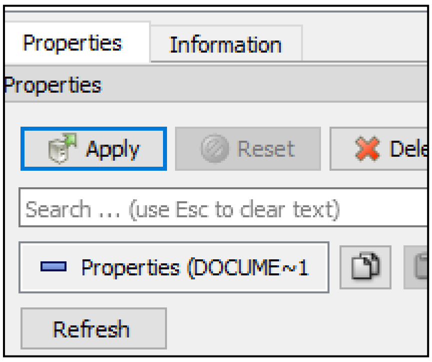
f. Type ‘paraFoam’ and click enter. This command will open the ‘ParaView’ software.

```
Ratish Dixit@LAPTOP-BEOHK820 MINGW64 OpenFOAM-
$ paraFoam
Created temporary 'DOCUME~1.foam'
|
```

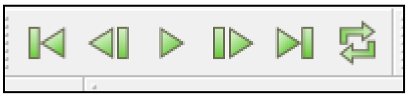
NOTE: When in command window, type a few alphabets of the command and press tab key. It will show all the possible commands you can choose from and then complete the command.

IV. POST-PROCESSING OF RESULTS

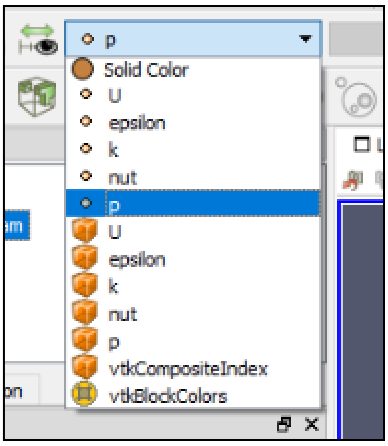
(i) Click ‘close’ in the dialogue box that opens when the software opens. Then click ‘Apply’.



(ii) Click ‘Play’ button then.

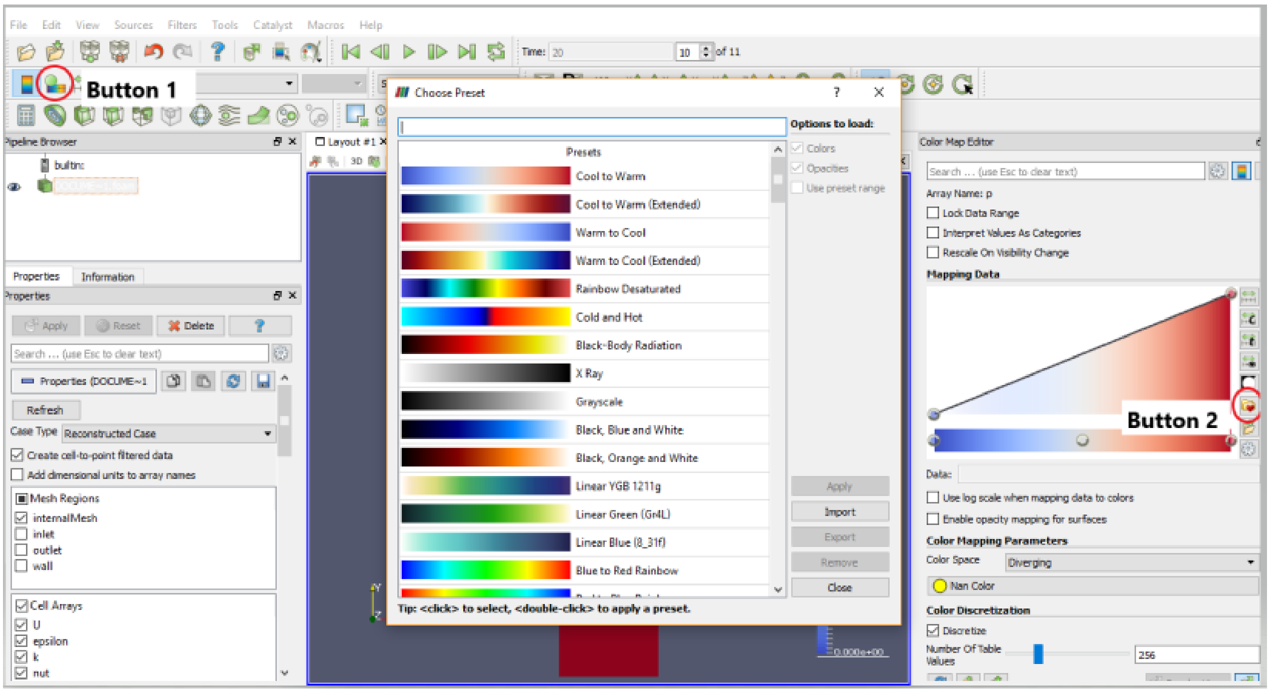


(iii) To change the vector quantity i.e. pressure and velocity and see their affects, toggle as below.

