CFD using OpenFOAM
Lecture 3: Essential Governing Laws & OpenFOAM Implementation

Part I : Computational Heat Conduction

Instructor : Sumant R Morab (Ph.D Research Scholar)
Co-ordinator : Prof. Janani S Murallidharan
Indian Institute of Technology, Bombay

Outline

Governing Laws

Energy Conservation

FVM Numerical Methodology

OpenFOAM Implementation

OpenFOAM Illustration

- Assuming Incompressible fluid : (CV $\rightarrow$ Control Volume)

- Assuming Incompressible fluid : (CV $\rightarrow$ Control Volume)
- Mass Conservation :
  " Net amount of mass entering the CV = Net amount of mass leaving the CV (over a period of time Δt) "

- Assuming Incompressible fluid : (CV $\rightarrow$ Control Volume)
- Mass Conservation :
  " Net amount of mass entering the CV = Net amount of mass leaving the CV (over a period of time Δt) "
- Momentum Conservation :
  " Net amount increase in momentum inside a CV = Net amount of advected momentum entering CV + Net amount of viscous and pressure impulse in positive x/y/z direction "

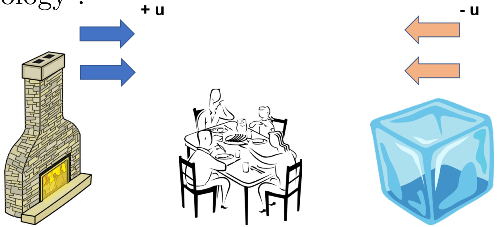
Governing Laws in CFD & HT

- Assuming Incompressible fluid : (CV $\rightarrow$ Control Volume)
- Mass Conservation :
  " Net amount of mass entering the CV = Net amount of mass leaving the CV (over a period of time Δt) "
- Momentum Conservation :
  " Net amount increase in momentum inside a CV = Net amount of advected momentum entering CV + Net amount of viscous and pressure impulse in positive x/y/z direction "
- Energy Conservation (1^{st} Law of Thermodynamics):
  " Increase in energy within CV = Net amount of energy entering into CV + amount energy generated within CV "

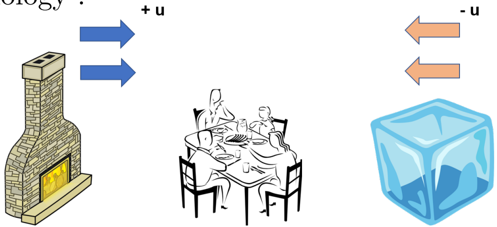
- Before going into mathematical definition for Energy conservation, let us look at some terminology :

- Before going into mathematical definition for Energy conservation, let us look at some terminology :

- Before going into mathematical definition for Energy conservation, let us look at some terminology :

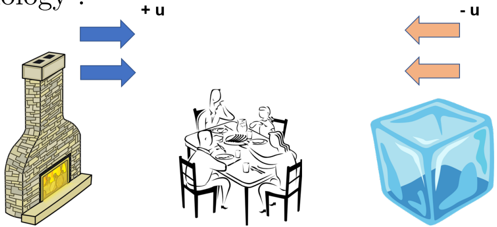


- Before going into mathematical definition for Energy conservation, let us look at some terminology :



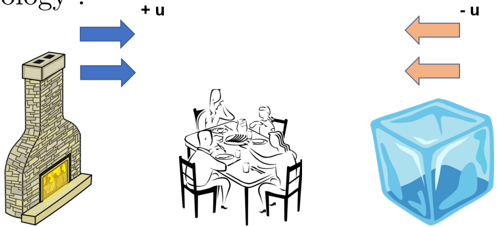
- Conduction : Energy transfer through random vibration of molecules.

- Before going into mathematical definition for Energy conservation, let us look at some terminology :



- Conduction : Energy transfer through random vibration of molecules.
- Advection : Energy transfer due to bulk motion of molecules.

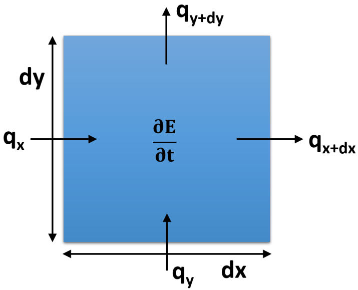
- Before going into mathematical definition for Energy conservation, let us look at some terminology :



- Conduction : Energy transfer through random vibration of molecules.
- Advection : Energy transfer due to bulk motion of molecules.
- Convection : Conduction + Advection

- Consider a Control Volume (CV) as shown in figure $\rightarrow$
- Physical Conservation Law : Net rate of convected (conduction + advection) thermal energy entering a CV in time-interval Δt = Rate of increase of enthalpy stored within CV

- Consider a Control Volume (CV) as shown in figure $\rightarrow$
- Physical Conservation Law : Net rate of convected (conduction + advection) thermal energy entering a CV in time-interval Δt = Rate of increase of enthalpy stored within CV

