

Chapter 1

Installing and Running OpenFOAM and ParaView

1.1 Introduction to OpenFOAM

OpenFOAM, which stands for Open source Field Operation And Manipulation, is a free and open source CFD toolbox. It is a C++ library / toolkit, used primarily to create executables, known as applications. These applications fall into two major categories:

- **Solvers** – designed to solve a specific problem in continuum mechanics
- **Utilities** - designed to perform tasks that involve data manipulation

OpenFOAM has a vast range of solvers spanning from complex fluid flows involving chemical reactions, turbulence and heat transfer, to solid dynamics and electromagnetics. These numerical solvers cover both 2-Dimensional and 3-Dimensional problems. A comprehensive structure of OpenFOAM can be given as 1.1:

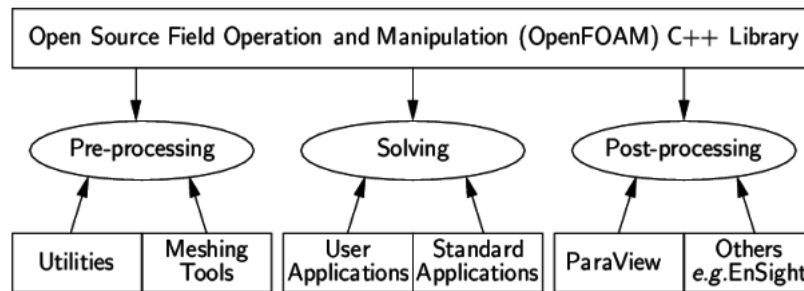


Figure 1.1: Overall structure of OpenFOAM

Nowadays, OpenFOAM is extensively used in academia and industry to solve wide variety of computational problems. The parallelization capability of OpenFOAM for both pre and post-processing of data provides the users with an efficient environment for working. One of the major advantage of using OpenFOAM is that in contrast to any proprietary software, this source code here is accessible and modifiable.

1.2 Introduction to ParaView

ParaView is a multi-platform, open-source software used for data analysis and scientific visualization application. It is built based on Visualization Tool-Kit (VTK) libraries. It uses qualitative and quantitative techniques to build the visualization. ParaView was largely developed to analyze huge data sets using distributed memory computing resources. It analyzes data interactively both in 3-Dimension as well as programmatically using its batch processing capabilities.

It is a user-friendly data visualization software which utilizes Cmake for its compilation. Paraview is predominantly used for post-processing in OpenFOAM due to its unique ability to run on distributed and shared memory, parallel and single processor systems.

OpenFOAM and Paraview can be very easily installed in your systems by mainly following the user manual for installation. There are many ways of installing OpenFOAM and ParaView in your systems. Here we will be discussing two basic ways of installation in a LINUX environment.

1.3 Installation of OpenFOAM and ParaView through Synaptic Package Manager

Here in this section we will discuss how to install OpenFOAM-2.4.0 and ParaView-4.1.0 using Synaptic Package Manager. To understand this chapter our readers should have a prior exposure to LINUX based operating systems along with its basic commands and also some trivial knowledge on Computational Fluid Dynamics (CFD).

In a LINUX operating system, on the left side of your computer screen you can see the Launcher with the list of softwares.

To install OpenFOAM and ParaView, click on the search box as shown in figure 1.2 on top of the Launcher and type Synaptic. This will display the Synaptic Package Manager. Click on it to open.



Figure 1.2: Search Icon on top of Launcher

Here you would be interrupted with the system authentication. After providing the system password press enter.

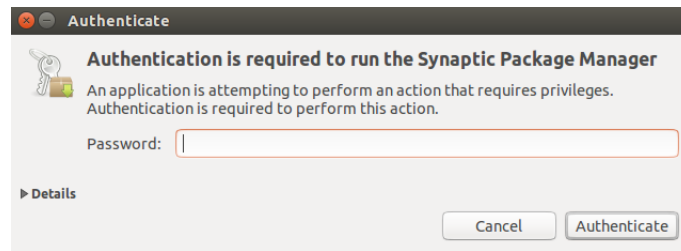


Figure 1.3: Enter the system password here

Once the Synaptic Package Manager is Opened, in the search box (as shown in figure 1.4) type OpenFOAM.

