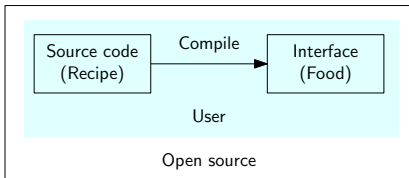# Introduction to `OpenFOAM`

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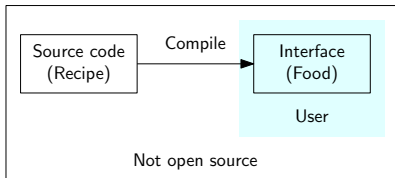- ▶ Open source $\implies$ source code is given to user
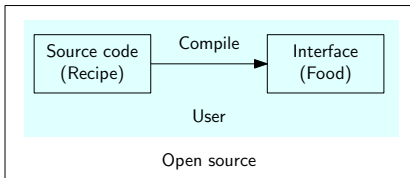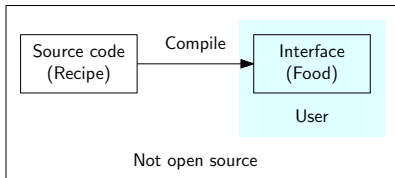
What

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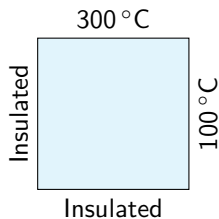

No GUI $\implies$ hard to learn

Why

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

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

Case 1
Heat conduction in a square plate

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

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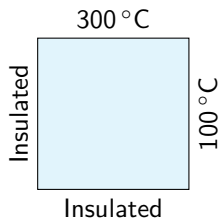- $D_T = 1\,\mathrm{m^2/s}$
- $L = 1\,\mathrm{m}$
- End time $5\,\mathrm{s}$

- `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 $0.01\,\mathrm{m^2/s}$ 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 $-100\,^{\circ}$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), compare it with diffusion time scale

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

- ▶ Try
  - Increase D_T to $10\,\mathrm{m}^2/\mathrm{s}$
 - 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.
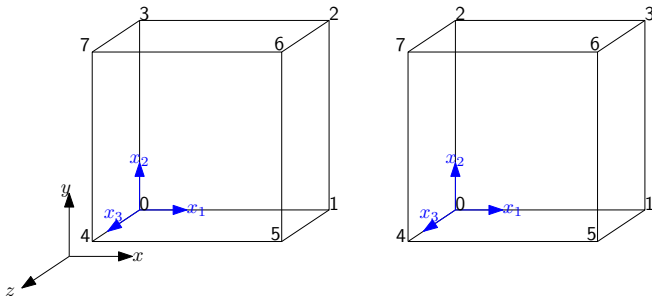
- ▶ Boundaries are defined using a list of points such that right hand thumb rule points outward the block

- ▶ The boundary names in `system/blockMeshDict` and `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
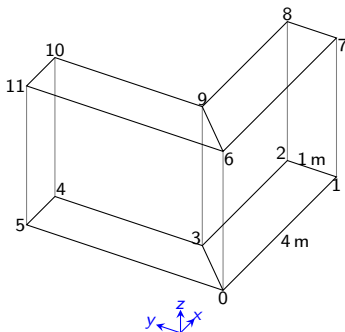
- ▶ This geometry can be constructed using two blocks
  1. Block 1: (0 1 2 3 6 7 8 9)
  2. Block 2: (0 3 4 5 6 9 10 11)
- ▶ Use single cell in z direction and perpendicular to clamp

- ▶ Tip: you can see the mesh in ParaView without running the simulation
- ▶ BCs
  - One end (4 5 11 10) of clamp at $100\,^{\circ}$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

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

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