

# 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)

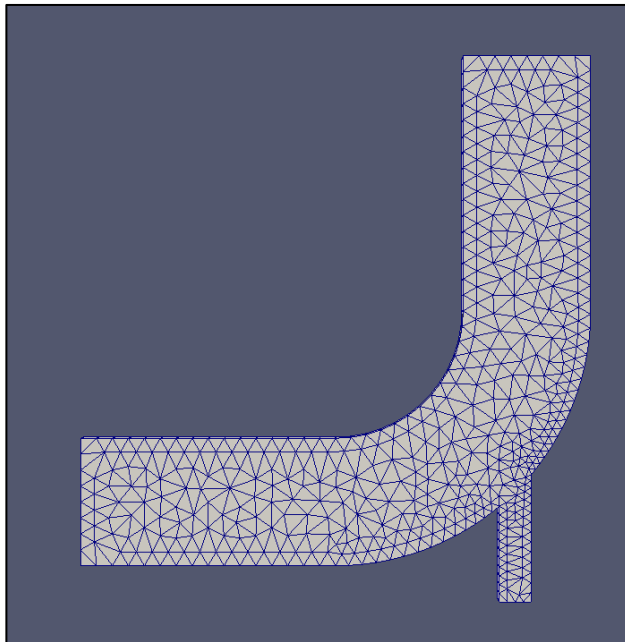
**Boundary conditions setting in  
text editor**

**Material assignment**

```
// *****  
nu          [0 2 -1 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;
```

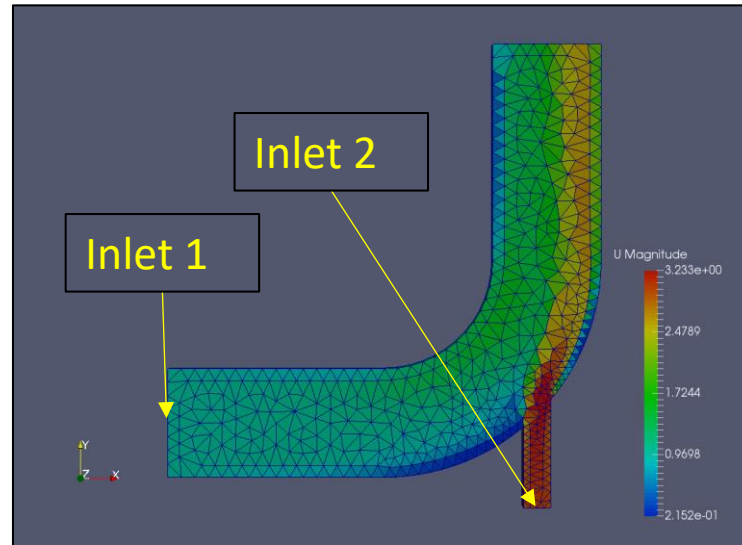


```
dimensions      [0 1 -1 0 0 0];  
internalField   uniform (0 0 0);  
boundaryField  
{  
  wall-4  
  {  
    type        noSlip;  
  }  
  velocity-inlet-5  
  {  
    type        fixedValue;  
    value       uniform (1 0 0);  
  }  
  velocity-inlet-6  
  {  
    type        fixedValue;  
    value       uniform (0 3 0);  
  }  
  pressure-outlet-7  
  {  
    type        zeroGradient;  
  }  
}
```

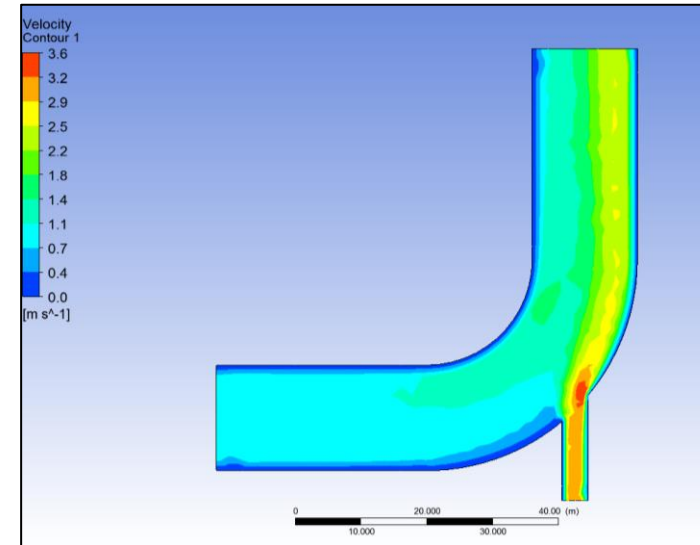
# Comparison of OpenFOAM and ANSYS-Fluent Simulation Results

2D elbow with a fluid ( $\nu = 0.01 \text{ m}^2/\text{s}$ ) 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

