Introduction to OpenFOAM

Vachan Potluri
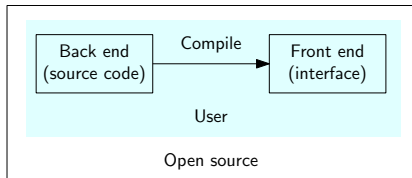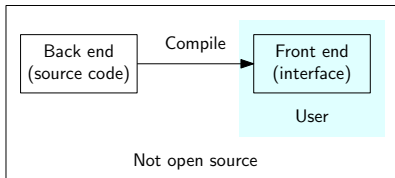
April 2023

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
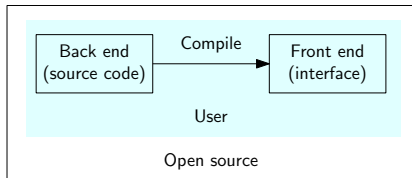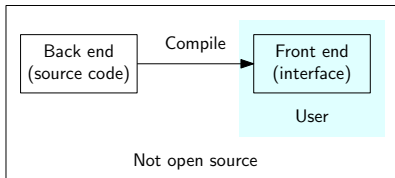- Open source $\implies$ source code is given to user

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
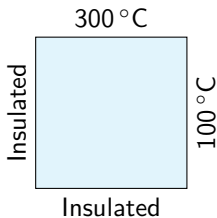

No GUI $\implies$ hard to learn

Why

- ▶ Free
- ▶ Fast
- ▶ User customisable

Case 1
Heat conduction in a square plate

$$\frac{\partial T}{\partial t} = D_T \left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} \right)$$

300 °C

Insulated

100 °C

Insulated

Other settings

- $D_T = 1\,\mathrm{m}^2/\mathrm{s}$
- $L = 1\,\mathrm{m}$
- End time 5 s

Case 1
Heat conduction in a square plate

$$\frac{\partial T}{\partial t} = D_T \left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} \right)$$
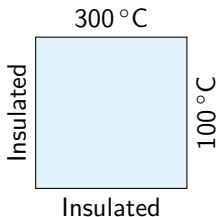

300 °C

Insulated

100 °C

Insulated

Other settings

- $D_T = 1\,\mathrm{m^2/s}$
- $L = 1\,\mathrm{m}$
- End time 5 s

- laplacianFoam is the "solver" to be used for heat conduction equation
- Visualise using paraFoam
    - Mesh representation
    - Data array selection
    - Navigating times

OpenFOAM's simulation setup

Case structure for laplacianFoam

```
case_name/
├── 0/
│   └── T
├── constant/
│   ├── physicalProperties
│   └── polyMesh/*
└── system/
    ├── controlDict
    ├── fvSchemes
    └── fvSolution
```

OpenFOAM's simulation setup

Case structure for `laplacianFoam`

```
case_name/
├── 0/
│   └── T
├── constant/
│   ├── physicalProperties
│   └── polyMesh/*
└── system/
    ├── controlDict
    ├── fvSchemes
    └── fvSolution
```

► Every simulation is setup using certain "setting" files

► These files are grouped into 3 folders: `0/`, `constant/` and `system/`

► All are text files

OpenFOAM's simulation setup

Case structure for `laplacianFoam`

```
case_name/
├── 0/
│   └── T
├── constant/
│   ├── physicalProperties
│   └── polyMesh/*
└── system/
    ├── controlDict
    ├── fvSchemes
    └── fvSolution
```

- ▶ Every simulation is setup using certain "setting" files

- ▶ These files are grouped into 3 folders: `0/`, `constant/` and `system/`

- ▶ All are text files

- ▶ Only mandatory files are shown here, there can be additional files also

- ▶ When a simulation is done, OpenFOAM generates additional time files

- ▶ We will learn about these files by doing some variations of the square plate simulation

Case 1, variation 1
Change the thermal diffusivity

- ▶ In `constant/physicalProperties`, you can change D_T

- ▶ Things to note
  - FoamFile "header"
  - Units of D_T

- ▶ Try
  - Reduce D_T to $0.01\,\mathrm{m^2/s}$ and see the solution evolution

- ▶ Tips
  - `foamListTimes -rm` deletes all time folders other than 0/
  - Reload files in ParaView by right-clicking

Case 1, variation 2

Change the boundary conditions

▶ In `0/T` you can set initial condition (IC) and boundary conditions (BCs)

▶ Things to note
  • `internalField` $\implies$ IC
  • Names of different boundaries
  • `type` of different boundaries $\implies$ type of BC

▶ Special BC type `empty`
  • By default `OpenFOAM` does 3d simulations
    ◦ Check this in `ParaView`
  • `empty` BC in a direction tells `OpenFOAM` to not consider that direction

▶ Try
  • Change bottom BC to $-100\,^{\circ}\text{C}$
  • Change IC
  • Do a 1d simulation by setting top and bottom boundaries as `empty`
    ◦ Verify the solution using "Plot Over Line" in `ParaView` or `gradTx` value

Case 1, variation 3
Change time settings

- ▶ `system/controlDict` contains all the main controls of the simulation

- ▶ Things to note
    - `startFrom`, `stopAt`
    - `startTime`, `endTime`
    - `deltaT`
    - `writeControl`, `writeInterval`

- ▶ For heat conduction equation
    - "Stable" time step value (Δt_s) satisfies $D_T \dfrac{\Delta t_s}{\Delta x^2} < 1$
    - To determine end time (t_e), compare it with diffusion time scale

    $$t_e \gg \frac{L^2}{D_T} \implies \text{Steady state reached}$$

- ▶ Try
    - Increase D_T to $10\,\mathrm{m^2/s}$
    - For correct simulation, time step and end time also have to be changed

Case 1, variation 4
Change the mesh

- Yo