



## 1 Online / Offline content

1. The online content of Spoken Tutorials can be accessed from :  
<http://spoken-tutorial.org/tutorial-search/>
2. You can also download the Spoken Tutorials for offline learning from :  
<http://spoken-tutorial.org/cdcontent/>
3. From this link download the FOSS categories in the language you wish to learn.
4. The Spoken Tutorial content will be downloaded as a zip file on your machine.
5. Extract the contents of the zip file & access them.  
(It is already available on your desktop)

## 2 Instructions to practise OpenFoam

1. Click on "Select FOSS" or "All FOSS Categories" drop-down and choose "OpenFOAM"
2. Click on "Select Language" or "All Languages" drop-down and choose the language (English, Hindi, Marathi ...) in which you wish to learn
3. Click on "Submit" button
4. You will see a list of tutorials based on your selection
5. Start with the first tutorial in the displayed list. You will typically do one tutorial at a time
6. Locate the topic, for example "Introduction to OpenFOAM" and click on it.
7. To view the tutorial, click on the Play icon which is located in the player.
8. The **Pre-requisite** will be visible below the player (only for Online contents).
9. **Outline, Assignments, Code Files and Slides** are available below the player.
10. Adjust the size of the browser in such a way that you are able to practice in parallel.
11. Strictly follow instruction sheet step by step. Updated instructions for latest software version are given in the tutorial videos.

12. Please do not copy paste any terminal commands from pdf to terminal. This causes errors. You have to manually type them in terminal prompt.
13. After following the instruction sheet and watching the tutorial, replicate the tutorial steps on your computer
14. Attempt the Assignments as instructed in the tutorial
15. Once the tutorial is complete, choose the next tutorial from the playlist which is located on the right side or below the player.

## 3 Installing and Running

1. If OpenFOAM is not installed, go through the installation instruction sheet to install latest version. Tutorial videos are updated to include changes for latest version.

## 4 System Requirements/Prerequisites

- Ubuntu Linux Operating System 14.04 and above
- OpenFOAM 5.0
- ParaView 5.0 and above

## 5 Source the Installation

- For multiple versions of OpenFOAM installed in same system, it is recommended to use an alias (an alternative name/label that refers to an item and can be used to locate/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. We have used the alias of5 for OpenFOAM 5 during installation
- Open the command terminal by pressing Ctrl+Alt+T keys together
- Type: of5 in the command terminal (For other versions of OpenFoam, commands will differ)