International Journal of Technical Innovation in Modern Engineering & Science (IJTIMES)
Impact Factor: 3.45 (SJIF-2015), e-ISSN: 2455-2584
Volume 2, Issue 12, December-2016

CFD ANALYSIS OF HEAT TRANSFER TO THE FLOWING FLUIDIN A CLOSE CONDUIT AT CONSTANT HEAT FLUX

Bhavin P. Pandya[1]

Assistant Professor, KJ Institute of Engineering and Technology, Savli, Vadodara, Gujarat, India.

Heat Transfer is the most important phenomena in engineering application. Complete recovery of heat is a challenging task. Different people opt for different option for heat recovery. The most important parameter in heat transfer is heat transfer coefficient. Higher the value of heat transfer coefficient higher is heat transfer from the system.

The objective of the dissertation is to study the effect on heat transfer in pipe and pipe having different flow cross section using ANSYS. To achieve this pipe of length 1.5 meter is taken. The fluid is flowing full in pipe. The different parameters like pressure, velocity, temperature, and heat flux and heat transfer coefficient are found out using simulation at various cross sections along the flow. The cross section is at 0.3, 0.6, 0.9, 1.2 and 1.5 meters respectively. The simulations are carried out at different mass flow rates.

1. Introduction

Heat transfer development is the process of improving the concert of a heat transfer system by increasing the heat transfer coefficient. In the past decades, heat transfer enhancement technology has been increasing and generally applied to heat exchanger application. An enlarge in heat transfer coefficient generally leads to another advantage of reducing the temperature forceful force, which increase the second law efficiency, and decreases entropy invention. Also heat augmentation techniques play a vital role for laminar flow, since the heat transfer coefficient is generally low in plain tubes. Use of software for simulation of various fluid flows – heat transfer processes is picking rapidly in recent times. The simulation reduces the time consumption for modeling using basic differential equations. One can simulate any engineering process using computational software by clearly specifying the physics of the problem and knowing use of various tools available in that software.

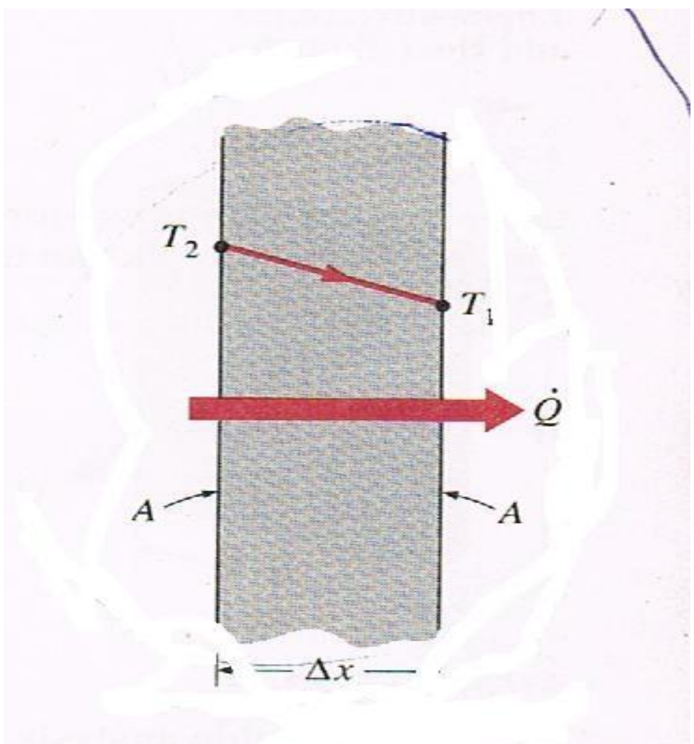

Fig 1.1 Heat conduction through a large plane wall of thickness Δx

or the area A normal to the direction of heat transfer is doubled, but is halved when the wall thickness L is doubled. Thus we conclude that the rate of heat conduction through a plane layer is proportional to the temperature differentiation across the layer and the heat transfer area, but is inversely relative to the thickness of layer. That is,

Rate of heat conduction ∞ (Area) (Temperature difference) / Thickness

OR,

$Q_{conduction}$=kA.ΔT/Δx (1.1)

Where the constant of proportionality k is the thermal conductivity of the material, which is a measure of the ability of a material to conduct heat. In the limiting case Δx→0, the equation above reduces to the differential form,

$\mathbf{Q}_{\mathbf{conduction}}$ = - kA. ΔT/Δx (1.2)

Which is called a Fourier's law of heat conduction.

