Hydrodynamic and Thermal Flows of Fluids

Nawfel Muhammed Baqer
M.Sc in Technical College, Engineering , Alfurat University, IRAQ
Corresponding : nawfelbaqer@yahoo.com

Abstract
The physical properties of materials for the fluid domains, properties including density, viscosity, thermal conductivity and specific heat capacity are required for the calculation purposes. The physical properties can be assumed as dependent or independent of temperature. When there is a large temperature difference between the fluid and the surface the assumption of constant thermo-physical fluid properties may cause some errors, because in reality the thermo-physical properties of the most of the fluids vary with temperature. It is also important to note that the Prandtl number of liquids also varies with temperature, similar to that of viscosity. These property variations, of course, will affect the velocity and the temperature profile of fluid in the tube. the thermo-physical properties of working fluids are assumed as temperature dependent throughout this paper. There is working fluids used in this review paper.
Keywords: therm , phys , Hagen, wall .

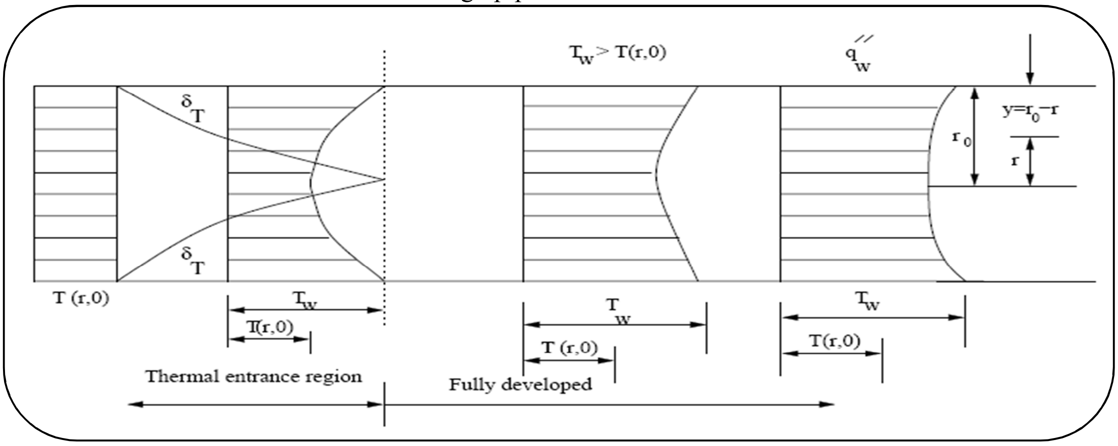
Introduction
Throughout the years extensive studies were performed on fluids flowing within tubes .these studies started as far back as 1883 when Osborne Reynolds introduced a dye flowing in water to distinguish between two distinct regimes he called "direct" and "sinuous" (Reynolds,1883) or in modern terms ,laminar and turbulent regimes .it was this groundbreaking work which led other researchers to pursue and demystify the true nature of these flow regimes .In 1839 and 1840 ,Hagen and Poiseuille ,respectively ,studied hydrodynamically fully developed viscous /laminar isothermal flows within tubes (white,1991).they showed that the pressure drop within a tube is directly proportional to the shear stress at the tube wall and inversely proportional to the diameter of the tube .this shear stress is non-dimensionalised with respect to the dynamic pressure to obtain a friction factor , one known as the fanning friction factor and the other the Darcy friction factor . these friction factors are widely used in the design of piping systems as well as heat exchangers to determine the pumping power consumption required for the system .Friction factors for pipe flow can be found on a Moody chart ,relating friction factors with Reynolds numbers . the chart is divided into four regions ;laminar ,critical ,transitional and turbulent regimes .the laminar regime extends up to a Reynolds number of somewhere between 2000 and 3000 within which there is a strong discontinuity at a Reynolds number of approximately 2200 .the discontinuity lies in the critical zone ,which is defined up to a Reynolds number of approximately 6000 ,after which it moves naturally into the transition zone and then into the fully turbulent region .the discontinuity ,though ,is a major problem for designers due to the paucity of data .An extensive amount of research work has been done regarding heat transfer in laminar flow .typical results obtained in most heat transfer texts are those for a uniform wall heat flux and for a constant wall temperature boundary condition .for a uniform heat flux boundary condition ,it can be shown that the Nusselt number reaches a constant value of 4.364 ,while for a constant wall temperature boundary condition ,a value of 3.662 is obtained (Mills,1999). These values are ,however ,only obtained for the very special case where the flow is fully developed (hydrodynamically as well as thermally) and any buoyancy –induced secondary flows are neglected .in 1883 ,Great z solved the problem for thermally developing low Prandtl number flows and in 1885 for high Prandtl number flows with the solutions being in the form of an infinite series (White ,1991).studies on different laminar flow problems such as combined hydrodynamically and thermally developing flows for different boundary conditions and different tube geometries are given in elaborate detail by shah and London (1978). Numerous research projects in the laminar regime have been undertaken pertaining to mixed convection (combined forced and natural convection) by kern and Othmer (1943),Jackson et al .(1961),Oliver (1962),Petukhov et al .(1969) and Shannon and Depew (1969) are to name but a few .Turbulent flow research within tubes has also enjoyed substantial attention .the first research to come up with a practical correlation were Dittus and Boelter (1985) ,who in 1930 stated that the heat transfer coefficient is proportional to ,and a strong function of ,the Reynolds number , as well as the Prandtl number to a lesser degree . by making use of the Reynolds analogy ,Colburn (1933) generated a very similar result .research into turbulent flow heat transfer within tubes is of such an extent that its effects can be predicted with fair accuracy

The following factors should be noted in analyzing heat transfer in internal flow:
1- Laminar vs. turbulent flow.
2-Entrance vs. fully developed region.
3-Surfase boundary conditions.

Types of Fully Developed Flows
a) Hydrodynamically fully developed flow
As fluid enters a pipe , the boundary layers keep on growing till they meet downstream from the entrance region. After this the velocity of fluid flow doesn't change in the direction of flow
(du/dz)=0
b) Thermally fully developed flow
When fluid enters a tube with wall temperature different from the temperature of the fluid, convective heat transfer occurs between the fluid and the wall , eventually a thermally fully developed condition is reached downstream of the entrance region

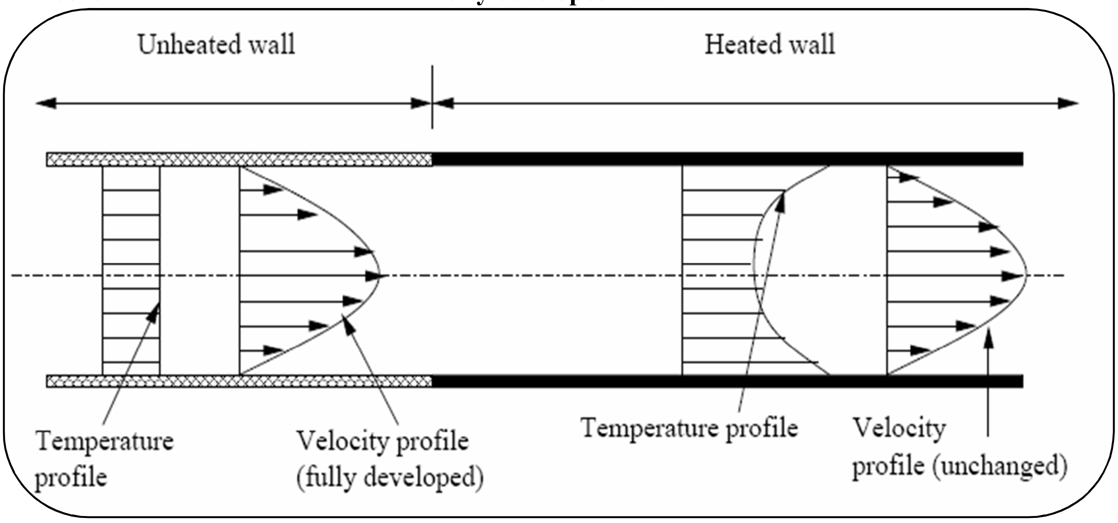
$$\frac{d}{dz}\left[\frac{T_w(x) - T(r,z)}{T_w(x) - T_m(z)}\right] = 0$$