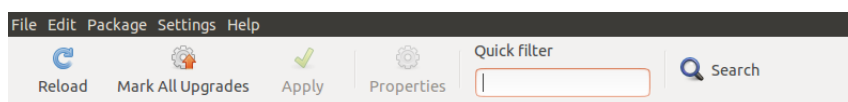


Figure 1.4: Type OpenFOAM in the search box

You will see both OpenFOAM-2.3.0 and Paraview-4.1.0, as shown in figure 1.5. Right click on both of them for installation and click Apply to install. It 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 1.5: Click on OpenFOAM and Paraview to Mark and Install

1.4 Installation of OpenFOAM and ParaView through OpenFOAM Website

OpenFOAM can also be downloaded and installed through the OpenFOAM website by carrying out the following steps:

1. On your browser type www.openfoam.com/download
2. Go to Ubuntu Debian Installation
3. 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 Launch bar

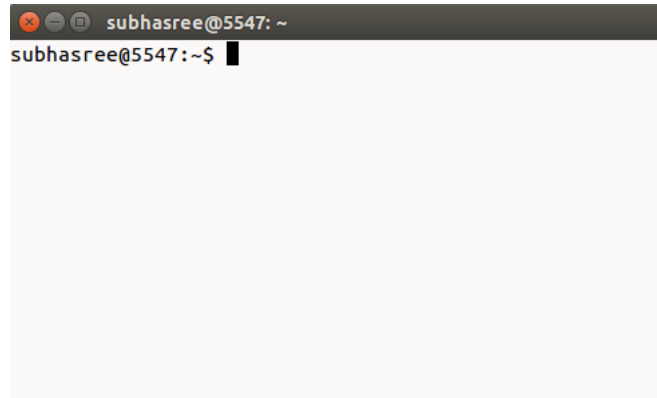


Figure 1.6: Press Ctrl+Alt+t keys to open up terminal window

4. Under the first point of Installation copy the command line from the website and paste it in your terminal window
5. For complete installation for OpenFOAM and Paraview follow the steps under installation

1.5 Configure OpenFOAM and ParaView

To configure the installed software we need to source this application by editing the `bashrc` file. To do this, open a new command terminal and type:

```
gedit ~/.bashrc
```

and press enter.



Figure 1.7: 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 command line

```
source /opt/openfoam230/etc/bashrc
```

under this heading and paste it at the bottom of the `bashrc` file. Save it and close the file.

1.6 Test the Installed Software

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, as shown in figure 1.8. This shows that the installation is done.

```
subhasree@5547:~$ icoFoam -help

Usage: icoFoam [OPTIONS]
options:
  -case <dir>          specify alternate case directory, default is the cwd
  -noFunctionObjects    do not execute functionObjects
  -parallel             run in parallel
  -roots <(dir1 .. dirN)>
                        slave root directories for distributed running
  -srcDoc              display source code in browser
  -doc                 display application documentation in browser
  -help                print the usage

Using: OpenFOAM-2.4.0 (see www.OpenFOAM.org)
Build: 2.4.0-dcea1e13ff76
```

Figure 1.8: Usage message

You may also install OpenFOAM and Paraview 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.

1.7 Creating User Working Directory

Before proceeding further with an example case file, we will first discuss how to create a working directory for the users.

Follow the steps given below:

1. Open up a new terminal and type the following and press enter

```
mkdir -p $FOAMRUN
```

2. After this type

```
cp -r $FOAM_TUTORIALS $FOAMRUN
```

This will copy the tutorials folder into the user's directory.

1.8 Running OpenFOAM and ParaView

Here in this section we will teach how to run openFOAM and ParaView using an example **Lid Driven Cavity** case file. This case file is already written in the OpenFOAM tutorial folder.

This example is a two dimensional problem where the upper plate moves at a mentioned user defined speed while the other three sides of the plate are stationary as shown in figure 1.9. The solver we use here is icoFoam which is an Transient solver for incompressible flow.

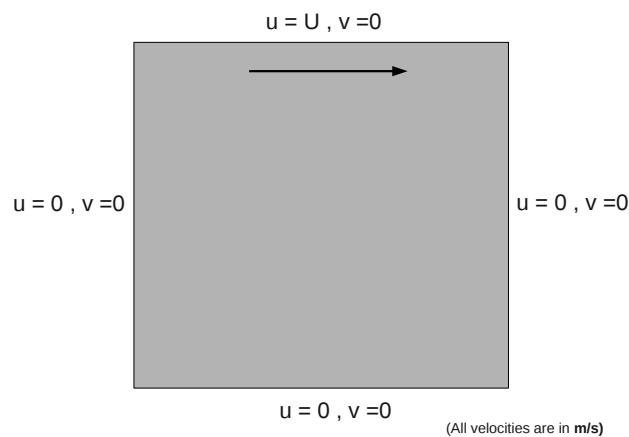


Figure 1.9: Example problem solved after OpenFOAM installation

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 and you will see the following details as shown in figure 1.10

```

subhasree@5547: ~/OpenFOAM/OpenFOAM-2.4.0/tutorials/incompressible/icoFoam/cavity
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
-----

