CFD using OpenFOAM
Lecture 4: Essential Governing Laws & OpenFOAM Implementation

Part II : Computational Heat Convection

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

Outline

Part I Recap

Convection - Introduction

Governing Law & PDE

FVM Numerical Methodology

OpenFOAM Implementation

OpenFOAM Illustration

- Governing Equations :

$$\frac{\partial(\rho C_p T)}{\partial t} = k\left[\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2}\right] \tag{1}$$

- Governing Equations :

$$\frac{\partial(\rho C_p T)}{\partial t} = k\left[\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2}\right] \tag{1}$$

- Discretization : Finite Volume Methodology

$$a_P T_P + a_W T_W + a_E T_E + b = 0 \tag{2}$$

- Governing Equations :

$$\frac{\partial(\rho C_p T)}{\partial t} = k\left[\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2}\right] \tag{1}$$

- Discretization : Finite Volume Methodology

$$a_P T_P + a_W T_W + a_E T_E + b = 0 \tag{2}$$

- Solution Methodology : Iterative scheme with implicit time-stepping

- Governing Equations :

$$\frac{\partial(\rho C_p T)}{\partial t} = k\left[\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2}\right] \tag{1}$$

- Discretization : Finite Volume Methodology

$$a_P T_P + a_W T_W + a_E T_E + b = 0 \tag{2}$$

- Solution Methodology : Iterative scheme with implicit time-stepping
- Illustrative Problem : Unsteady 2D heat conduction in a metallic slab.

- Consider flow over bike engine as shown

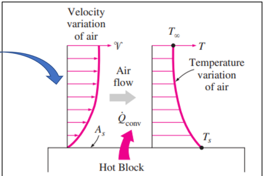
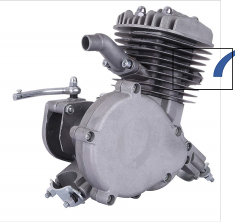


FIGURE 1–31
Heat transfer from a hot surface to air by convection.

Source : Cengel, Heat Transfer [2]

- Consider flow over bike engine as shown



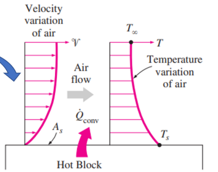


FIGURE 1–31
Heat transfer from a hot surface to air by convection.

Source : Cengel, Heat Transfer [2]

- What if bike stopped and air speed is 0 ? - pure conduction.

- Consider flow over bike engine as shown

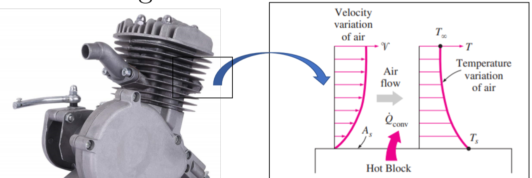


FIGURE 1–31
Heat transfer from a hot surface to air by convection.

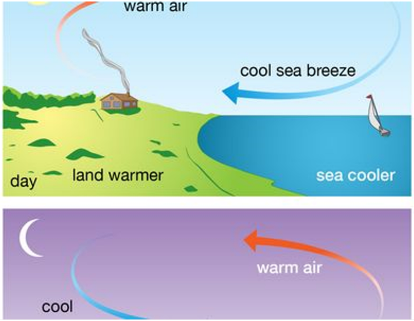
Source : Cengel, Heat Transfer [2]

- What if bike stopped and air speed is 0 ? - pure conduction.
- ***Convection*** - “ Energy transfer between fluid/solid interfaces in motion due vibration of molecules(conduction) & bulk motion of fluid(Advection) ”

1. **Day and Night Breeze** : The temperature of land surface is maintained due to convection between land surface and air along with ocean surface and air as shown

1. **Body Temperature Regulation** : the heat generated by cells tissue cells is carried by blood which moves through arteries and veins.

- Variables Involved : Temperature + Flow (velocities)



Source : Britannica [3]

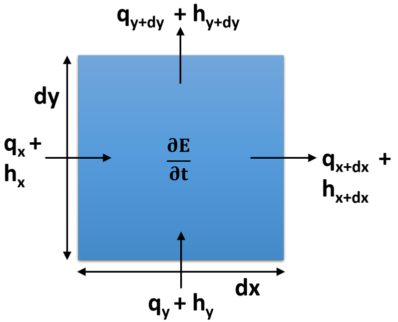
Energy Conservation with Convection

- Governing Law : Over a time-interval Δt, net amount of convected thermal energy entering the C.V + heat generated within C.V = amount of increase of Enthalpy(ΔE) stored within C.V

$$\Longrightarrow \left[Q^{net,in}_{cond} + Q^{net,in}_{adv}\right] + Q_{gen,vol} = \frac{\partial E}{\partial t} \quad (3)$$

$$\Longrightarrow \frac{\partial E}{\partial t} = -\frac{\partial(q_x + h_x)}{dx} - \frac{\partial(q_y + h_y)}{dy} \quad (4)$$

$$\Longrightarrow \frac{\partial(\rho C_p T)}{\partial t} + \frac{\partial h_x}{\partial x} + \frac{\partial h_y}{\partial y} = -\frac{\partial q_x}{\partial x} - \frac{\partial q_y}{\partial y} \quad (5)$$



