Special Issue Published in International Journal of Trend in Research and Development (IJTRD), ISSN: 2394-9333, www.ijtrd.com

Heat Transfer in Computational Fluid Dynamics

[1]T. Mahalakshmi, [2]M. Zainub
[1]Assistance professor, [2]M.Phil Scholar
[1,2]Department of Mathematics, St. Josephs College of Arts and Science for Women-Hosur, India
[2]Department of Mathematics, Morappur Kongu College of Arts and Science, India

Abstract: Computational fluid dynamics (CFD) is the science of predicting fluid flow [7], heat transfer, mass transfer, chemical reactions, and related phenomena by solving the mathematical equations which govern these processes using a numerical process. Here, in this paper I have discussed about heat transfer take place in CFD. This includes models of heat transfer, enthalpy equation, periodic heat transfer, fluid properties, convection heat transfer etc. Heat transfer analyses using computational fluid dynamics and CFD calculation of convective heat transfer coefficients has been discussed briefly.

Keywords: CFD, heat transfer, conduction, convection, radiation.

I. INTRODUCTION

Heat transfer is the physical act of thermal energy being exchanged between two systems by dissipating heat. Temperature and the flow of heat are the basic principles of heat transfer. Typical design problems involve the determination of, overall heat transfer coefficient, e.g. for a car radiator. Helps to increase or decrease temperature in a system, e.g. in a gas turbine, chemical reaction vessels, food ovens. To know the temperature distribution (related to thermal stress), e.g. in the walls of a spacecraft. Temperature response in time dependent heating/cooling problems, e.g. engine cooling, or how fast does a car heat up in the sun and how is it affected by the shape of the windshield? Etc.

II. MODELS OF HEAT TRANSFER

On a microscopic scale, the kinetic energy of molecules is the direct relation to thermal energy. As temperature rises, the molecules increase in thermal agitation manifested in linear motion and vibration. Regions that contain higher kinetic energy transfer the energy to region with lower kinetic energy. Heat can be transferred from one place to another by three methods: conduction in solids, convection of liquids (liquids or gases), and radiation through anything that will allow radiation to pass. The method used to transfer heat is usually the one that is the most efficient.[2]

III. CONDUCTION

Conduction transfers heat via direct molecular collision. An area of greater kinetic energy will transfer thermal energy to an area of lower kinetic energy. Higher-speed particles will collide with slower speed particles. The slower speed particles will increase in kinetic energy as a result. Conduction is most common form of heat transfer and occurs via physical contact.

Example: place your hand against a window or place metal into an open flame.

IV. HEAT CONDUCTION – FOURIER'S LAW

The heat flux is proportional to the temperature gradient: $\frac{Q}{A} = q = -k\nabla T$ where k(x,y,z,T)is the thermal conductivity. In most practical situations conduction, convection, and radiation appear in combination. Also for convection, the heat transfer coefficient is important, because a flow can only carry heat away from a wall when that wall is conducting.

V. CONVECTION

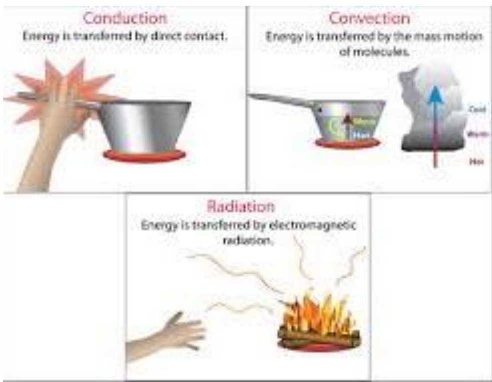
When a fluid, such as air or a liquid, is heated and then travels away from the sources, it carries the thermal energy along. This type of heat transfer is called convection. The fluid about a hot surface expands, becomes less dense, and rises. Convective heat transfer is tightly coupled to the fluid flow solution.

Example: a space heater is a classic convection example. As the space heater heats the air surrounding it near the floor, the air will increase in temperature, expand, and rise to the top of the room. This forces down the cooled air so that it becomes heated.