Thermal considerations in Fluid flow through pipe

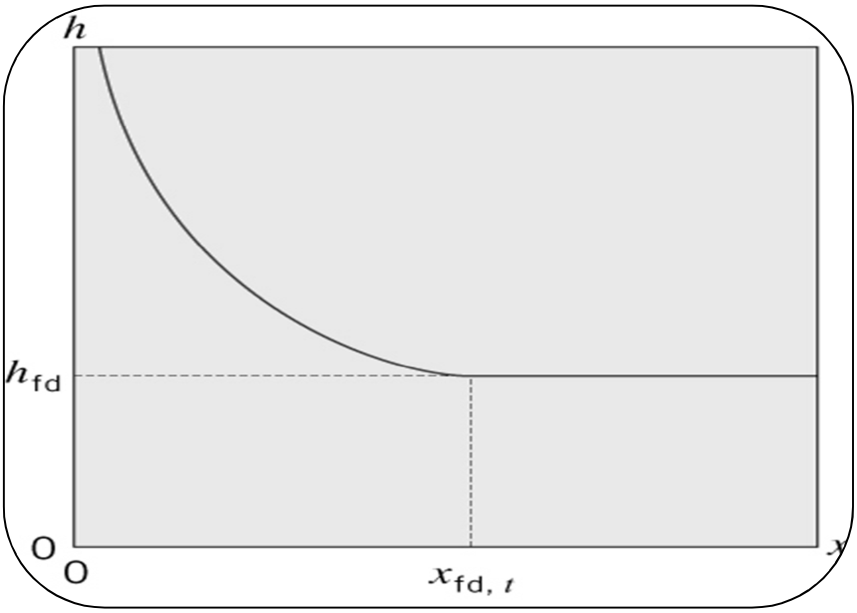


Schematic representing a thermally developing flow

hz ≠ f z **Fully developed flow**
hz = constant **Fully developed flow**



Developing Temperature and Velocity profiles
Variation of *h in entrance and* fully developed regions:



– Hydrodynamic L h turbulent = 1.359 Re D 1/4
– Thermal (approximate) L h turbulent ≈ L t turbulent ≈ 10 D
1.2 Enhancement heat transfer.
Classification of Augmentation Techniques:
They are broadly classified into three different categories:
1. Passive Techniques
2. Active Techniques
3. Compound Techniques.
1) Passive Techniques: These techniques do not require any direct input of external power; rather they use it from the system itself which ultimately leads to an increase in fluid pressure drop. They generally use surface or geometrical modifications to the flow channel by incorporating inserts or additional devices. They promote higher heat transfer coefficients by disturbing or altering the existing flow behaviour except for extended surfaces. Heat transfer augmentation by these techniques can be achieved by using;
(i) Treated Surfaces: Such surfaces have a fine scale alteration to their finish or coating which may be continuous or discontinuous. They are primarily used for boiling and condensing duties.
(ii) Rough surfaces: These are the surface modifications that promote turbulence in the flow field in the wall region, primarily in single phase flows, without increase in heat transfer surface area.
(iii) Extended surfaces: They provide effective heat transfer enlargement. The newer developments have led to modified finned surfaces that also tend to improve the heat transfer coefficients by disturbing the flow field in addition to increasing the surface area.
(iv) Displaced enhancement devices: These are the inserts that are used primarily in confined forced convection, and they improve energy transport indirectly at the heat exchange surface by displacing the fluid from the heated or cooled surface of the duct with bulk fluid from the core flow.
(v) Swirl flow devices: They produce and superimpose swirl flow or secondary recirculation on the axial flow in a channel. These include helical strip or cored screw type tube inserts, twisted tapes. They can be used for single phase and two-phase flows.
(vi) Coiled tubes: These lead to relatively more compact heat exchangers. It produces secondary flows and vortices which promote higher heat transfer coefficients in single phase flows as well as in most regions of boiling.
(vii) Surface tension devices: These consist of wicking or grooved surfaces, which direct and improve the flow of liquid to boiling surfaces and from condensing surfaces.
(viii) Additives for liquids: These include the addition of solid particles, soluble trace additives and gas bubbles in single phase flows and trace additives which usually depress the surface tension of the liquid for boiling systems.
(ix) Additives for gases: These include liquid droplets or solid particles, which are introduced in single-phase gas flows either as dilute phase (gas-solid suspensions) or as dense phase (fluidized beds).

2) Active Techniques: In these cases, external power is used to facilitate the desired flow modification and the concomitant improvement in the rate of heat transfer. Augmentation of heat transfer by this method can be achieved by

(i) Mechanical Aids: Such instruments stir the fluid by mechanical means or by rotating the surface. These include rotating tube heat exchangers and scrapped surface heat and mass exchangers.

(ii) Surface vibration: They have been applied in single phase flows to obtain higher heat transfer coefficients.

(iii) Fluid vibration: These are primarily used in single phase flows and are considered to be perhaps the most practical type of vibration enhancement technique.

(iv) Electrostatic fields: It can be in the form of electric or magnetic fields or a combination of the two from dc or ac sources, which can be applied in heat exchange systems involving dielectric fluids. Depending on the application, it can also produce greater bulk mixing and induce forced convection or electromagnetic pumping to enhance heat transfer.

(v) Injection: Such a technique is used in single phase flow and pertains to the method of injecting the same or a different fluid into the main bulk fluid either through a porous heat transfer interface or upstream of the heat transfer section.

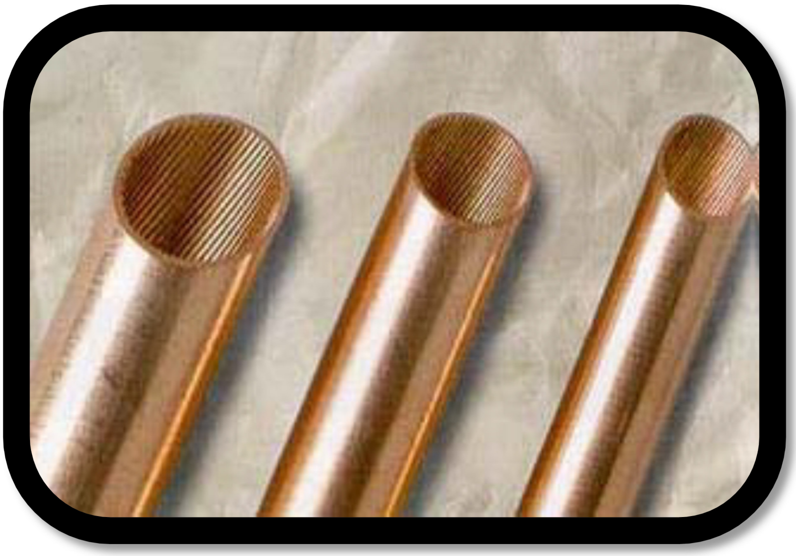
(vi) Suction: It involves either vapor removal through a porous heated surface in nucleate or film boiling, or fluid withdrawal through a porous heated surface in single-phase flow.