2. COMPUTATIONAL FLUID DYNAMICS

Flows and related phenomena can be described by partial differential equations, which cannot be solved analytically except in special cases. To obtain an approximated solution numerically, we have to use a discretization method, which approximates the degree of difference equations by a system of numerical equation, which can then be solved by computer. The approximations are applied to small domains in space and/ or time so the numerical solution provides outcome at discrete locations in space and time. Much as the correctness of experimental data depends on the value of the tools used, the accuracy of numerical solutions is dependent on the quality of discretization s used. Here we shall be concerned with methods designed to solve the equations of fluid motion in two or three dimensions. These are the methods used in unusual application, by which we mean applications for which solutions cannot be found in textbooks or handbooks. While these methods have been used in high-technology engineering from the very beginning, they are being used more frequently in fields of engineering where the geometry is complicated or some important feature cannot be dealt with by standard methods. CFD is finding its way into process, chemical, civil, and environmental engineering. Optimization in these areas can produce large savings in equipment and energy costs and in reduction of environmental pollution.

2.1 POSSIBILITIES AND LIMITATIONS OF NUMERICAL METHODS

If we want to suggest the flow around a affecting car in a wind tunnel, we need to fix the car model and blow air at it – but the ground has to move at the air momentum, which is difficult to do. It is not difficult to do in a algebraic simulation. Other types of boundary surroundings are easily prescribed in computations; if we solve the unsteady three-dimensional Navier-Stokes equations accurately, we obtain a complete data set from which any quantity of physical importance can be derived. There are reasons for differences between computed results and 'reality' i.e. errors arise from each part of the process used to produce numerical solutions:

1. The differential equations may contain approximations.
2. Approximations are made in the discretization process.
3. In solving the discretized equations, iterative methods are used. Unless they are run for a very long time, the exact solution of the discretized equations is not produced.

When the governing equations are known accurately solutions of any desired accuracy can be achieved in principle. However, for many phenomena for example turbulence, combustion and multiphase flow the correct equations are either not available or algebraic solution is not possible. This makes introduction of models a necessity. Even if we solve the equations exactly, the solution would not be a correct representation of reality. In order to validate the models, we have to rely on experimental data. Even when the exact treatment is possible, models are often needed to reduce the cost. Wonderful color pictures make a great impression but are of no value if they are not quantitatively correct. Results must be examined very critically before they are believed.

Numerical Grid

The mathematicalframework defines the separate locations at which the variables are to be calculated, which is essentially a separate representation of the numericalarea on which the problem is to be solved. It divides the solution area into a finite number of sub domains. (Elements, control volumes etc.). Some of the options available are the following:

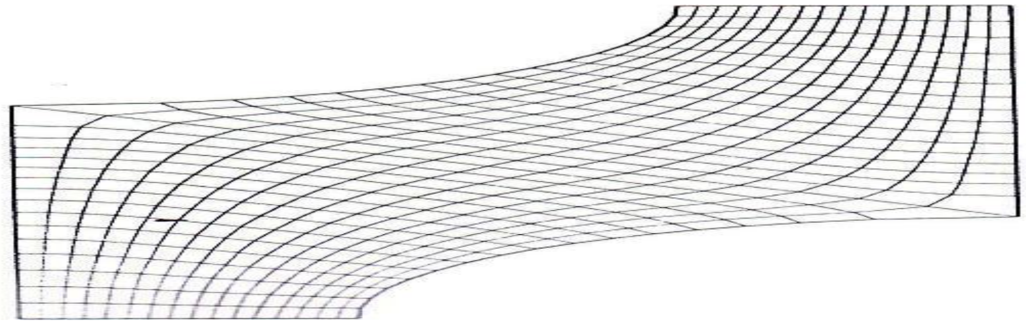

FIGURE 2.1 2-D, STRUCTURED, NON ORTHOGONAL GRID

[a] controlled (regular) grid – Regular or controlled grids consist of families of network lines with the possessions that members of a single family do not cross each other and cross each member of the other family only once. This allows the lines of a given set to be numbered consecutively. The position of any network point within the domain is uniquely recognized by a set of two (2D) or three (3D) indices. This is the simplest grid structure, since it is logically equivalent to a Cartesian grid. prepared grids may be of H-, O-, or C- type; the names are derived from the shapes of the grid lines. Figure 1.1 shows an H-type grid which, when map onto a oblong, has distinct east, west, north and south boundaries.