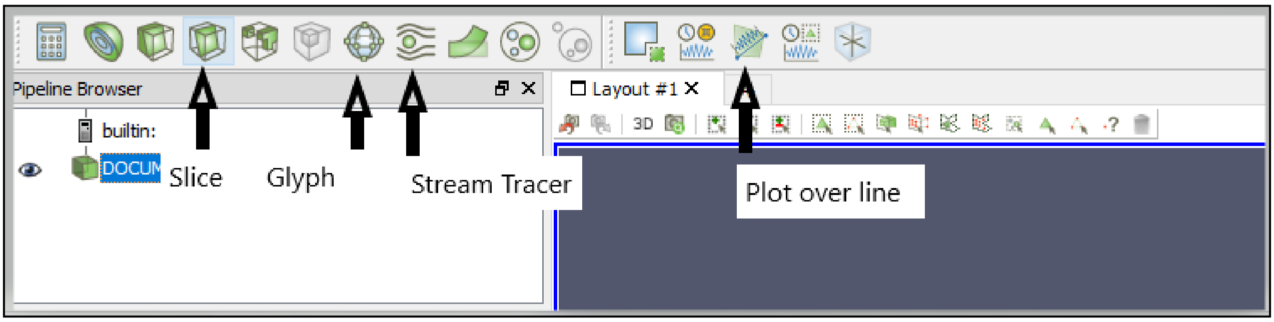


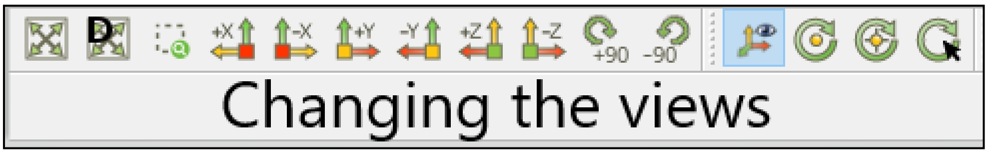
(iv) To change the color scheme of the results or to enhance the readability in terms of colors do as following.

First click, button 1 as in figure then click button 2. Select the color scheme you want, click apply and close. And close the ‘color map window’ that opened.



(v) Controls of the ParaView software





(vi) Applying ‘Slice’

a. First, select the box on the left side of the window which has to be sliced

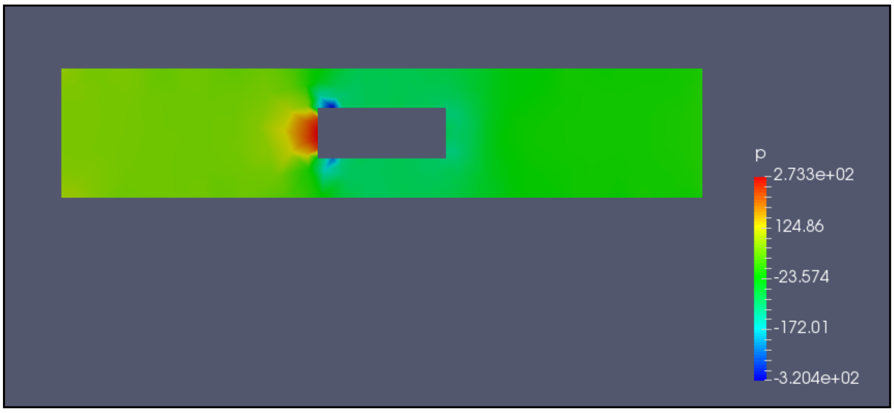
b. Click on the ‘slice’ button

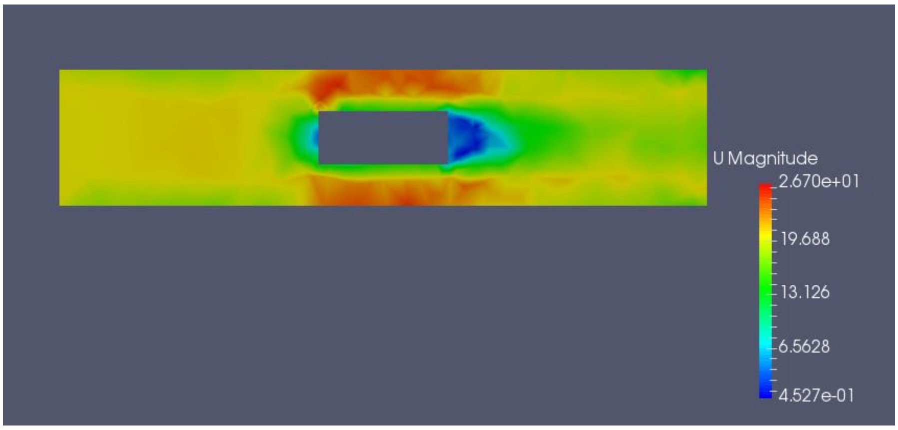
c. Select the direction in which you want to see the plane

d. To change the position of the plane to be viewed, click and drag the red boundary of the plane.

e. Click ‘Apply’.