(vii) Jet impingement: It involves the direction of heating or cooling fluid perpendicularly or obliquely to the heat transfer surface.

3) Compound Techniques: When any two or more of these techniques are employed simultaneously to obtain enhancement in heat transfer that is greater than that produced by either of them when used individually, is termed as compound enhancement. This technique involves complex design and hence has limited applications [1].

To increase heat transfer, internally micro fin tubes are widely used in commercial HVAC applications. It is commonly understood that the micro- fin enhances heat transfer but at the same time increase the pressure drop as well. Defined the micro fin tube to have a height less than 0.03Di,where Di is the inside diameter and e is the fin height .Basically ,such kind of tube is widely used in high flow rate application because the heat transfer enhancement in high flow rates (turbulent region) is more pronounced than in the low flow rates (laminar region). Internally finned tubes have acquired importance over the years in a variety of heat transfer applications including compact heat exchangers. Fins increase the effective heat transfer area and convective heat transfer thus improving the performance of the heat exchanger. Various configurations of internally finned tubes like longitudinal, helical, annular have been studied both experimentally and numerically, and have demonstrated enhancement in heat transfer. Various cross sections of fins like triangular and trapezoidal have also been studied and their performances have been validated numerically. From the investigations for a longitudinal fin of a trapezoidal cross section, the parameters influencing the performance of the heat exchanger were found to be the thermal conductivity of the fin, height of the fin, and number of fins.

Extended surfaces are provided in order to enhance the heat transfer rate; and there are many methods which are adopted for heat transfer enhancement. Some of the most commonly applied methods are providing fins, coiled tubes, and swirl flow devices. Plate fins and tube fins constitute the fin type of heat transfer enhancement. The plate fin type in turn has several other forms such as plain fins, offset strip fins, wavy fins, perforated fins, and pin fins. Of the above mentioned types of heat transfer enhancement methods the offset strip fin is the most widely applied



During the last few years the Computational Fluid Dynamics becomes one of the most powerful and useful tools for predicting the internal flow behavior in fin tube. Any design modification required can be performed with the available workbench software and can be analyzed again with the available CFD tool. Thus, repeating this cycle enables to reach an optimized design with less running cost. Therefore, a complete numerical analysis is used in the present work to simulate the flow in fin tube model and predict the enhancement in heat transfer in this tubes . In this chapter theoretical base of heat transfer and fluid flow of fin tube discussed minutely, and the main details concerning with analysis CFD modeling were discussed as possible to introduce a good solution for the present design and to be easier guide for future similar work.

Description of the Problem and Geometry

Heat transfer enhancement has been an important factor in obtaining energy efficiency improvements in refrigeration and air-conditioning applications. The fin tube geometries are neither a classic ''integral roughness'' (little surface area increase), nor an internally finned tube (significant surface area increase, but no flow separation), as illustrated by Webb and Kim [48].

The diameter of tube are 14 mm and number of fins 20.while remain the dimension of tubes in the below table 3.1. The basic algebraic dimensions that define the fin geometry are shown in Fig 3The fin layout parameters characterize by the fin height (e), pitch normal to the fins (P_f) and the helix angle (β). The number of starts is given by $N_s = \pi d_i \cos \beta / P_f$. The fin shape parameters are defined by the fin base thickness (t_b), and the apex angle of the fin (α). The axial pitch is defined geometrically by $p_a = \pi d_i/(N_s \tan\beta)$. The total surface area of the fin tube (A_{actu}) relative to its smooth area (A_s) based on the fin root diameter (d_i), is given by :

$$A_{actu}/A_s = 1+2[\sec(\frac{\alpha}{2})-\tan(\frac{\alpha}{2})]*\frac{e}{P_f} \quad (3.1)$$

For a constant tube diameter, a decrease of the fin pitch normal (P_f) will increase the helix angle (β) which provides increased A_{actu}/A_s. and also will increase the apex angle (α) which provides decreased A_{actu}/A_s.

Table (1): show the dimension of geometry

Tube model	d_i(mm)	β(deg)	α(deg)	e(mm)	P_a(mm)	P_f(mm)	A_{actu}/A_s
1	14	30	0	1	3.807	1.905	2.049995538
2	14	30	10	1	3.807	1.905	1.962186373
3	14	30	30	1	3.807	1.905	1.805800623
4	14	30	53.13	1	3.807	1.905	1.656277977
5	14	50	0	1	1.8452	1.414	2.41412713
6	14	50	10	1	1.8452	1.414	2.29586632
7	14	50	30	1	1.8452	1.414	2.085247015
8	14	50	53.13	1	1.8452	1.414	1.883870892
9	14	70	0	1	0.8	0.753	3.654632333
10	14	70	10	1	0.8	0.753	3.432630392
11	14	70	30	1	0.8	0.753	3.037250933
12	14	70	53.13	1	0.8	0.753	2.659222992
13	14	30	0	0.8	3.807	1.905	1.83999643
14	14	30	10	0.8	3.807	1.905	1.769749098
15	14	30	30	0.8	3.807	1.905	1.644640498
16	14	30	53.13	0.8	3.807	1.905	1.525022382
17	14	50	0	0.8	1.8452	1.414	2.131301704
18	14	50	10	0.8	1.8452	1.414	2.036693056
19	14	50	30	0.8	1.8452	1.414	1.868197612
20	14	50	53.13	0.8	1.8452	1.414	1.707096714
21	14	70	0	0.8	0.8	0.753	3.123705867
22	14	70	10	0.8	0.8	0.753	2.946104313
23	14	70	30	0.8	0.8	0.753	2.629800746
24	14	70	53.13	0.8	0.8	0.753	2.327378394
25	14	30	0	0.6	3.807	1.905	1.629997323
26	14	30	10	0.6	3.807	1.905	1.577311824
27	14	30	30	0.6	3.807	1.905	1.483480374
28	14	30	53.13	0.6	3.807	1.905	1.393766786
29	14	50	0	0.6	1.8452	1.414	1.848476278
30	14	50	10	0.6	1.8452	1.414	1.777519792
31	14	50	30	0.6	1.8452	1.414	1.651148209
32	14	50	53.13	0.6	1.8452	1.414	1.530322535
33	14	70	0	0.6	0.8	0.753	2.5927794
34	14	70	10	0.6	0.8	0.753	2.459578235
35	14	70	30	0.6	0.8	0.753	2.22235056
36	14	70	53.13	0.6	0.8	0.753	1.995533795