Energy Conservation (Convection) - Continuous PDE Form

- Using Fourier law and advected enthalpy flux expansion, the continuous form of energy conservation in terms of Temperature can be derived as follows :

- Using Fourier law and advected enthalpy flux expansion, the continuous form of energy conservation in terms of Temperature can be derived as follows :

$$q_x = -k\frac{\partial T}{\partial x} \tag{6}$$

$$q_y = -k\frac{\partial T}{\partial y} \tag{7}$$

- Using Fourier law and advected enthalpy flux expansion, the continuous form of energy conservation in terms of Temperature can be derived as follows :

$$q_x = -k\frac{\partial T}{\partial x} \quad (6)$$

$$q_y = -k\frac{\partial T}{\partial y} \quad (7)$$

$$h_x = m_x C_p T = \rho u_x C_p T \quad (8)$$

$$h_y = m_y C_p T = \rho u_y C_p T \quad (9)$$

- Using Fourier law and advected enthalpy flux expansion, the continuous form of energy conservation in terms of Temperature can be derived as follows :

$$q_x = -k\frac{\partial T}{\partial x} \quad (6)$$

$$q_y = -k\frac{\partial T}{\partial y} \quad (7)$$

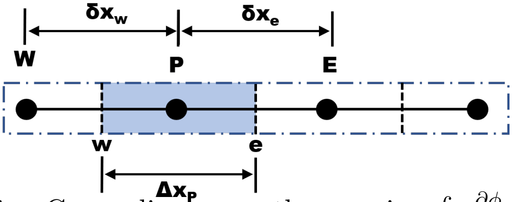
$$h_x = m_x C_p T = \rho u_x C_p T \quad (8)$$

$$h_y = m_y C_p T = \rho u_y C_p T \quad (9)$$

$$\frac{\partial(\rho C_p T)}{\partial t} + \frac{\partial(\rho u_x C_p T)}{\partial x} + \frac{\partial(\rho u_y C_p T)}{\partial y} = k\left[\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2}\right] \quad (10)$$

- Along with Temperature, flow variables need to be calculated at faces of C.V

- Consider a 1D domain as shown. It is required to obtain algebraic equation for steady state energy conservation at CV - ‘P’ (Assume that x-velocity ‘u’ is known).



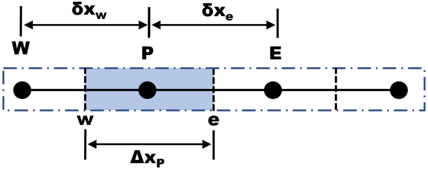
$$k\frac{\partial^2 T}{\partial x^2} = \rho C_p u \frac{\partial T}{\partial x}$$

$$\implies \int_V k\frac{\partial^2 T}{\partial x^2} dV = \rho C_p \int_V u\frac{\partial T}{\partial x} dV$$

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

$$\implies \int_w^e k\frac{\partial T}{\partial x} dy.dz - \rho C_p \int_w^e uT dy.dz = 0$$

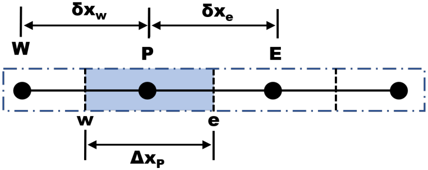
- Consider a 1D domain as shown. It is required to obtain algebraic equation for steady state energy conservation at CV - 'P'.



$$k\left[\left(\frac{\partial T}{\partial x}\right)_e - \left(\frac{\partial T}{\partial x}\right)_w\right] - \rho C_p\left[(uT)_e - (uT)_w\right] = 0$$

$$\implies k\left[\frac{T_E - T_P}{\delta x_e} - \frac{T_P - T_W}{\delta x_w}\right] = \rho C_p\left[u_e T_e - u_w T_w\right]$$

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

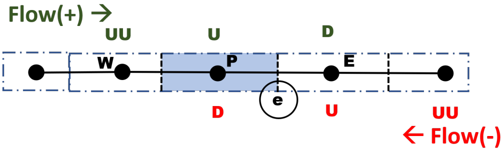


$$k\left[\left(\frac{\partial T}{\partial x}\right)_e - \left(\frac{\partial T}{\partial x}\right)_w\right] - \rho C_p\left[(uT)_e - (uT)_w\right] = 0$$

$$\implies k\left[\frac{T_E - T_P}{\delta x_e} - \frac{T_P - T_W}{\delta x_w}\right] = \rho C_p\left[u_e T_e - u_w T_w\right]$$

- How do you find face center values of u & T ?
- Option 1 : Average of common cells i.e, $T_e = 0.5*(T_E + T_P)$

- The averaged variable does not work when gradient of flow is very large.