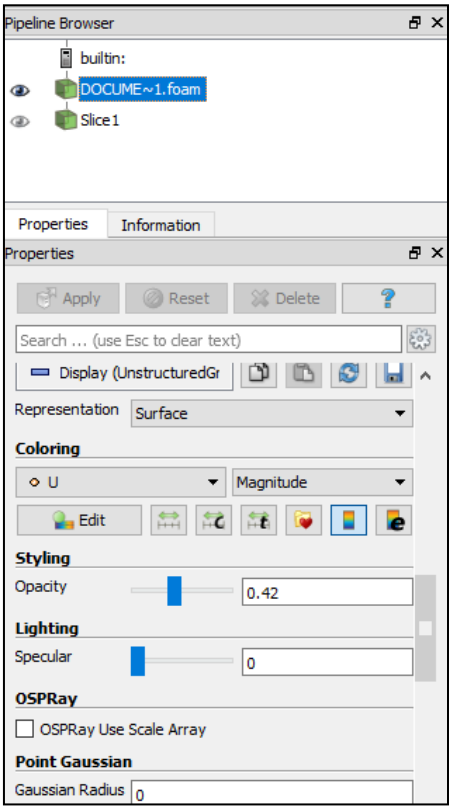
f. Default it will show pressure variation. To see the velocity profile, change ‘p’ to ‘U’ as shown in step(iii)

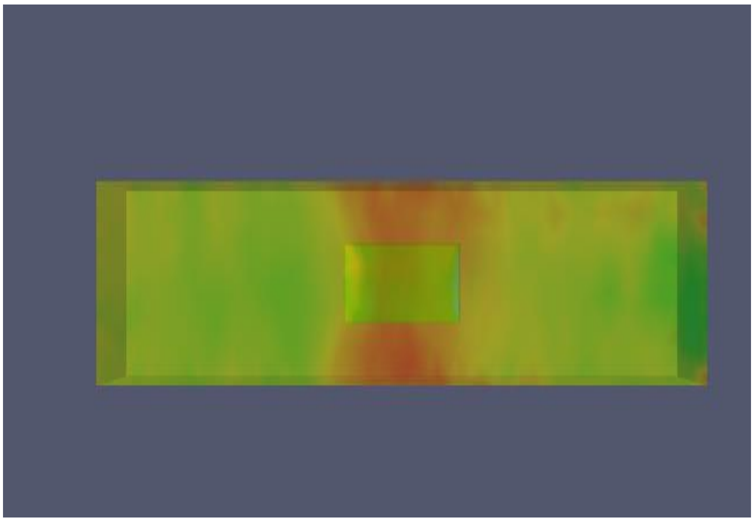




(vii) Applying 'stream tracer'

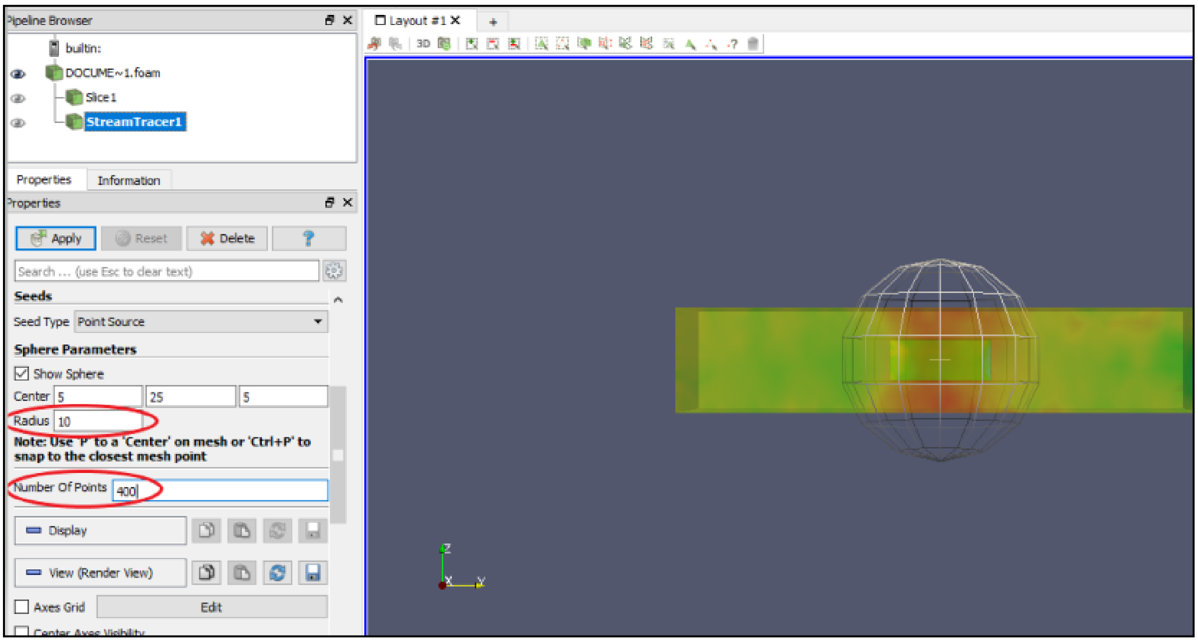
a. Select the box from 'pipeline browser' window and decrease the opacity of the box as shown below.



