

OpenFOAM

- CFD based on FVM
- Open source Field Operation And Manipulation.
- C++ toolbox
- Open source and free
- Commercial and academic
- Chemical reactions, turbulence and heat transfer, to acoustics, solid mechanics and electromagnetics.
- Every six months in June and December.

Abbreviations

CFD: [Computational fluid dynamic](#)

FVM: [Finite volume method](#)

Is OF for Hrvoje Jasak (Thesis) or not?

Guide for Installing OpenFOAM by various methods

1. [On windows OS](#)

1.1. [Direct](#)

1.2. [Virtual machine](#)

2. [On ubuntu OS](#)

3. [On Mac OS](#)

4. [Dual boot](#)

Abbreviations

OS: [Operating System](#)

Mac: [Macintosh](#)

Install OF by VM method

1. [Download a VM \(VMware workstation\)](#)
2. [Install VM on a OS \(VMware workstation\)](#)
3. [Install Ubuntu on VM \(VMware workstation\)](#)
4. [Install OF6 on Ubuntu](#)
 - 4.1. [Online](#)
 - 4.2. [Source Pack](#)

Note #1: [Which OF version can be installed on a OS?](#)

Note #2: Foamentu (Ubuntu 18.04 LTS + OF6 + PV + S4F + PF) can be installed instead of Ubuntu.

Abbreviations

VM: [Virtual Machine](#)

OF: [OpenFOAM](#)

PV: [ParaView](#)

S4F: [Swak4foam](#)

PF: [PyFOAM](#)

Difference between OF and Foam-extend

- FE and OF's foundation are not the same.
- It is an open project welcoming and integrating contributions from all users and developers.
- There are no significant differences in their codes.
- Main features of FE compared with OF (FEM, FSI)

Abbreviations

FE: [Foam-extend](#)

FEM: [Finite volume method](#)

FSI: [Fluid-structure interaction](#)

Learning sources for OF

1. [Nima Samkhaniani's Book](#)
2. [Chalmers university of technology \(Hakan Nilsson\)](#)
3. [Teymour Javaherchi's cases](#)
4. [OpenFOAM documentations](#)
5. [CFD direct](#)

Debugging sources

1. [CFD Online's Forums](#)
2. [OpenFOAM](#)
3. [OpenFOAM wiki](#)