

# OpenFOAM through Spoken Tutorial



Prepared by

CFD Team

FOSSEE, IIT Bombay



Indian Institute of Technology Bombay

September , 2015

# Contents

Table of contents	i
List of Figures	ii
1 Introduction	1
2 Installing OpenFOAM and Paraview	2

# List of Figures

2.1	Search Icon on top of Launcher . . . . .	2
2.2	Enter the system password here . . . . .	2
2.3	Type OpenFOAM in the search box . . . . .	3
2.4	Click on OpenFOAM and Paraview to Mark and Install . . . . .	3
2.5	Press Ctrl+Alt+t keys to open up terminal window . . . . .	3
2.6	Open the bashrc file . . . . .	4
2.7	Usage message appears which shows OpenFOAM is correctly installed . .	4
2.8	Example problem solved after OpenFOAM installation . . . . .	5
2.9	blockMesh utility for Meshing in OpenFOAM . . . . .	6
2.10	Terminal shows the iterations running . . . . .	6
2.11	Paraview window . . . . .	7
2.12	Geometry seen in the Paraview Window . . . . .	7
2.13	Uncheck the internal mesh option . . . . .	8
2.14	Check for the moving and fixedWalls in paraview window . . . . .	8
2.15	FixedWalls appear in the paraview window . . . . .	9

# Chapter 1

## Introduction

OpenFOAM is a free and open source CFD toolbox. OpenFOAM stands for Open source Field Operation And Manipulation. It is first and foremost a C++ library / toolkit, used primarily to create executables, known as applications. The applications fall into two categories. Solvers, that are each designed to solve a specific problem in continuum mechanics and Utilities, that are designed to perform tasks that involve data manipulation. The OpenFOAM distribution contains numerous solvers and utilities covering a wide range of problems of two Dimensions and three dimensions. It is used in academia and industry to solve wide variety of computational problems. In contrast to any proprietary software, the source code here is accessible and modifiable.

# Chapter 2

## Installing OpenFOAM and Paraview

The First chapter deals with Installing OpenFOAM and Paraview. We are using Linux Operating System for installation and OpenFOAM-2.3.0 and Paraview-4.1.0.

The user of OpenFOAM is expected to have some basic knowledge of Computational Fluid Dynamics ( CFD ) and should be able to use basic Linux Commands.

OpenFOAM and Paraview can be installed using Synaptic Package Manager. On the left side of your computer screen you can see the Launcher with the list of softwares.

Click on the search box ,Fig.2.1 on top of the Launcher and type Synaptic. This will display the Synaptic Package Manager. Click on it to open.



Figure 2.1: Search Icon on top of Launcher

You will be interrupted to enter the system password and press Enter.

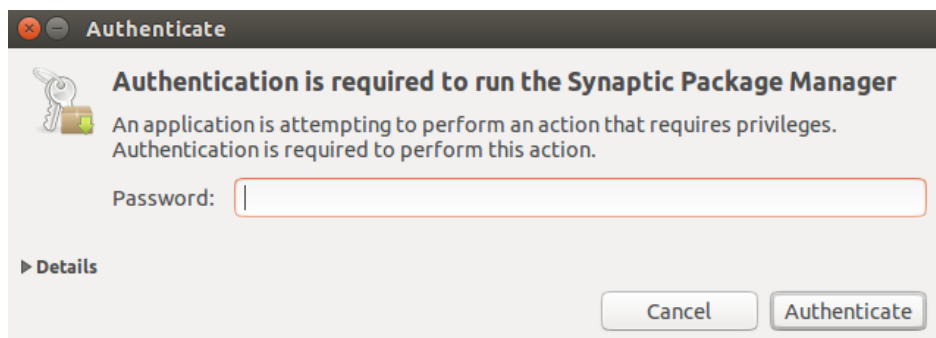


Figure 2.2: Enter the system password here

Once the Synaptic Package Manager is Opened, in the search box type OpenFOAM.