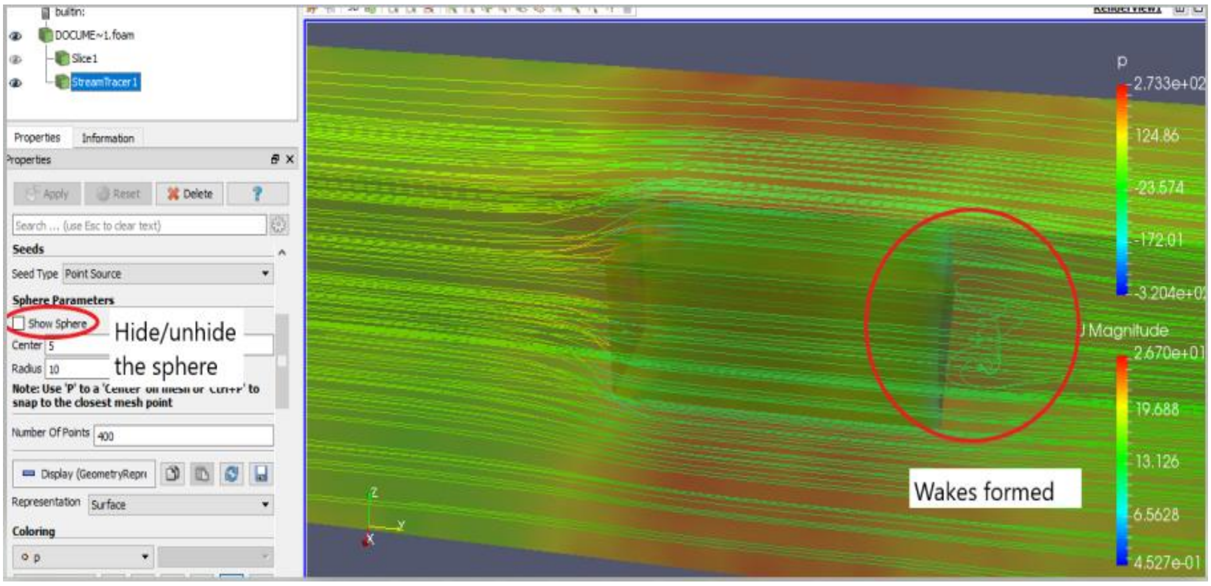


b. Click 'Stream tracer' button

c. Set the sphere around the object for which you want to see streamlines around. Change the radius of the sphere to completely surround the smaller box. Also change the number of points to see more streamlines



d. Click 'Apply'

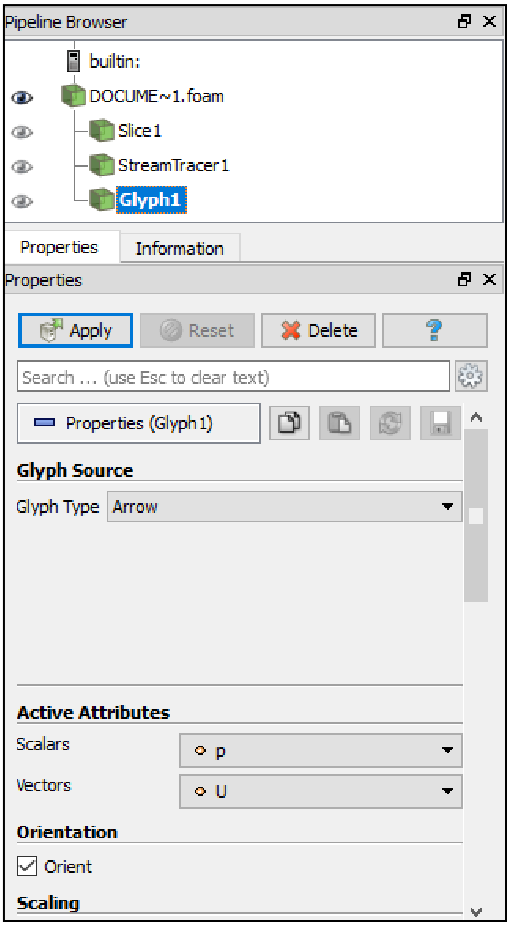


(viii) Applying 'Glyphs'

