EXAMPLES OF SETUP AND SIMULATION USING ALTERNATIVE CFD SOLVERS

OpenFOAM

Solidworks Flow Simulation

COMSOL Multiphysics

OpenFOAM Example 1

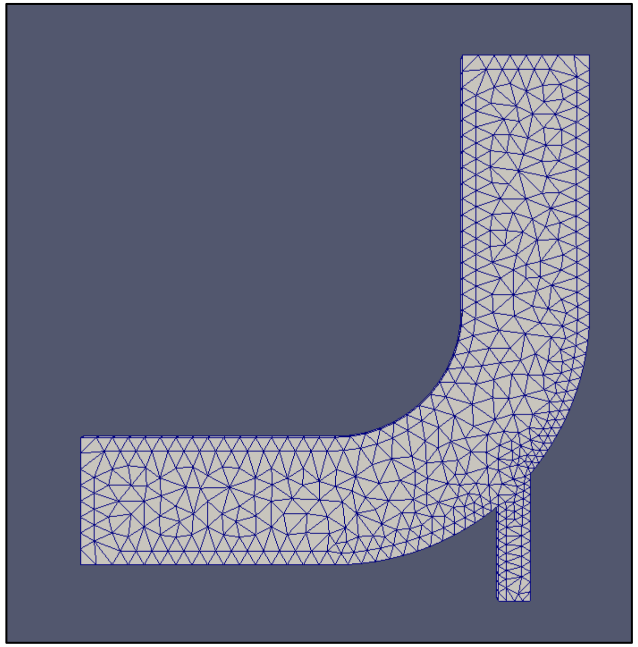
Aditya Vipradas

Using OpenFOAM for CFD

Aditya Vipradas

Platform: Linux
Solver: icoFoam
Model: Transient incompressible laminar flow,
Viewer: Paraview

Mesh: Generated in Fluent imported (fluentMeshToFoam)



Boundary conditions setting in text editor

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

internalField   uniform (0 0 0);

boundaryField
{
    wall-4
    {
        type            noSlip;
    }

    velocity-inlet-5
    {
        type            fixedValue;
        value           uniform (1 0 0);
    }

    velocity-inlet-6
    {
        type            fixedValue;
        value           uniform (0 3 0);
    }

    pressure-outlet-7
    {
        type            zeroGradient;
    }
```

Material assignment

```
// * * * * * * * * * * * * * * * * * * * * * * * * *

nu              [0 2 -1 0 0 0 0] 0.01;

// ******************************************
```

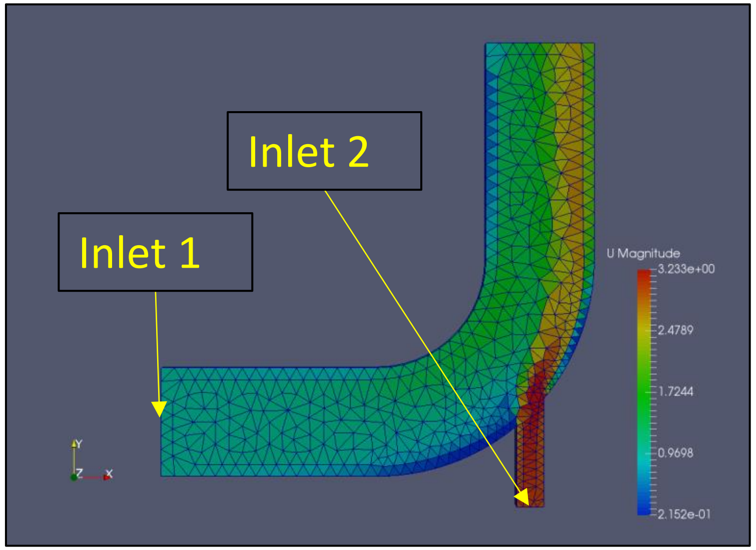
Time step definition

```
application     icoFoam;

startFrom       latestTime;

startTime       0;

stopAt          endTime;

endTime         75;

deltaT          0.05;

writeControl    timeStep;