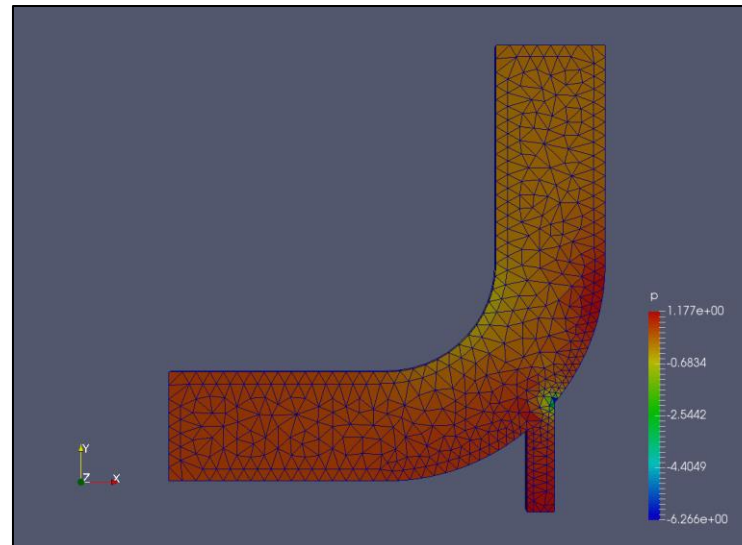
OpenFOAM velocity plot at 75s



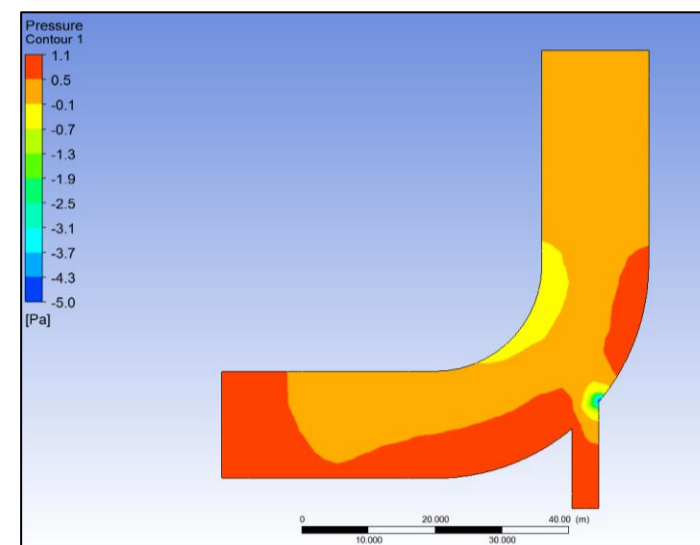
Fluent velocity plot at 75s



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.

Name	Date modified	Type	Size
bonds.foam	11/22/2016 2:29 AM	FOAM File	0 KB
Allclean	11/22/2016 1:42 AM	File	1 KB
Allrun	11/22/2016 1:42 AM	File	1 KB
elbow.msh	11/22/2016 1:42 AM	MSH File	32 KB
74	11/22/2016 >:0 AM	File folder	

## 2. Meshing

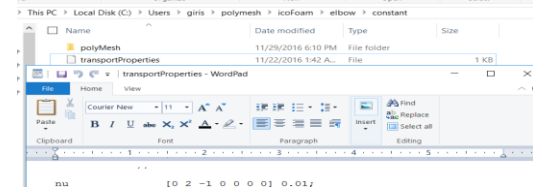
This can be done using OpenFOAM itself using the .msh file created from third party software. Following command needs to be used to do the mesh. Refinement of mesh can also be done using a suitable command.

Select OpenFOAM\_Start

```
[ofuser@default elbow]$ fluentMeshToFoam elbow.msh
=====
// / F ield      OpenFOAM: The Open Source CFD Toolbox
// / O peration  Version: v1606+
// / A nd        Web: www.OpenFOAM.com
// / M anipulation
```

