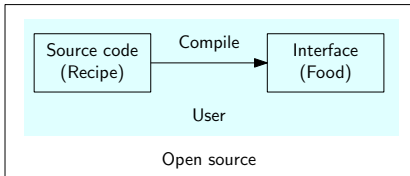# Introduction to `OpenFOAM`

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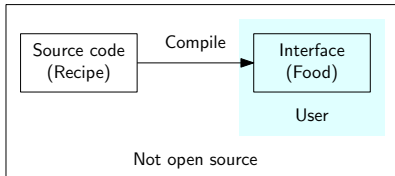- ▶ Open source $\implies$ source code is given to user
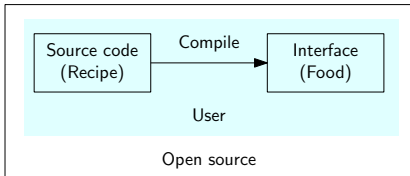
What

- ▶ 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: solve any equation you desire

What

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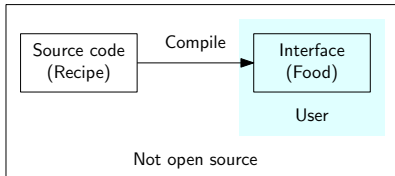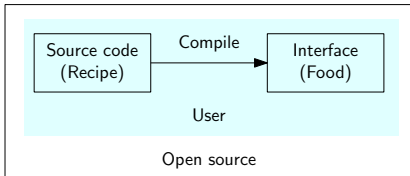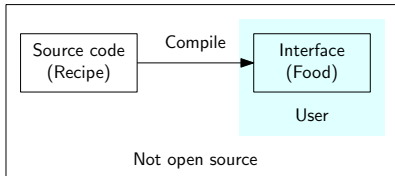- ▶ Open source $\implies$ source code is given to user

No GUI $\implies$ hard to learn

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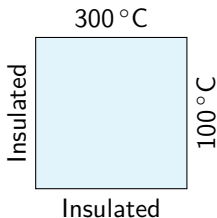
- ▶ Free
- ▶ Fast
- ▶ User customisable: solve any equation you desire

Let's jump right into doing some simulations

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

- $D_T = 1\,\mathrm{m}^2/\mathrm{s}$
- $L = 2\,\mathrm{m}$
- End time $10\,\mathrm{s}$

Case 1
Heat conduction in a square plate

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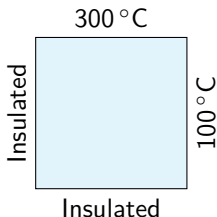
Other settings

- $D_T = 1\,\text{m}^2/\text{s}$
- $L = 2\,\text{m}$
- End time $10\,\text{s}$

- `laplacianFoam` is the "solver" to be used for heat conduction equation
- Visualise using `parafoam -builtin`
  - Mesh representation
  - Data array selection
  - Navigating times
  - Changing color map

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
- Units in OpenFOAM are specified using powers of 7 fundamental units: [kg m s K mol A cd]
  - [1 1 -2 0 0 0 0] is kg m/s^2 $\implies$ unit of force
  - [0 0 1 0 0 1 0] is A s $\implies$ unit of charge
  - [1 2 2 -1 -1 0 0] is J/mol K $\implies$ unit of universal gas constant

▶ Try
- Reduce D_T to $0.01\,\text{m}^2/\text{s}$ and see the solution evolution
- Can you guess whether the solution will evolve faster or slower?

▶ 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 $-100\,^{\circ}\mathrm{C}$
  - 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 $D_T \dfrac{\Delta t_s}{\Delta x^2} < 1$
  - To determine end time (t_e) for steady state:

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

- ▶ Try
  - Increase D_T to $10 \, \text{m}^2/\text{s}$
 - For correct simulation, time step also has 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 (p_0 p_1 p_2 p_3 p_4 p_5 p_6 p_7 p_8) 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

Suppose a block is defined using (0 1 2 3 4 5 6 7).
Which one of these is wrong?
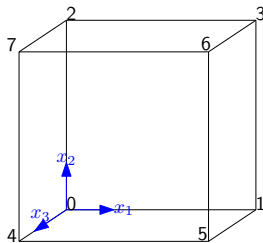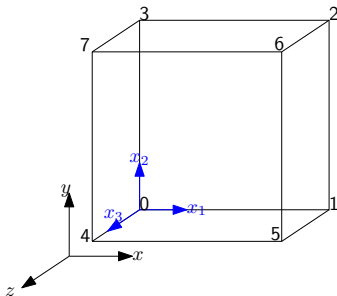
- ▶ Boundaries are defined using a list of points such that right hand thumb rule points outward the block

- ▶ The boundary names in `system/blockMeshDict` and `0/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
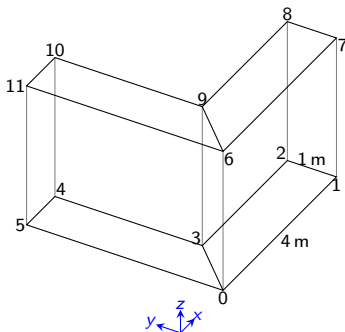
▶ This geometry can be constructed using two blocks
  ❶ Block 1: (0 1 2 3 6 7 8 9)
  ❷ Block 2: (0 3 4 5 6 9 10 11)

▶ 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 $100\,^\circ$C
  • Other end (1 7 8 2) at $0\,^\circ$C
  • All other boundaries insulated
  • empty BC for top and bottom planes

▶ Use $D_T = 1\,\mathrm{m^2/s}$; set end time and time step accordingly

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

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