VI. RADIATION

Transfer of energy by electromagnetic waves between surfaces with different temperatures, separated by a medium that is at least partially transparent to the (infrared) radiation. Radiation is especially important at high temperatures, e.g. during combustion processes, but can also have a measurable effect at room temperatures. All materials radiate thermal energy based on their temperature. The hotter an object, the more it will radiate.

Example: the sun is a clear example of heat radiation that transfers heat across the solar system.

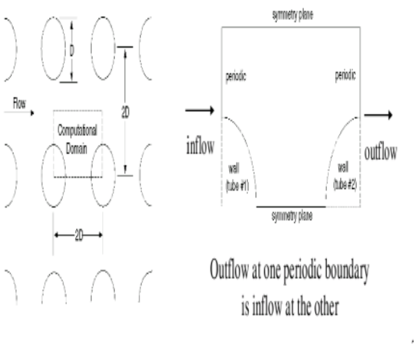


VII. ENTHALPY EQUATION

In CFD it is common to solve the enthalpy equation, subject to a wide range of thermal boundary conditions. Energy sources due to chemical reaction are included for reacting flows. Energy sources due to species diffusion are included for multiple species flows. The energy source due to viscous heating describes thermal energy created by viscous shear in the flow. This is important when the shear stress in the fluid is large (e.g. lubrication) and/or in high-velocity, compressible flows. Often, however, it is negligible. In solid regions, a simple conduction equation is usually solved, although convective terms can also be included for moving solids. [3]

VIII. PERIODIC HEAT TRANSFER

Also known as stream wise-periodic or fully-developed flow. Used when flow and heat transfer patterns are repeated, e.g.: Compact heat exchangers. Flow across tube banks. Geometry and boundary conditions repeat in stream wise direction.



IX. FLUID PROPERTIES

Fluid properties such as heat capacity, conductivity, and viscosity can be defined as: Constant, Temperature-dependent, Composition-dependent, Computed by kinetic theory and Computed by user-defined functions.

Density can be computed by ideal gas law. Alternately, density can be treated as: Constant (with optional Boussinesq modelling), Temperature-dependent, Composition-dependent and User defined functions.

X. CONVECTION HEAT TRANSFER

Convection is movement of heat with a fluid. E.g., when cold air sweeps past a warm body, it draws away warm air near the body and replaces it with cold air.

Convection in coffee to spoon Natural convection

Natural convection (from a heated vertical plate). As the fluid is warmed by the plate, its density decreases and a buoyant force arises which induces flow in the vertical direction. The force is proportional to $(\rho - \rho_\infty)g$. The dimensionless group that governs natural convection is the Rayleigh number: $Ra = Gr.Pr = \frac{g\beta\Delta TL^3}{\alpha\nu}$. Typically, $Nu \propto Ra^x$ $\frac{1}{4} < x < \frac{1}{3}$ [8] & [3]

NATURAL CONVECTION AROUND A PERSON:

Light weight warm air tends to move upward when surrounded by cooler air. Thus, warm-blooded animals are surrounded by thermal plumes of rising warm air. This plume is made visible by means of a Schlieren optical system that is based on the fact that the refraction of light through a gas is dependent on the density of the gas. Although the velocity of the rising air is relatively small, the Reynolds number for this flow is on the order of 3000.

XI. NATURAL CONVECTION- BOUSSINESQ MODEL

Makes simplifying assumption that density is uniform. Except for the body force term in the momentum equation, which is replaced by: $(\rho - \rho_o)g = -\rho_o\beta(T - T_o)g$

Valid when density variations are small (i.e. small variations in T). Provides faster convergence for many natural-convection flows than by using fluid density as function of temperature because the constant density assumptions reduces non-linearity. Natural convection problems inside closed domains: For steady-state solver, Boussinesq model must be used. Constant density ρ_o allows mass in volume to be defined. For unsteady solver, Boussinesq model or ideal gas law can be used. Initial conditions define mass in volume.