writeInterval   20;
```

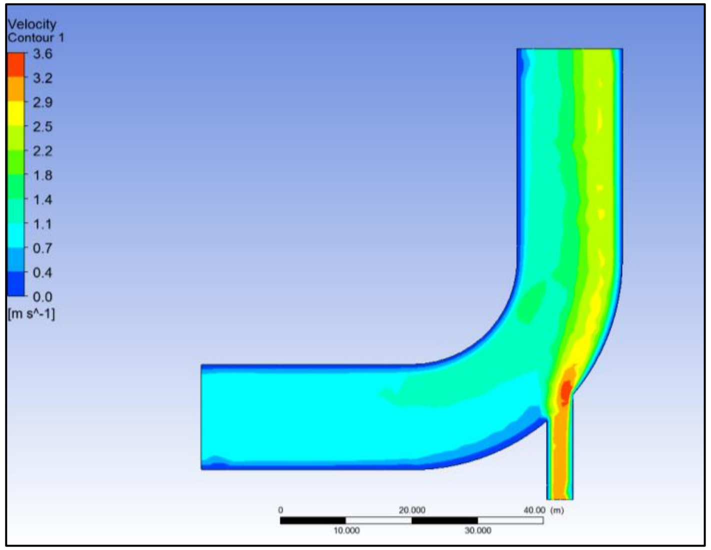
Comparison of OpenFOAM and ANSYS-Fluent Simulation Results

2D elbow with a fluid (nu = 0.01 m^2/s) flowing with velocity of 1m/s in inlet 1 and 3m/s in inlet 2. Transient laminar analysis is performed in OpenFOAM and ANSYS-Fluent using PISO scheme

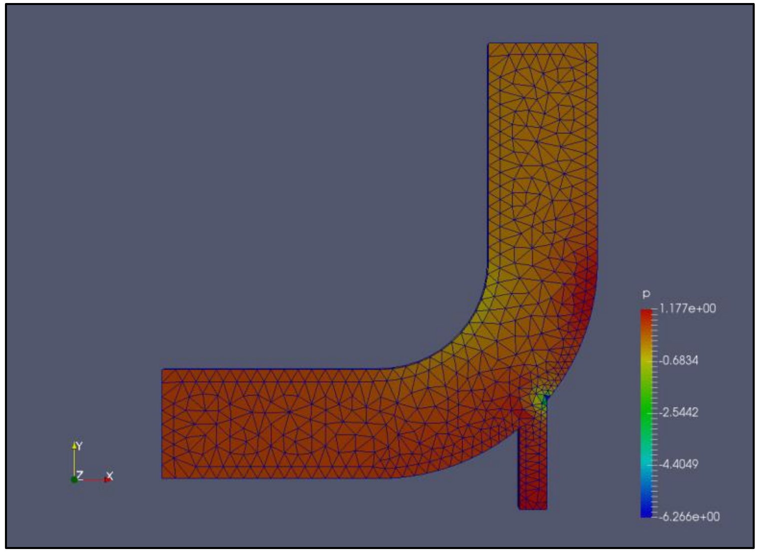
OpenFOAM velocity plot at 75s



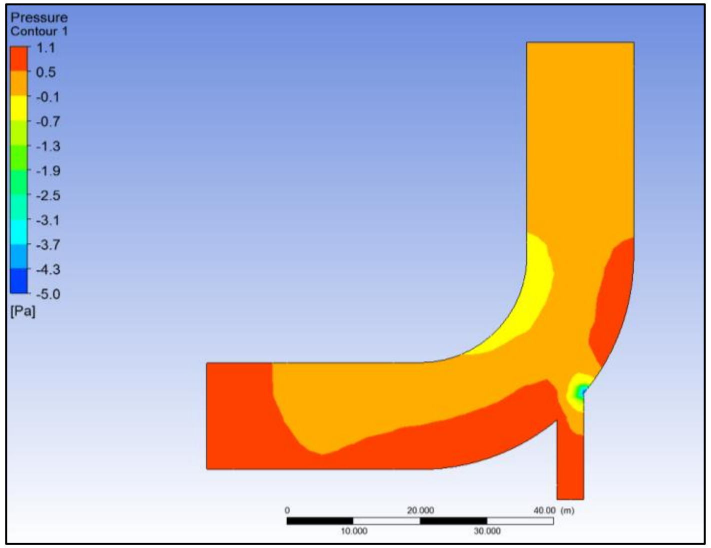
Fluent velocity plot at 75s



OpenFOAM pressure plot at 75s



Fluent pressure plot at 75s



The results generated by OpenFOAM and Fluent are in good agreement.

OpenFOAM Example 2

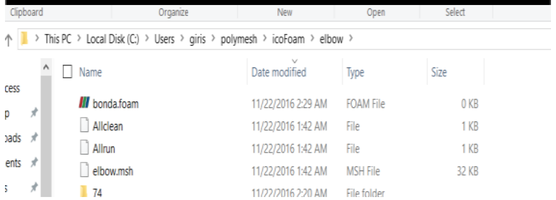
Girish Nigamanth Raghunathan

Openfoam

Girish Nigamanth Raghunathan

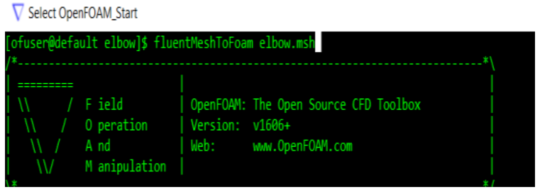
1. Creating Geometry

This can be done using a third party software like ICEM, SolidWorks or even ANSYS. The final .msh file needs to be saved into the working directory. Boundaries also can be specified in the software for ease of use.



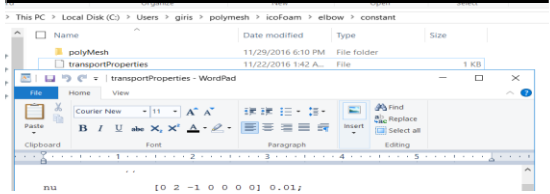
2. Meshing

This can be done using OpenFOAM itself using the .msh file created from third party software. Following command needs to be used to do the mesh. Refinement of mesh can also be done using a suitable command.



3. Problem Setup - Material Properties

Material Properties can be specified under transportproperties, in this case kinematic viscosity is specified and rest are taken as default values.



3. Problem Setup - Boundary Condition

Boundary conditions can be specified from the folder 0. The file p specifies pressure at boundaries which are 0 in all of them. The file U specifying velocity is adjusted accordingly in vector form.

```
velocity-inlet-5
{
    type            fixedValue;
    value           uniform (0.4 0 0);
}

velocity-inlet-6
{
    type            fixedValue;
    value           uniform (0 1.2 0);
}
```

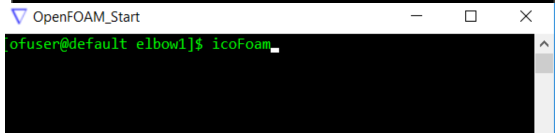
3. Problem Setup - Iterations, Time-step etc

For this the controldict file under system was opened and edited for the number of seconds to be computed and also the time step size for transient case.