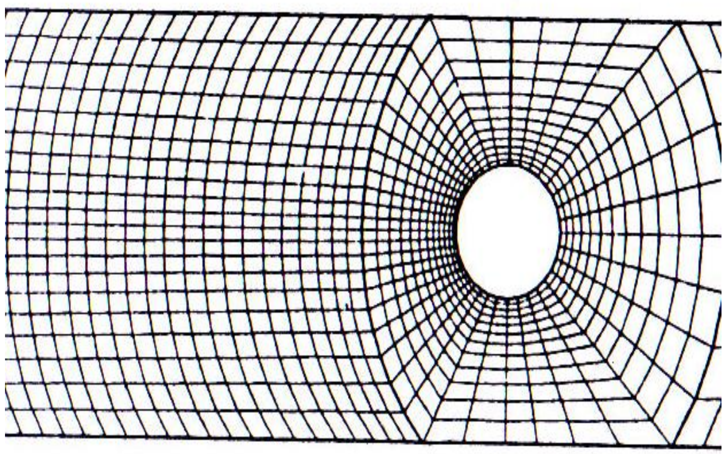

FIGURE 2.2 2-D BLOCK STRUCTURED GRID MATCHES AT INTERFACES

Figure 2.2 shows an O-type structured grid around a cylinder. In this type of grid, one set of grid lines in “endless” ; if the grid lines are treated as coordinate lines and we follow the coordinate around the cylinder, it will continuously increase and, to avoid a problem, one must introduce an artificial “cut” at which the coordinate jumps from a finite value to a zero.

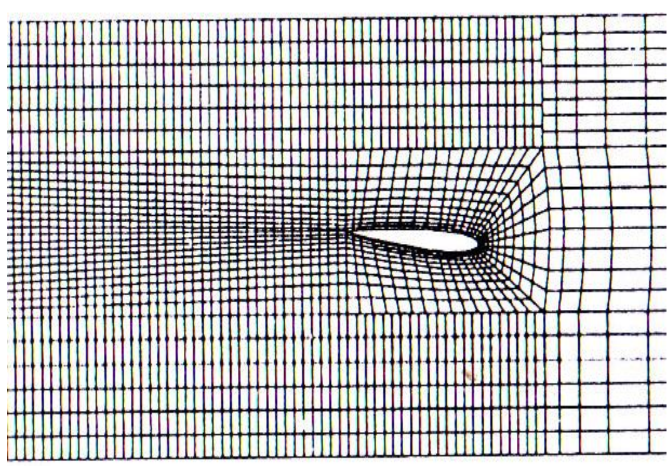

FIGURE 2.3 2-D BLOCK STRUCTURED GRID DOES NOT MATCH AT INTERFACE

The block grid around hydrofoil in figure 3.3 is of C-type. In this type of grid, points on portions of one grid line coincide, requiring the introduction of a cut similar to the ones found in O-type grids. This type of grid is often used for bodies with sharp edges for which they are capable of good grid quality.

[b] Block controllednetwork: In a block-structured grid, there is a two level subdivision of solution domain. On the common level, there are blocks, which are relatively large segments of the domain; their arrangement may be irregular and they may or may not overlap. Figure 2.2 a block-structured grid with matching at the interface is shown it is considered for the calculation of 2D flow around a cylinder in a channel and contains three blocks.

[c] Unstructured Grids:
For very complex geometries, the most elastic type of grid is one which can fit an arbitrary solution area boundary. In principle, such grid could be used with any discretization scheme, but they are best modified to the finite volume and finite element approach. In practice, grids made of triangles or quadrilaterals in 2D, and tetrahedron or hexahedra in 3D are most often used. Existing algorithms can generate such grids automatically.