XII. NEWTON'S LAW OF COOLING

Newton described the cooling of objects with an arbitrary shape in a pragmatic way. He postulated that the heat transfer Q is proportional to the surface area A of the object and a temperature difference ΔT. The proportionality constant is the heat transfer coefficient h(W/m²-K). This empirical constant lumps together all the information about the heat transfer process that we don't know or don't understand.

$$Q = qA = \bar{h}A(T_{body} - T_\infty) = \bar{h}A\Delta T$$

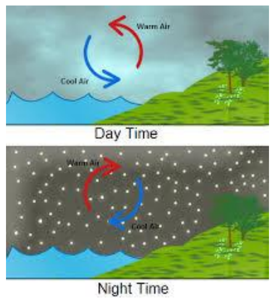
$\bar{h}$ = average heat transfer coefficient (W/m²-K).

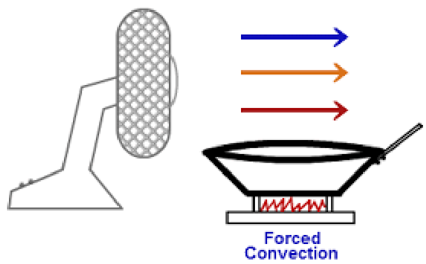
h is not a constant, but h = h(Δ T).

There are three types of convection :

Forced convection: fluid is forced. $h \propto \Delta T^x$ $\frac{1}{4} < x < \frac{1}{3}$,

Natural or free convection: fluid is induced by temperature difference. h=const Boiling convection: body is hot enough to boil liquid. $h \propto \Delta T^2$





XIII. HEAT TRANSFER ANALYSES USING COMPUTATIONAL FLUID DYNAMICS

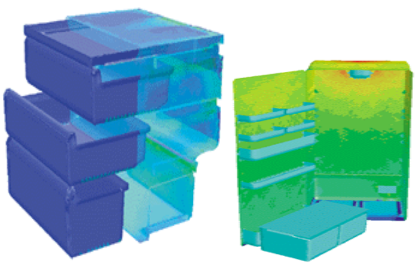
The rate of heat transfer depends on airflow conditions, and local variations in the heat transfer coefficients are expected along the surface; these variations produce local differences in temperature. In food freezing, the air velocity distribution determines the efficiency and the homogeneity of the treatments to which the product is being submitted. In equipment used in food processing, the airflow is generally turbulent and transient. Free-stream turbulence influences the transfer phenomena in porous media. Heat transfer heterogeneity is related to the local velocity and turbulence intensity, which influences the heat transfer coefficients in food processing. The measurement of local heat transfer coefficients for a complex geometry is difficult, and values associated with the different geometric shapes, surfaces and package space arrays have not been determined. For the freezing of foodstuffs in boxes, buckets and drums, the literature values show that the surface heat transfer coefficients vary when the measurements are made in different locations in the stacking. Thus, the values of the coefficients are different between the layers of product in the stack; therefore, studies that ignore these variations should be used only with due care. Computational fluid dynamics (CFD) can be used to simulate the local surface heat transfer coefficients on the surfaces. On the packages' surfaces, the heat transfer coefficients vary from one cross-section to another and within each cross-section. The use of simulated CFD data allows the packages to be divided into subsections and the heat transfer to be averaged over each subsection. The methods of predicting freezing time could be used to validate the average heat transfer coefficients. [3]

XIV. APPLICATIONS

- *Manufacturing Sectors*:

