Instruction Sheet for OpenFoam
Spoken Tutorials in Linux OS

Spoken Tutorial Team
IIT Bombay

1 The procedure to practice

1. You have been given a set of spoken tutorials and files
2. You will typically do one tutorial at a time
3. You may listen to a spoken tutorial and reproduce all the commands shown in the video
4. If you find it difficult to do the above, you may consider listening to the *whole* tutorial once and then practice during the second hearing

2 OpenFoam

1. Double-click on the file named `index.html`, to open this file in the browser.
2. Read the instructions given.
3. You will see five topics in the left-hand-side panel `Installing and running OpenFoam and Paraview, Creating Simple Geometry in OpenFoam, Creating Curved Geometry in OpenFoam, Simulating Flow in a LID-driven cavity, Supersonic Flow over a Wedge`
4. The first topic is `Installing and running OpenFoam and Paraview`, which teaches how to install `OpenFoam` and `Paraview`.
5. For the purpose of this workshop, `OpenFoam` and `Paraview` have already been installed on the machines.
6. Hence, skip this topic.
7. Listen to the video of Running OpenFOAM and Paraview from 11:50,this will help you in starting OpenFOAM and how to access the directory.
8. Please click on the next topic `Creating Simple Geometry in OpenFoam`.

3 Creating Simple Geometry in OpenFoam

1. Please locate the topic `Creating Simple Geometry in OpenFoam`.
2. Click on it.
3. You will see the video in the centre.
4. Click on the play button on the player to play the tutorial.
5. To view the tutorial in a bigger player, right-click on the video and choose `View Video` option.
6. Adjust the size of the player in such a way that you are able to practice in parallel.
7. Please follow the tutorial and `reproduce` all the activities as shown in the tutorial.
8. At 08:13, pause the tutorial.
9. After the command `type wall;` type the command `faces;`
10. Resume the tutorial.

3.1 Instructions for Practice and Assignments

(a) We recommend that you practice in parallel as you watch the video.

(b) However, since the assignments are long, we recommend you attempt to do them later.

(c) Attempt all the given assignments for your own benefit.

(d) Save your work in the same working directory as shown in the video.

(e) After you finish this tutorial, locate the next topic in the left-hand-side panel.

3.2 Creating Curved geometry in OpenFOAM

(a) Please make a note that in `blocks` you have to enter the number of grid points in x,y, and z direction in bracket.It is missing in the tutorial.

(b) When you try to mesh the geometry using the `blockMesh` command there may be some error in the command terminal.

(c) This is a small bug which tell you to change the allowSystemOperations in controldict to be changed from 0 to 1.

(d) To make these changes you may need to go through the root directory of linux.

(e) In the command terminal type `sudo -i` and press enter.

(f) In the next line type the password for your system.

(g) Now type cd (space)..(dot dot) ,press enter and go to the directory where openfoam is installed i.e. `opt`

(h) Go to the OpenFOAM folder by typing cd (space) .. openfoam211 and press enter.

(i) Type ls to see the list of things inside `openfoam`

(j) Type cd (space) `etc` and press enter.

(k) In this you can see the `controlDict` file.

(l) Open it with `gedit controlDict` and scroll down to line number 43.

(m) Change allowSystemOperations from 0 to 1 and save the file.

(n) Exit the root mode by pressing `Ctrl+z` and resume with the Meshing.

4 Similarly, remember to follow these three simple steps for all the tutorials

1. Go to the next topic or module in the left-hand-side panel.
2. Practice in parallel as you watch the video.
3. Attempt all the given assignments.
4. Save your work in the same working directory as shown in the video.