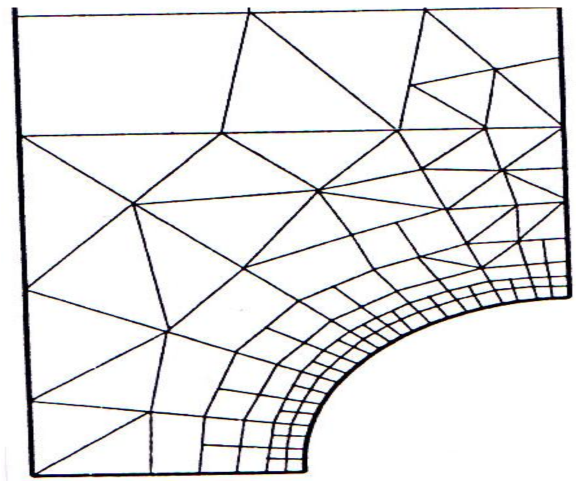

FIGURE 2.4 2-D UNSTRUCTURED GRID

2.2.5 Finite Approximation
Following the choice of network type, single has to select the approximation to be used in the discretization process. In a finite difference method, approximations for the derivatives at the grid points have to be selected. In finite volume method, one has to select the methods of approximating surface and volume integrals. The choice influences the accuracy of the approximation. It also affects the difficulty of developing the solution method, coding it, debugging it, and the speed of the code. More accurate approximations involve more nodes and give fuller coefficient matrices. The increased memory requirement may require using coarser grid, partially offsetting the advantage of higher accuracy. A compromise between difficulty, ease of implementation, accuracy and computational efficiency has to be made.

2.2.6 Solution Method
Discretization yields a large system of non-linear algebraic equations. The method of solution depends on the problem. For unsteady flows, methods based on those used for initial value problems for ordinary differential equations are used. Pseudo-time marching or an equivalent iteration scheme usually solves steady flow problems. Since the equations are non-linear, an iteration scheme is used to solve them.

2.2.7 Convergence Criteria
Finally, one needs to set the convergence criteria for the iterative method. Usually, there are two level of iterations: internal iterations, within which the linear equations are solve, and outer iterations, that deal with the non-linearity and combination of the equations. Deciding when to stop the iterative process on each level is significant, from both the accuracy and efficiency points of view.

3. RESULT & DISCUSSION

There are three different configuration of pipe, which are taken for analysis. After carrying out simulation the results that are obtained are as follow:

3.1 CASE STUDY

The analysis is done on single smooth pipe having length L= 1.5 m and diameter d = 0.0254 m. Constant heat flux is given to the pipe to determine the effect on the increase of temperature of fluid flowing through the pipe. Mass flow rate is varied to determine the effect of it on other parameters. Due to the change in mass flow rate effect of the parameter like pressure (p), Temperature (T), Velocity (v), Heat transfer coefficient (h) is determined

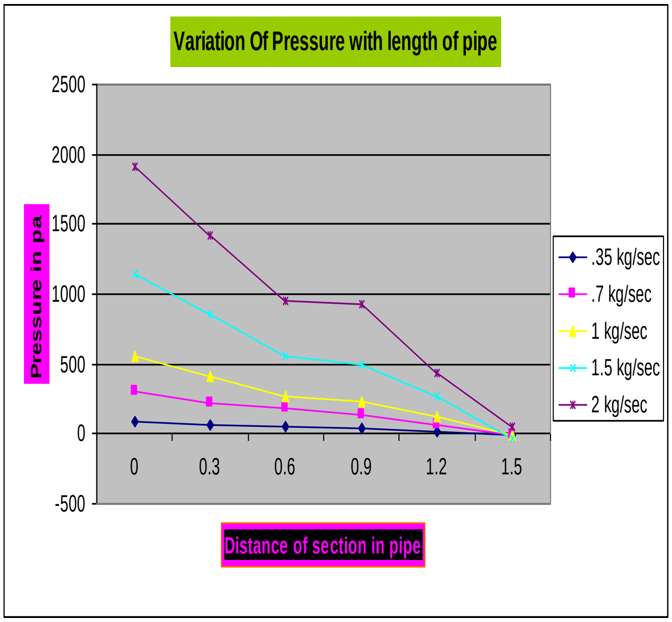

Fig 3.1 Variation in pressure with length of pipe in single tube.

