

Introduction to OpenFOAM

Vachan Potluri

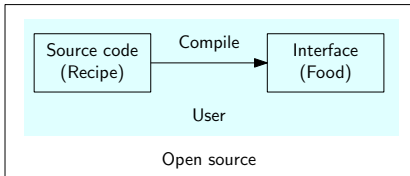
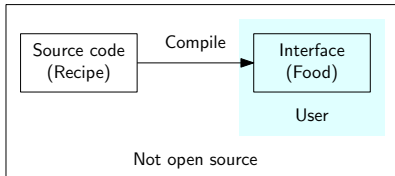
April 2023

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

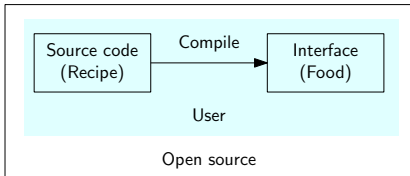
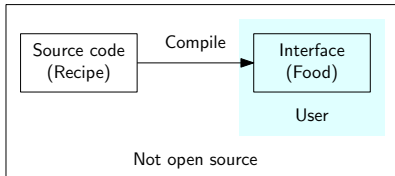
What

- ▶ CFD software (but without GUI)
- ▶ **Open** source **Field** **Operation** **And** **Manipulation**
- ▶ 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



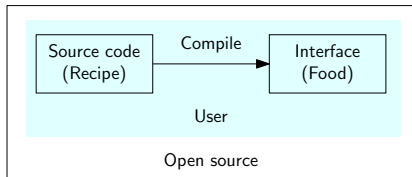
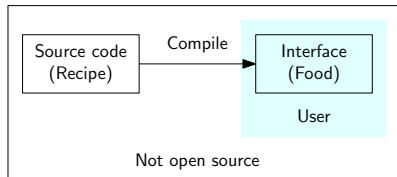
No GUI \implies hard to learn

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

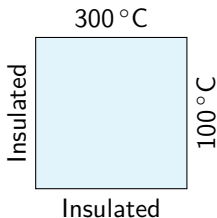
- ▶ Free
- ▶ Fast
- ▶ User customisable: solve any equation you desire

Let's jump right away into doing some simulations

Case 1

Heat conduction in a square plate

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



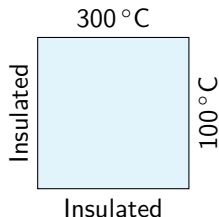
Other settings

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

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

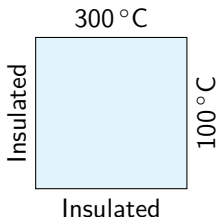
- ▶ $D_T = 1 \text{ m}^2/\text{s}$
- ▶ $L = 2 \text{ m}$
- ▶ End time 10 s

- ▶ `laplacianFoam` is the “solver” to be used for heat conduction equation

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

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

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

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 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

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

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`

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) for steady state:

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

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) 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

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

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

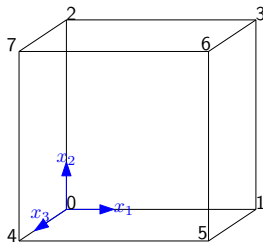
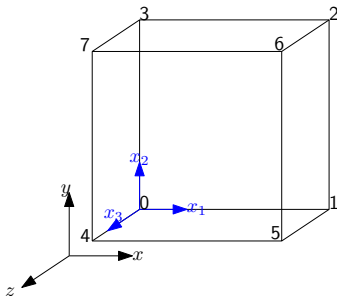
Case 1, variation 4

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?

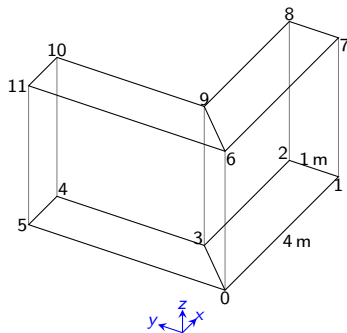


- ▶ Boundaries are defined using a list of points such that right hand thumb rule points outward the block
- ▶ The boundary names in `system/blockMeshDict` and O/T must match

- ▶ 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

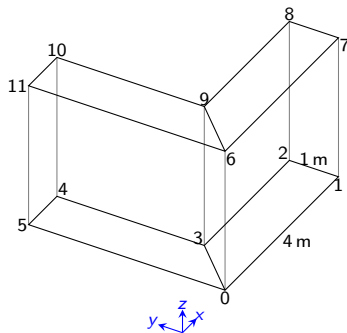
Case 2

Heat conduction in an L-clamp



Case 2

Heat conduction in an L-clamp



- This geometry can be constructed using two blocks

① Block 1:

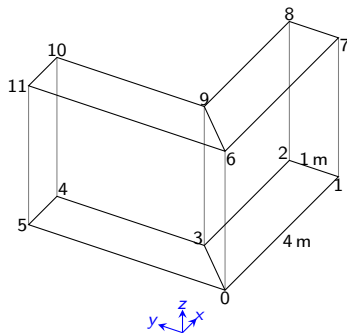
$(p_0 \ p_1 \ p_2 \ p_3 \ p_6 \ p_7 \ p_8 \ p_9)$

② Block 2:

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

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:

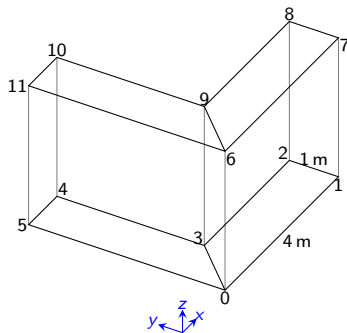
$(p_0 \ p_3 \ p_4 \ p_5 \ p_6 \ p_9 \ p_{10} \ p_{11})$

► BCs

- One end ($p_4 \ p_5 \ p_{11} \ p_{10}$) of clamp at 100°C
 - Other end ($p_1 \ p_2 \ p_8 \ p_7$) at 0°C
 - All other boundaries insulated
 - empty BC for top and bottom planes
- Tip: you can see the mesh in ParaView without running the simulation

Case 2

Heat conduction in an L-clamp



- This geometry can be constructed using two blocks

❶ Block 1:

(p_0 p_1 p_2 p_3 p_6 p_7 p_8 p_9)

❷ 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°C
- Other end (p_1 p_2 p_8 p_7) at 0°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

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

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