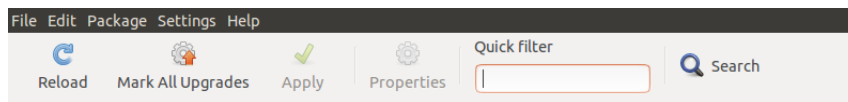


Figure 2.3: Type OpenFOAM in the search box

You will see both OpenFOAM-2.3.0 and Paraview-4.1.0. Right Click Both of them for installation and click Apply to install, Fig 2.4. This will take a while for installation depending upon your internet speed.

S	Package	Installed Version	Latest Version	Size	Description
<input checked="" type="checkbox"/>	openfoam231		0-1		OpenFOAM
<input type="checkbox"/>	openfoam240		0-1		OpenFOAM
<input checked="" type="checkbox"/>	foam-extend-3.1	3.1-2	3.1-2	537 MB	foam-extend, community fork of the OpenFOAM(R) CFD library
<input checked="" type="checkbox"/>	paraviewopenfoam410		0-1		Paraview visualisation application
<input type="checkbox"/>	libfreefoam1		0.1.0+dfsg-1build1		libraries for Computational Fluid Dynamics (CFD)
<input type="checkbox"/>	freefoam-user-doc		0.1.0+dfsg-1build1		software for Computational Fluid Dynamics - user documentation
<input type="checkbox"/>	freefoam-dev-doc		0.1.0+dfsg-1build1		software for Computational Fluid Dynamics - developers documentation
<input type="checkbox"/>	libfreefoam-dev		0.1.0+dfsg-1build1		libraries for Computational Fluid Dynamics (CFD) - development files
<input type="checkbox"/>	freefoam		0.1.0+dfsg-1build1		programs for Computational Fluid Dynamics (CFD)

Figure 2.4: Click on OpenFOAM and Paraview to Mark and Install

OpenFOAM can also be downloaded and installed using the OpenFOAM website as follows.

- On your browser type **www.openfoam.com/download**
- Go to Ubuntu Debian Installation
- Under the first point of Installation copy the command line and paste this in your terminal window
- Open the terminal window by pressing **Ctrl+Alt+t** keys simultaneously on your keyboard or you can also open it using the search icon on top of the Launchbar

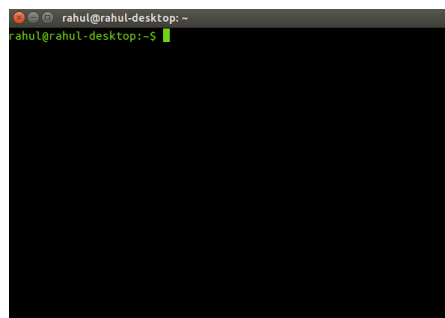


Figure 2.5: Press Ctrl+Alt+t keys to open up terminal window

- For complete installation for OpenFOAM and Paraview follow the steps under installation

To configure the installed software we need to edit the bashrc file. To do this open a new command terminal and type **gedit ~/.bashrc** and press enter

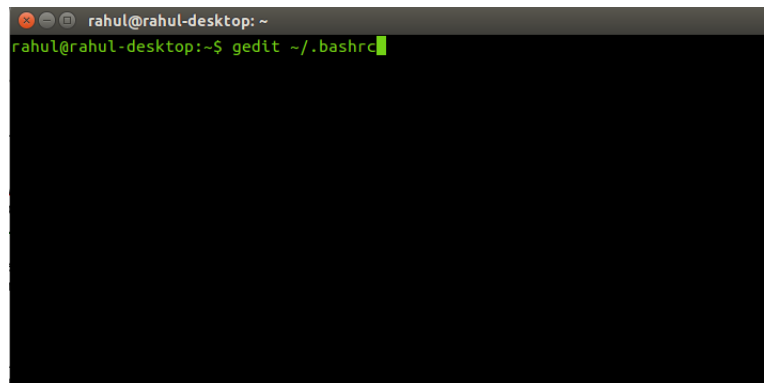


Figure 2.6: Open the bashrc file

After the bashrc file is opened scroll down to the bottom of the file. Then go back to your browser (OpenFOAM download page) and scroll down to **User Configuration**.