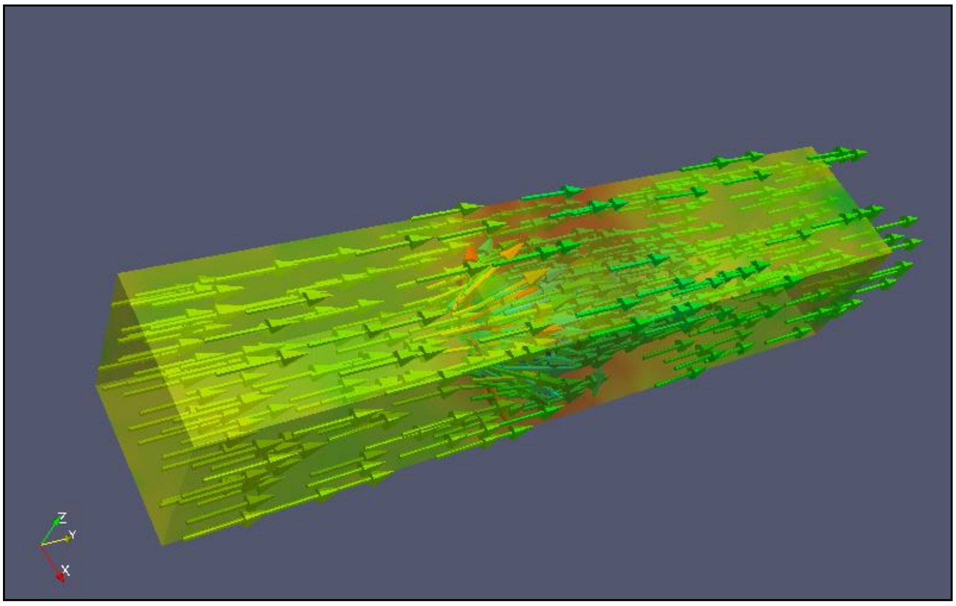
a. Select the box from 'pipeline browser' window and decrease the opacity of the box as shown below

b. Click the 'glyph' button

c. Set it to 'Arrow' type. Click 'Apply'.