using Computational Fluid Dynamics. As shown in figure 3.1 variations of pressure along the length of the pipe for different mass flow rate is obtained. The pipe is of total length 1.5 meter and five sections are made along the length of the pipe at which pressure is determined. The cross- sections are at 0.3, 0.6, 0.9, 1.2 and 1.5 respectively. As the fluid is going to flow through the pipe due to frictional effect, the pressure is going to decrease. The same phenomenon is observed in which the pressure is decreasing along the length of the tube. The decrease in pressure is due to loss of energy against friction.

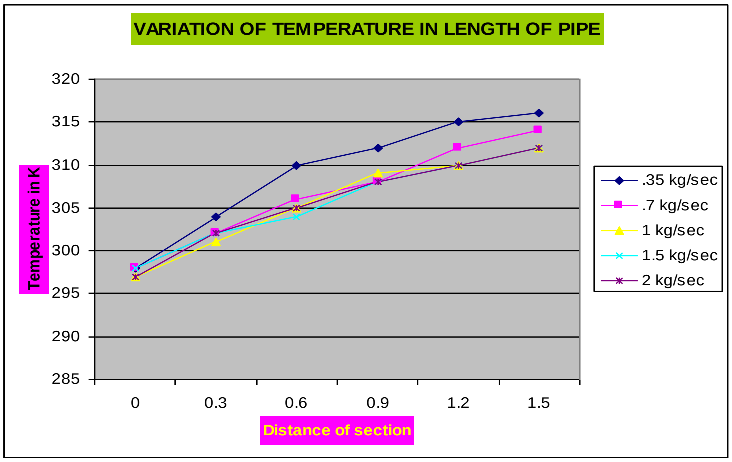

Fig 3.2 Variation in Temperature with length of pipe in single tube.

The pipe is at constant heat flux. The fluid entering the pipe is at atmospheric temperature. Figure 3.2 shows the variations in temperature of the fluid flowing along the length of the pipe. It is observed that there is an increase in temperature of the fluid while flowing through the pipe. As the mass flow rate of the fluid is increased, the rise in temperature is less as compare to less mass flow rate. This is due to less time availability at higher mass flow rates so less heat is transferred to fluid. Temperature is determined at five cross-sections that are at 0.3, 0.6, 0.9, 1.2 and 1.5 respectively.

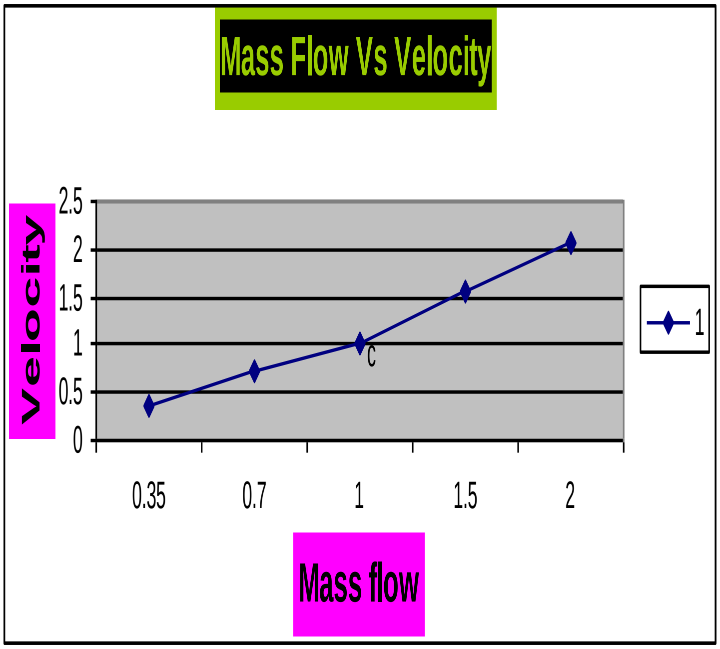

Fig 3.3 Variation in velocity with length of pipe in single tube

Figure 3.3 shows that, the variation in velocity in the pipe when the mass flow rate is varied. For a given mass flow, rate velocity is determined at five cross-sections. The cross-sections are at 0.3, 0.6, 0.9, 1.2 and 1.5 meter respectively. As shown in figure 5.3 velocity is increasing as we increase the mass flow rate through the tube.

