Introduction to OpenFOAM

Vachan Potluri

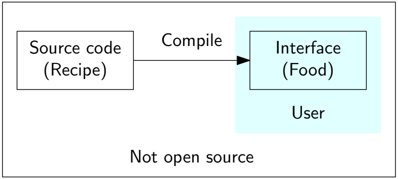
April 2023

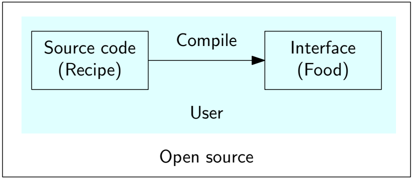
What

- 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

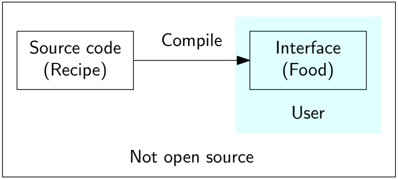
- CFD software (but without GUI)
- **Open** source **F**ield **O**peration **A**nd **M**anipulation
- Open source $\implies$ source code is given to user

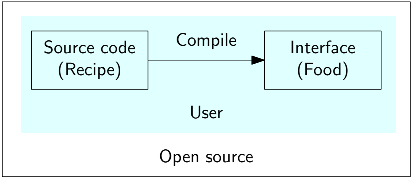




What

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





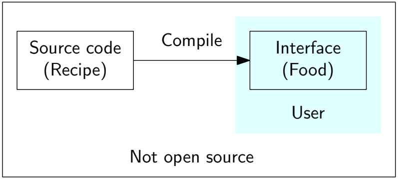
No GUI $\implies$ hard to learn

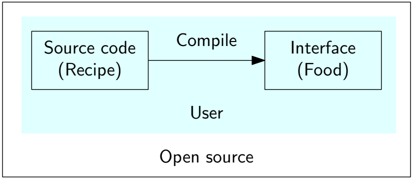
Why

- 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

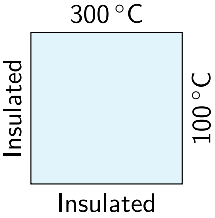
- Free
- Fast
- User customisable: solve any equation you desire

Let's jump right away into doing some simulations

Case 1

Heat conduction in a square plate

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



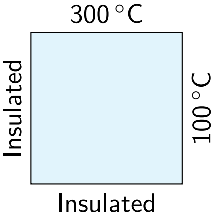
Other settings

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

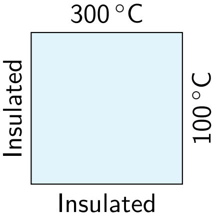
- $D_T = 1\,\mathrm{m^2/s}$
- $L = 2\,\mathrm{m}$
- End time 10 s

- `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

- $D_T = 1\,\mathrm{m}^2/\mathrm{s}$
- $L = 2\,\mathrm{m}$
- 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 $\mathrm{kg\,m/s^2} \implies$ force
    - [0 0 1 0 0 1 0] is $\mathrm{A\,s} \implies$ charge
    - [1 2 2 -1 -1 0 0] is $\mathrm{J/mol\,K} \implies$ 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 $\mathrm{kg\,m/s^2} \implies$ force
    - [0 0 1 0 0 1 0] is $\mathrm{A\,s} \implies$ charge
    - [1 2 2 -1 -1 0 0] is $\mathrm{J/mol\,K} \implies$ universal gas constant
- Try
  - Reduce D_T to $0.01\,\mathrm{m^2/s}$ 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` $\Longrightarrow$ IC
  - Names of different boundaries
  - `type` of different boundaries $\Longrightarrow$ 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` $\Longrightarrow$ IC
  - Names of different boundaries
  - `type` of different boundaries $\Longrightarrow$ 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 `0/T` you can set initial condition (IC) and boundary conditions (BCs)
- Things to note
  - `internalField` $\Longrightarrow$ IC
  - Names of different boundaries
  - `type` of different boundaries $\Longrightarrow$ 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`

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:

