

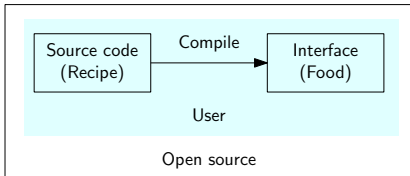
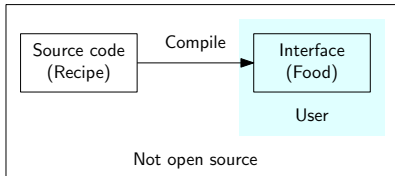
Introduction to OpenFOAM

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

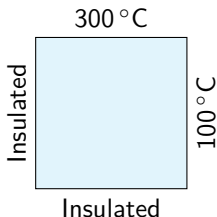
- ▶ Free
- ▶ Fast
- ▶ User customisable: solve any equation you desire

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°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 $D_T \frac{\Delta t_s}{\Delta x^2} < 1$
 - 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

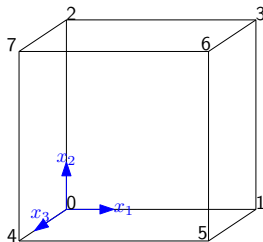
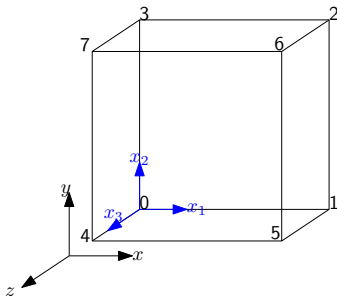
Case 1, variation 4

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

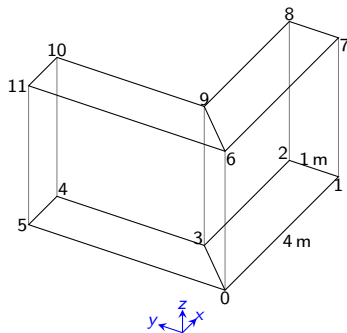
Suppose a block is defined using (p_0 p_1 p_2 p_3 p_4 p_5 p_6 p_7).
Which one of these is correct?



- ▶ 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°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

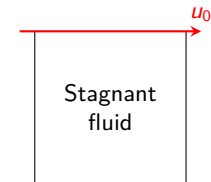
- Use $D_T = 1 \text{ m}^2/\text{s}$; set end time and time step accordingly

$$D_T \frac{\Delta t_s}{\Delta x^2} \leq 1$$

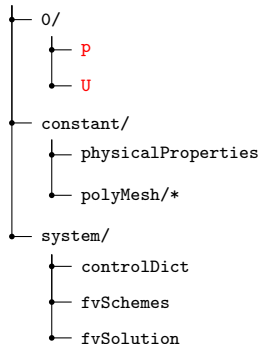
$$t_e \gg \frac{L^2}{D_T}$$

Case 3

Lid driven cavity



case_name/



$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$$

$$\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = -\frac{\partial(p/\rho)}{\partial x} + \nu \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right)$$

$$\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} = -\frac{\partial(p/\rho)}{\partial y} + \nu \left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right)$$

- ▶ icoFoam is one solver for “transient laminar” incompressible flow
 - p in icoFoam is p/ρ
- ▶ We will setup this case from scratch using OpenFOAM’s “tutorial cases”
- ▶ And learn some techniques in ParaView
 - Extracting 2d slice of a 3d domain
 - Plot over line
 - Visualising streamlines

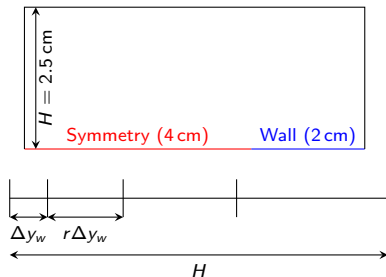
Case 4

Incompressible flat plate boundary layer

Incompressible flow simulation of air over a flat plate

$$\rho_{\infty} = 0.09719 \text{ kg/m}^3, \quad u_{\infty} = 149.3 \text{ m/s}$$

$$p_o = 63.71 \text{ kPa}, \quad \nu = 1.493 \times 10^{-4} \text{ m}^2/\text{s}$$