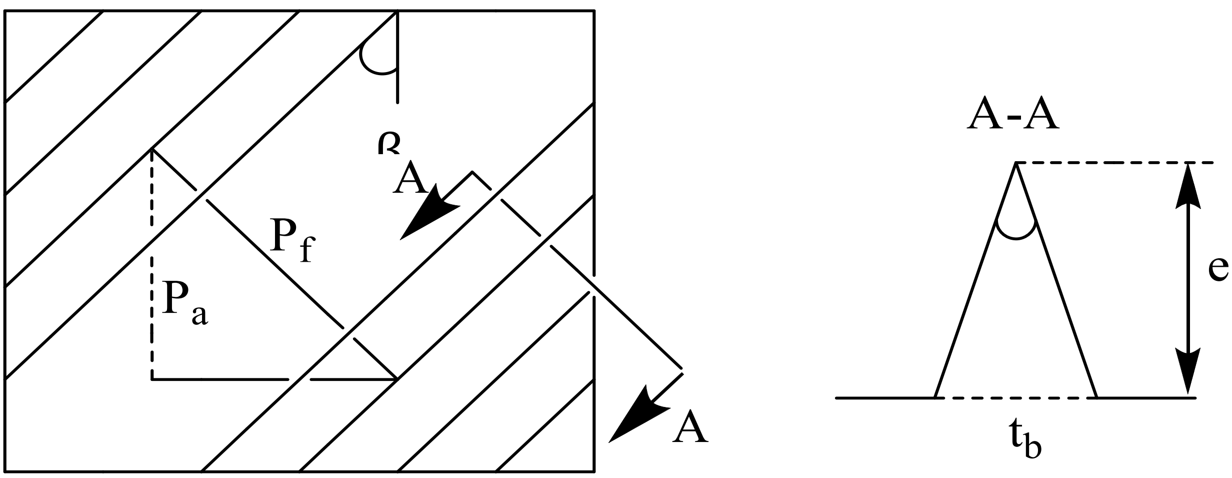


Fig.(4) Geometric parameters of the fin surface

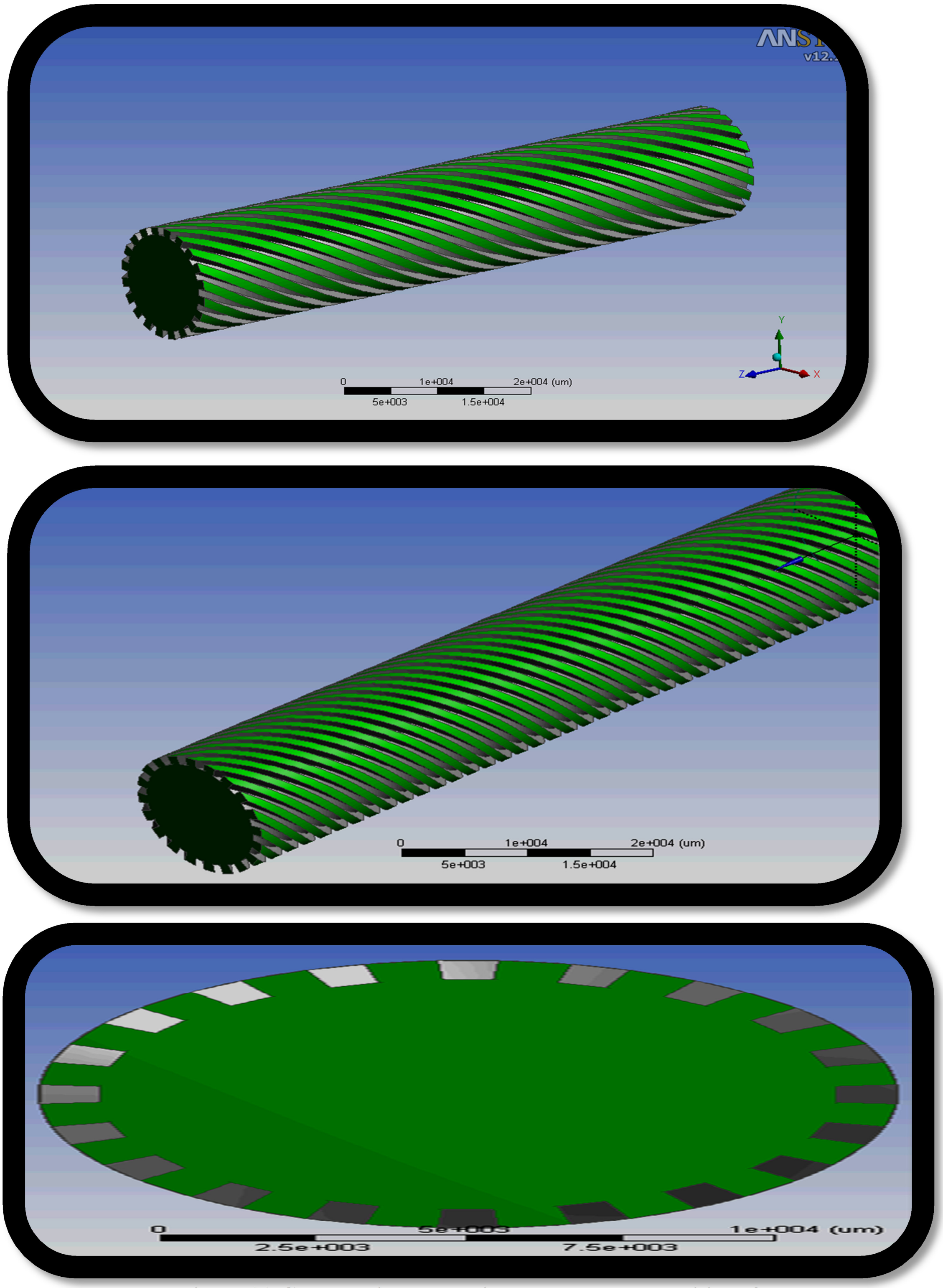


Figure (5) Computational domain and boundary conditions for tube.

Basic Governing Equations

ANSYS CFX solves the governing equations numerically for fluid flow using finite volume methods. mass transport equation and three-dimensional momentum transport equations are the fundamental governing equations solved in the CFD code. Energy equation is included for problems involving heat transfer. Turbulence models require the transport equations for the turbulence flow variables in addition to the Navier-Stokes equations. The governing equations are subjected to the boundary conditions of the problem to formulate a solution.

Continuity Equation :

$$\frac{\partial u}{\partial z}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial x}=0 \qquad (2)$$

Momentum Equation

x-component:

$$\rho\left(u\frac{\partial}{\partial z}u+v\frac{\partial}{\partial y}u+w\frac{\partial}{\partial x}u\right)=-\frac{\partial P}{\partial z}+\mu\left(\frac{\partial^2 u}{\partial z^2}+\frac{\partial^2 u}{\partial y^2}\ \frac{\partial^2 u}{\partial x^2}\right) \qquad (.3)$$

y-Component:

$$\rho\left(u\frac{\partial}{\partial z}v+v\frac{\partial}{\partial y}v+w\frac{\partial}{\partial x}v\right)=-\frac{\partial P}{\partial y}+\mu\left(\frac{\partial^2 v}{\partial z^2}+\frac{\partial^2 v}{\partial y^2}\ \frac{\partial^2 v}{\partial x^2}\right) \qquad (4)$$

z-Component:

$$\rho\left(u\frac{\partial}{\partial z}w+v\frac{\partial}{\partial y}w+w\frac{\partial}{\partial x}w\right)=-\frac{\partial P}{\partial x}+\mu\left(\frac{\partial^2 w}{\partial z^2}+\frac{\partial^2 w}{\partial y^2}\ \frac{\partial^2 w}{\partial x^2}\right) \qquad (5)$$

The Energy Equation:

$$\rho C_p\left(u\frac{\partial}{\partial z}T+v\frac{\partial}{\partial y}T+w\frac{\partial}{\partial x}T\right)=\lambda\left(\frac{\partial^2 T}{\partial z^2}+\frac{\partial^2 T}{\partial y^2}\ \frac{\partial^2 T}{\partial x^2}\right) \qquad (6)$$

Equations of Turbulence

Turbulence consists of small scale fluctuations in the flow characteristics over time. It is a complex process, mainly because it is three dimensional, unsteady and chaotic, and it can have a significant effect on the characteristics of the flow. Turbulence occurs when the inertia forces in the fluid become significant compared to viscous forces, and is characterized by a high Reynolds Number.

