Instruction Sheet for OpenFOAM 5.0 and ParaView 5.4.2 Installation
CFD team
FOSSEE and Spoken Tutorials
IIT Bombay

Procedure to install OpenFOAM 5.0 and ParaView 5.4.2 for Ubuntu Linux Operating Systems 14.04 and above

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

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 5 (5 in the name refers to version 5.0) which also installs ParaView 5.4.2 (paraviewopenfoam54) as a dependency by typing the following in terminal prompt.

```
sudo apt-get -y install openfoam5
```

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. For multiple versions of OpenFOAM installed in same system, it is better to use an alias (an alternative name or label that refers to a file, command, address, or other item, and can be used to locate or access it) for each version installed. The alias of a particular version will have to be entered in the terminal prompt to start the respective OpenFOAM version.

   For OpenFOAM 5.0 we use the alias: **of5**

   In the last line of the bashrc file, copy and paste the following line

```
alias of5='source /opt/openfoam5/etc/bashrc'
```

   Save it and close.

7. Type the following in terminal prompt.

```
source ~/.bashrc
```

8. **Close the terminal**. With this OpenFOAM 5.0 and ParaView 5.4.2 will be installed.

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. First we type the alias for OpenFOAM 5.0 in terminal prompt
   `of5`

2. 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 openfoam5 folder.

   ```
   mkdir -p $FOAM_RUN
   ```

3. Now we go to **run** folder. Here cd is the terminal command used for navigating to the folder.

   ```
   cd $FOAM_RUN
   ```

4. Now we 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 .
   ```

5. Now we will go to the cavity case directory
   `cd tutorials/incompressible/icoFoam/cavity/cavity`

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

7. We run the solver icoFoam which is an incompressible transient flow solver, by typing
   `icoFoam`

8. For visualization we type the following command to open ParaView 5.4.2
   `paraFoam`

9. Once ParaView window is opened we click on the Apply button view the geometry and mesh.