

# OpenFOAM Tutorials

## 1 Intro

1. Install a Linux distribution in your computer configuration (preferably the latest [Ubuntu LTS](#) version), either alongside with Windows (dual boot) or using a Virtual Machine. The next step is to familiarize with the Linux terminal [\[1, 2\]](#) and the Linux OS in general.
2. Successfully install the latest version of the OpenFOAM © open-source CFD software and the post-process tool Paraview © from the official website (<https://openfoam.org/>), following the installation instructions.
3. With the help of the latest version of the OpenFOAM User Guide, which can be found in the website, run the incompressible solver `icoFoam` to solve the lid-driven cavity case. In order to achieve that, wither manually copy the directory with the corresponding tutorial case, which can be found in the following path (e.g. for the OpenFOAM 8 version)

`/opt/openfoam8/tutorials/incompressible/icoFoam/cavity/cavity`

into your personal OpenFOAM directory that was created in the installation process, or run the command

```
$ cp -r $FOAM_TUTORIALS/incompressible/icoFoam/cavity/cavity $FOAM_RUN
```

To solve the cavity tutorial, run the following commands:

```
$ cd $FOAM_RUN/cavity
$ blockMesh
$ icoFoam
$ paraFoam
```

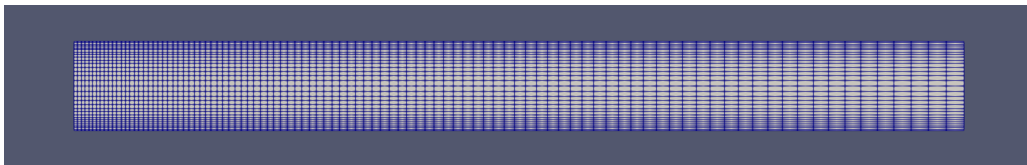
More information about the commands and their usage, can be found in the User Guide.

After the successful solution of the cavity case, you can run different simulations by altering the problem's geometry, the flow parameters `./constant/transportProperties`, the boundary conditions (`./0/` directory), the solution time step  $\Delta t$ , the solution writing times (`./system/controlDict`), etc. Also, familiarize yourselves with the ParaView post-processing tool, and its capabilities.

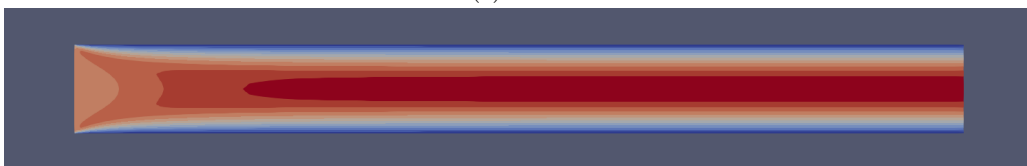
4. Modify the cavity case, from the previous step, in order to simulate the 2D laminar incompressible flow between parallel plates. It must be reminded that the maximum exit velocity must be equal to the 3/2 of the inlet velocity.

As for the boundary conditions, the velocity at the walls must be zero (`noSlip`), `uniform` (Dirichlet BC) at the inlet and `zeroGradient` (Neumann BC) at the outlet of the pipe. The pressure must be `zeroGradient` everywhere at the walls and inlet, except the pipe's outlet, where must be zero, as the fluid exits the pipe into the atmosphere (relative pressure).

The inlet and outlet boundaries type must be defined as 'patch' in the `blockMeshDict` file, while all walls could be defined as either 'patch' or 'wall'. Modify the mesh in the `blockMeshDict` so that it is denser towards the walls, and towards the pipe's inlet.



(a) Mesh



(b) Velocity

5. Extend the previous case, into a 3D square pipe.