```

Figure 1.10: BlockMesh utility for Meshing in OpenFOAM

Once meshing is done we will now run the solver by typing icoFoam in the command terminal. The iteration running can be seen in the terminal window, as shown in figure 1.11. Thus in this section we have learned how to solve a problem case in OpenFOAM.

```

subhasree@5547: ~/OpenFOAM/OpenFOAM-2.4.0/tutorials/incompressible/icoFoam/cavity
Time = 0.495

Courant Number mean: 0.222158 max: 0.852134
smoothSolver: Solving for Ux, Initial residual = 2.46591e-07, Final residual = 2.6591e-07, No Iterations 0
smoothSolver: Solving for Uy, Initial residual = 5.36152e-07, Final residual = 5.6152e-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 = -2.04992e-18
DICPCG: Solving for p, Initial residual = 8.33045e-07, Final residual = 8.33045e-07, No Iterations 0
time step continuity errors : sum local = 8.59385e-09, global = 5.07889e-19, cumulative = -1.54203e-18
ExecutionTime = 0.14 s ClockTime = 1 s

Time = 0.5

Courant Number mean: 0.222158 max: 0.852134
smoothSolver: Solving for Ux, Initial residual = 2.32737e-07, Final residual = 2.2737e-07, No Iterations 0
smoothSolver: Solving for Uy, Initial residual = 5.07002e-07, Final residual = 5.7002e-07, No Iterations 0
DICPCG: Solving for p, Initial residual = 1.0281e-06, Final residual = 2.77237e-06, 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 = -2.48344e-18
ExecutionTime = 0.14 s ClockTime = 1 s

End

```

Figure 1.11: Terminal shows the iterations running

Now we will learn how to post-process this data. The solution can be Visualized using Paraview. To open ParaView window type **paraFoam** in your terminal and press enter. This will open up the paraview window, as shown in Figure 1.12.

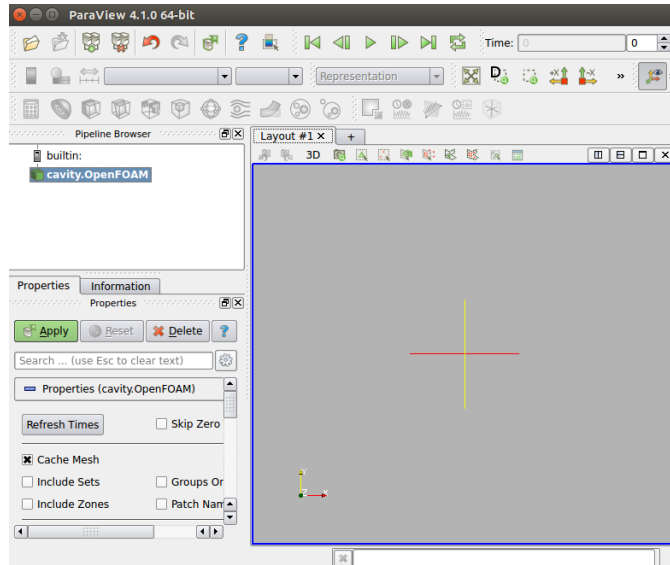


Figure 1.12: Paraview window

Click on the Apply button on the left hand side of the Object Inspector Menu to view the Geometry, as shown in figure 1.13.

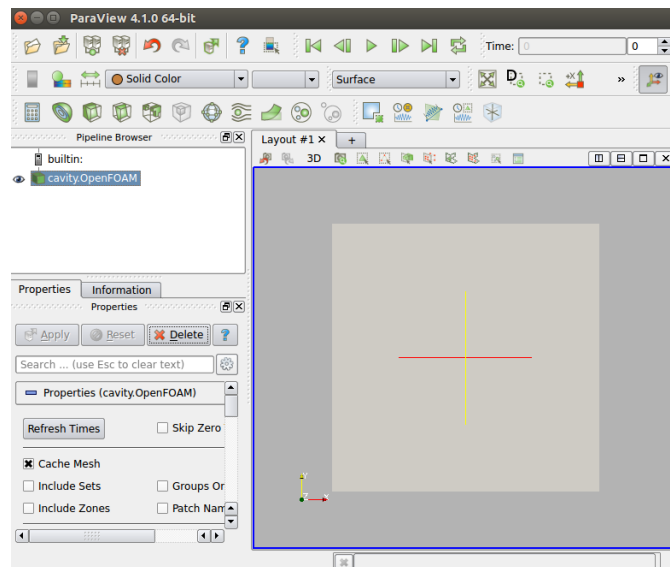


Figure 1.13: Geometry seen in the Paraview Window

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 will see that the geometry disappears, as shown in figure 1.14.