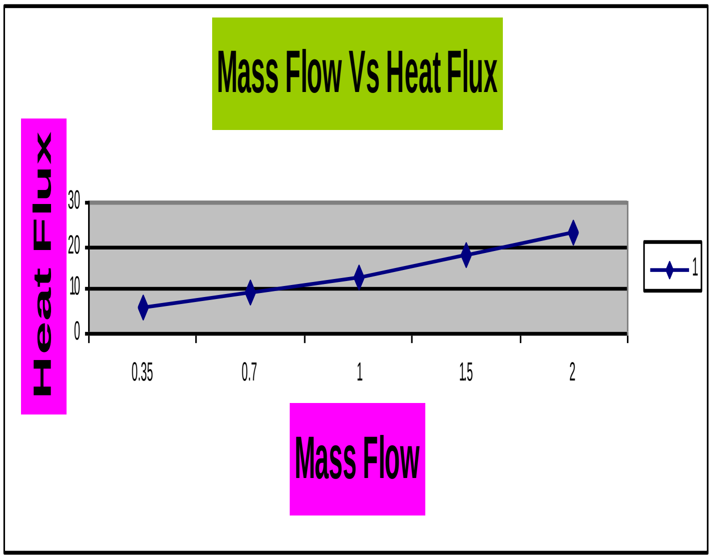

Fig 3.4 Variation in Heat Flux with length of pipe in single tube.

Heat flux is the amount of heat flowing into the pipe per meter square area of the pipe. Figure 3.4 shows the variations in heat flux as the mass flow rate in the pipe is changed. It is observed that as we increase the mass flow rate heat flux is also increasing. This is due to liquid coming in contact with the surface of the pipe. As the liquid is moving fast as compare to lower mass flow rate more heat is given to the fluid as compare to the lower mass flow rate.

Figure 3.5 shows the variation of heat transfer coefficient with the variation in mass flow rate. To determine the heat transfer coefficient is an important parameter in heat transfer mechanism. Many researchers had done number of experiments to determine the value of heat transfer coefficient. Here with the help of software the value of heat transfer coefficient is found out for a particular mass flow rate. As we increase the mass flow rate, the value of heat transfer coefficient is increasing.

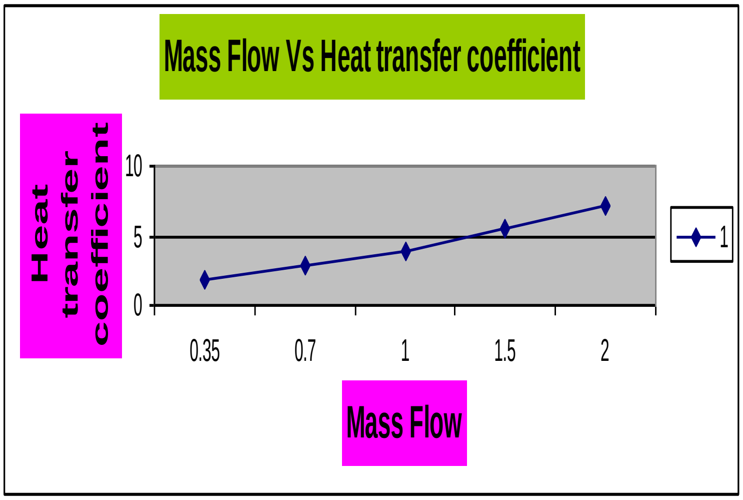

Fig 3.5 Variation in heat transfer coefficient in a single tube.