Copy the second point **source /opt/openfoam230/etc/bashrc** under this heading and paste it at the bottom of the bashrc file. Save it and close the file.

To check whether OpenFOAM is installed properly open a new command terminal and type **icoFoam -help** and press enter. You will see a "Usage" message on your terminal screen, Fig 2.7 which shows that the installation is done.

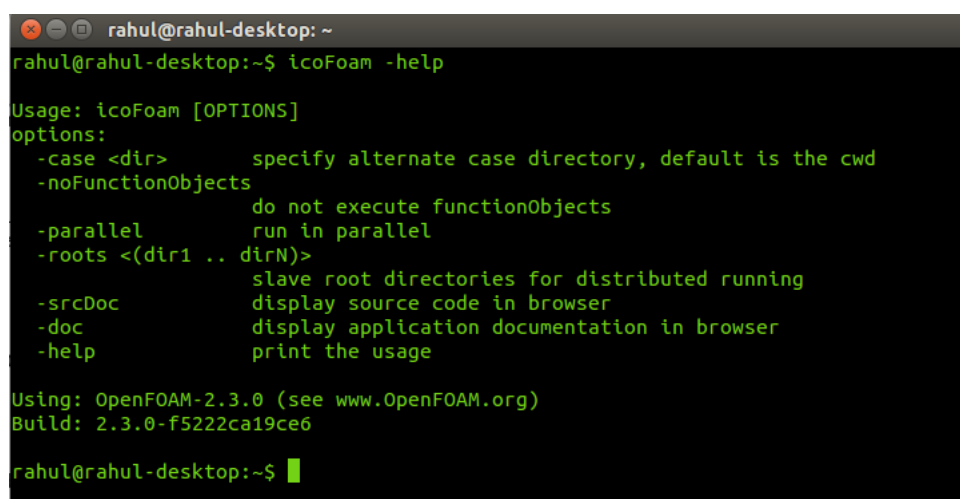


Figure 2.7: Usage message appears which shows OpenFOAM is correctly installed

Now we will set up the working directory and copy the tutorial folder. Follow the steps given below.

1. Open up a new terminal and type **mkdir -p \$FOAM\_RUN** and press enter
2. Now type **cp -r \$FOAM\_TUTORIALS \$FOAM\_RUN** and press enter. This will copy the tutorials folder into the run directory.

Alternate way to install OpenFOAM and Paraview is by Compiling the Source code available under the header of **Source Pack** Installation on the OpenFOAM website. Download the tar files. Create a folder in your Home directory by the name OpenFOAM and paste the tar files in that folder. Extract the files in that folder.

Follow the steps given on the OpenFOAM source pack installation page to complete the installation. Since we compile the source code it might take a few hours to complete.

We will solve an example problem here by the name Lid Driven Cavity. It is a two dimensional problem where the upper plate moves and other three sides of the plate are fixed, 2.8. The solver we use here is icoFoam which is an Transient solver for incompressible flow.

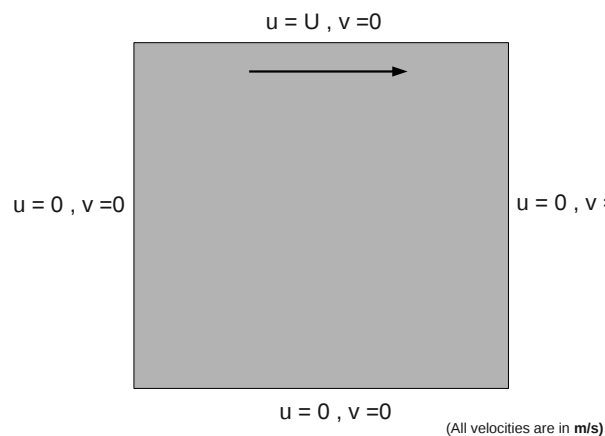


Figure 2.8: Example problem solved after OpenFOAM installation

In the terminal copy paste the path given below :

```
cd OpenFOAM/OpenFOAM-2.3.0/run/tutorials/incompressible/icoFoam/cavity
```

