

1 Online / Offline content

1. The online content of Spoken Tutorials can be accessed from:
<https://spoken-tutorial.org/tutorial-search/>
2. You can also download the Spoken Tutorials for offline learning from:
<https://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.

2 The procedure to practise

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 steps 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 practise during the second hearing.

3 OpenFOAM version 7

1. Click on "Select FOSS" or "All FOSS Categories" drop-down and choose "OpenFOAM version 7".
2. Click on "Submit" button.
3. You will see a list of tutorials based on your selection.
4. Start with the first tutorial in the displayed list.

4 First tutorial: Overview of OpenFOAM

1. Locate the topic Overview of OpenFOAM and click on it.
2. To view the tutorial, click on the Play icon which is located in the player.

5 Common Instructions

1. The **Pre-requisite** will be visible below the player (only for Online contents).

2. **Outline, Assignments, Code Files** and **Slides** are available below the player.
3. Adjust the size of the browser in such a way that you are able to practise in parallel.
4. Instructions to use Code files in *any* tutorial:
 - (a) Click on the link "**Code Files**" located below the player and save it in your folder.
 - (b) Extract the downloaded ZIP file.
 - (c) You will see all the code/source files used in the particular tutorial.
 - (d) Use these files as per the instructions given in the particular tutorial.
5. Attempt the **Assignments** as instructed. Save your work in your folder.
6. The **Additional reading material** section contains the theory and calculation of various parameters in the respective tutorial.
7. Whenever asked to refer to the **Additional reading material**, pause the tutorial and refer to relevant portion of the **Additional reading material** before continuing with the rest of the tutorial.
8. Throughout this series, the **Username**, the **Hostname** and the full path to the **Working directory** are not shown in the **Linux terminal**.
9. A typical folder structure of an OpenFOAM case folder is given at the end of this document. Please refer to it when navigating between OpenFOAM files/folders while practising the tutorials.

6 Second tutorial: Installing OpenFOAM in Ubuntu Linux

1. If you are using Linux, locate the topic **Installing OpenFOAM in Ubuntu Linux**, and click on it.
2. Follow the instructions carefully in the tutorial to install OpenFOAM 7 on Ubuntu Linux OS.
3. The OpenFOAM 7 installation also installs ParaView 5.6.0 as a dependency.

7 Third tutorial: Setting-up a Test Case in OpenFOAM

1. Since it is advisable not to run case files directly in installation directory, we create a folder **run** in home directory to which we will copy the **tutorials** cases.
2. The lid-driven cavity case is available in the **tutorials** directory.

3. The working directory during the tutorial changes.

- (a) At 1:50 - The `make` directory command (`mkdir`) creates a `run` directory whose path is `/home/username/OpenFOAM/username-7/run`
- (b) At 2:00 - The change directory command (`cd`) makes the `run` directory the working directory. The `run` directory can be made the working directory using the command
`cd $FOAM_RUN`
- (c) At 2:32 - The copy command (`cp`) copies the `cavity` folder from the `tutorials` directory into the current working directory, i.e. the `run` directory.
The path of the `tutorials` directory is `/opt/openfoam7/tutorials`
The path of the `cavity` folder in the `tutorials` directory is `tutorials/incompressible/icoFoam/cavity/cavity`
- (d) At 3:28 - The working directory is changed to `0` folder in the `cavity` folder. The path of the current working directory is `/home/username/OpenFOAM/username-7/run/cavity/0`
- (e) At 5:06 - The working directory goes a level back in the directory hierarchy using the command `cd..`.
Now, the `cavity` folder is the working directory.
- (f) At 6:17 - The working directory is changed to `system` folder in the `cavity` folder. The path of the current working directory is `/home/username/OpenFOAM/username-7/run/cavity/system`
- (g) At 8:44 - The working directory goes a level back in the directory hierarchy to the `cavity` folder.

4. All the utility and solver commands such as `blockMesh`, `icoFoam`, `paraFoam` etc. can be executed from the OpenFOAM cases' parent directory, in this case the `cavity` folder.

8 Fourth tutorial: Creating 2D Channel Geometry and Mesh in OpenFOAM

1. The content of the `blockMesh` file used in this tutorial is available in the `Code Files` section.
2. Download extract the contents of the ZIP file.
3. Open the text file `blockMesh.txt`.
4. Copy the contents from the text file into the `blockMeshDict` file that you had opened earlier in a text editor.

9 Seventh tutorial: Simulating Hagen-Poiseuille flow through a pipe in OpenFOAM

1. The `blockMeshDict` file used in this tutorial is the same as the one in the tutorial `Creating 3D Pipe Geometry and Mesh`.
2. The contents of the `p`, `U` and `controlDict` files used in this tutorial are available in the `Code Files` section.
3. Copy the contents from each of the text files from the `Code Files` section into the respective file opened by you in a text editor.

