

Instruction Sheet for OpenFOAM 7 and ParaView 5.6.0
Installation CFD team
FOSSEE and Spoken Tutorials
IIT Bombay

Procedure to install OpenFOAM 7 and ParaView 5.6.0 for Ubuntu Linux Operating Systems 16.04 and above

NOTE: You should have a good internet connection to install OpenFOAM 7. Otherwise, your software may not be installed properly.

1. Open the terminal by typing **Ctrl+Alt+T**
2. **Copy and paste** the following in the **terminal prompt** to add dl.openfoam.org to the list of software repositories for apt to search, and to add the public key (gpg.key) for the repository to enable package signatures to be verified.

```
sudo sh -c "wget -O - http://dl.openfoam.org/gpg.key | apt-key add -"
```

```
sudo add-apt-repository http://dl.openfoam.org/ubuntu
```

3. Update the apt package list to account for the new download repository location by typing the following in terminal prompt.

```
sudo apt-get update
```

4. Install OpenFOAM 7 which also installs ParaView 5.6.0 (paraviewopenfoam56) as a dependency by typing the following in terminal prompt.

```
sudo apt-get -y install openfoam7
```

5. Open the .bashrc file in the user home directory in an editor, e.g. by typing the following in terminal prompt. (note the dot)

```
gedit ~/.bashrc
```

6. At the bottom of the bashrc file, copy and paste the following line

```
source /opt/openfoam7/etc/bashrc
```

7. Save it and close.

Checking the installation:

To check the software open a new command terminal (Ctrl+Alt +T) and run a lid-driven cavity case using icoFoam solver by typing the following in the command terminal.

1. Since it is advisable not to run case files directly in installation directory, we create a folder run in home directory where we will copy the tutorials case directory from openfoam7 folder.

```
mkdir -p $FOAM_RUN
```

2. Now go to run folder. Here cd is the terminal command used for navigating to the folder.

```
cd $FOAM_RUN
```

3. Now copy the tutorials folder from installation directory to run folder. Here cp -r is the command used to copy directory from old location to new location. While copying command note the space and . (dot) after the word TUTORIALS

```
cp -r $FOAM_TUTORIALS .
```

4. Now go to the cavity case directory

```
cd tutorials/incompressible/icoFoam/cavity/cavity
```

5. Now run blockMesh utility present in OpenFOAM to do the meshing of the geometry, by typing

```
blockMesh
```

6. Run the solver icoFoam which is an incompressible transient flow solver, by typing

```
icoFoam
```

7. For visualization type the following command to open ParaView 5.6.0

```
paraFoam
```

8. Once ParaView window is opened click on the Apply button view the geometry.