```
application     icoFoam;
startFrom       latestTime;
startTime       0;
stopAt          endTime;
endTime         75;
deltaT          0.05;
writeControl    timeStep;
writeInterval   20;
purgeWrite      0;
```

4. Running and Post-Processing

For running the code, we should use the terminal to get into the folder then type icoFOAM. For post-processing a software called paraview is used to open the controldict file which then is used to display the results of the simulation.

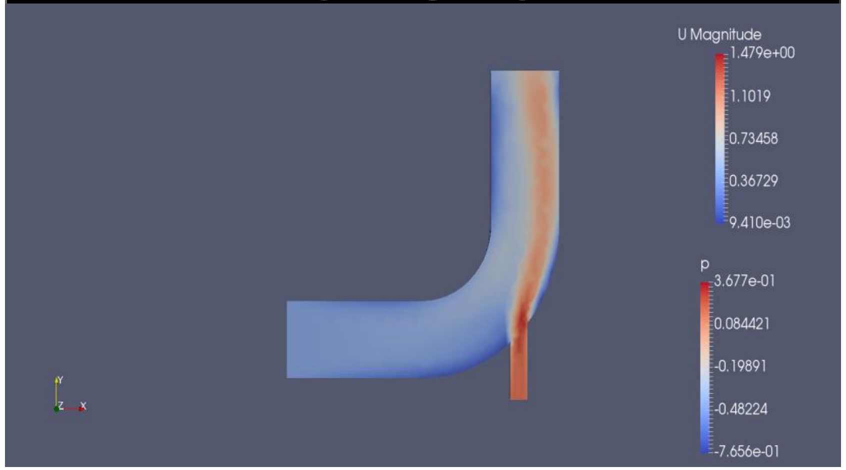


Openfoam: Result of Simulation

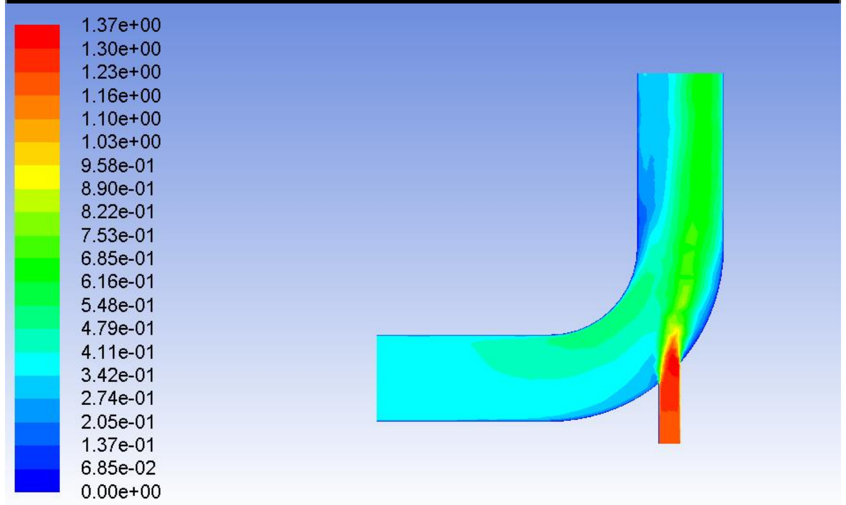
1. Simulation result as in OpenFOAM

```
Time = 75

Courant Number mean: 0.0323681 max: 0.205575
smoothSolver:  Solving for Ux, Initial residual = 0.000135391, Final residual = 2.77072e-08, No Iterations 1
smoothSolver:  Solving for Uy, Initial residual = 9.08303e-05, Final residual = 2.02958e-08, No Iterations 1
DICPCG:  Solving for p, Initial residual = 0.000594351, Final residual = 2.28137e-05, No Iterations 8
DICPCG:  Solving for p, Initial residual = 0.000111398, Final residual = 4.75451e-06, No Iterations 54
DICPCG:  Solving for p, Initial residual = 2.3665e-05, Final residual = 1.08852e-06, No Iterations 7
time step continuity errors : sum local = 5.61354e-11, global = 6.04081e-12, cumulative = 6.38588e-09
DICPCG:  Solving for p, Initial residual = 1.18536e-05, Final residual = 8.58743e-07, No Iterations 6
DICPCG:  Solving for p, Initial residual = 2.60562e-06, Final residual = 7.05292e-07, No Iterations 1
DICPCG:  Solving for p, Initial residual = 7.78236e-07, Final residual = 7.78236e-07, No Iterations 0
time step continuity errors : sum local = 4.01337e-11, global = 7.28764e-12, cumulative = 6.39316e-09
ExecutionTime = 20.61 s  ClockTime = 42 s