In principle, the Navier-Stokes equations describe both laminar and turbulent flows without the need for additional information. However, turbulent flows at realistic Reynolds numbers span a large range of turbulent length and time scales and would generally involve length scales smaller than the smallest finite volume mesh which can be practically used in a numerical analysis.

Shear Stress Transport (SST)

The - based SST model accounts for the transport of the turbulent shear stress and gives highly accurate predictions of the onset and the amount of flow separation under adverse pressure gradients [49]. The BSL model combines the advantages of the Wilcox and the - model, but still fails to properly predict the onset and amount of flow separation from smooth surfaces [50]. The reasons for this deficiency are given in detail in Menter [51]. The main reason is that both models do not account for the transport of the turbulent shear stress. This results in an overprediction of the eddy-viscosity. The proper transport behavior can be obtained by a limiter to the formulation of the eddy-viscosity:

The two-equation model (written in conservation form) is given by the

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial(\rho ujk)}{\partial xj}=P-\beta^{*}\rho\omega k+\frac{\partial}{\partial xj}\left[\left[(\mu+\sigma_k\mu_t)\frac{\partial k}{\partial xj}\right]\right. \qquad (7)$$

$$\frac{\partial(\rho\omega)}{\partial t}+\frac{\partial(\rho uj\omega)}{\partial xj}=\frac{\gamma}{v_t}P-\beta\rho\omega^2+\frac{\partial}{\partial xj}\left[(\mu+\sigma_\omega\mu_t)\frac{\partial\omega}{\partial xj}\right]+2(1-F1)\frac{\rho\sigma_{\omega 2}}{\omega}\frac{\partial k}{\partial xj}\frac{\partial w}{\partial xj} \qquad (.8)$$

Where :

k for the turbulent kinetic energy

ω turbulent frequency

Blending Functions

The blending functions are critical to the success of the method. Their formulation is based on the distance to the nearest surface and on the flow variables. and the turbulent eddy viscosity is computed from[52]:

$$P=\tau_{ij}\frac{\partial ui}{\partial xj} \qquad (9)$$

$$\tau_{ij}=\mu_t\left(2Sij-\frac{2}{3}\frac{\partial uk}{\partial xk}\delta ij\right)-\frac{2}{3}\rho k\delta ij \qquad (10)$$

$$S_{ij}=\frac{1}{2}\left(\frac{\partial ui}{\partial xj}+\frac{\partial uj}{\partial xi}\right) \qquad (11)$$

Each of the constants is a blend of an inner (1) and outer (2) constant, blended by:

$$\phi=F_1\,\phi_1+(1-F_1)\phi_2 \qquad (12)$$

where

ϕ_1 represents constant 1

ϕ_2 represents constant 2.

Additional functions are given by:

$$F_1=\tanh\left(arg_1^4\right) \qquad (13)$$

$$arg_2=max\left(2\frac{\sqrt{k}}{\beta^{*}wd},\frac{500v}{d^2\omega}\right) \qquad (14)$$

$$arg_1=\min\left[max\left(\frac{\sqrt{k}}{\beta^{*}wd},\frac{500v}{d^2\omega}\right),\frac{4\rho\sigma\omega 2k}{CDk\omega d^2}\right] \qquad (15)$$

$CD_{kw} = \max\left(2\rho\sigma\omega 2 \, \frac{1}{\omega} \frac{\partial k}{\partial xj} \frac{\partial \omega}{\partial xj}, 10^{-20}\right)$ (16)

$F_2 = \tanh(arg_2^2)$ (17)

and the turbulent eddy viscosity is computed from:

$\mu t = \frac{\rho a1k}{max(a1\omega, \Omega F2)}$ (18)

ρ is the density

$v_t = \mu_t/\rho$ is the turbulent kinematic viscosity,

μ is the molecular dynamic viscosity,

d is the distance from the field point to the nearest wall,

$\Omega = \sqrt{2wijwij}$ is the vorticity magnitude, with

$W_{ij} = \frac{1}{2}\left(\frac{\partial ui}{\partial xj} - \frac{\partial uj}{\partial xi}\right)$ (19)

It is generally recommended to use a production limiter. the term P in the k-equation is replaced by

min (P ,$20\beta^* \rho\omega k$) (20)

$\gamma 1 = \frac{\beta 1}{\beta *} - \frac{\sigma_{w1}k2}{\sqrt{\beta *}}$ $\gamma 2 = \frac{\beta 2}{\beta *} - \frac{\sigma_{w2}k2}{\sqrt{\beta *}}$

σ_{k1} =0.85	$\sigma_{\omega 1}$ =0.5	β_1 =0.075
σ_{k2} =1.0	$\sigma_{\omega 2}$ =0.856	β_2 =0.0828
β^* =0.09	k=0.41	a_1 =0.31

Modeling Flow Near the Wall

An important issue in the accurate prediction of industrial turbulent flows is the formulation and the numerical treatment of the equations in regions close to solid walls. The near-wall formulation determines the accuracy of the wall-shear-stress and the wall heat transfer predictions and has an important influence on the development of boundary layers, including the onset of separation. Near a no-slip wall, there are strong gradients in the dependent variables and viscous effects on the transport processes are large.

In the present study the wall-function method is employed to model the flow in the near-wall region. In the wall-function approach, the viscosity affected sublayer region is bridged by employing empirical formulas to provide near-wall boundary conditions for the mean flow and turbulence transport equations. These formulas connect the wall conditions (e.g. the wall-shear-stress) to the dependent variables at the near-wall grid node which is presumed to lie in the fully-turbulent region of the boundary layer. The major advantage of the wall-function approach is that it conserves valuable computer resources, and it avoids the need to account for viscous effects in the turbulence model. An alternative approach to the use of wall-functions is to use a fine-grid analysis in which computations are extended through the viscosity affected sublayer close to the wall.

Experiments have shown that the near-wall region can be subdivided into two layers. In the innermost layer, the so-called “viscous sublayer”, the flow is almost laminar-like, and the (molecular) viscosity plays a dominant role in momentum and heat transfer. In the outer layer, called the “fully-turbulent layer”, turbulence plays a major role. Finally there is a region between the viscous sublayer and the fully turbulent layer where the effects of molecular viscosity and turbulence are of equal importance. This region is called the log-law region. The wall-function approach is an extension of the method of Launder and Spalding (1974)[53]. In the log-law region, the near wall tangential velocity is related to the wall-shear-stress, τω, by means of a logarithmic relation. Assuming that the logarithmic profile reasonably approximates the velocity distribution near the wall, it provides a means to numerically compute the fluid shear stress as a function of the velocity at a given distance from the wall. This is known as a ‘wall function’ and the logarithmic nature gives rise to the well known ‘log law of the wall’. Wall functions are still the most popular way to account for wall effects. In the present study Scalable Wall Functions is used for turbulence model based on the SST.