## 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 o. 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);
}
```

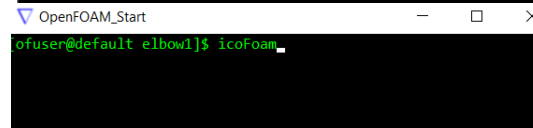
## 3. Problem Setup - Iterations, Time-step etc

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

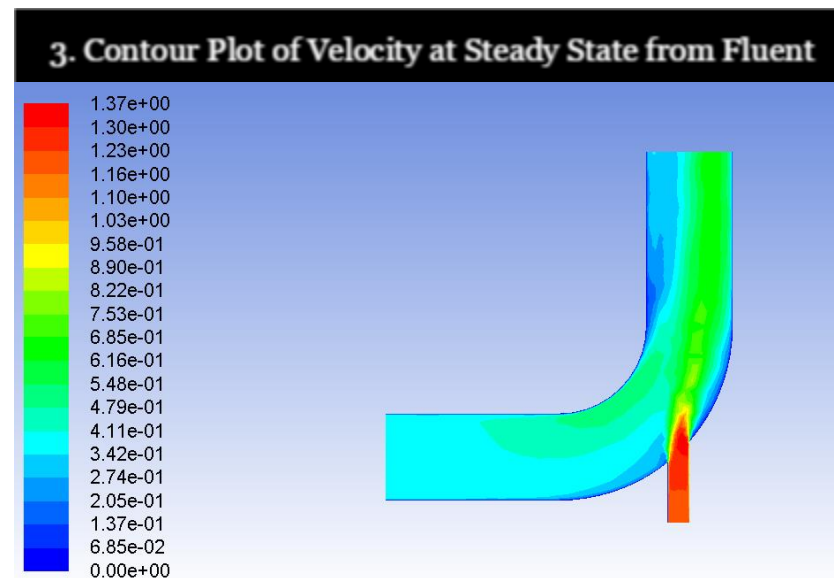
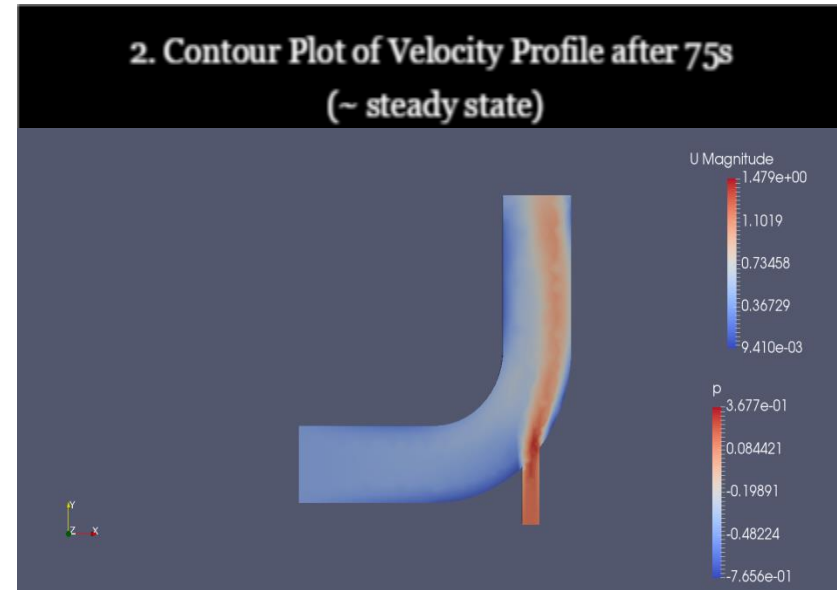
```
1. Simulation result as in OpenFOAM

