Exploring Computational Fluid Dynamics – a state of the art engineering tool in thermal-fluid flows

Summer Undergraduate Research Experience (SURE 2021)

Chance Parrotte, Dominik Walle, Ethan Chandler; Advisor: Dr. Tat Acharya

Introduction

Computational fluid dynamics (CFD) is a technology that uses data structures to solve thermal-fluid engineering problems that involve flow and heat transfer.

In this project, the commercial CFD software ANSYS Fluent is used to perform numerical simulations to study heat transfer in two and three dimensions. CFD is a state-of-the-art technology widely used in the energy industry.

CFD simulations can replace experiments which are often expensive to set up. Therefore, CFD can potentially reduce expenditure by the industries.

The technology involves solving set of non-linear partial differential equations called as Navier Stokes equations using the finite volume method. These equations include mass, momentum, and energy conservation equations as shown in the section on materials and methods.

Objective

The objective of this project is to model one dimensional and two-dimensional heat conduction with temperature (Dirichlet) and heat flux (Neumann) boundary conditions

Materials and Methods

The commercial computational fluid dynamics (CFD) code ANSYS Fluent 18.2 is used.

The code solves Navier Stokes equations using finite volume method. Equation (1) shows the mass conservation or continuity equation:

Continuity:
$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho v)}{\partial y} = 0 \qquad (1)$$

Equations (2) and (3) show the momentum conservation equations in x and y directions

X-Momentum:
$$\frac{\partial (\rho v)}{\partial t} + \frac{\partial (\rho u v)}{\partial x} + \frac{\partial (\rho v^2)}{\partial y} = -\frac{\partial p}{\partial y} + \frac{1}{Re_r}\left(\frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y}\right) \qquad (2)$$

Y-Momentum:
$$\frac{\partial (\rho u)}{\partial t} + \frac{\partial (\rho u^2)}{\partial x} + \frac{\partial (\rho u v)}{\partial y} = -\frac{\partial p}{\partial x} + \frac{1}{Re_r}\left(\frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y}\right) \qquad (3)$$

where Coordinates: (x,y); Time: t; Pressure: p; x velocity component: u; y velocity component: v; Density: ρ; Shear Stress: ζ ; Reynolds Number: Re_r

Equation (4) shows the energy equation:

Energy:
$$\frac{\partial (E_t)}{\partial t} + \frac{\partial (uE_t)}{\partial x} + \frac{\partial (vE_t)}{\partial y} = -\frac{\partial (up)}{\partial x} - \frac{\partial (vp)}{\partial y} + \frac{1}{Re_r Pr_r}\left(\frac{\partial q_x}{\partial x} + \frac{\partial q_x}{\partial y}\right) + \frac{1}{Re_r}\left(\frac{\partial}{\partial x}(u\tau_{xx} + v\tau_{xy}) + \frac{\partial}{\partial y}(u\tau_{xy} + v\tau_{yy})\right) \qquad (4)$$

where Total Energy: E_t; and Prandtl Number: Pr

Results and Discussion

Task #1 Mesh Independence Test

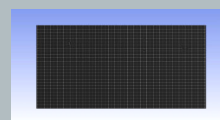

Fig. 1 Meshed Geometry

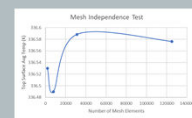

Fig. 2 Mesh Independence Test

Uniform mesh is used. The acceptable size of mesh is 0.008 m. The number of mesh elements is 31250.

Task #2 Heat Conduction with Dirichlet Boundaries (2D Geometry)

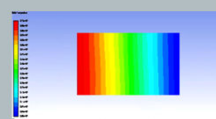

Fig. 3 Temperature contour profile on front face (2D geometry)

Fig 3 shows 1-D heat conduction with Dirichlet boundaries. The hot face is at 373 K and the cold face is at 300 K. The top and bottom surfaces are thermally insulated.

Task #3 Heat Conduction with Dirichlet Boundaries (3D Geometry)

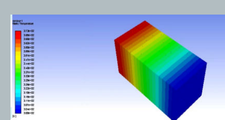

Fig 4. Temperature contour profile (3D)

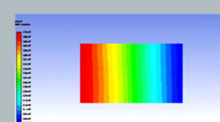

Fig 5. Temperature contour profile on front face (3D geometry)

The avg top surface temperature on the 3D geometry is 336.57 K while the top surface temperature on the 2D geometry is 336.59 K. Results suggest the 2D geometry is sufficient to model 1D heat conduction.

Task #4 Heat Conduction with Dirichlet and Neumann Boundaries

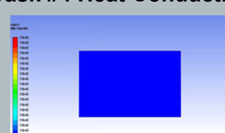

Fig 6. Temperature contour profile

Fig. 6 shows the geometry with Dirichlet, and Neumann boundaries specified on the left and right surfaces, respectively. While the left surface has a temperature of 373 K, the right surface has zero heat flux. The top and bottom surfaces are insulated as earlier cases.

Task #5 2D Heat Conduction

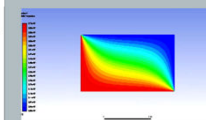

Fig 7. Temperature contour profile

Fig. 7 shows temperature contour profile when the top and right surfaces are at 300 K and the left and bottom surfaces are at 373 K.

Task #6 Convection Boundaries

The left surface is at 373 K, while the right surface is at 300 K. The bottom surface is insulated, and the top surface is subjected to convection heat transfer. The convection heat transfer coefficients (h) are varied.

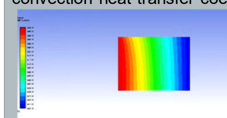

Fig 8. h = 0.01 W/m²K

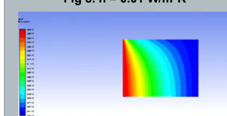

Fig 9. h = 0.1 W/m²K

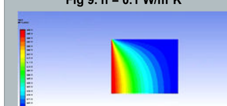

Fig 10. h = 10 W/m²K

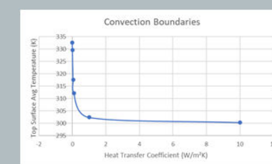

Fig 11. Heat transfer coefficient vs average temperature

Fig 11 shows the average temperature versus the heat coefficient on the top surface. The average temperature reduces with increase in heat transfer coefficient.

Conclusion

- The structure could be modeled as a 2D geometry when the walls in the y and z directions were thermally insulated
- With increase in convection heat transfer coefficient over the top surface, the average top surface temperature reduces
- Computational heat transfer can be used to model heat conduction

References

1. Bergman, T.L, Incropera, F.P., Lavine, A.S., & DeWitt, D.P. (2011). Introduction to heat transfer. John Wiley & Sons
2. Ozisik, M.N. (1987). Basic heat transfer. Robert E. Krieger

Acknowledgments

California State University, Bakersfield
Chevron Corporation
SURE 2021