

Introduction to OpenFOAM

Vachan Potluri

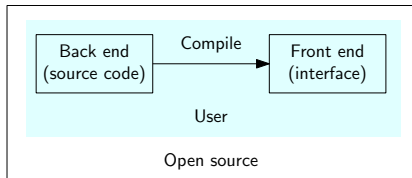
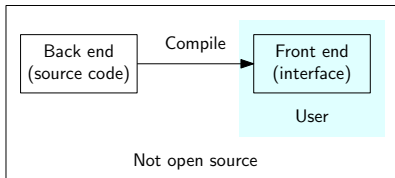
April 2023

What

- ▶ CFD software (but without GUI)
- ▶ **Open** source **F**ield **O**peration **A**nd **M**anipulation
- ▶ Open source \implies source code is given to user

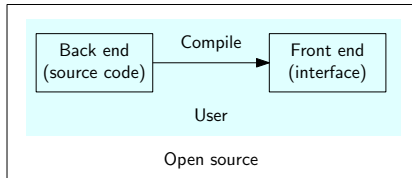
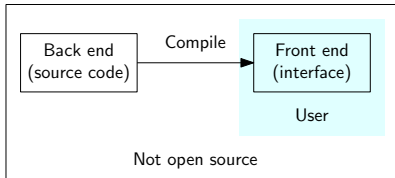
What

- ▶ CFD software (but without GUI)
- ▶ **Open** source **F**ield **O**peration **A**nd **M**anipulation
- ▶ Open source \implies source code is given to user



What

- ▶ CFD software (but without GUI)
- ▶ **Open** source **F**ield **O**peration **A**nd **M**anipulation
- ▶ Open source \implies source code is given to user



No GUI \implies hard to learn

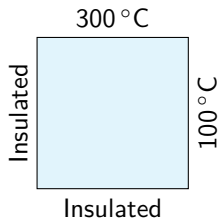
Why

- ▶ Free
- ▶ Fast
- ▶ User customisable

Case 1

Heat conduction in a square plate

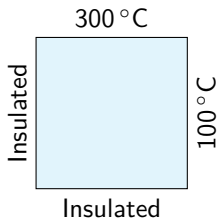
$$\frac{\partial T}{\partial t} = \nabla \cdot (D_T \nabla T)$$



Case 1

Heat conduction in a square plate

$$\frac{\partial T}{\partial t} = \nabla \cdot (D_T \nabla T)$$



- ▶ `laplacianFoam` is the “solver” to be used for heat conduction equation
- ▶ Visualise using `paraFoam`

OpenFOAM's case setup

For doing `laplacianFoam` simulation, create a folder with this structure

Case structure for `laplacianFoam`