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]$
```



# Solidworks Flow Simulation Example 1

Mohammed Mehdi



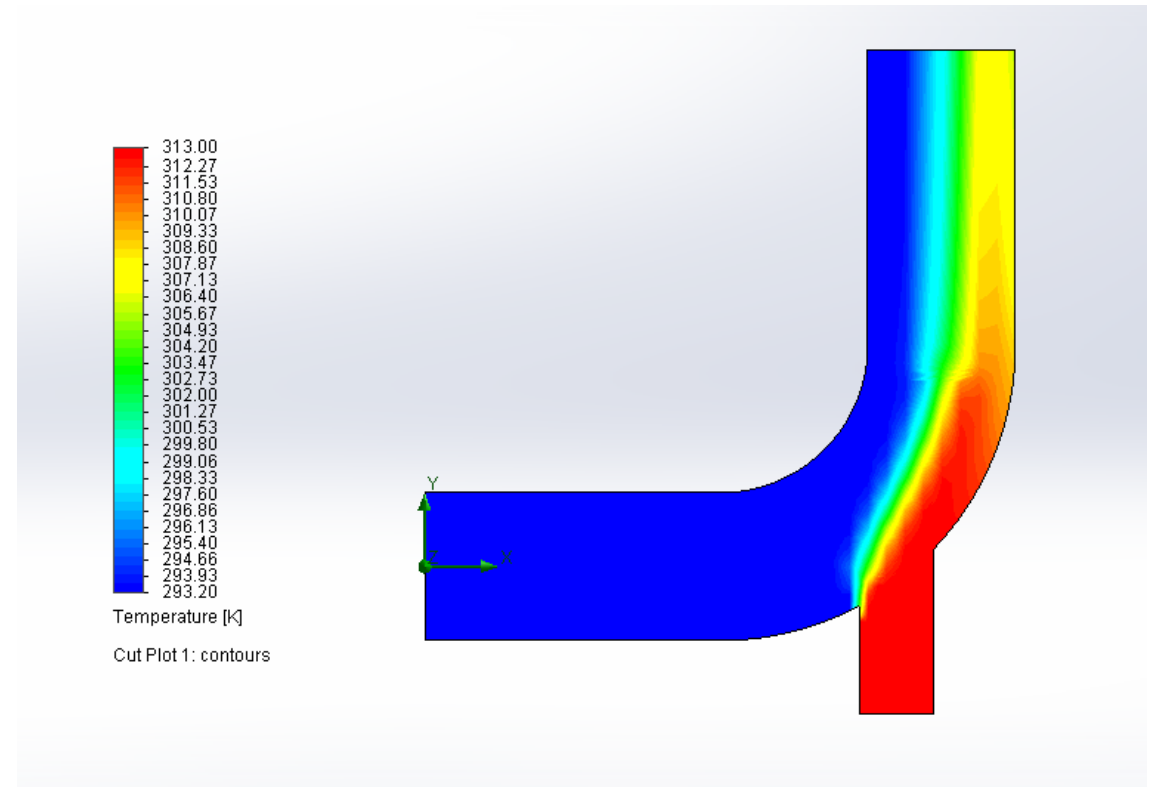
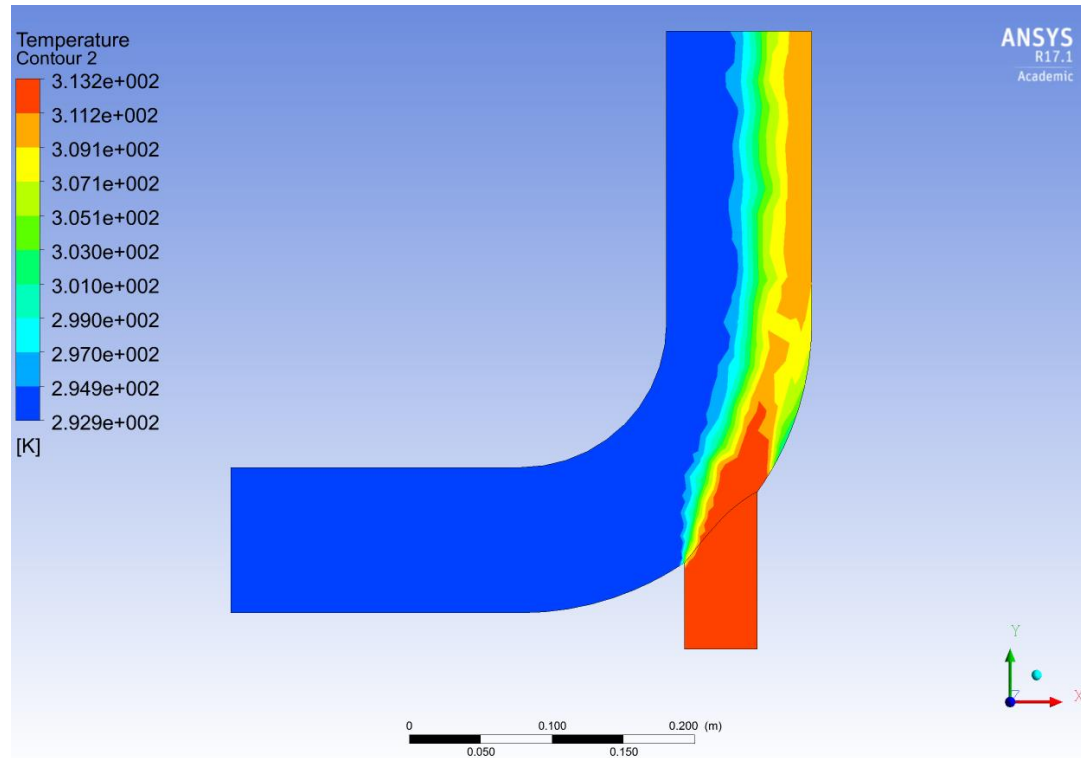
# SOLIDWORKS Flow Simulation: Setup

Mohammed Mehdi

- Simply and quick setup:
  - Solidworks add-ins
  - Solidworks 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

The image displays the SolidWorks software interface during the Flow Simulation setup process. The top ribbon shows the 'SOLIDWORKS Flow Simulation' tab selected. Below the ribbon, the 'SOLIDWORKS Add-Ins' menu is open, with the 'Wizard' option highlighted. The 'Wizard - Default Fluid' dialog box is visible, showing a list of fluid types (Gases, Liquids, Non-Newtonian Liquids, Compressible Liquids, Real Gases, Steam) and a 'Default Fluid' section set to 'Water (Liquids)'. The 'Flow type' is set to 'Laminar and Turbulent'. The 'Project Fluids' section shows 'Water (Liquids)' as the default fluid. The 'Flow Characteristic' section shows 'Cavitation' as the selected value. The 'Results (Not loaded)' tree is visible on the right, showing a list of post-processing tools: Cut Plots, Surface Plots, Isosurfaces, Flow Trajectories, Particle Studies, Point Parameters, Surface Parameters, Volume Parameters, XY Plots, Goal Plots, and Report. The 'Project(1)' tree on the left shows the simulation setup, with 'Boundary Conditions' expanded to show 'Inlet Velocity 1', 'Inlet Velocity 2', and 'Environment Pressure 1'. The 'Run...' option is highlighted in the context menu.

# SOLIDWORKS Flow Simulation: Results and conclusion



# COMSOL Multiphysics Example 1

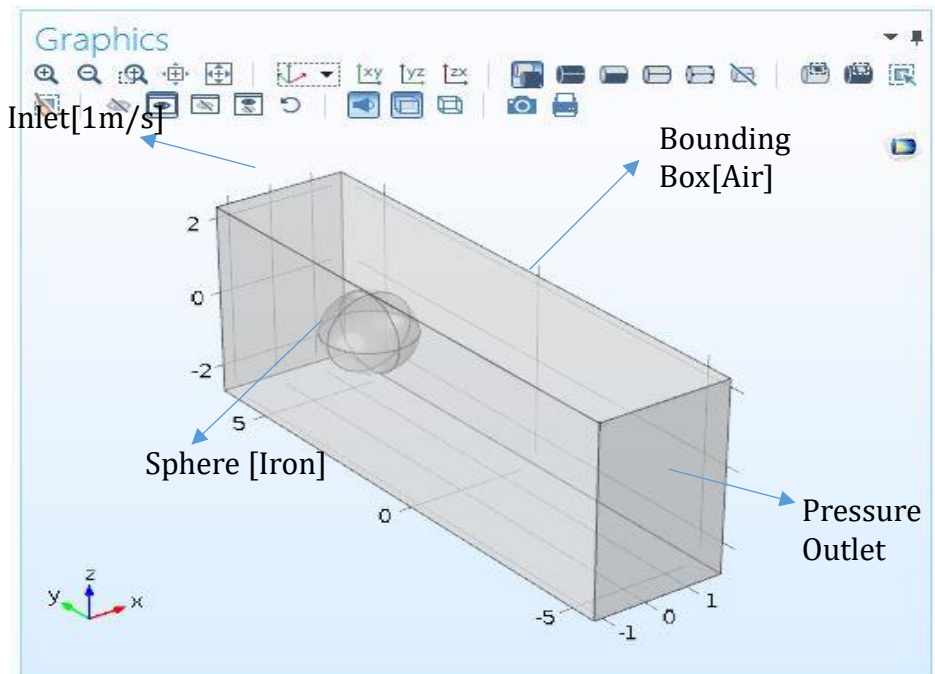
Sujal Tipnis

# 3D Simulation using COMSOL Multiphysics 5.2

Sujal Tipnis

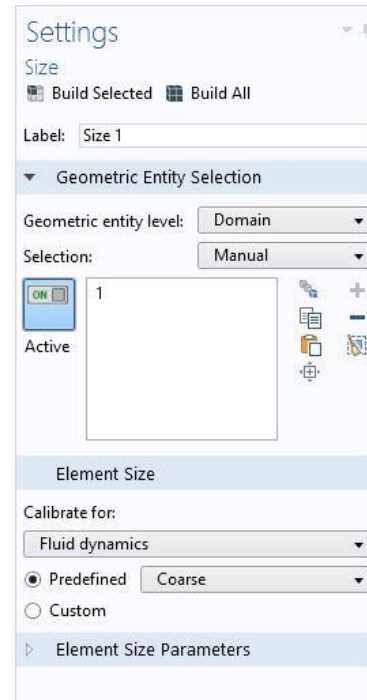
## 1. Geometry and Boundary Conditions

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



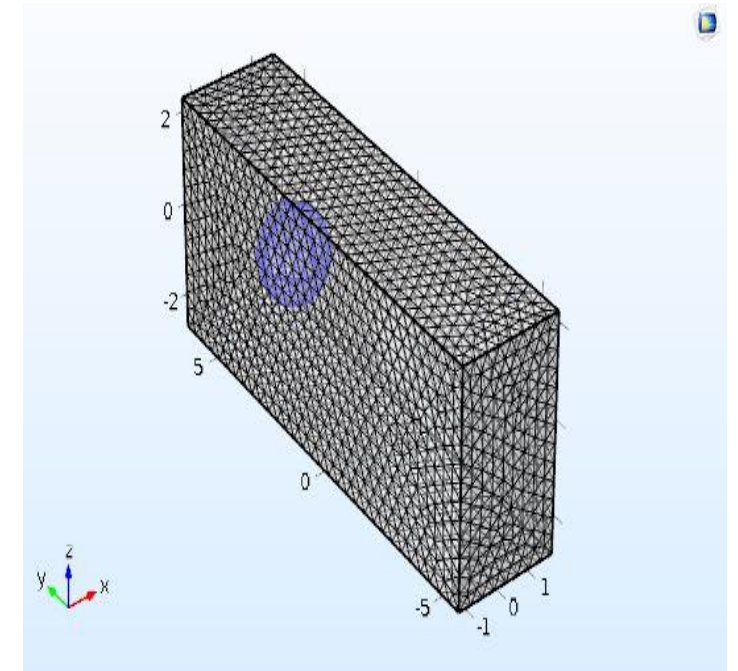