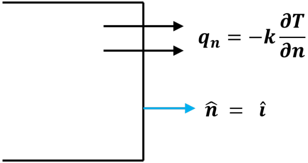


$$\frac{\partial E}{\partial t} = \frac{q_x - q_{x+dx}}{dx} + \frac{q_y - q_{y+dy}}{dy} \quad (1)$$

$$\implies \frac{\partial E}{\partial t} = -\frac{\partial q_x}{\partial x} - \frac{\partial q_y}{\partial y} \quad (2)$$

- In-order to convert energy equation in terms of temperature, we need Fourier Law :

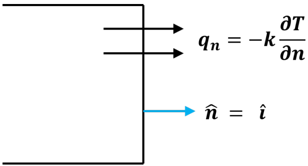
- In-order to convert energy equation in terms of temperature, we need Fourier Law :



$$q_n = -k\frac{\partial T}{\partial n} \tag{3}$$

where k $\rightarrow$ Thermal Conductivity, n $\rightarrow$ normal direction

- In-order to convert energy equation in terms of temperature, we need Fourier Law :

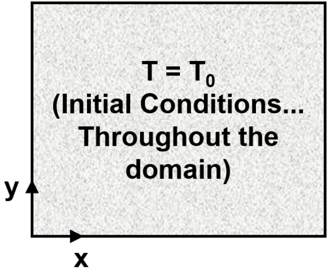


$$q_n = -k\frac{\partial T}{\partial n} \tag{3}$$

where k → Thermal Conductivity, n → normal direction

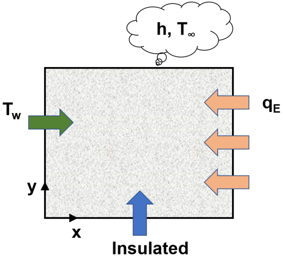
- Also, Assuming total energy E = $\rho C_p T$ (for incompressible solids/fluids)

$$\frac{\partial(\rho C_p T)}{\partial t} = k\left[\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2}\right] \tag{4}$$

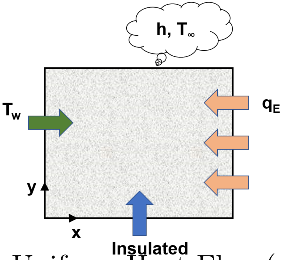


- Initial Conditions (IC) $\rightarrow$ The variable values at t= 0 (starting of simulations) for whole domain.

Initial and Boundary Conditions



- Initial Conditions (IC) → The variable values at t= 0 (starting of simulations) for whole domain.
- Boundary Conditions (BC) → condition of variables all over boundary at all time-steps.



- Initial Conditions (IC) $\rightarrow$ The variable values at t= 0 (starting of simulations) for whole domain.
- Boundary Conditions (BC) $\rightarrow$ condition of variables all over boundary at all time-steps.

1. Uniform Heat Flux (q_n) :

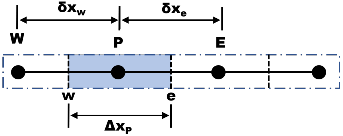
$$\frac{\partial T}{\partial n} = 0 \tag{5}$$

2. Insulated $q_n = 0$
3. Convective Boundary :

$$-k\frac{\partial T}{\partial n} = h(T - T_\infty) \tag{6}$$

- Consider a 1D domain as shown. It is required to obtain algebraic equation for steady state energy conservation at CV - 'P'.

$$k\frac{\partial^2 T}{\partial x^2} + \dot{Q}_{gen,vol} = 0$$

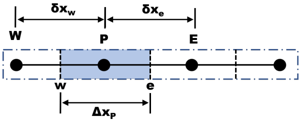


$$\implies \int_V k\frac{\partial^2 T}{\partial x^2}dV + \int_V \dot{Q}_{gen,vol}dV = 0$$

Using Gauss-divergence theorem i.e, $\int_V \frac{\partial \phi}{\partial n}dV = \int_S \phi\hat{n}.dS$

$$\implies \int_w^e k\frac{\partial T}{\partial x}dy.dz + \dot{Q}_{gen,vol}\Delta x.1.1 = 0$$

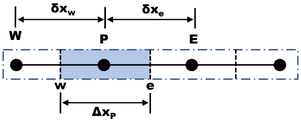
- Consider a 1D domain as shown. It is required to obtain algebraic equation for steady state energy conservation at CV - ‘p’.



$$k\left[\left(\frac{\partial T}{\partial x}\right)_e - \left(\frac{\partial T}{\partial x}\right)_w\right] + \dot{Q}_{gen,vol}\Delta x = 0$$

$$\implies k\left[\frac{T_E - T_P}{\delta x_e} - \frac{T_P - T_W}{\delta x_w}\right] + \dot{Q}_{gen,vol}\Delta x = 0$$

- Consider a 1D domain as shown. It is required to obtain algebraic equation for steady state energy conservation at CV - 'p'.



$$k\left[\left(\frac{\partial T}{\partial x}\right)_e - \left(\frac{\partial T}{\partial x}\right)_w\right] + \dot{Q}_{gen,vol}\Delta x = 0$$

$$\implies k\left[\frac{T_E - T_P}{\delta x_e} - \frac{T_P - T_W}{\delta x_w}\right] + \dot{Q}_{gen,vol}\Delta x = 0$$