The logarithmic relation for the near wall velocity is given by:

$u^+ = \frac{U_t}{u_\tau} = \frac{1}{K} ln(y^+) + C$ (21)

Where:

$y^+ = \frac{\rho \Delta y \, u_\tau}{\mu}$ (22)

$u_\tau = \left(\frac{\tau\omega}{\rho}\right)^{1/2}$ (23)

u^+ is the near wall velocity

u_τ is the frictional velocity

U_t is the known velocity tangent to the wall at a distance of Δy from the wall

y^+ is the dimensionless distance from the wall

τ_w is the wall shear stress

κ is the Von Karman constant
C is a log-layer constant depending on wall roughness.
Scalable Wall-Functions:
The below Equation has the problem that, it becomes singular at separation points when the near wall velocity, U_t , approaches zero. In the logarithmic region, an alternative velocity scale, u* can be used instead of u+:

$$u^* = C_\mu^{1l4} k^{-1/2} \quad (23)$$

This scale has the useful property that it does not go to zero if U_t goes to zero (in turbulent flow k is never completely zero). Based on this definition, the following explicit equation for u_τ can be obtained:

$$u_\tau = \frac{U_t}{\frac{1}{K} ln(y^+) + C} \quad (24)$$

The absolute value of the wall shear stress τ_w , is then obtained from:
Where:

$$\tau_w = (\rho u^* u\tau) \quad (25)$$

$$y^* = (\rho u^* \Delta y)/\mu \quad (26)$$

One of the major drawbacks of the wall-function approach is that the predictions depend on the location of the point nearest to the wall and are sensitive to the near-wall meshing; refining the mesh does not necessarily give a unique solution of increasing accuracy Grotjans and Menter (1998)[54]. The problem of inconsistencies in the wall-function in the case of fine grids can be overcome with the use of the scalable wall-function formulation developed by the computational fluid dynamics code CFX (version 12.1) (2009). It can be applied on arbitrarily fine grids and allows you to perform a consistent grid refinement independent of the Reynolds number of the application. The basic idea behind the scalable wall-function approach is to assume that the surface coincides with the edge of the viscous sublayer, which is defined to be at y+ = 11.06 This is the intersection between the logarithmic and the linear near wall profile. The computed y+ is not allowed to fall below this limit. Therefore, all grid points are outside the viscous sublayer and all fine grid inconsistencies are avoided.

Heat flux in the Near –Wall Region
The thermal boundary layer is modeled using the thermal law –of –the-wall function of B.A Kader [55] .Heat flux at the wall can be modeled using a wall function approach or the automatic wall treatment .The non –dimensional near –wall temperature profile follows a universal profile through the viscous sub layer and the logarithmic region . The non-dimensional temperature T^+,is defined as :

$$T^+ = \frac{\rho\, c_p\, u^* (T_w - T_f)}{q_w} \quad (27)$$

Where
T_w is the temperature at the wall
T_f the near –wall fluid temperature
C_p the fluid heat capacity
q_w the heat flux at the wall
Turbulent fluid flow and heat transfer problems without conjugate heat transfer objects require the specification of the wall heat flux ,q_w or the wall temperature ,T_w .The energy balance for each boundary control volume is completed by multiplying the wall heat flux by the surface area and adding to the corresponding boundary control volume energy equation .

Assumptions
Assumptions about the fluid and the analysis are as follows:
1. The fluid is Newtonian, incompressible flow.
2. Variable physical properties with temperture.
3. The flow is assumed to be three-dimensional
4. steady state.
5. Turbulent fully developed.

Based on above assumptions, the problem is defined by the laws of mass, momentum and energy which are stated in the following sections. The presented study stretches from the turbulent range flow 10000≤ Re ≤ 50000 range flow. Equations that govern the problem are that of a model turbulent for the flow in the range. The turbulence model were tested in the present study for the flow in the transitional range namely. The commercial CFD code CFX-12.1 was used to carry out the computations.

MATERIAL PROPERTIES
One of the important steps in the set up of the numerical model is the definition of the physical properties of materials. for the fluid domains, properties including density, viscosity, thermal conductivity and specific heat

capacity are required for the calculation purposes. The physical properties can be assumed as dependent or independent of temperature. When there is a large temperature difference between the fluid and the surface the assumption of constant thermo-physical fluid properties may cause some errors, because in reality the thermo-physical properties of the most of the fluids vary with temperature. It is also important to note that the Prandtl number of liquids also varies with temperature, similar to that of viscosity. These property variations, of course, will affect the velocity and the temperature profile of fluid in the tube. the thermo-physical properties of working fluids are assumed as temperature dependent throughout this thesis. There is working fluids used in this thesis. The water. Since variation of the temperature along the tube is considered to be influencing the thermo-physical properties, algebraic equations, are derived as a function of temperature by the interpolation method from the data obtained from references within a certain temperature range. The equations obtained for a temperature range of 273.15 – 368.15 K are given below in data [56].

For Water

1-Thermal Conductivity [W/mK]

$$k(T) = \{(T-T_1/T_3-T_1)*(k_3-k_1)\} + k_1 \qquad [28]$$

2- Density [kg/m3]

$$\rho(T) = \{(T-T_1/T_3-T_1)*(\rho_3-\rho_1)\} + \rho_1 \qquad [29]$$

3- Dynamic Viscosity [kg/ms]

$$\mu(T) = \{(T-T_1/T_3-T_1)*(\mu_3-\mu_1)\} + \mu_1 \qquad [30]$$

4- Specific Heat Capacity [j/kgK]

$$Cp(T) = \{(T-T_1/T_3-T_1)*(Cp_3-Cp_1)\} + Cp_1 \qquad [31]$$

Mesh Generation

from previous explained, in order to perform numerical calculations, the representative domain of the considered problem should be divided into several small sub domains which are called as meshes or grids. The considered problem is analyzed for two different flow regimes, laminar and turbulence. In this thesis, only one meshing operation to numerically analyze the problem for both regimes is applied; although, two different meshing operations are in general required in order to analyze the problem for the two different regimes. Compared to the laminar flow simulations, the turbulent flow is more challenging in many ways, and it needs some extra care for grid generation. In this thesis, for easier data management, grid appropriate for the turbulence case is applied to laminar case as well. For turbulent case, wall function approach is used to resolve the viscosity-affected region between the walls of tube and fully-turbulent region in flow, and the mean quantities in turbulent flow have larger gradients than in laminar flow. For that reason some extra care is taken for grid generation as Tetrahedral volume mesh element that is used[57]

1. Tetra Generation Steps

The steps involved in generating a Tetra mesh are:

- Repairing/cleaning up the geometry
- Specifying geometry details
- Specifying sizes on surfaces/curves
- Meshing inside small angles or in small gaps between objects
- Desired mesh region
- Computing the mesh
- Checking the mesh for errors
- Editing the mesh to correct any errors
- Smoothing the mesh to improve quality

The mesh is then ready to apply loads, boundary conditions, etc., and for writing to the desired solver.

2. Tetrahedral Mesh

The **Tetra** mesher can use different meshing algorithms to fill the volume with tetrahedral elements and to generate a surface mesh on the object surfaces. You must define prescribed curves and points to determine the positions of edges and vertices in the mesh. For improved element quality, the **Tetra** mesher incorporates a powerful smoothing algorithm, as well as tools for local adaptive mesh refinement and coarsening [58].

3. Mesh Topology

The Build Topology operation will find holes and gaps in the geometry. It should display yellow curves where there are large (in relation to a user-specified tolerance) gaps or missing surfaces.

During the Tetra process any leakage path (indicating a hole or gap in the model) will be indicated. The problem can either be corrected on a mesh level, or the geometry in that vicinity can be repaired and the meshing process repeated. For further information on the process of interactively closing holes.