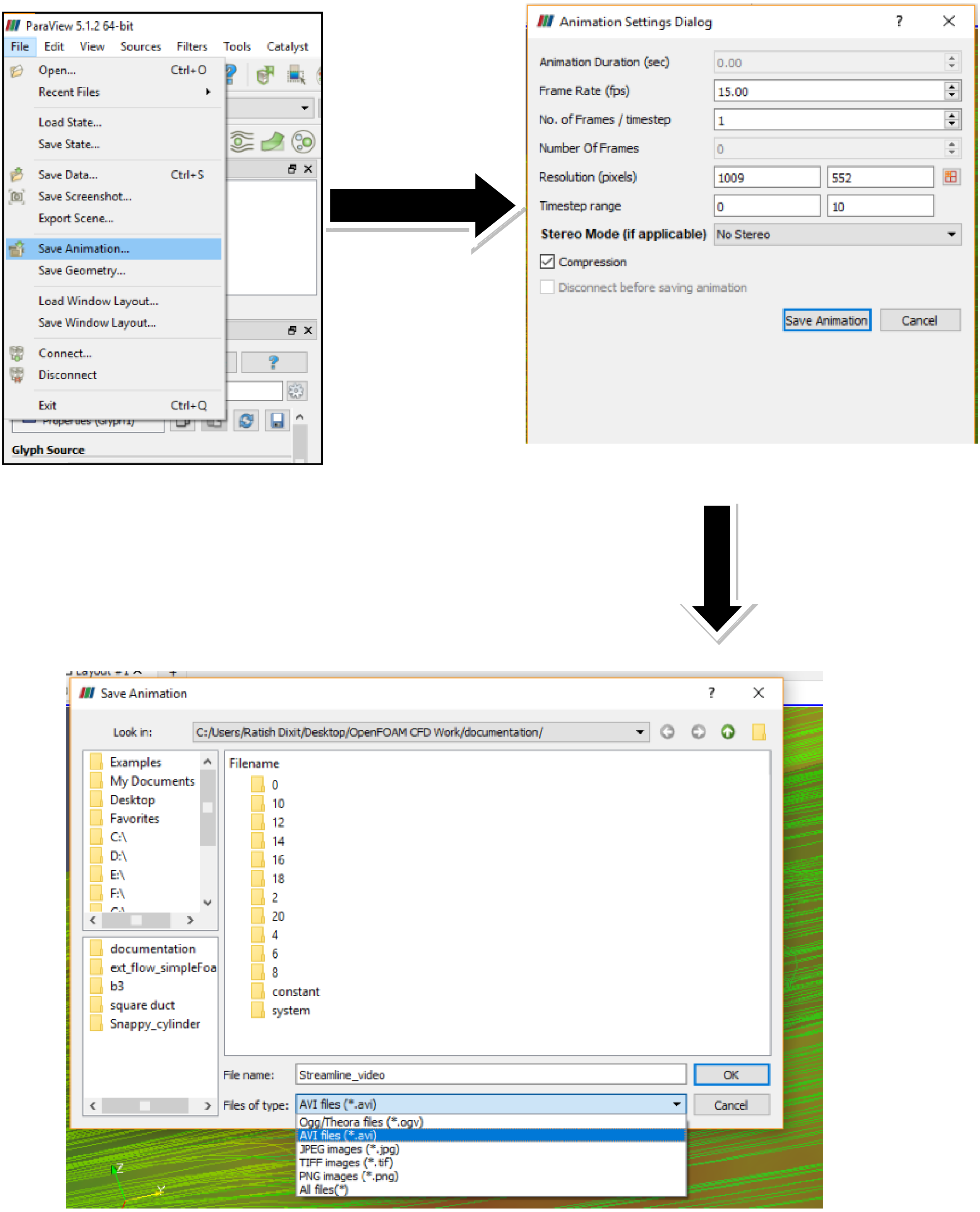


d. It will arrows in the direction from inlet to outlet



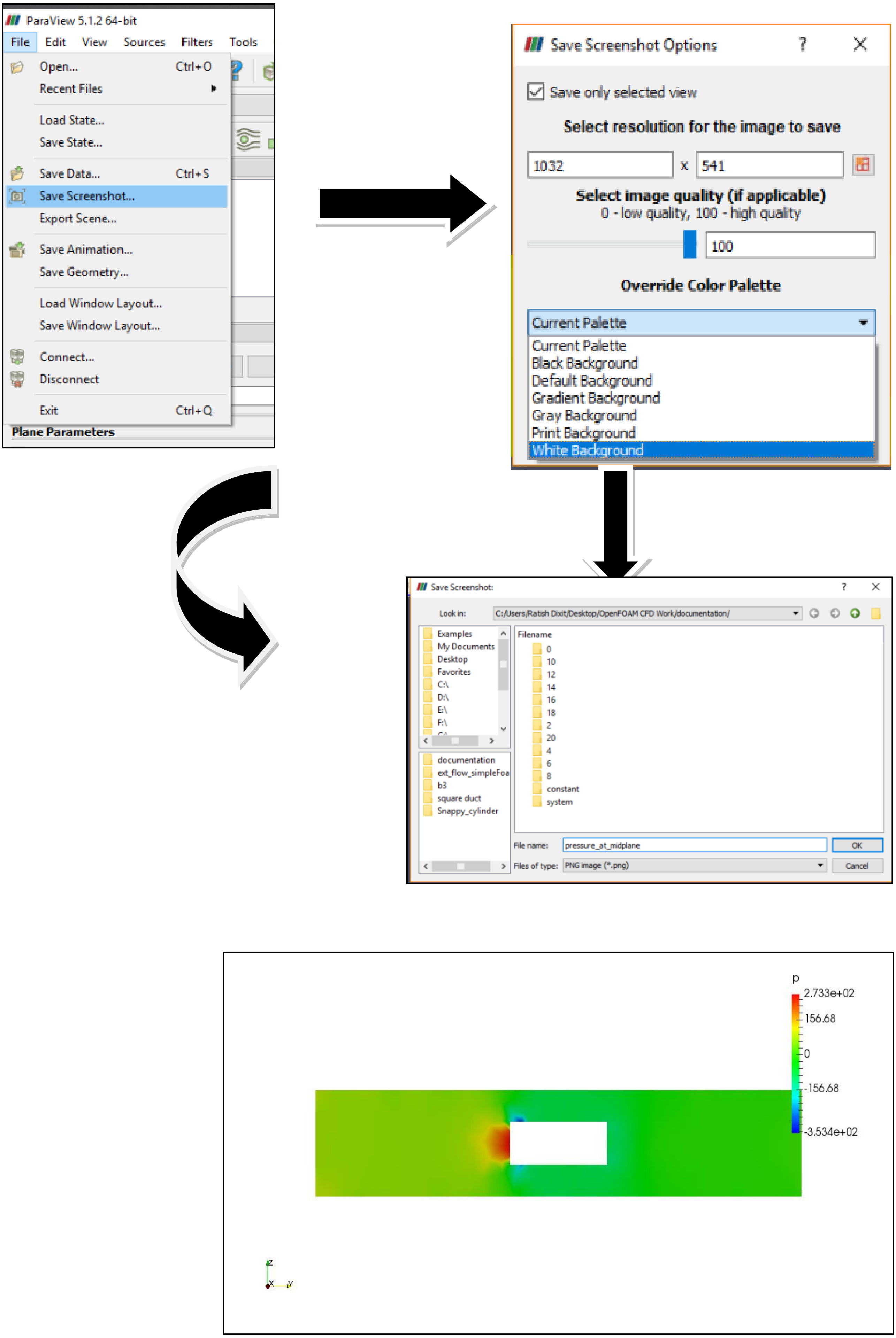
(ix) To save animation