- Estimate BL thickness

$$\frac{\delta}{x} = \frac{5}{\sqrt{\text{Re}_x}} \approx 0.7 \text{ mm}$$

- Use $\Delta y_w \approx 0.1 \text{ mm}$ and grade the mesh away from wall

$$\frac{H}{\Delta y_w} = \frac{r^n - 1}{r - 1}$$

- Also grade in x direction to have nearly square cells at leading edge
- Time step and end time:

$$\frac{u\Delta t_s}{\Delta x} < 1, \quad t_e \gg \frac{L}{u_{\infty}}$$

- ▶ OpenFOAM has many “post-processing” tools
- ▶ Post-processing is what you do after a simulation
- ▶ Suppose we wish to compare these results with Blasius solution

$$c_f := \frac{\tau_w}{\frac{1}{2}\rho_\infty u_\infty^2} = \frac{0.664}{\sqrt{\text{Re}_x}}$$
$$\delta_2 := \int_0^\infty \frac{u}{u_\infty} \left(1 - \frac{u}{u_\infty}\right) = 0.665 \sqrt{\frac{\nu x}{u_\infty}}$$

- ▶ We need two things
 - ① Compute shear stress from velocity data
 - ② Obtain the simulation data on wall and at a given x location
- ▶ Use `postProcess -func "grad(U)"` for the first
- ▶ For the second, use `postProcess -func -<filename>` with the corresponding file system/
 - Alternatively, ParaView can be used to export “Plot Over Line” data

fvSchemes and fvSolution

```
case_name/  
├── 0/  
│   └── T  
├── constant/  
│   ├── physicalProperties  
│   └── polyMesh/*  
└── system/  
    ├── controlDict  
    ├── fvSchemes  
    └── fvSolution
```

$$\frac{\partial T}{\partial t} = D_T \nabla^2 T$$

Discretisation

$$AT = b$$

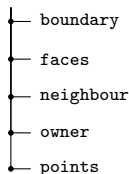
Linear algebra
solution

Solution

- ▶ fvSchemes: discretisation procedure
- ▶ fvSolution: linear algebra solution procedure
- ▶ Not a good idea to change these
- ▶ Simply use the settings from tutorial cases

polyMesh folder

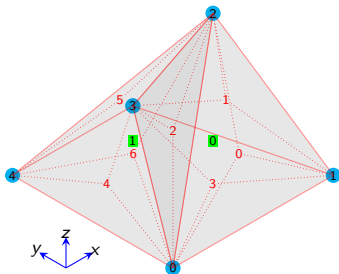
polyMesh/



► OpenFOAM-specific mesh format

- Each cell can have any number of faces
- Each face can have any number of edges
- Highly flexible!

- **points**: contains a list of all points in the mesh
- **faces**: list of all faces in the mesh
 - Every face is generated using a list of points
 - Ordering is such that normal points away from **owner** cell
 - For a face common to two cells, OpenFOAM assigns cell with smallest id as “owner”
- **owner**: list of cell ids that “own” the faces
- **neighbour**: list of cell ids that are neighbour to the faces
- **boundary**: boundary faces information



points	faces	owner	neighbour
(0,0,0) // 0	(0,1,2) // 0	0	-1
(1,0,0)	(1,3,2)	0	-1
(0,-0.25,1.5)	(2,3,0)	0	1
(0.25,0.5,0.5)	(0,3,1)	0	-1
(0,1,0) // 4	(0,4,3)	1	-1
	(2,3,4)	1	-1
	(0,2,4) // 6	1	-1

► boundary file needs to be edited when using empty BC

Resources

- ▶ <https://openfoam.org/download/windows/>: installation on Windows
 - A sure-shot procedure
 - Follow every line of this guide with utmost attention
- ▶ OpenFOAM has many versions. Two major ones are
 - <https://openfoam.org/> (I was using this)
 - <https://www.openfoam.com/>
- ▶ Both these websites have links to many useful resources
 - [Basic linux guide](#)
 - [User guide](#). Also available as pdf in your installation at `$FOAM_INST_DIR/OpenFOAM-10/doc/Guides/`
- ▶ User guide is the most resourceful place to start for a beginner
- ▶ Some youtube channels are very helpful
 - [Wolf Dynamics OpenFOAM-9 tutorials](#) (linear, very detailed)
 - [CFD basics](#), [CFD intermediate](#) and [many other playlists](#) by József Nagy (be careful with versions while following these)
- ▶ When you face an error, search for its solution in these Q&A websites
 - <https://askubuntu.com/>: Ubuntu related issues
 - <https://www.cfd-online.com/Forums/openfoam/>: an unofficial OpenFOAM user forum (some posts are decades old here!)