End

[ofuser@default elbow]$
```

2. Contour Plot of Velocity Profile after 75s (~ steady state)



3. Contour Plot of Velocity at Steady State from Fluent



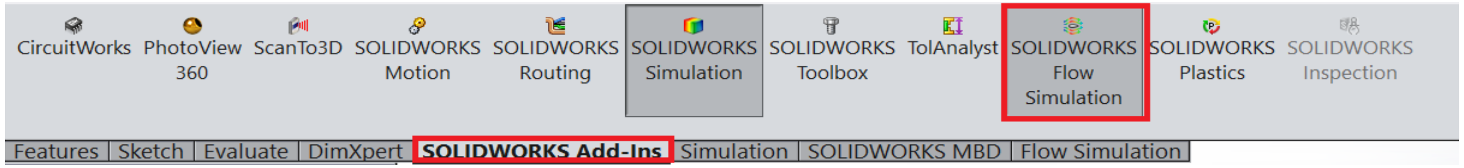
Solidworks Flow Simulation Example 1

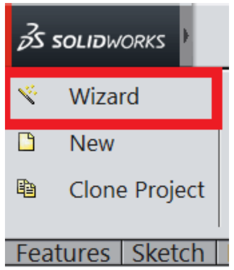
Mohammed Mehdi

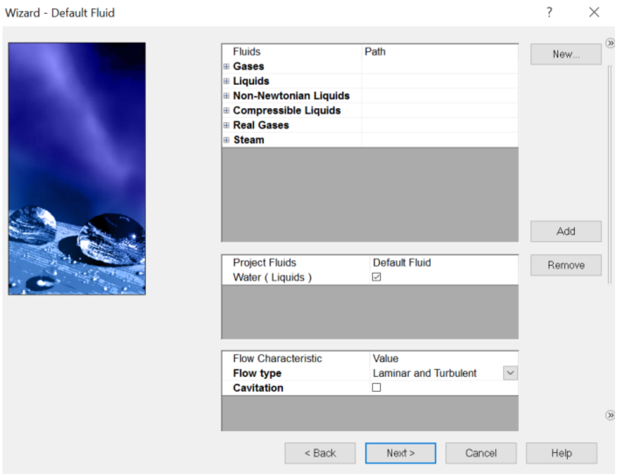
SOLIDWORKS Flow Simulation: Setup

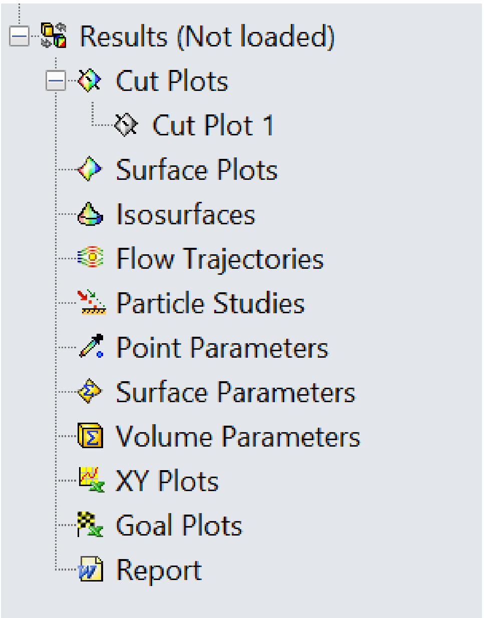
Mohammed Mehdi

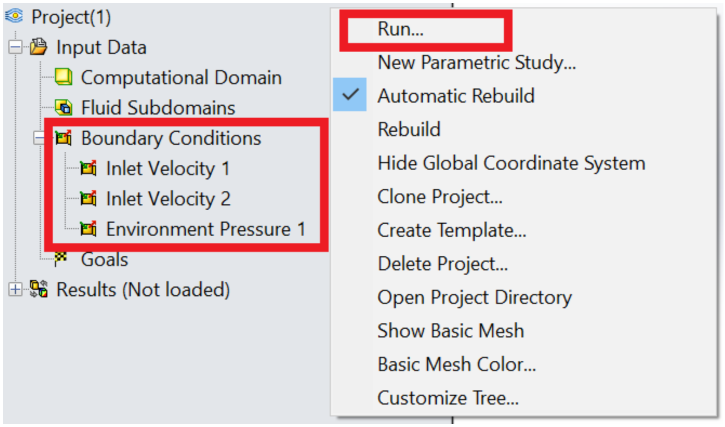
- Simply and quick setup:
  - Solidworks add-ins
  - Solidworrks flow simulation
