

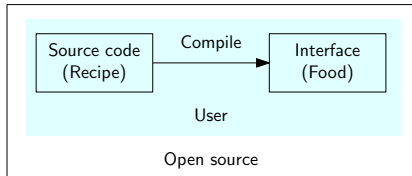
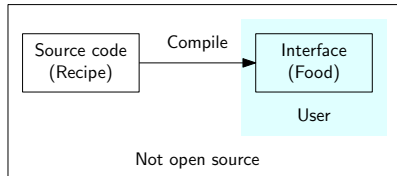
# Introduction to OpenFOAM

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



No GUI  $\implies$  hard to learn

## Why

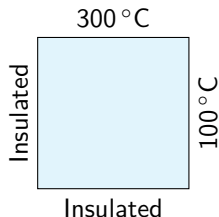
- ▶ Free
- ▶ Fast
- ▶ User customisable: solve any equation you desire

Let's jump right away 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

- ▶  $D_T = 1 \text{ m}^2/\text{s}$
- ▶  $L = 2 \text{ m}$
- ▶ End time 10 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 the setting files 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  $\text{kg m/s}^2 \implies$  force
    - $[0\ 0\ 1\ 0\ 0\ 1\ 0]$  is  $\text{A s} \implies$  charge
    - $[1\ 2\ 2\ -1\ -1\ 0\ 0]$  is  $\text{J/mol K} \implies$  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 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  $-100^{\circ}\text{C}$
  - 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 ( $\Delta t_s$ ) satisfies  $D_T \frac{\Delta t_s}{\Delta x^2} < 1$
  - 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

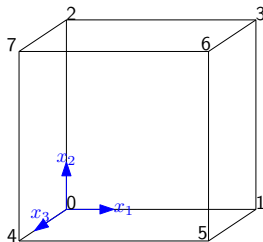
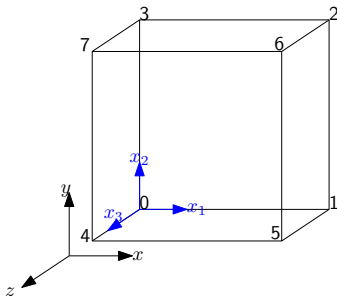
## Change the mesh

- ▶ 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 ( $p_0$   $p_1$   $p_2$   $p_3$   $p_4$   $p_5$   $p_6$   $p_7$ ) ordered in a specific way
  - ① Point  $p_0$  is the origin of LCS
  - ② Line  $p_0$ - $p_1$  is along  $x_1$  direction
  - ③ Line  $p_1$ - $p_2$  is along  $x_2$  direction
  - ④ Points  $p_0$ - $p_3$  define plane  $x_3 = 0$
  - ⑤ Points  $p_4$ - $p_7$  are obtained by translating points  $p_0$ - $p_3$  in  $x_3$  direction

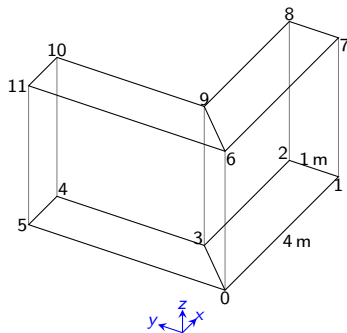
Suppose a block is defined using ( $p_0$   $p_1$   $p_2$   $p_3$   $p_4$   $p_5$   $p_6$   $p_7$ ).  
Which one of these is correct?



- ▶ 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:

( $p_0$   $p_1$   $p_2$   $p_3$   $p_6$   $p_7$   $p_8$   $p_9$ )

② Block 2:

( $p_0$   $p_3$   $p_4$   $p_5$   $p_6$   $p_9$   $p_{10}$   $p_{11}$ )

### ► BCs

- One end ( $p_4$   $p_5$   $p_{11}$   $p_{10}$ ) of clamp at  $100^\circ\text{C}$
- Other end ( $p_1$   $p_2$   $p_8$   $p_7$ ) at  $0^\circ\text{C}$
- All other boundaries insulated
- empty BC for top and bottom planes

- Tip: you can see the mesh in ParaView without running the simulation

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

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

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