The geometry now needs to be meshed. This can be done using the blockMesh utility of OpenFOAM. In the command terminal type **blockMesh** and press enter which completes the meshing, Fig 2.9



```

Creating curved edges
Creating topology blocks
Creating topology patches
Creating block mesh topology
Check topology

Basic statistics
  Number of internal faces : 0
  Number of boundary faces : 6
  Number of defined boundary faces : 6
  Number of undefined boundary faces : 0
  Checking patch -> block consistency

Creating block offsets
Creating merge list .

Creating polyMesh from blockMesh
Creating patches
Creating cells
Creating points with scale 0.1
  Block 0 cell size :
    i : 0.005 .. 0.005
    j : 0.005 .. 0.005
    k : 0.01 .. 0.01

Writing polyMesh
-----
Mesh Information
-----
  boundingBox: (0 0 0) (0.1 0.1 0.01)
  nPoints: 882
  nCells: 400
  nFaces: 1640
  nInternalFaces: 760
-----
Patches
-----
  patch 0 (start: 760 size: 20) name: movingWall
  patch 1 (start: 760 size: 60) name: fixedWalls
  patch 2 (start: 840 size: 800) name: frontAndBack
End

```

Figure 2.9: blockMesh utility for Meshing in OpenFOAM

Once meshing is done we now run the solver by typing **icoFoam** in the command terminal. The iteration running can be seen in the terminal window, Fig 2.10. We have reached the solving point.

```

time step continuity errors : sum local = 6.42122e-09, global = 6.90699e-20, cumulative = -1.51597e-18
ExecutionTime = 0.1 s ClockTime = 0 s

time = 0.49
Courant Number mean: 0.222158 max: 0.852134
smoothSolver: Solving for Ux, Initial residual = 2.58149e-07, Final residual = 2.58149e-07, No Iterations 0
smoothSolver: Solving for Uy, Initial residual = 5.65518e-07, Final residual = 5.65518e-07, No Iterations 0
DICPCG: Solving for p, Initial residual = 8.90806e-07, Final residual = 8.90806e-07, No Iterations 0
time step continuity errors : sum local = 9.06241e-09, global = 2.49643e-19, cumulative = -1.26633e-18
DICPCG: Solving for p, Initial residual = 1.85743e-06, Final residual = 2.83778e-07, No Iterations 1
time step continuity errors : sum local = 4.09215e-09, global = -5.29644e-19, cumulative = -1.79597e-18
ExecutionTime = 0.1 s ClockTime = 0 s

time = 0.495
Courant Number mean: 0.222158 max: 0.852134
smoothSolver: Solving for Ux, Initial residual = 2.46591e-07, Final residual = 2.46591e-07, No Iterations 0
smoothSolver: Solving for Uy, Initial residual = 5.36152e-07, Final residual = 5.36152e-07, No Iterations 0
DICPCG: Solving for p, Initial residual = 6.20776e-07, Final residual = 6.20776e-07, No Iterations 0
time step continuity errors : sum local = 6.85402e-09, global = -2.53944e-19, cumulative = -2.84992e-18
DICPCG: Solving for p, Initial residual = 8.33045e-07, Final residual = 8.33045e-07, No Iterations 0
time step continuity errors : sum local = 8.59305e-09, global = 5.07089e-19, cumulative = -1.54203e-18
ExecutionTime = 0.1 s ClockTime = 0 s

time = 0.5
Courant Number mean: 0.222158 max: 0.852134
smoothSolver: Solving for Ux, Initial residual = 2.32737e-07, Final residual = 2.32737e-07, No Iterations 0
smoothSolver: Solving for Uy, Initial residual = 5.07002e-07, Final residual = 5.07002e-07, No Iterations 0
DICPCG: Solving for p, Initial residual = 1.0281e-06, Final residual = 2.77237e-07, No Iterations 1
time step continuity errors : sum local = 4.0374e-09, global = -9.0204e-19, cumulative = -2.44407e-18
DICPCG: Solving for p, Initial residual = 5.31987e-07, Final residual = 5.31987e-07, No Iterations 0
time step continuity errors : sum local = 6.12557e-09, global = -3.93738e-20, cumulative = -2.44407e-18