  - Wizard
  - Only thing that needs to be done in wizard is:
    - Choosing fluid
    - Choosing model, although default will work for any simulation
  - Specify boundary conditions
    - Do this by right clicking and pressing “add boundary condition” then selecting surfaces
  - Right click on “project” and click “Run…”
  - To view results expand the “Results” menu and you will be able to choose from a variety of post-processing tools
- Mesh automatically generated
- Models limited to turbulent or laminar or both



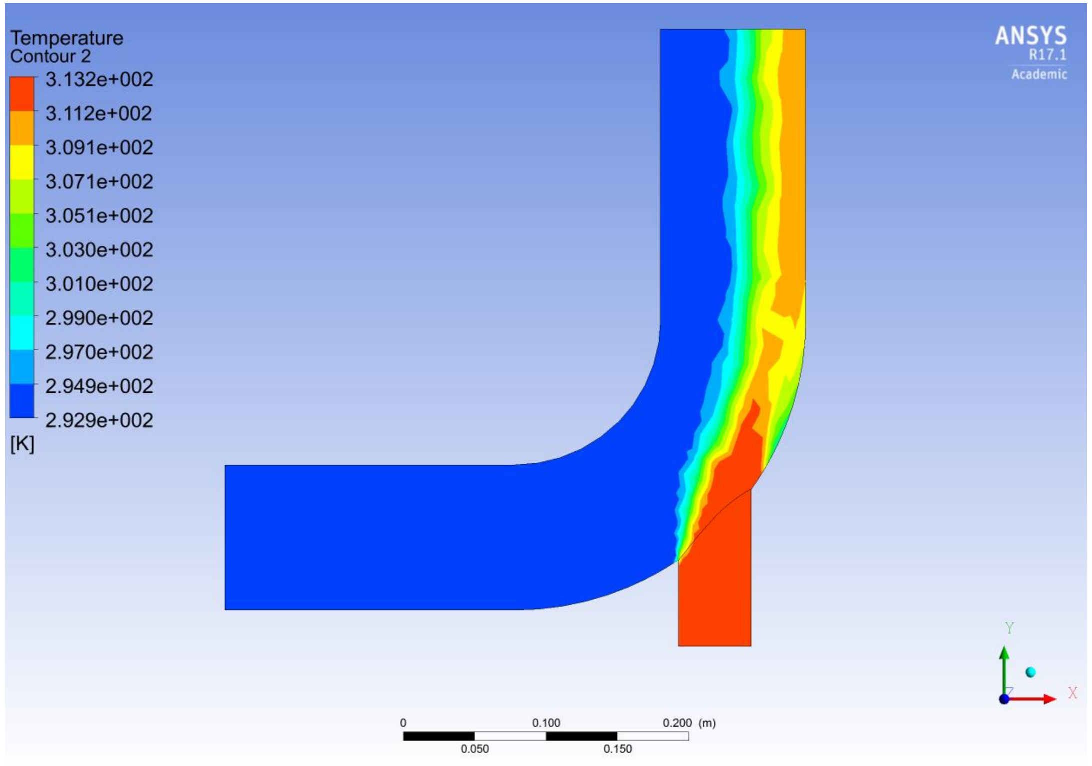


