

A quick guide to getting started with OpenFOAM

Vachan Potluri

March 9, 2023

Contents

1	Introduction	1
1.1	OpenFOAM is a software ...	1
1.2	...for performing numerical simulation	2
2	Installation	2
3	The first simulation	2

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.

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 it 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 in detail.

- For Ubuntu: <https://openfoam.org/download/10-ubuntu/>.
- For other Linux distributions: <https://openfoam.org/download/10-linux/>.
- For Windows: <https://openfoam.org/download/windows/>. Additionally, the [video tutorial by József Nagy](#) may also be helpful.

3 The first simulation