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

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 $10\,\mathrm{m^2/s}$
  - 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

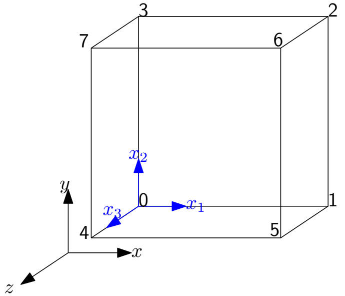
Change the mesh

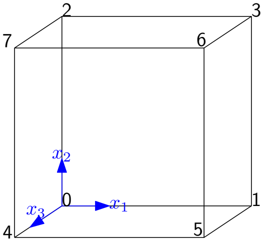
- Multiple ways to create mesh in `OpenFOAM`
  - Create in a different software (e.g. ANSYS) and import to `OpenFOAM`
  - Use `OpenFOAM`'s tools to create mesh
- `blockMesh` is one such tool in `OpenFOAM` to create meshes
- `blockMesh` reads an additional file `system/blockMeshDict` to create mesh

- 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?



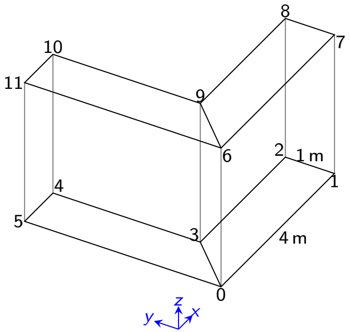


- Boundaries are defined using a list of points such that right hand thumb rule points outward the block
- The boundary names in `system/blockMeshDict` and `0/T` must match

- 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 `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`

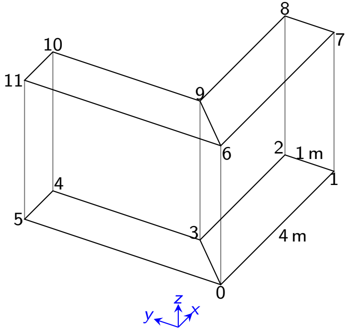
Case 2

Heat conduction in an L-clamp



Case 2

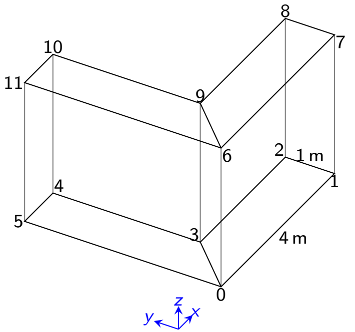
Heat conduction in an L-clamp



- 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

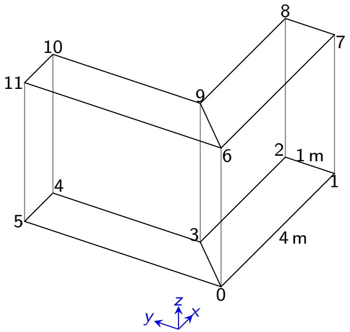


- This geometry can be constructed using two blocks
  1. Block 1: $(p_0 \; p_1 \; p_2 \; p_3 \; p_6 \; p_7 \; p_8 \; p_9)$
  2. Block 2: $(p_0 \; p_3 \; p_4 \; p_5 \; p_6 \; p_9 \; p_{10} \; p_{11})$

- BCs
  - One end $(p_4 \; p_5 \; p_{11} \; p_{10})$ of clamp at $100\,^{\circ}\mathrm{C}$
  - Other end $(p_1 \; p_2 \; p_8 \; p_7)$ at $0\,^{\circ}\mathrm{C}$
  - 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
  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 $100\,^\circ\mathrm{C}$
  - Other end $(p_1\ \ p_2\ \ p_8\ \ p_7)$ at $0\,^\circ\mathrm{C}$
  - All other boundaries insulated
  - `empty` BC for top and bottom planes
- Tip: you can see the mesh in `ParaView` without running the simulation
- Use $D_T = 1\,\mathrm{m}^2/\mathrm{s}$; set end time and time step accordingly

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

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