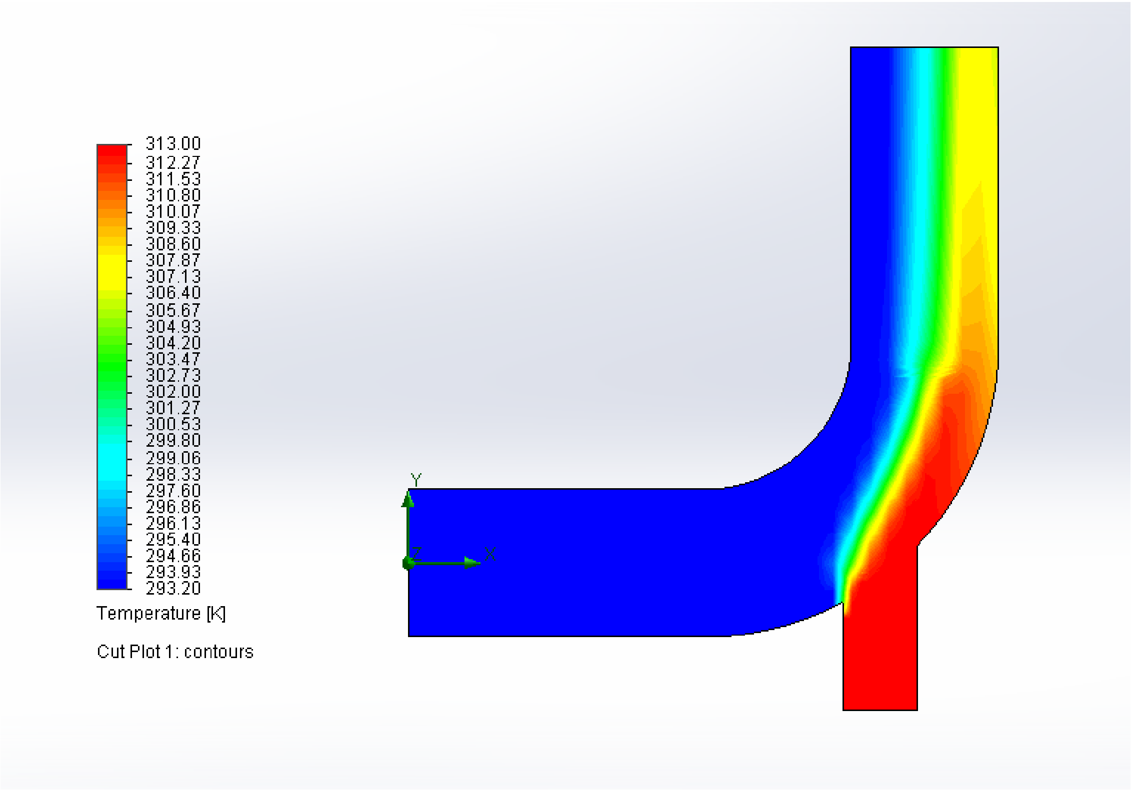






SOLIDWORKS Flow Simulation: Results and conclusion





COMSOL Multiphysics Example 1

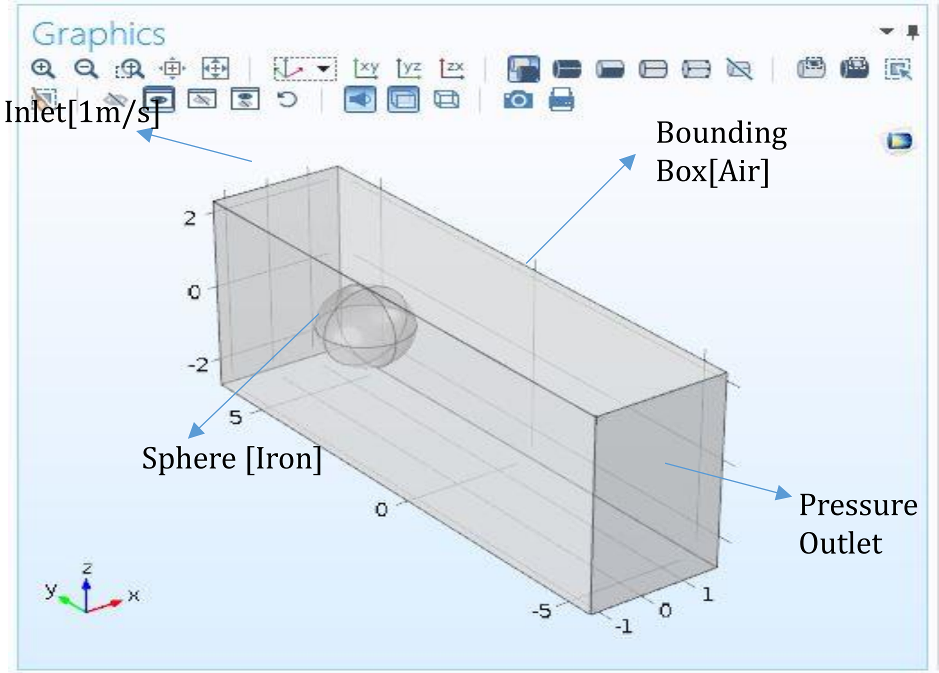
Sujal Tipnis

3D Simulation using COMSOL Multiphysics 5.2

Sujal Tipnis

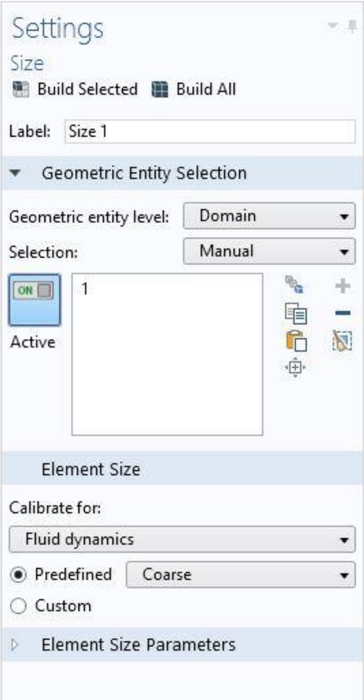
1. Geometry and Boundary Conditions

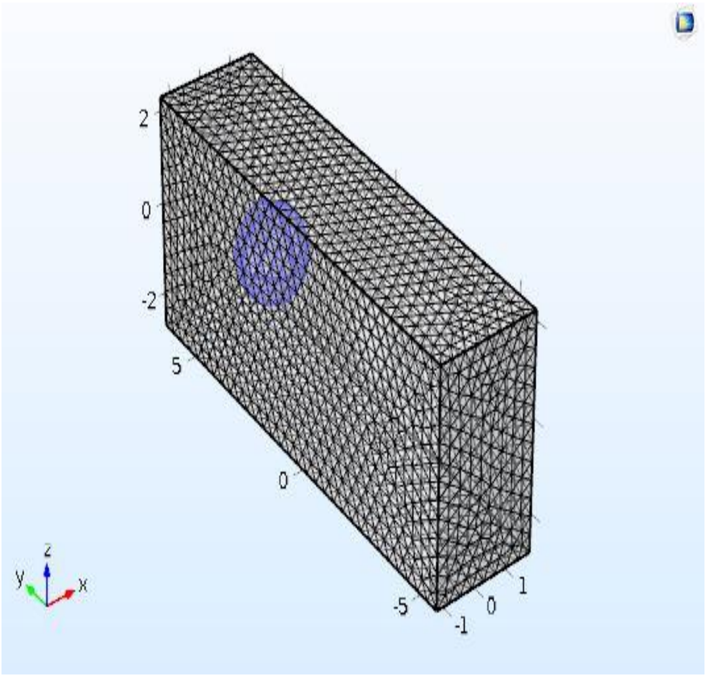
- Ran the simulation on Microsoft Windows used
- Used primitives to construct geometry



2. Mesh

- Two Domains
- Free Tetrahedral Mesh
- Calibrated for Fluid Dynamics