```

Figure 2.10: Terminal shows the iterations running

The results can be Visualized using Paraview. To open paraview in your terminal type **paraFoam** and press enter. This will open up the paraview window, Fig 2.11.

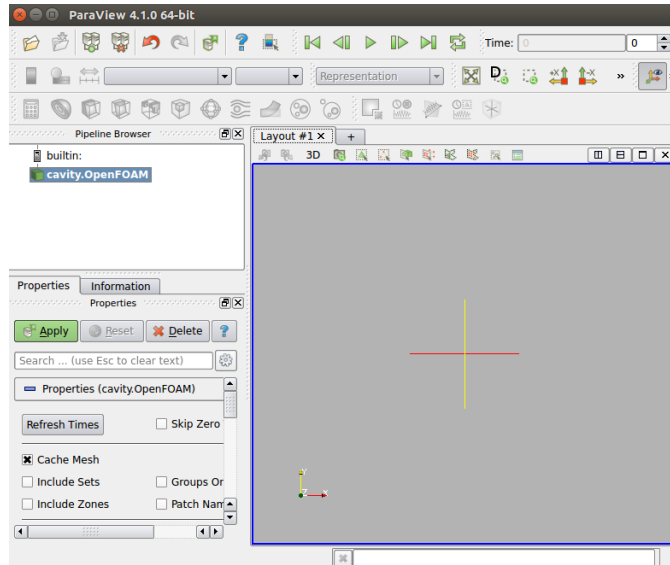


Figure 2.11: Paraview window

Click on the Apply button on the left hand side of the **Object Inspector** Menu to view the Geometry, Fig2.12.

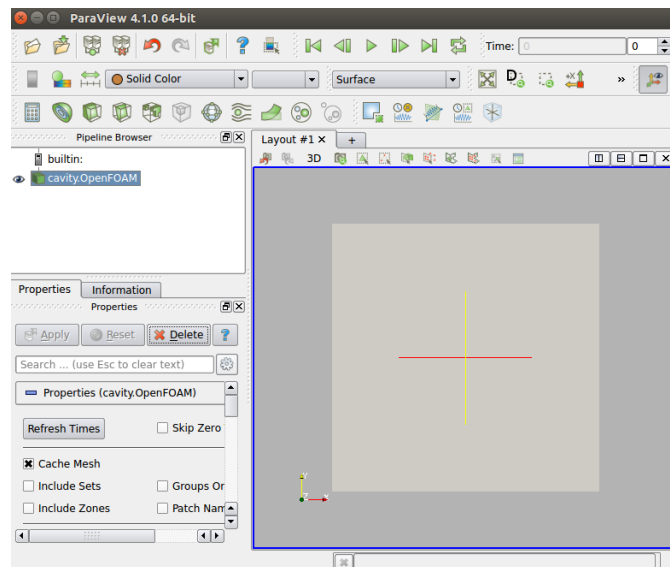


Figure 2.12: Geometry seen in the Paraview Window

To check the Boundary conditions scroll down the object inspector menu and go down to mesh parts and uncheck internal mesh option and click Apply. You can see that the geometry disappears, Fig 2.13.