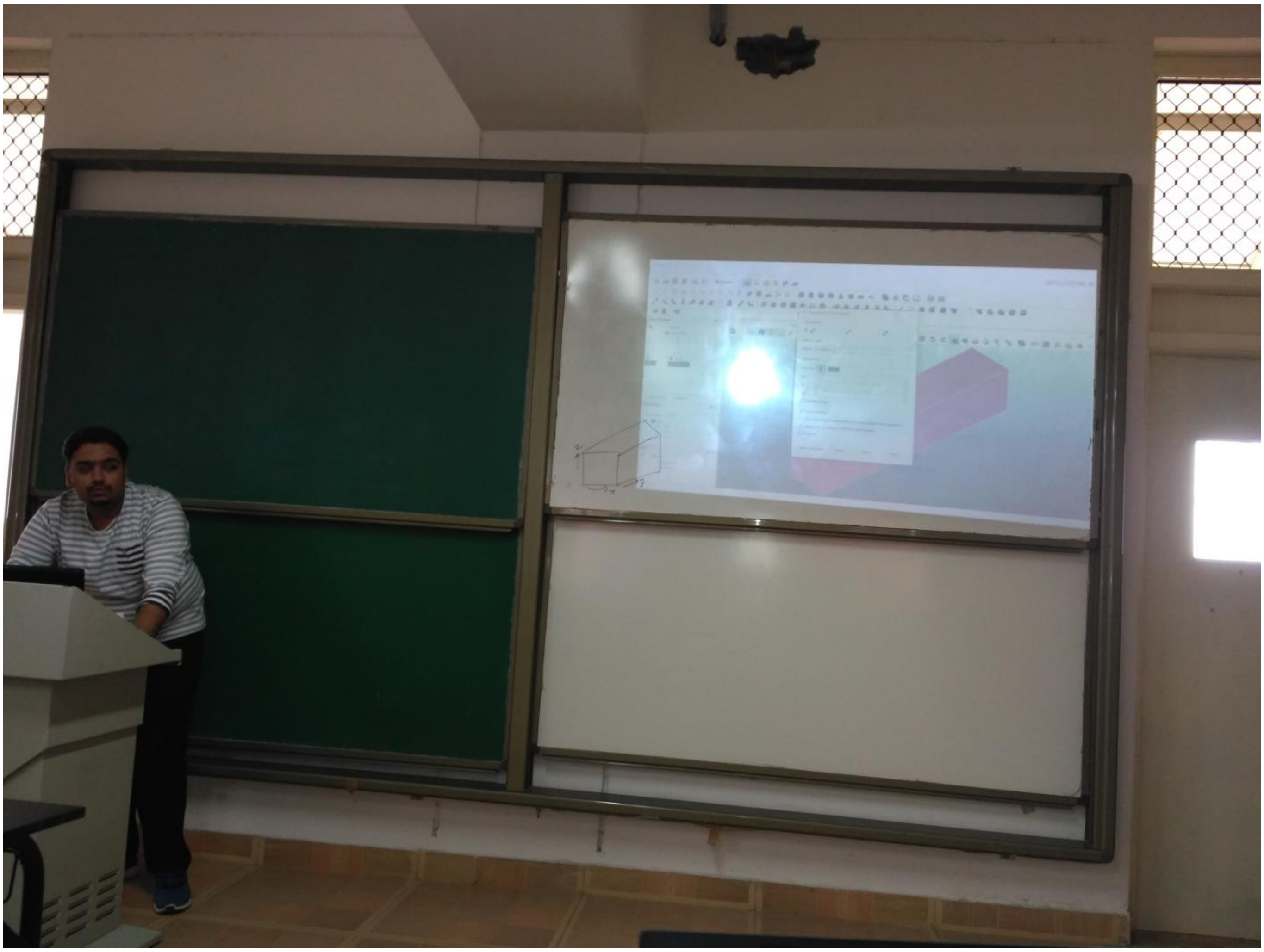
Click File → Save Animation → in the dialogue box that opens set desired frame rate→Save animation in .avi format.

