

OpenFOAM

PENG

October 30, 2026

Contents

1	Introduction	1
2	Tutorials	1
3	Applications and Libraries	2
4	cases	2
5	Mesh Generation and Conversion	2
6	Boundary and Initial Conditions	2
7	Models and Physical Properties	3
8	Post-processing	3
A	Tools	3

1 Introduction

Definition 1.1 (openfoam). *<https://openfoam.org/resources/>*

OpenFOAM is a free, open source computational fluid dynamics software to solve complex fluid flows involving turbulence and solid dynamics.

Components of OpenFOAM:

- solvers: perform calculation in fluid
- utilities: config simulation, geometry, mesh, and process results

2 Tutorials

Definition 2.1 (environment).

```
# tutorial directory
$FOAM_TUTORIALS
```

```
# user run directory
$FOAM_RUN
mkdir -p $FOAM_RUN
```

3 Applications and Libraries

4 cases

5 Mesh Generation and Conversion

Definition 5.1 (mesh). *OpenFOAM always operates in a three dimensional Cartesian coordinate system and all geometries are generated in three dimensions.*

*OpenFOAM solves cases in two dimensions by specifying **empty** boundary condition on boundaries normal to the third dimension for which no solution is required.*

Definition 5.2 (blockMesh). *blockMesh generates meshes from a blockMeshDict file located in system directory for a given case.*

```
// -*- C++ -*-
// Website: https://openfoam.org
// Version: openfoam-dev

FoamFile // sub-dictionary
{
    format    ascii
    class     dictionary
    location  "system"
    object    blockMeshDict
}

units [mm]; // optional
// the generated mesh is written in the base units for the underlying unit system, i.e. meter

vertices
(
    (x1 y1 z1) // vertex 1
    ...
);

blocks
(
    hex (v1 ... v8)
      (cells_x cells_y cells_z)
      simpleGrading (g_x g_y g_z)
      ...
);

boundary
(
    name
    {
        type T;
        faces
        (
            (v1 ... v4)
            ...
        );
    }
);
```

6 Boundary and Initial Conditions

Definition 6.1 (initial field).

```
// case starts at t = 0
// initial field data is stored in a 0 sub-directory
```

Definition 6.2 (p).

7 Models and Physical Properties

8 Post-processing

Definition 8.1 (paraview).

```
touch foam.foam
```

```
// Camera Parallel Projection
```

A Tools

Definition A.1 (installation). *<https://openfoam.org/download/source/>*

```
cd ~/OpenFOAM/  
git clone git@github.com:OpenFOAM/OpenFOAM-dev.git --depth=1  
git clone git@github.com:OpenFOAM/ThirdParty-dev.git --depth=1  
# git fetch --depth=1 && git reset --hard origin
```

```
cd OpenFOAM-dev  
source "$HOME/OpenFOAM/OpenFOAM-dev/etc/bashrc"
```

```
./Allwmake
```