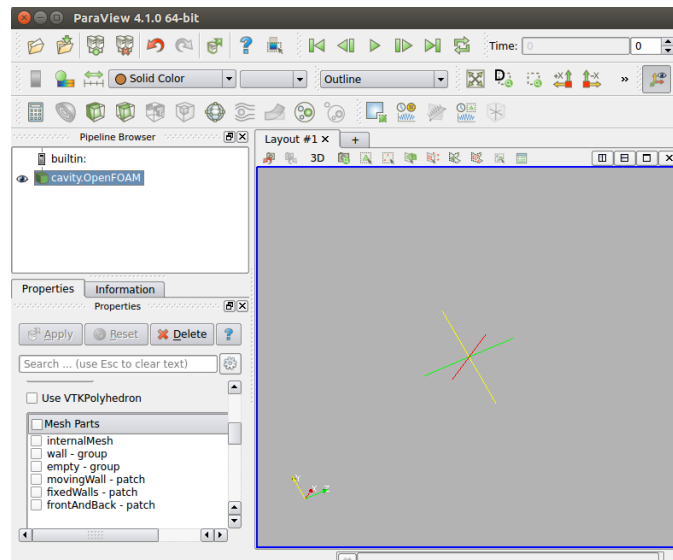


Figure 2.13: Uncheck the internal mesh option

Now click on the checkbox for movingWall and fixedWall and click the Apply button. You can see the moving wall and fixedWall in the paraview window, Fig 2.14. Also uncheck the movingWall to see the fixedWalls in the paraview, Fig 2.15.

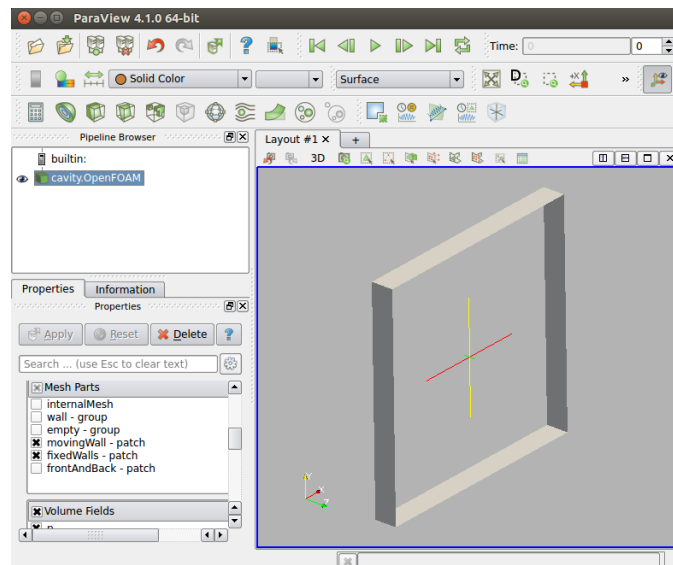


Figure 2.14: Check for the moving and fixedWalls in paraview window

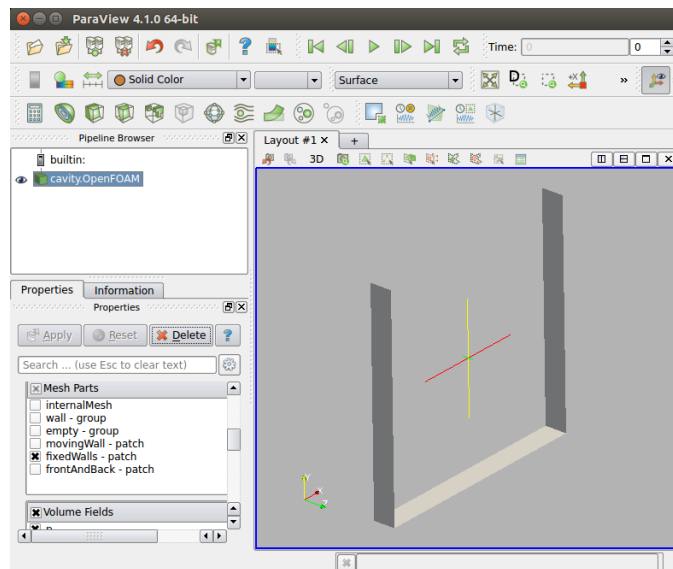


Figure 2.15: FixedWalls appear in the paraview window