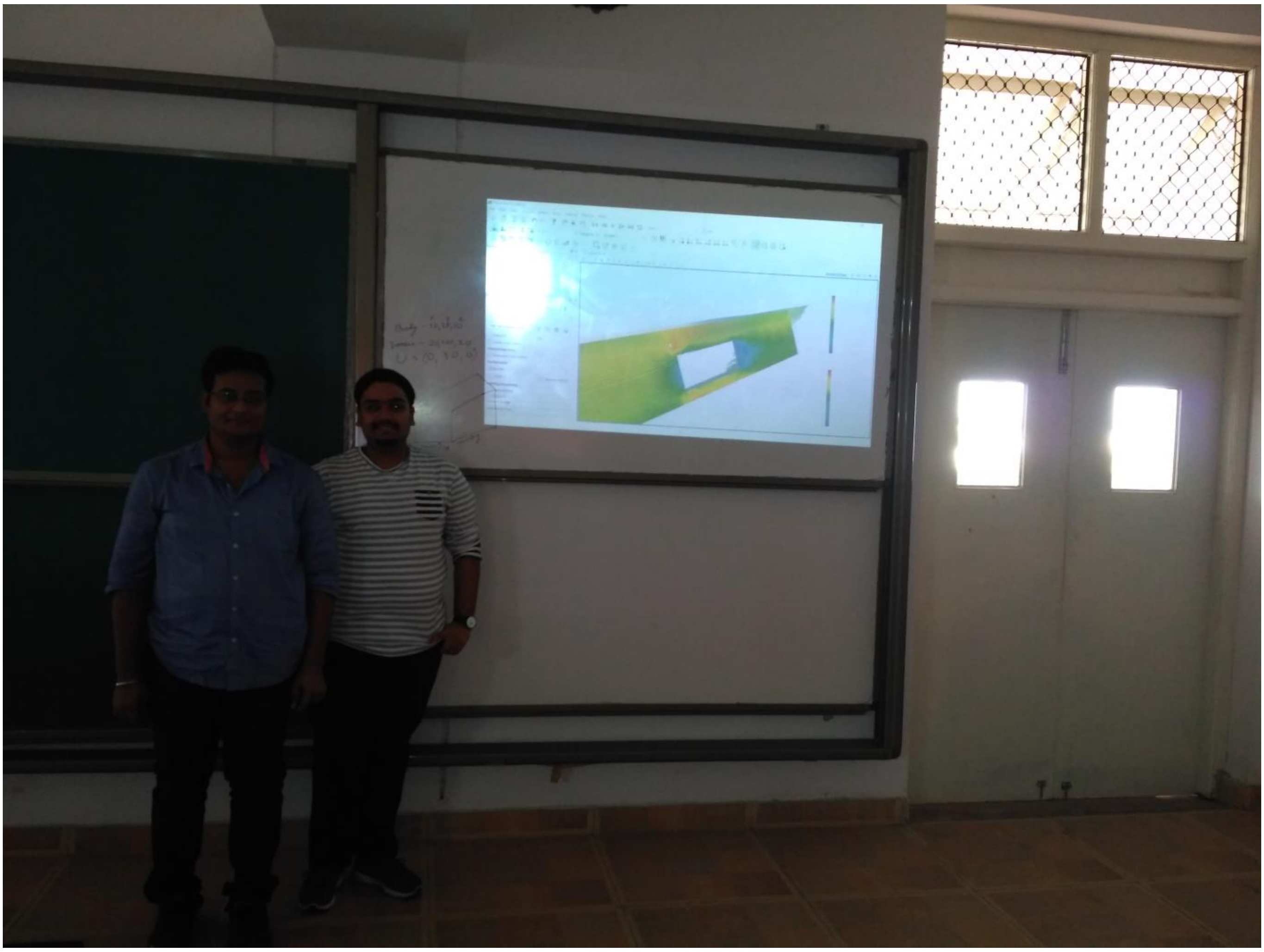


(x) To save screenshot

Click File → Save Screenshot →choose resolution and white background → Rename image → Save image


ParaView 5.1.2 64-bit
File
Edit
View
Sources
Filters
Tools
Open... Ctrl+O
Recent Files
Load State...
Save State...
Save Data... Ctrl+S
Save Screenshot...
Export Scene...
Save Animation...
Save Geometry...
Load Window Layout...
Save Window Layout...
Connect...
Disconnect
Exit Ctrl+Q
Plane Parameters
Save Screenshot Options
Save only selected view
Select resolution for the image to save
1032
x
541
Select image quality (if applicable)
0 - low quality, 100 - high quality
100
Override Color Palette
Current Palette
Current Palette
Black Background
Default Background
Gradient Background
Gray Background
Print Background
White Background
Save Screenshot:
Look in:
C:/Users/Ratish Dixit/Desktop/OpenFOAM CFD Work/documentation/
Examples
My Documents
Desktop
Favorites
C:\
D:\
E:\
F:\
documentation
ext_flow_simpleFoa
b3
square duct
Snappy_cylinder
Filename
0
10
12
14
16
18
2
20
4
6
8
constant
system
File name:
pressure_at_midplane
OK
Files of type:
PNG image (*.png)
Cancel
p
2.733e+02
156.68
0
-156.68
-3.534e+02
Z
X
Y





19/12/18. OpenFOAM CFD Club

Ratish Dixit. A seminar on external flow using openfoam and salome.

Sl. no.	Name.	Reg. no	Signature
1.	Prateek Srivastava	18BME1079	Srivastava
2	Chamala Vaishnavi	18BME1008	J.Ch[illegible]
3.	Meethaveerappan	18BME1083	AL. Muthu
4.	Atul kumar	18MCD1004	Atul
5.	Abhin. m. V	18MCD1041	[illegible]
6.	Nijon josu	18MCD1040	[illegible]
7	S. Bhagyenath	18MCD1037	[illegible]
8	Mathew George	18MCD1023	[illegible]
9	M. Devesh	16BME1139	[illegible]
10.	Akhil Kalaga	16BME1211	Vsakhil
11.	Para Akhil	16BME1204	P. Akhil
12.	Antim Gupta	16BME1163	Antim
13.	Ratish Dixit	16BME1093	Ratish Dixit