References

1. Abu Madi, M., Johns, R. A. and Heikal, M. R. (1998): *Performance characteristics correlation for round tube and plate finned heat exchanger*, Int. J. Refrig. Vol. 21, No. 7, pp. 507-517.
2. Aknipar, E.K., Bicer, Y., Yildiz, C., and Pehlivan, D. (2004): *Heat Transfer Enhancements in a Concentric Double Pipe Exchanger Equipped with Swirl Elements*, Int. Communications in Heat and Mass Transfer, Vol. 31, No. 6, pp. 857-868.
3. ASHRAE Handbook *Fundamentals* (1997): American Society of Heating, Refrigeration and Air-Conditioning Engineers, Atlanta, GA.
4. Babus'sHaq, F.R., Akintude, K., and Probert, D,S. (1995): *Thermal Performance of a Pin-Fin Assembly*, Int. J. Heat Fluid Flow, 16, pp. 50-55.
5. Baehr, H., Stephan, K. (2004): *Wärme- und Stoffübertragung*, 4th ed., Springer Verlag, Berlin.
6. Bell, K. J. (1963): *Final report of the cooperative research programm on shell and tube heat exchangers*, University of Delaware, Engineering Experimental Station, Bulletin No. 5.
7. Bergelin, O. P., Brown, G. A., and Doberstein, S. C. (1952): *Heat Transfer and Fluid Friction During Across Banks of Tubes*, Trans. ASME, 74, pp. 953-960.
8. Bergles, A. E. (1999): *The Imperative to Enhance Heat Transfer*, Heat Transfer Enhancement of Heat Exchangers (eds. Kakac, S., Bergles, A.E., Mayinger, F., and Yüncü), Kluwer Academic Publishers, Dordrecht.
9. Bergles, A. E. (1988): *Augmentation of heat transfer*, Heat Exchanger Design Handbook, Vol. 2, Hemisphere, Washington, D.C.
10. Bergles, A. E., Jensen, M. K., and Shome, B. (1995): *Bibliography on Enhancement of Convective Heat and Mass Transfer*, Report HTL-23, Heat Transfer Laboratory, Rensselaer Polytechnic Institute, Troy, NY.
11. Bergles, A.E. (1985): *Techniques to Augment Heat Transfer*, Handbook of Heat Transfer Applications, (eds., Rosenhow, W.M., Hartnett, J.P., and Ganic, E.N.), Chapter 3, McGraw-Hill, New York.
12. Blevins, D. R. (1992): *Applied Fluid Dynamics Handbook*, Krieger, Malabar, FL.
13. Camci, C., Uzol, O. (2001): *Elliptical pin fins as an alternative to circular pin fins for gas turbine blade cooling applications*, ASME paper 2001-GT-0180, ASME Int. Turbine Conference, New Orleans.
14. Chang, J. Y. and Wang, Ch. Ch. (1997): *A generalised heat transfer correlation for louver fin geometry,* Int. J. Heat Mass Transfer, 40, pp. 533-544.
15. Chang, J. Y, Hsu, Ch. K., Lin, T. Y. and Wang, Ch. Ch. (2000): *A generalised friction correlation for louver fin geometry*, Int. J. Heat Mass Transfer, 43, pp. 2237-2243. 9 References 135
16. Chen, Z., Li, Q., Meier, D., Warnecke, H. J. (1997): *Convective heat transfer and pressure loss in rectangular ducts with drop-shaped pin fins*, Heat and Mass Transfer 33, pp.219-224.
17. Chyu, M. K. (1990): *Heat Transfer and Pressure Drop for Short Pin- Fin Arrays with Pin- Endwall Fillet*, ASME J. of Heat Transfer, Vol. 112, Nov., pp. 926-932.
18. Chyu, M.K., Natarajan, V., and Metzger, D.E. (1992): *Heat/Mass Transfer From Pin-Fin Arrays with Perpendicular Flow Entry*, ASME HTD, Vol. 226, Fundamentals and Applied Heat Transfer Research for Gas Turbine Engines.
19. Colburn, A.P. (1942): *Heat Transfer by Natural and Forced Convection, Engineering Bulletin, Purdue University*, Research Series No. 84, Volume 26, pp. 47-50.
20. Cowell, T. A., Heikel, M. R., and Achaichia, A. (1995): *Flow and Heat Transfer in Compact Louvered Fin Surfaces*, Experimental and Thermal Fluid Science, 10, pp. 192-199.
21. Cowell, T. A. (1990): *A general method for the comparison of compact heat transfer surfaces*, Trans. ASME, Vol. 112, pp. 288-294.
22. DeJong, N. C. and Jacobi, A. M. (2003a): *Flow, heat transfer, and pressure drop in the near-wall region of louvered –fin arrays*, Experimental Thermal and Fluid Science, 27, pp. 237-250
23. DeJong, N. C. and Jacobi, A. M. (2003b): *Localized flow and heat transfer interactions in louvered-fin arrays*, Int. J. Heat Mass Transfer, 46, pp. 443-455.
24. Dewan, A., Mahanta, P., Raju, K.S., and Suresh Kumar, P. (2004): *A Review of Passive Heat Transfer Augmentation Techniques*, Proc. Institution of Mechanical Engineers: Part A J. of Power and Energy, Vol. 218, pp. 509-527
25. DIN 24 163-2 (1985): *Ventilatoren, Leistungsmessung, Normprüfstände*, Beuth Verlag, Berlin.
26. DIN EN 308 (1997): *Wärmetauscher, Prüfverfahren zur Bestimmung der Leistungskriterien von Luft/Luft- und Luft/Abgas- Wärmerückgewinnungsanlagen*, Beuth Verlag, Berlin.
27. DIN EN 1216 (1999): *Wärmetauscher, Luftkühler und Lufterhitzer für erzwungene Konvektion*, Beuth Verlag, Berlin.
28. Dogruoz, B., Urdaneta, M., Ortega, A. (2002): Experiments *and Modeling of the Heat Transfer of In-Line Square Pin Fin Heat Sinks with Top By-Pass Flow*, Proceedings of IMECE 2002, ASME Int. Mechanical Engineering Congress & Exposition, New Orleans, LA.

