A quick guide for getting started with OpenFOAM

Vachan Potluri

March 9, 2023

Contents

1 Introduction

OpenFOAM is a software for performing numerical simulations.

1.1 OpenFOAM is a software …

More specifically, it is a free and **open source** software. That means the underlying files that are used to develop the software are given to users. These "underlying files" are called the **source code** . Advanced users often modify this source code for their custom applications.

open source

source code

OpenFOAM doesn't have a graphical user interface (GUI). The user communicates with OpenFOAM through files and commands. This makes it less suitable

for Windows users who are more familiar with GUI. However, OpenFOAM commands are very simple to use, and you can very quickly get familiar with them.

Since OpenFOAM does not have a GUI, it does not have a visualisation interface of its own, but is uses a software named ParaView for this purpose. OpenFOAM installs ParaView by itself without requiring any additional installation steps.

1.2 ...for performing numerical simulation

OpenFOAM can perform simulation of various kinds of physical problems: heat conduction, fluid flow, elastic deformation, combustion etc. OpenFOAM has different **solvers** for simulating each of these physical applications.

solvers

In addition to solvers, OpenFOAM also has some tools that are useful before and after a simulation. For example, if you want to simulate flow over a F1 race car, then OpenFOAM can also generate the mesh required for the simulation. After the simulation, it can calculate the net drag force on the car. Such tools are called **applications** .

applications

2   Installation

These links contain instructions for installation of OpenFOAM (version 10) in detail.

- Link for Ubuntu.

- Link for other Linux distributions.

- Link for Windows. For Windows, OpenFOAM runs within an application called "Windows Subsystem for Linux" (WSL). WSL is a virtual environment that allows running Linux softwares on Windows. These video tutorials by József Nagy are very helpful for installation and getting familiar with WSL: for Windows 10, for Windows 11.

3   The first simulation

First, create a new directory (folder) for performing simulations. We will call this the "run" directory.

```
> mkdir -p $FOAM_RUN
```

`$FOAM_RUN` is an "environment variable" that is assigned by OpenFOAM during installation. The command `mkdir` ("make directory") creates this directory.

Now, copy a heat conduction tutorial case into this run directory.

```
> cp -r $FOAM_TUTORIALS/basic/laplacianFoam/flange \
$FOAM_RUN/
```

`cp` (stands for "copy") copies files from a source to a destination. `$FOAM_TUTORIALS` is a directory where many tutorial cases provided by OpenFOAM reside.

Now navigate to this newly copied folder.

```
> cd $FOAM_RUN/flange
```

`cd` (stands for "change directory"). This command sets the folder copied above as the working directory.

You can notice that this folder contains an ANSYS mesh file with the name `flange.ans`. We will use this mesh for simulation. First, convert it into Open-FOAM's format.

```
> ansysToFoam flange.ans
```

`ansysToFoam` is an OpenFOAM application for reading ANSYS meshes.

`laplacianFoam` is the solver name for heat conduction simulation. Execute the solver to perform the simulation.

```
> laplacianFoam
```

Now, to visualise the results.

```
> paraFoam
```

This launches the ParaView software and also opens the results in the current directory.