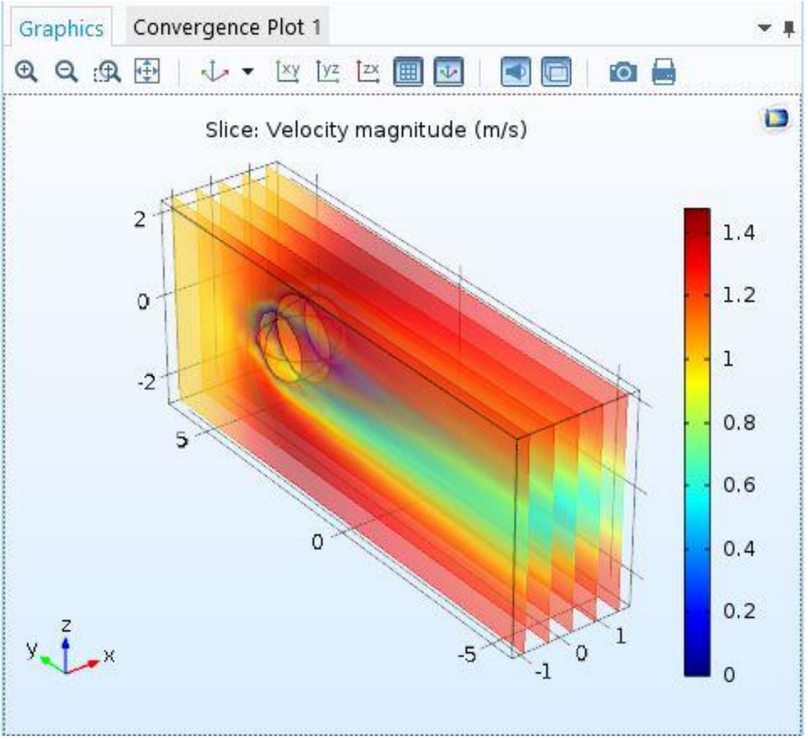


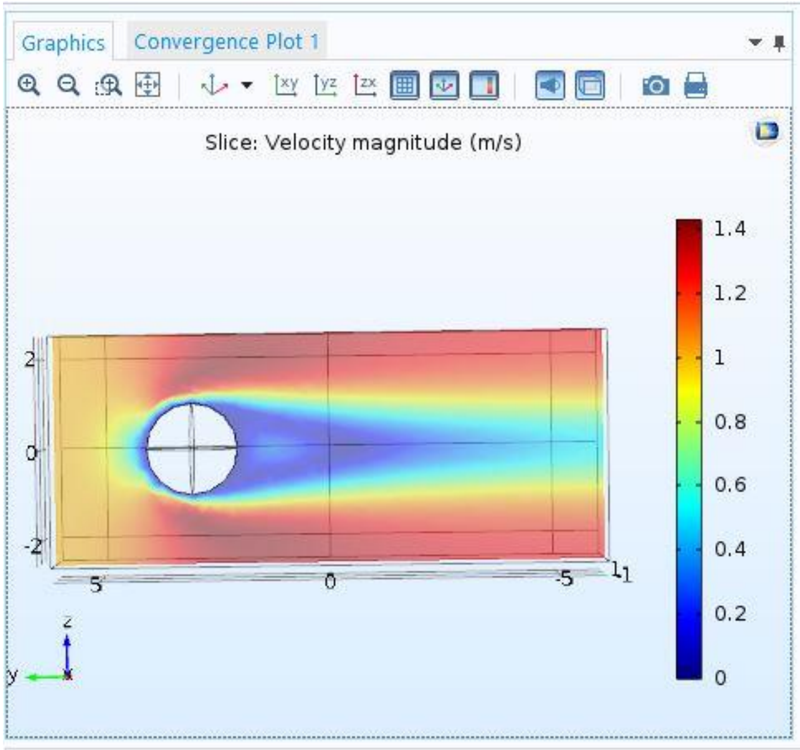


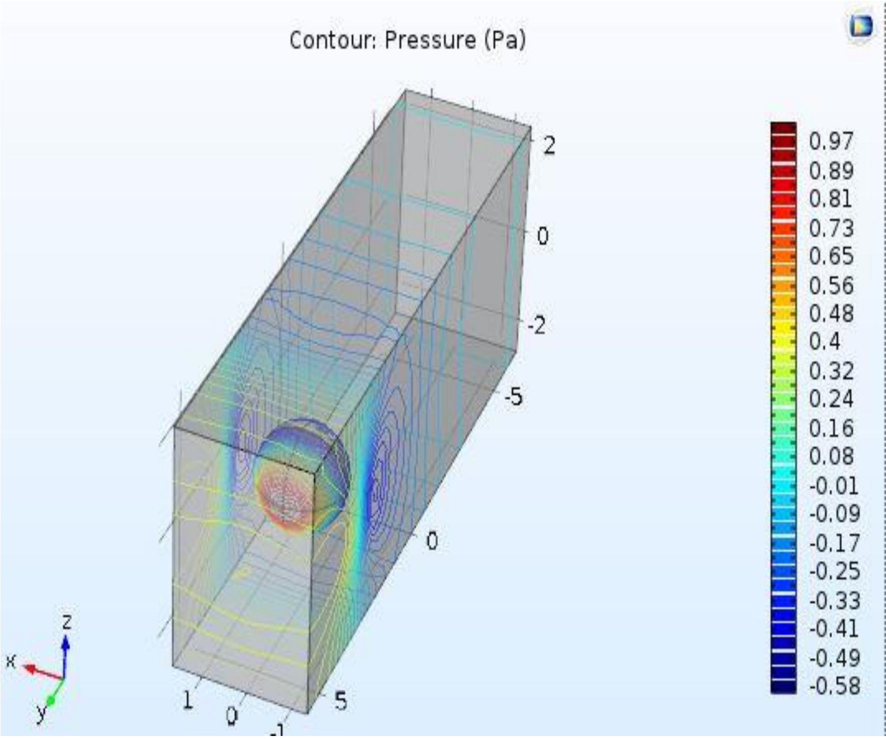
3. Setup

- Laminar Incompressible Flow
- Bounding Box Walls were set as No Slip
- Inlet: Normal Inflow Velocity
- Outlet: Pressure Outlet [Backflow supressed]

COMSOL Multiphysics 5.2: Simulation Results







- Simulation is similar to the Task 1 of Project 3 [3D Model is used here instead, with varying dimensions]
- Velocity and Pressure Contours suggest that the Reynolds number is very low.