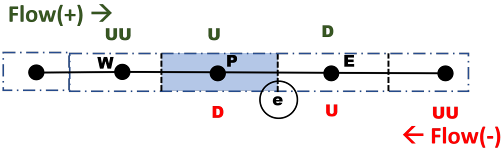


$$T_f = w_1 T_{f,D} + w_2 T_{f,U} + w_3 T_{f,UU} \quad (11)$$

The values of weight function are available in literature ([1]).

Advection Schemes

- The averaged variable does not work when gradient of flow is very large.



$$T_f = w_1 T_{f,D} + w_2 T_{f,U} + w_3 T_{f,UU} \quad (11)$$

The values of weight function are available in literature ([1]).

- Based on values of weights, Advection schemes are classified as Second Order Upwind (SOU), First Order Upwind (FOU), Quadratic Interpolation for Kinematic Convection (QUICK), Central Difference (CD) etc..

OpenFOAM Format of Equations

- Let us check general Scalar Transport Model solution implementation in OpenFOAM
- Go to → /opt/openfoam7/applications/solvers/basic/scalarTransportFoam
- Open scalarTransportFoam.C file

OpenFOAM Format of Equations

- Let us check general Scalar Transport Model solution implementation in OpenFOAM
- Go to → /opt/openfoam7/applications/solvers/basic/scalarTransportFoam
- Open scalarTransportFoam.C file

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

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

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

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

```
divSchemes
{
    default         none;
    div(phi,T)      Gauss linearUpwind grad(T);
}

laplacianSchemes
{
    default         none;
    laplacian(DT,T) Gauss linear corrected;
}
```

```
solvers
{
    T
    {
        solver          PBiCGStab;
        preconditioner  DILU;
        tolerance       1e-06;
        relTol          0;
    }
}
```

Illustration : 2D Unsteady State Convection in backward facing step

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

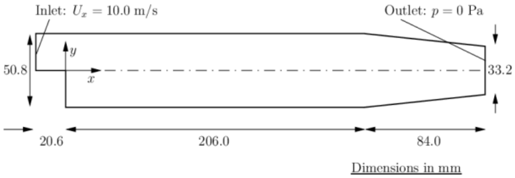
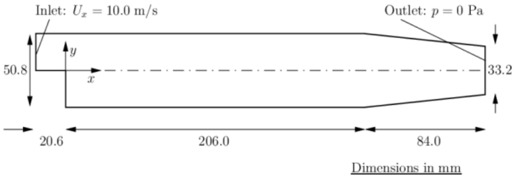


Illustration : 2D Unsteady State Convection in backward facing step

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



Parameter	Value
Inlet Temperature	1 units
Diffusivity (α = k/ρCp)	0.01
Wall	zeroGradient
Outlet	zeroGradient
Inlet Velocity	10 units

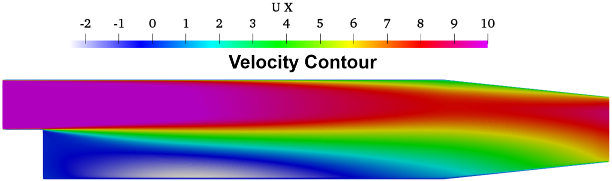
Illustration : 2D Unsteady State Convection in backward facing step

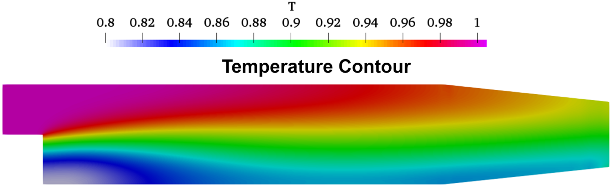
- Download the zip file given with video & extract the files
- In the terminal, type 'blockMesh' to generate mesh file

- Download the zip file given with video & extract the files
- In the terminal, type ‘blockMesh’ to generate mesh file
- Run ‘scalarTransportFoam’ solver and check the results using ‘paraFoam’.

Illustration : 2D Unsteady State Convection in backward facing step

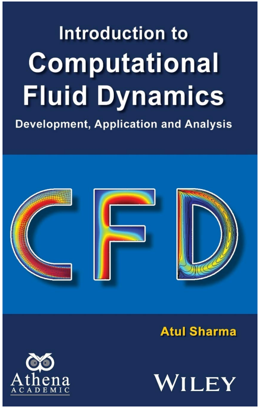
- Download the zip file given with video & extract the files
- In the terminal, type ‘blockMesh’ to generate mesh file
- Run ‘scalarTransportFoam’ solver and check the results using ‘paraFoam’.





References

1. Sharma, A. (2016). Introduction to computational fluid dynamics: development, application and analysis. John Wiley & Sons.
2. https://www.openfoam.com/
3. Britannica, T. Editors of Encyclopaedia (2020, March 10). land breeze. Encyclopedia Britannica. https://www.britannica.com/science/land-breeze



Thank you for listening!

Sumant Morab