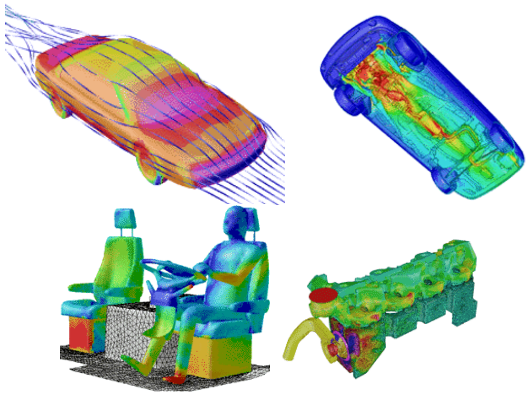
In more mundane matters CFD is also used extensively in the Chemical/Manufacturing sector as an alternative to pilot plant testing and to determine potential efficiencies of competing designs. A background in Chemical Engineering means that you have working knowledge of heat and mass transfer processes as well as fluid dynamics and are therefore pretty well positioned to tackle CFD as far as theory goes. However, with the latest progresses in GPU and Processor design, it is becoming increasingly apparent that the next generation of applied CFD engineers must have sufficient knowledge of parallel programming with at least a working familiarity in multi-threading techniques to take full advantage of the insane amounts of computing power that is becoming readily available. So the answer is yes on both counts. It is certainly helpful to have CFD as your major research area. Additionally, if you are looking into applied methods, then yes there are extensive commercial applications out there as well. However, the international market for CFD engineers has not been showing sustained growth. So it would be worth the effort to differentiate yourself in the marketplace. Any-one can set up a velocity-inlet, pressure-outlet scheme in Fluent. It takes slightly more effort to run a 7 million cell simulation of a nuclear reactor, while carrying out parametric studies and post-processing results to check solution progress during run-time.

- *Appliances:*



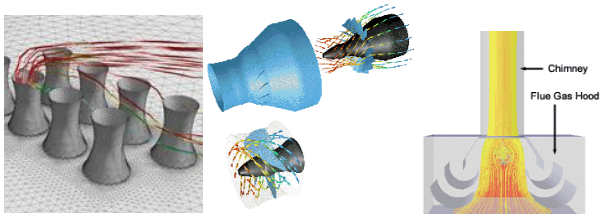
Surface-heat-flux plots of the No-Frost refrigerator and freezer compartments helped BOSCH-SIEMENS engineers to optimize the location of air inlets.

- *Automotive:*



External Aerodynamic, Undercarriage Aerodynamics, Interior Ventilation, Engine Cooling.

- *Power Generation:*



Flow around cooling towers, Flow pattern through a water turbine, Path lines from the inlet colored by temperature during standard operating conditions.

CONCLUSION

Heat transfer is the study of thermal energy (heat) flows: conduction, convection, and radiation. The fluid flow and heat transfer problems can be tightly coupled through the convection term in the energy equation and when physical properties are temperature dependent. Chemical reactions, such as combustion, can lead to source terms to be included in the enthalpy equation. While analytical solutions exist for some simple problems, we must rely on computational methods to solve most industrially relevant applications.

"MATHEMATICS IS THE ONLY SUBJECT IN WHICH WE NEVER KNOW WHAT WE ARE TALKING ABOUT OR WHETHER WHAT WE ARE SAYING IS TRUE.IN MATHEMATICS, YOU DON'T UNDERSTAND THINGS, YOU JUST GET USED TO THEM."

References

[1] "Computational Fluid Dynamics And Heat Transfer" By Anderson D.A., Tenehill J.C. And Pletcher R.H

[2] Http://Www.Bakker.Org ©Andrébakker (2002-2006) ©Fluent Inc. (2002)

[3] Www.Abeq.Org.Br/Bjche Brazilian Journal Of Chemical Engineering "Heat Transfer Analyses Using Computational Fluid Dynamics In The Air Blast Freezing Of Guava Pulp In Large Containers" W. M. Okita, M. J. Reno, A. P. Peres And J. V. Resende*

[4] Ferziger, J.H., Perić, M. "Computational Methods For Fluid Dynamics." Springer, 3rd Edition, 58-60, 2002.

[5] Fluent 6.1 User's Guide, 2003.

[6] Lienhard Iv, J.H., Lienhard V, J.H. "A Heat Transfer Textbook", Phlogiston Press, 2006.

[7] "Introduction To Computational Fluid Dynamics" By M.Zainub And T.Mahalakshmi In Ijraset Journal.

[8] The Free Wikipedia For Convection Heat Transfer.