10 Ninth tutorial: Simulation of a 2D Turbulent Flow in a Channel using OpenFOAM

1. The case files used in this tutorial is available in the `Code Files` section.
2. Download the file and extract it.
3. Once extracted, copy the extracted folder named `"kepsilon"` into the `run` directory using the `cp` command. The command will change depending upon the extracted location of the downloaded folder.

11 Tenth tutorial: Turbulence Modelling in OpenFOAM

1. The case files used in this tutorial is available in the `Code Files` section.
2. There are three cases used in this tutorial whose folders are namely, `kepsilon`, `komega` and `komegasst`
3. Once downloaded and extracted, the extracted location contains 3 ZIP files named `"kepsilon.zip"`, `"komega.zip"` and `"komegasst.zip"`
4. Extract each of the three ZIP files again such that the extracted location contains three folders of the names: `"kepsilon"`, `"komega"` and `"komegasst"`.
5. Copy each of the folders from the extracted location into the `run` directory using the appropriate command whenever instructed in the tutorial.

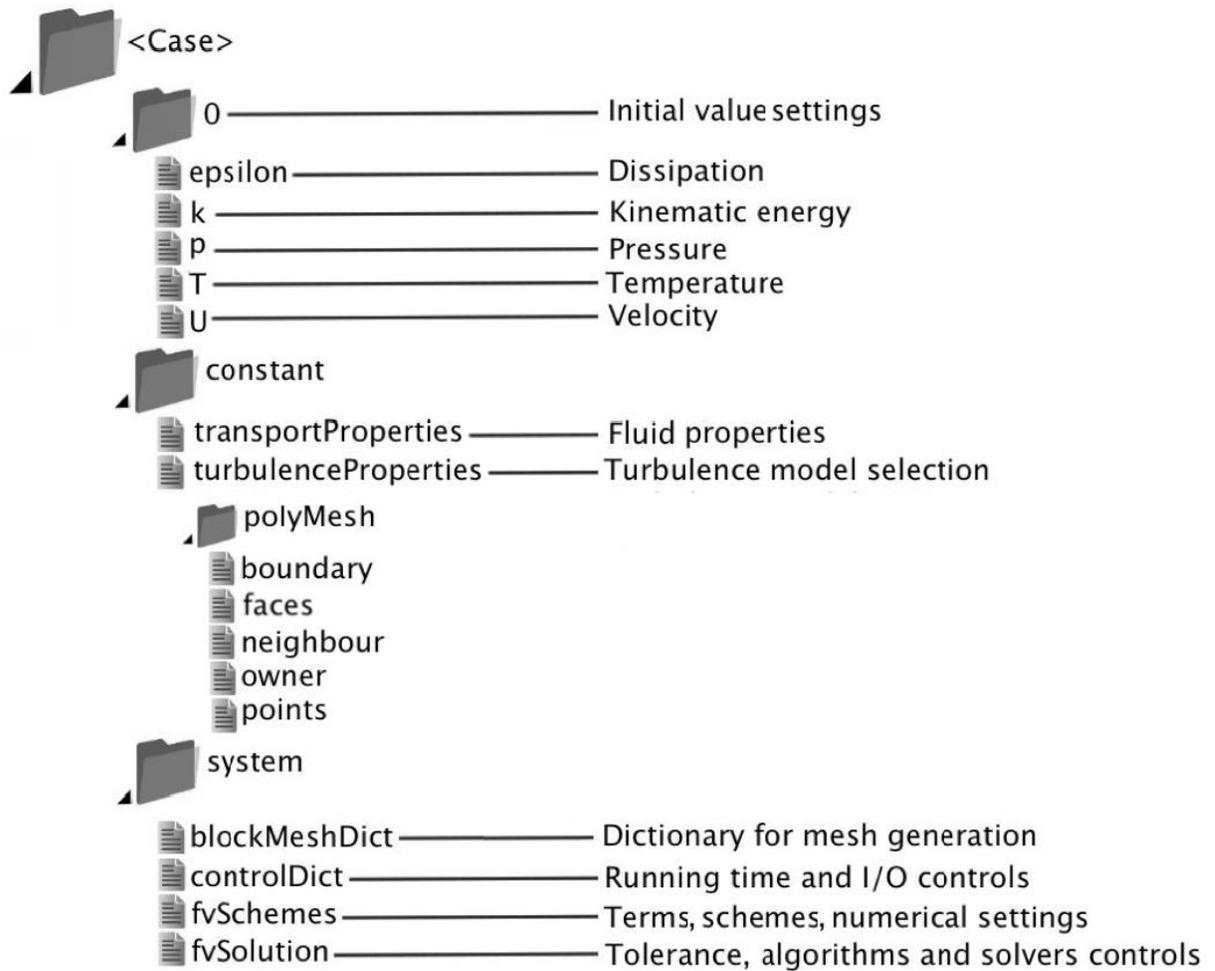


Figure 1: A typical file structure of an OpenFOAM case.
 (Reference: <https://www.cfd.at/>)