29. Eckert, E. (1966): *Wärme- und Stoffaustausch*, Springer Velag, Berlin
30. Elsner, N., Fischer, S., Huhn, J. (1993): *Grundlagen der technischen Thermodynamik*, Bd. 2, 8 Auflage, Akademie Verlag, Berlin.
31. Ferzinger, J. H. and Peric, M. (2002): *Computational Methods for Fluid Dynamics*, 3 ed, Springer Verlag, Berling.
32. Gram, A. J., Mackey, C. O and Monroe, E. S. (1958): *Convection Heat Transfer and Pressure Drop of Air Flowing Across In-Line Tube Banks*, Part II, Trans. ASME, Vol. 80, pp. 24-35.
33. Greiner, M., Fischer, P.F., and Tufo, H.M. (2002): *Two-Dimensional Simulations of Enhanced Heat Transfer in an Intermittently Grooved Channel*, Trans. ASME J. of Heat Transfer, Vol. 124, pp. 538-545.
34. Haaland, S. E. (1983): Simple *and Explicit Formulas for the Friction Factor in Turbulent Pipe Flow, J. Fluid Engineering*, Vol. 105, pp. 89-90.
35. Han, C. J., Dutta, S., Ekkad, S. (2000): *Gas Turbine Heat Transfer and Cooling Technology*, Taylor & Francis, New York.
36. Harper, D. R. and Brown, W. B. (1922): *Mathematical equations for heat conduction in the fins of air-cooled engines*, NACA Report No. 158, Washington, DC.
37. Heggs, P. J. (1999): Extended *Surface Heat Transfer in Heat Exchangers and Performance Measurements*, in Heat Transfer Enhancement of Heat Exchangers (ed. Kakac, S., Bergles, A. E., Mayinger, F. and Yüncü, H.), NATO ASI Series, Serie E: Applied Sciences, Vol. 355, pp. 49-65.
38. Hewitt, G. F., Shires, G. L., Bott, T. R. (1994): *Process Heat Transfer*, CRC Press, Boca Raton, FL.
39. Holman, J. P. (1999): *Heat Transfer*, 8th ed., McGraw-Hill, New York.
40. Holmann, J. P. (2001): *Experimental Methods for Engineers*, McGraw-Hill, Boston.
41. Hu, S. and Herold, K. E. (1995a): *Prandtl number effect on offset fin heat exchanger performance: experimental results,* Int. J. Heat Mass Transfer, 38, pp. 1053-1061.
42. Hu, S. and Herold, K. E. (1995b): *Prandtl number effect on offset fin heat exchanger performance: predictive models for heat transfer and pressure drop. Int. J. Heat Mass Transfer*, 38, pp. 1043-1051.
43. Hwang, J. J., Lui, Ch. Ch. (2002): Measurement *of endwall heat transfer and pressure drop in a pin fin wedge duct*, Int. J. Heat Mass Transfer 45, pp. 877-889.
44. Hwang, J.J., and Chau, Ch. L. (1990): *Detailed heat transfer characteristic comparison in straight and 90-deg turned trapezoidal ducts with pin fin arrays*, Inter. J. Heat Mass Transfer 42, pp. 4005-4016.
45. Incropera, F., DeWitt, D. (2002): *Introduction to Heat Transfer*, 4th ed., Wiley, New York.
46. Isachenko, V., Osipova, V., Sukomel, A. (1969): *Heat Transfer*, Mir, Moscow. 9 References 137
47. Issa, J., Ortega, A. (2002): *Experimental Measurements of the Flow and heat Transfer of a Square Jet Impinging on an Array of Square Pin Fins*, Proceedings to IMECE 2002, ASME Int. Engineering Congress & Exposition, New Orleans, LA.
48. Jacobs, N. E. (1931). *Tests of six symmetrical airfoils in the variable density wind tunnel*, NACA TN No. 385, Washington, D.C.
49. Jacobs, N. E., Ward, K.E. and Pinkerton, R. M. (1933): *The characteristics of 78 related airfoil sections from tests in the variable-density wind tunnel*, NACA Report 460, Washington, D.C.
50. Jaluria, Y. (1998): *Design and Optimisation of Thermal Systems*, McGraw-Hill, New York.
51. Jones, C. E. and Monroe, E. S. (1958): *Convection Heat Transfer and Pressure Drop of Air Flowing Across In-Line Tube Banks*, Part I, Trans. ASME, Vol. 80, pp. 18-24.
52. Joshi, H. M, and Webb, R. I. (1987): *Heat Transfer and Friction in the Offset Strip-Fin Heat Exchangers. Int. J. Heat Mass Transfer*, 30, pp. 69-84.
53. Kays, W. M., London, A. L. (1950a): *Heat-Transfer and Flow-Friction Characteristics of Some Compact Heat-Exchanger Surfaces-Part I*, Trans. ASME, vol. 72, pp. 1075-1085.
54. Kays, W. M., and London, A. L., (1950b): *Heat-Transfer and Flow-Friction Characteristics of Some Compact Heat-Exchanger Surfaces-Part2*, Trans. ASME, 72, pp. 1087-1097.
55. Kays, W. M. (1955): *Pin-Fin Heat-Exchanger Surfaces*, Trans. ASME, Vol. 77, pp.471-483.
56. Kays, W. M. (1950c): *Loss Coefficients for Abrupt Changes in Flow Cross Section With Low Reynolds Number Flow in Single and Multiple-Tube Systems*, Transactions of ASME, Vol. 72, pp. 1067-1074.
57. Kays, W. M. and London, A. L. (1954): *Heat-Transfer and Friction Characteristics for Gas Flow Normal to Tube Banks - Use of a Transient-Test Technique*, Transactions of ASME, Vol. 76, pp. 387-396.
58. Kays, W. and Kraford, W. (1993): *Convective Heat Transfer*, 3rd ed., McGraw-Hill, New York.
59. Kays, W.M. and London, A.L. (1964): *Compact Heat Exchangers*, 2rd ed., McGraw-Hill, New York.
60. Kays, W.M. and London, A.L. (1998): *Compact Heat Exchangers*, 3rd Reprinted ed., Krieger, Malabar, FL.
61. Kim, H. M. and Bullard, W. C. (2002). *Air-side thermal hydraulic performance of multilouvered fin aluminium heat exchangers*, Int. J. Refrigeration, 25, pp. 390-400.
62. Kline, S.J. and McClintock, F.A. (1953). *Describing Uncertainties in Single-Sample Experiments, Mechanical Engineering*, Vol. 75, pp. 3-8.

63. Kraus, A., Aziz, A., Welty, J. (2001): *Extended Surface Heat Transfer*, Wiley, New York.
64. LaHaye, P. G., Neugebauer, F. J., Sakkhuja, R. K. (1974): *A Generalized Prediction of Heat Transfer Surfaces*, ASME J. of Heat Transfer, Vol. 96, pp. 511-517.
65. Leong, C. K and Toh, C. K. (1999): *An experimental investigation of heat transfer and flow friction characteristics of louvered fin surfaces by the modified single blow technique*, Heat and Mass Transfer, 35, pp. 53-65.
66. Leu, Sh. J., Min, Sh. Liu, Liaw, S. J. and Wang, Ch. Ch. (2001): *A numerical investigation of louvered fin – and –tube heat exchangers having circular and oval tube configurations*, Int. J. Heat Mass Transfer, 44, pp. 4235-4243.
67. Li, Q. Chen, Zh., Flechtner, U., Warnecke, H.-J. (1996): *Konvektive Wärme/Stoff- Übertragung und Druckverlust in Rohrbündeln bestehend aus tropfenförmigen Rohren*, Chemie-Ingenieur Technik (68), pp. 1299-1302.
68. Li, Q., Chen, Zh., Flechtner, U., Warnecke, H.-J. (1998): *Heat transfer and pressure drop characteristics in rectangular channels with elliptic pin fins*, Int. J. Heat Fluid Flow 19, pp. 245-250.
69. Lienhard, J. H. (1981): *A Heat Transfer Textbook*, Prentice Hall, Englewood Cliffs, NJ.
70. London, A. L., Ferguson, C. K. (1949): *Test Results of High-Performance Heat- Exchanger Surfaces Used in Aircraft Intercoolers and Their Significance for Gas-Turbine Regenerator Design*, Trans. ASME, Vol. 71, pp. 17-26.
71. Lozza, G. and Merlo, U. (2001): *An experimental investigation of heat transfer and friction losses of interrupted and wavy fins for fin-and-tube heat exchangers*, Int. J. Refrig. 24, pp. 409-416.
72. Manglick, R. M. (2003): *Heat Transfer Enhancement*, in Heat Transfer Handbook (Bejan, A. and Kraus, D. A., eds.), Wiley, New York..
73. Manglik, M. R. and Bergles, A. (1995): *Heat Transfer and Pressure Drop Correlations for the Rectangular Offset Strip Fin Compact Heat Exchangers*, Exp. Thermal Fluid Science, 10, pp. 171-180.
74. Maveety, J.G., Jung, H. H. (2000): *Design of an optimal pin fin heat sink with air impingement cooling*, Int. Commun. Heat Mass Transfer Vol.27 No.2, pp. 229-240.
75. Merker, G. P. and Eiglmeier, C. (1999): *Fluid- und Wärmetransport, Wärmeübertragung*, Teubner Verlag, Stuttgart.