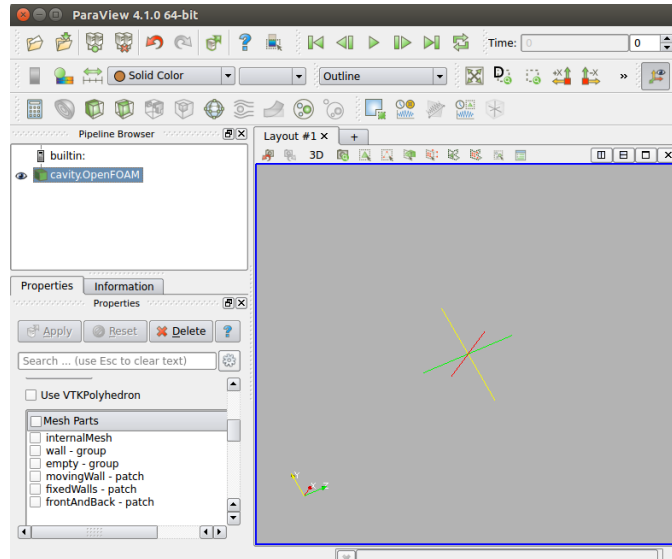


Figure 1.14: Uncheck the internal mesh option

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, as shown in figure 1.15. You can also uncheck the movingWall option to see the fixedWalls in the paraview window, as shown in Figure ??.

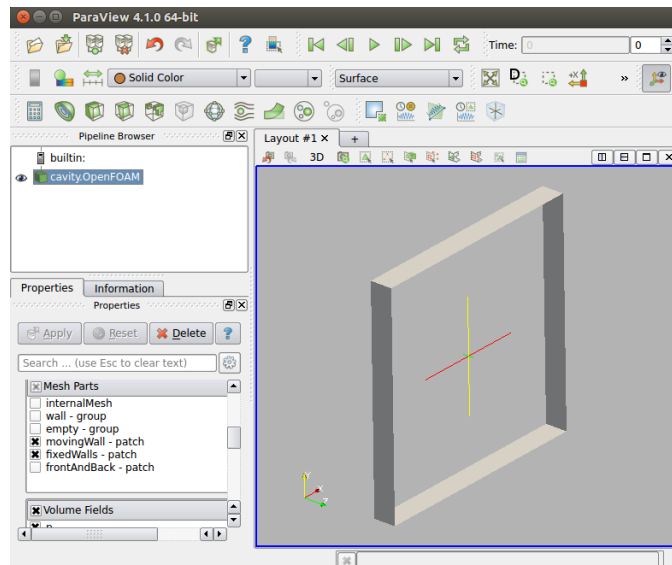


Figure 1.15: Check for the moving and fixedWalls in paraview window

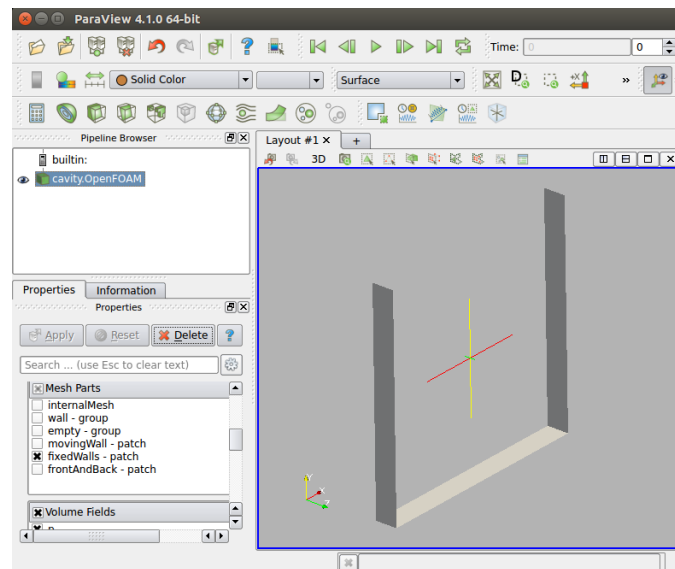


Figure 1.16: FixedWalls appear in the paraview window