CONCLUDING

In the said work following are some concluding remarks:

1. When fluid is flowing full through the pipe ie. The cross section of the flow is circular; the variation of different parameters obtained shows the general phenomenon. The pressure is decreasing; velocity is increasing, as it is heat transfer with constant heat flux the temperature of fluid is increasing along the length of the flow.
2. After putting restriction in the flow by dividing the flow area into two equals half. It is observed that the pressure drop is more as compare to point 1. Similarly other parameter also shows the positive changes respectively.
3. Further dividing the flow area into four equal parts more pressure drop takes place as more surface area comes in contact with the fluid. Due to constant heat flux on the surface the temperature rise is also comparatively high. It is observed that by dividing the flow area into four parts heat transfer coefficient is increased almost by 40%.
4. CFD is the best tool for observing what is happening inside the tube when the fluid is flowing and along with that heat transfer is taking place.

REFERENCE

1. R. A. Seban and T. T. Shimazaki, Heat transfer to a fluid flowing turbulently in a smooth pipe with wall at constant temperature, J. *Heat* Transfer 73, 803 (1951).

2. N. Z. Azer and B. T. Chao, Turbulent heat transfer in liquid metals-fully developed pipe flow with constant wall temperature, J.H mass Transfer 3, 77-83 (1961).

3. T. Y. Na and I. S. Habib, Heat transfer in turbulent pipe flow base on a new mixing length model, *Appl. Sci. Res.* 28,302-314 (1973).

4. F. Kreith, D. Margolis, Heat transfer and frictionin turbulent vortex flow, Applied Science Resource 8 (1959) 457–473.W.R. Gambill, R.D. Bundy, High flux heat transfer characteristics in turbulent swirl flow subjected to large transverse temperature gradients, AICTE Journal 91 (1963) 55–59.

5. N.V. Zozulya, I.Y. Shkuratov, Effect of length of twisted-tape turbulence promoter and its initial twisting pitch on augmentation of heat transfer inside a tube, Heat Transfer. Soviet Research 6 (1974) 98–100.

6. A.E. Bergles, in: W.M. Rosennow, et al., (Eds.), Techniques to Augment Heat Transfer Reading: Handbook of Heat Transfer Applications McGraw-Hill, New York, 1985, Chapter 3.

7. C. W. Tan and C. J. Hsu, Low Peclet number mass transfer in laminar flow through circular tubes, *Int. J. Heat Mas* Transfer 15, 2187-2201 (1972).

8. F. Kreith, D. Margolis, Heat transfer and friction in turbulent vortexflow Applied Science Resource 8 (1959) 457–473.

9. S. AI-Fahed, W. Chakroun, Effect of tube tap clearance on heattransferfor fully developed turbulent flow in a horizontal isothermaltube, International Journal Heat and Fluid Flow 17 (2) (1996) 173–178.

10. R.M. Manglik, A.E. Bergles, Fully developed laminar heat transfer in circular-segment ducts withuniform wall temperature, Numerical Heat Transfer. Part A, Applications 26 (1994) 499–519.

11. S. Eiamsa-ard, P. Promvonge, Experimental investigation of heattransfer and friction characteristics in a circular tube fitted with nozzleturbulators,International Communication Heat and Mass Transfer 33 (2006) 591–600.

12. Eimsa-ard S, Thianpong C, Promvonge P. Experimental investigation of heat transfer and flow friction in a circular tube fitted with regularly spaced twisted tape elements. Int Commun Heat Mass Transfer 2006,1225–33.

13. Moody LF. Friction factors for pipe flow. Trans ASME 1944;66:671–84.

14. Gnielinski V. New equations for heat and mass transfer in turbulent pipe flow and channel flow. Int Chem Eng 1976;16:359–68.

15. A.E. Bergles, Techniques to augment heat transfer, in: W.M Rohsenow, J.P. Hartnett, E. Ganie (Eds.), Handbook of Heat Transfer Application, McGraw-Hill, New York, 1985.

16. E. Smithberg, F. Landis, Friction and forced Convection heat transfercharacteristics in tubes fitted with twisted tape swirl generators, ASME Journal of Heat Transfer 2 (1964) 39–49.

17. R.M. Manglik, A.E. Bergles, A correlation for laminar flow enhanced heat transfer in uniform wall temperature circular tubes with twistedtape inserts, Advances in Heat Transfer-1987, HTD vol. 1987, Springer, New York, 1987, pp. 19–25.