## 2. Mesh

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

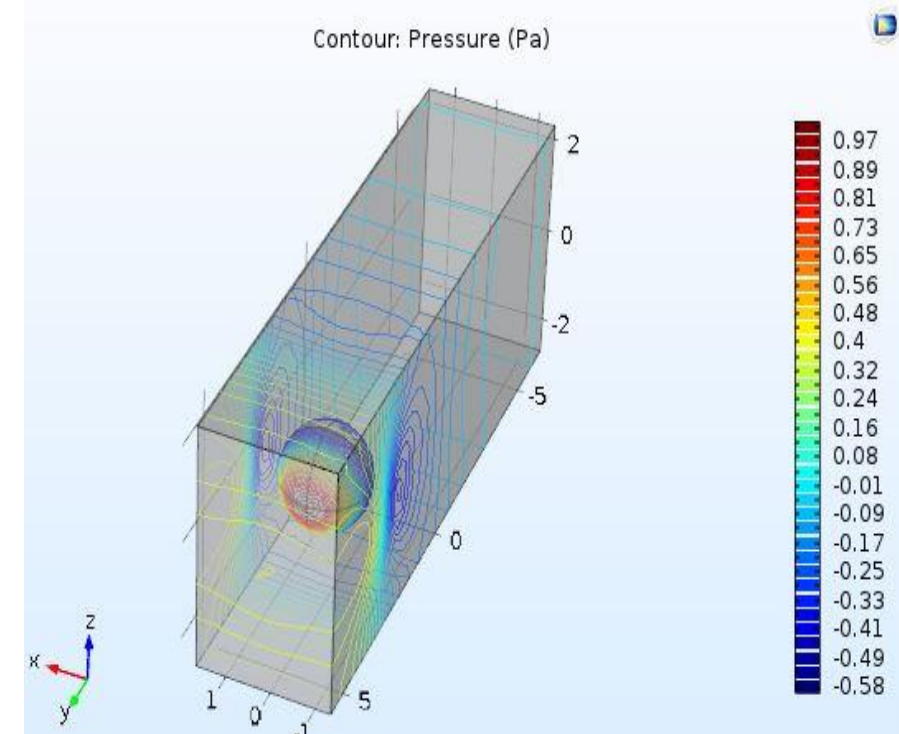
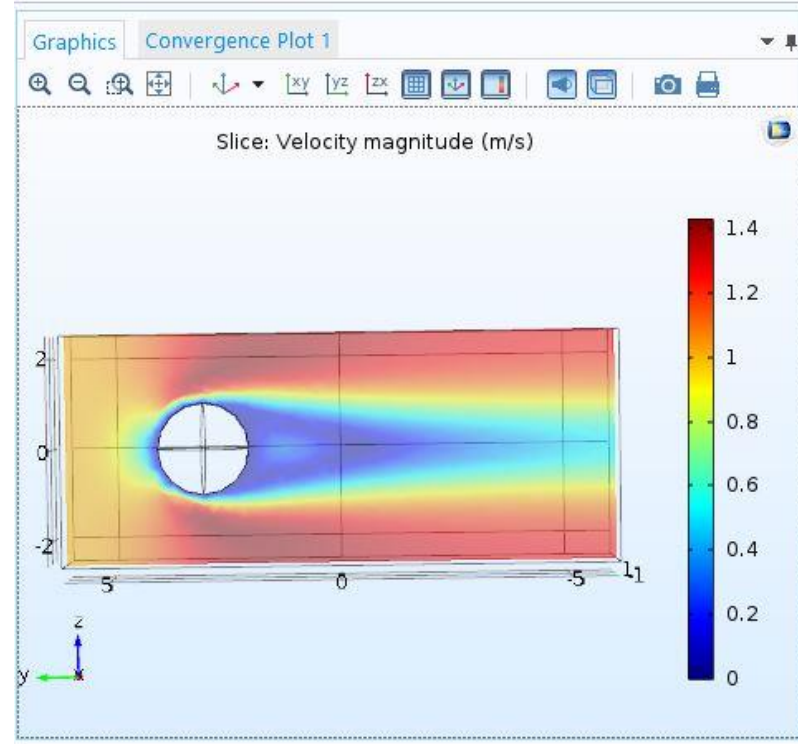
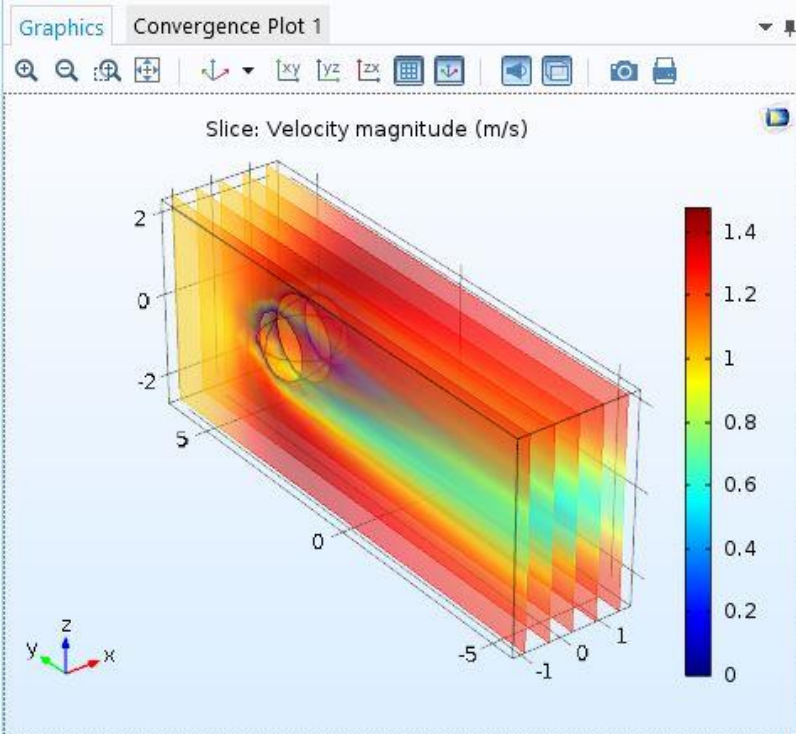


## 3. Setup

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



# 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

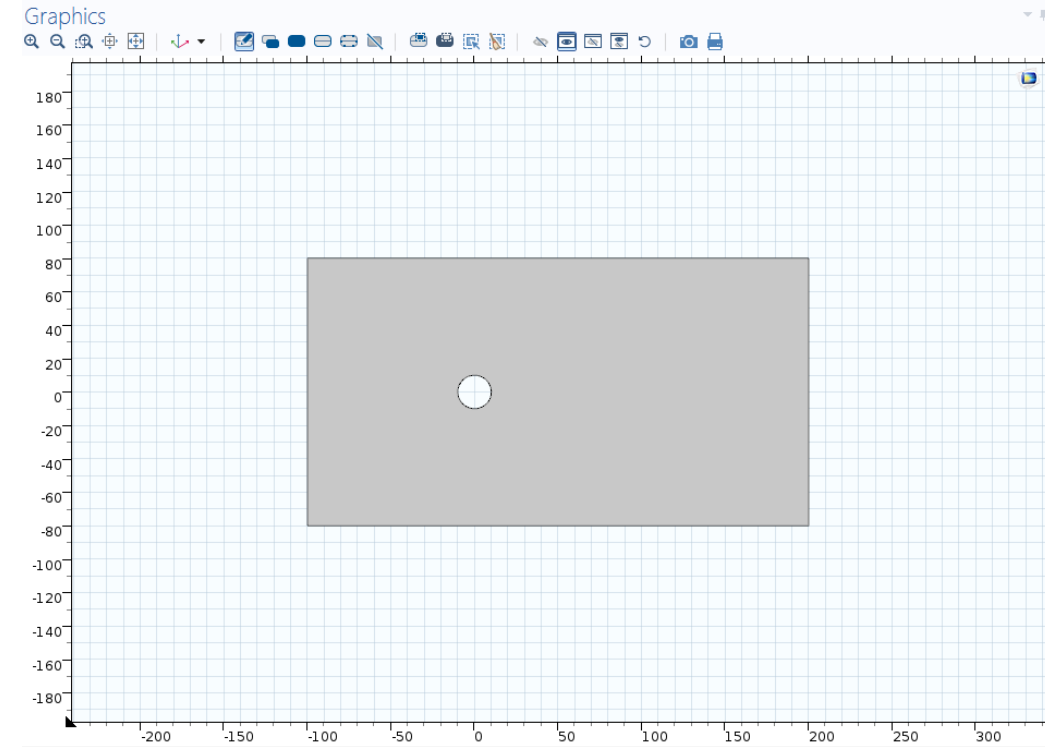
Vijay Jangid

# COMSOL Multiphysics

Vijay Jangid

## Procedure followed for analysis:

The screenshot shows the COMSOL Model Builder interface. The left pane (Model Builder) displays a tree view with several items circled in red: 'Circle geometry' (containing 'Circle 1 (c1)', 'Rectangle 