

Introduction to OpenFOAM

Vachan Potluri

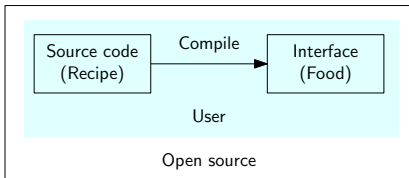
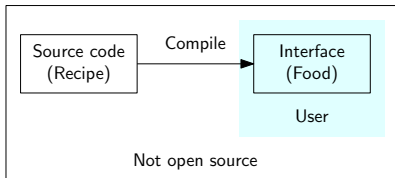
April 2023

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

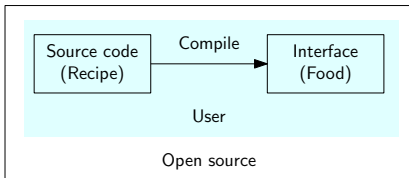
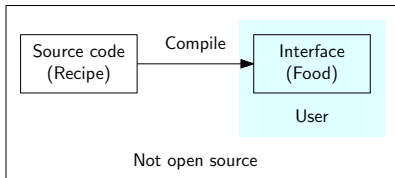
What

- ▶ 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 **Field** **O**peration **A**nd **M**anipulation
- ▶ Open source \implies source code is given to user



No GUI \implies hard to learn

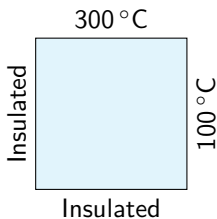
Why

- ▶ Free
- ▶ Fast
- ▶ User customisable: solve any equation you desire

Case 1

Heat conduction in a square plate

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



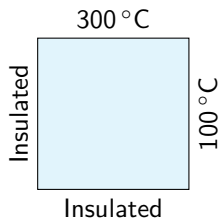
Other settings

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

Case 1

Heat conduction in a square plate

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



Other settings

- ▶ $D_T = 1 \text{ m}^2/\text{s}$
- ▶ $L = 1 \text{ m}$
- ▶ 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
```

- ▶ 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 corresponding 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 $0.01 \text{ m}^2/\text{s}$ 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 O/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°C
 - Change IC
 - Do a 1d simulation by setting top and bottom boundaries as `zeroGradient`
 - 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 $D_T \frac{\Delta t_s}{\Delta x^2} < 1$
 - 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 $10 \text{ m}^2/\text{s}$
 - For correct simulation, time step and end time also have to be changed

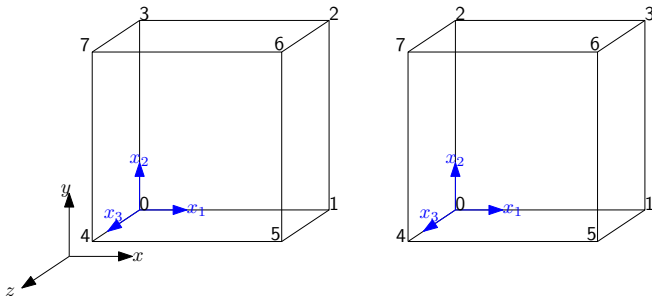
Case 1, variation 4

Change the mesh

- ▶ Multiple ways to create mesh in OpenFOAM
 - Create in a different software (e.g. ANSYS) and import to OpenFOAM
 - Use OpenFOAM's tools to create mesh
- ▶ blockMesh is one such tool in OpenFOAM to create meshes
- ▶ blockMesh reads an additional file system/blockMeshDict to create mesh
- ▶ system/blockMeshDict has 3 components
 - ① Specify points or vertices
 - ② Create "blocks" using these vertices
 - ③ Define boundaries using the vertices

- OpenFOAM defines a local coordinate system (LCS) for every block
- Blocks are created using a list of points ordered in a specific way
 - ① Point 0 is the origin of (LCS)
 - ② Line 0-1 is along x_1 direction
 - ③ Line 1-2 is along x_2 direction
 - ④ Points 0-3 define plane $x_3 = 0$
 - ⑤ Points 4-7 are obtained by translating points 0-3 in x_3 direction

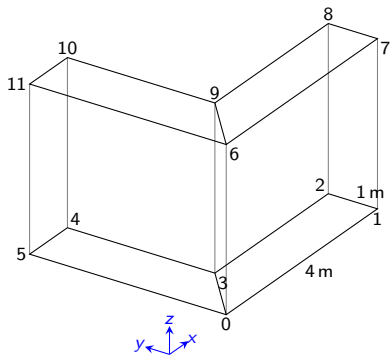
One of these is wrong.



- ▶ Boundaries are defined using a list of points such that right hand thumb rule points outward the block
- ▶ The boundary names in `system/blockMeshDict` and O/T must match
- ▶ Try
 - Increasing mesh resolution
 - Using mesh grading
 - Changing geometry
- ▶ Tips
 - Run `blockMesh` command to update mesh
 - Run `checkMesh` command to check if the mesh has no issues
 - You can view different mesh regions (e.g. boundaries) individually in ParaView

Case 2

Heat conduction in an L-clamp



- This geometry can be constructed using two blocks
 - ① Block 1: (0 1 2 3 6 7 8 9)
 - ② Block 2: (0 3 4 5 6 9 10 11)
- Use single cell in z direction and perpendicular to clamp

- Tip: you can see the mesh in ParaView without running the simulation
- BCs
 - One end (4 5 11 10) of clamp at 100°C
 - Other end (1 7 8 2) at 0°C
 - All other boundaries insulated
 - empty BC for top and bottom planes
- Use $D_T = 1 \text{ m}^2/\text{s}$; set end time and time step accordingly

$$D_T \frac{\Delta t_s}{\Delta x} \leq 1$$

$$t_e \gg \frac{L^2}{D_T}$$