- Setting up the simulation is easier than that in FLUENT, mainly due to all components of simulation being incorporated into one.

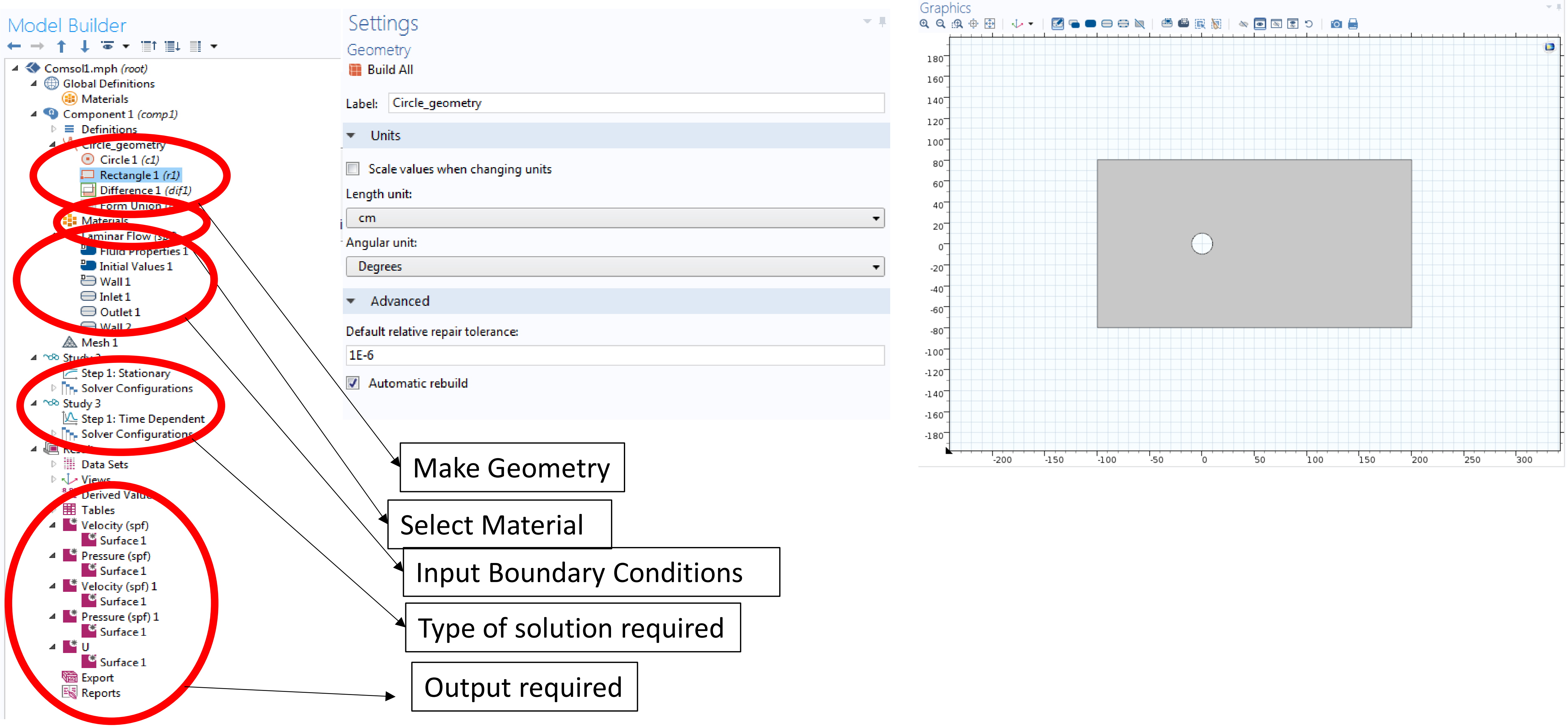
COMSOL Multiphysics Example 2

Vijay Jangid

COMSOL Multiphysics

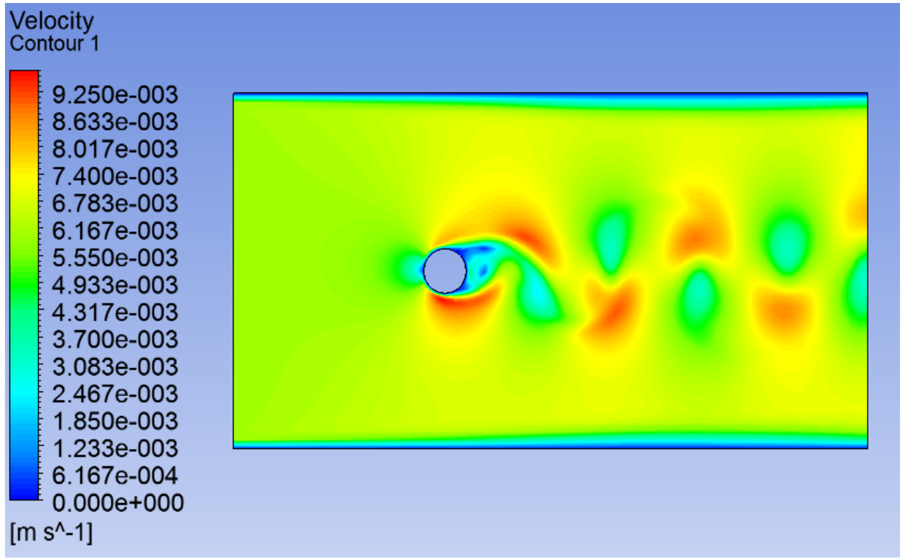
Vijay Jangid

Procedure followed for analysis:

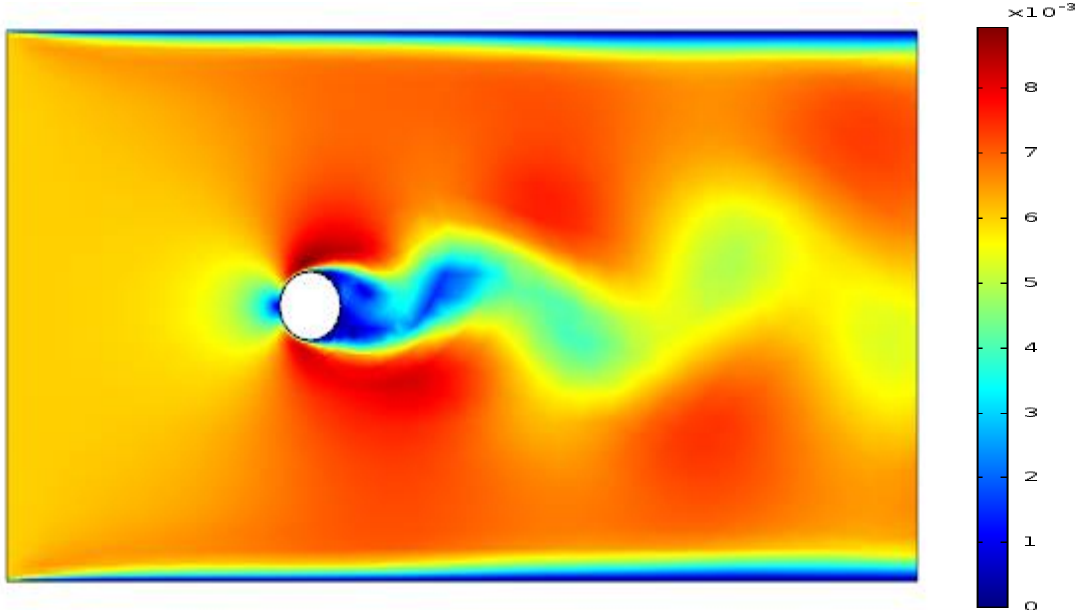


COMSOL Multiphysics: Simulation Results

Ex: Project 3: Task 1-a



Ansys Solution



Comsol Solution

The two solutions look very similar. The maximum velocity reached in the two solutions is also almost same.

Vijay Jangid