1 (r1)', and 'Difference 1 (dif1)'), 'Materials' (containing 'Laminar Flow (spf)' and 'Fluid Properties 1'), 'Study 3' (containing 'Step 1: Time Dependent'), and 'Results' (containing various data sets and plots). The right pane (Settings) shows the 'Geometry' settings for 'Circle\_geometry', including 'Units' (Length unit: cm, Angular unit: Degrees) and 'Advanced' (Automatic rebuild checked).



Make Geometry

Select Material

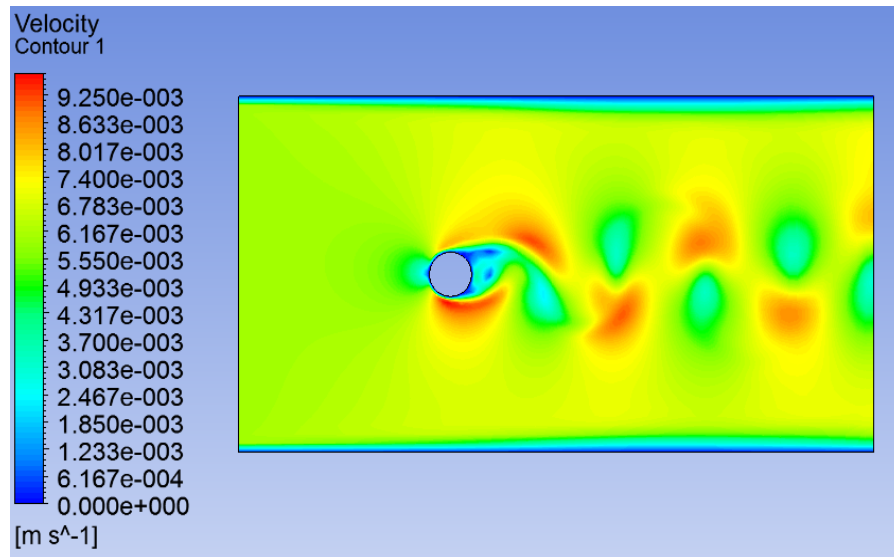
Input Boundary Conditions

Type of solution required

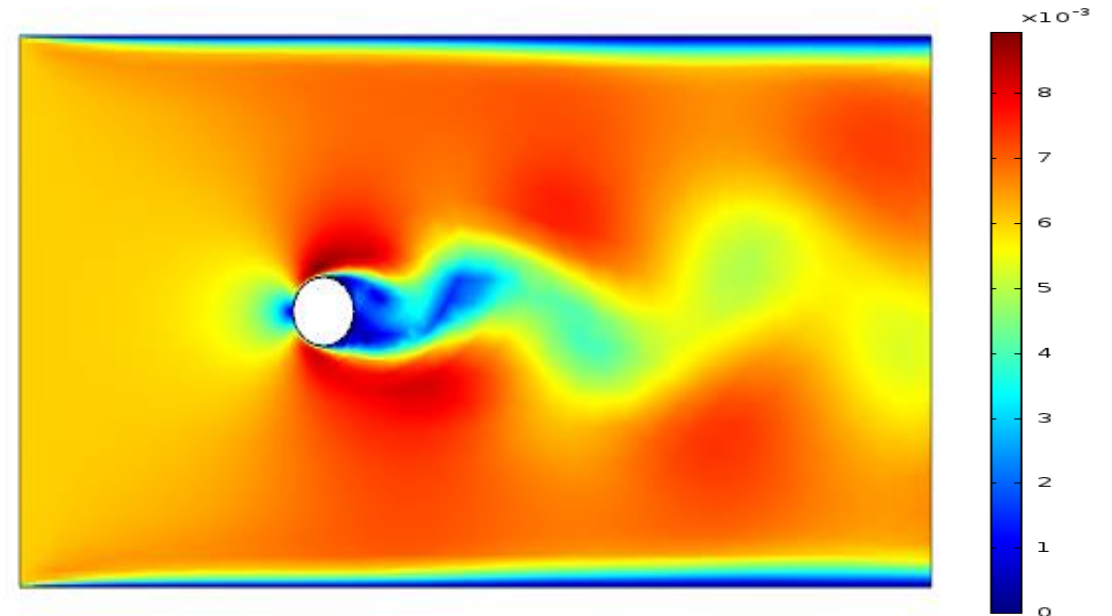
Output required

# 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.