- Using the above formulation, Linear Algebraic Equation can be obtained as follows :

$$a_P T_P + a_W T_W + a_E T_E + b = 0 \tag{7}$$

OpenFOAM Format of Equations

- Let us check general Laplacian solution implementation in OpenFOAM
- Go to → /opt/openfoam7/applications/solvers/basic/laplacianFoam
- Open laplacianFoam.C file

OpenFOAM Format of Equations

- Let us check general Laplacian solution implementation in OpenFOAM
- Go to → /opt/openfoam7/applications/solvers/basic/laplacianFoam
- Open laplacianFoam.C file

```
while (simple.correctNonOrthogonal())
{
    fvScalarMatrix TEqn
    (
        fvm::ddt(T) - fvm::laplacian(DT, T)
    ==
        fvOptions(T)
    );

    fvOptions.constrain(TEqn);
    TEqn.solve();
    fvOptions.correct(T);
}
```

- ‘**fvm**’ option is used create matrix of co-efficients i.e, a_P, a_W, a_E .... through FV discretisation technique discussed earlier

- Let us consider a tutorial example to understand solution schemes used by OpenFOAM
- Go to → /opt/openfoam7/tutorials/basic/laplacianFoam/flange/system

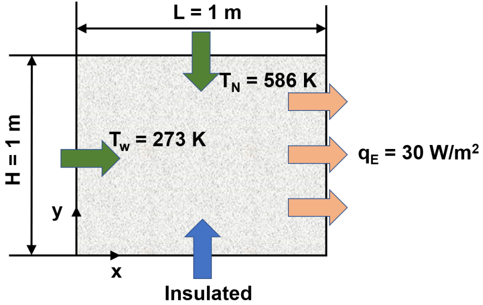
- Let us consider a tutorial example to understand solution schemes used by OpenFOAM
- Go to → /opt/openfoam7/tutorials/basic/laplacianFoam/flange/system
- Open fvSolution & fvSchemes files

```
solvers
{
    T
    {
        solver          PCG;
        preconditioner  DIC;
        tolerance       1e-06;
        relTol          0;
    }
}
```

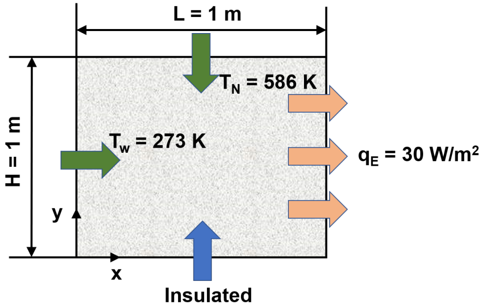
```
ddtSchemes
{
    default         Euler;
}

gradSchemes
{
    default         Gauss linear;
    grad(T)         Gauss linear;
}
```

- Consider a 2D Unsteady state heat-conduction problem as shown :



- Consider a 2D Unsteady state heat-conduction problem as shown :



Parameter	Value
Thermal Conductivity (k)	16.2 W/mK
Density (ρ_s)	7750 kg/m^3
Heat capacity (Cp)	500 J/kg.K

- How to run ?
- Download the zip file given with video
- Extract the files

- How to run ?
- Download the zip file given with video
- Extract the files
- In the terminal, type ‘blockMesh’ to generate mesh file

- How to run ?
- Download the zip file given with video
- Extract the files
- In the terminal, type ‘blockMesh’ to generate mesh file
- Run ‘laplacianFoam’ solver and check the results using ‘paraFoam’ command.

- How to run ?
- Download the zip file given with video
- Extract the files
- In the terminal, type ‘blockMesh’ to generate mesh file
- Run ‘laplacianFoam’ solver and check the results using ‘paraFoam’ command.
- Check 0/T file to see the implementation of various boundary conditions.

- How to run ?
- Download the zip file given with video
- Extract the files
- In the terminal, type 'blockMesh' to generate mesh file
- Run 'laplacianFoam' solver and check the results using 'paraFoam' command.
- Check 0/T file to see the implementation of various boundary conditions.

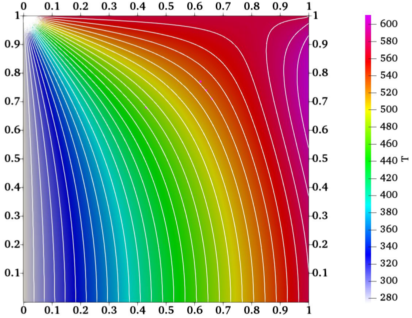
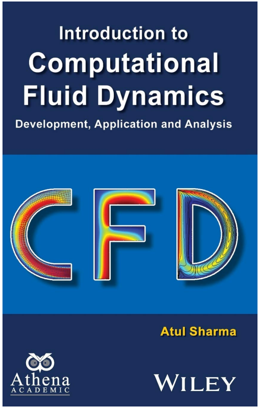


Figure: Steady State Temperature Contour

References

1. Sharma, A. (2016). Introduction to computational fluid dynamics: development, application and analysis. John Wiley & Sons.
2. https://www.openfoam.com/



Thank you for listening!

Sumant Morab