SATHYABAMA
INSTITUTE OF SCIENCE AND TECHNOLOGY
(DEEMED TO BE UNIVERSITY)
Accredited "A" Grade by NAAC | 12B Status by UGC | Approved by AICTE
www.sathyabama.ac.in

SCHOOL OF MECHANICAL ENGINEERING

DEPARTMENT OF AERONAUTICAL ENGINEERING

UNIT – I – Heat Transfer Techniques For Aerospace Applications – SAE1306

UNIT – I

CONDUCTION

1.1 BASIC CONCEPT IN HEAT TRANSFER

1.1.1 Heat Energy and Heat Transfer

Heat is a form of energy in transition and it flows from one system to another, without transfer of mass, whenever there is a temperature difference between the systems. The process of heat transfer means the exchange in internal energy between the systems and in almost every phase of scientific and engineering work processes, we encounter the flow of heat energy.

1.1.2 Importance of Heat Transfer

Heat transfer processes involve the transfer and conversion of energy and therefore, it is essential to determine the specified rate of heat transfer at a specified temperature difference. The design of equipments like boilers, refrigerators and other heat exchangers require a detailed analysis of transferring a given amount of heat energy within a specified time. Components like gas/steam turbine blades, combustion chamber walls, electrical machines, electronic gadgets, transformers, bearings, etc require continuous removal of heat energy at a rapid rate in order to avoid their overheating. Thus, a thorough understanding of the physical mechanism of heat flow and the governing laws of heat transfer are a must.

1.1.3 Modes of Heat Transfer

The heat transfer processes have been categorized into three basic modes: Conduction, Convection and Radiation.

Conduction – It is the energy transfer from the more energetic to the less energetic particles of a substance due to interaction between them, a microscopic activity.

Convection - It is the energy transfer due to random molecular motion a long with the macroscopic motion of the fluid particles.

Radiation - It is the energy emitted by matter which is at finite temperature. All forms of matter emit radiation attributed to changes m the electron configuration of the constituent atoms or molecules the transfer of energy by conduction and convection requires the presence of a material medium whereas radiation does not. In fact radiation transfer is most efficient in vacuum.

All practical problems of importance encountered in our daily life Involve at least two, and sometimes all the three modes occurring simultaneously When the rate of heat flow is constant, i.e., does not vary with time; the process is called a steady state heat transfer process. When the temperature at any point in a system changes with time, the process is called unsteady or transient process. The internal energy of the system changes in such a process when the temperature variation of an unsteady process describes a particular cycle (heating or cooling of a budding wall during a 24 hour cycle), the process is called a periodic or quasi-steady heat transfer process.

Heat transfer may take place when there is a difference in the concentration of the mixture components (the diffusion thermo effect). Many heat transfer processes are accompanied by a transfer of mass on a macroscopic scale. We know that when water evaporates, the heal transfer is accompanied by the transport of the vapor formed through an air-vapor mixture. The transport of heat energy to steam generally occurs both through molecular interaction and convection. The combined molecular and convective transport of mass is called convection mass transfer and with this mass transfer, the process of heat transfer becomes more complicated.

1.1.4 Thermodynamics and Heat Transfer-Basic Difference

Thermodynamics is mainly concerned with the conversion of heat energy into other useful forms of energy and IS based on (i) the concept of thermal equilibrium (Zeroth Law), (ii) the First Law (the principle of conservation of energy) and (iii) the Second Law (the direction in which a particular process can take place). Thermodynamics is silent about the heat energy exchange mechanism. The transfer of heat energy between systems can only take place whenever there is a temperature gradient and thus. Heat transfer is basically a non-equilibrium phenomenon. The Science of heat transfer tells us the rate at which the heat energy can be transferred when there IS a thermal non-equilibrium. That IS, the science of heat transfer seeks to do what thermodynamics is inherently unable to do.

However, the subjects of heat transfer and thermodynamics are highly complementary.

Many heat transfer problems can be solved by applying the principles of conservation of energy (the First Law)

1.1.5 Dimension and Unit

Dimensions and units are essential tools of engineering. Dimension is a set of basic entities expressing the magnitude of our observations of certain quantities. The state of a system is identified by its observable properties, such as mass, density, temperature, etc. Further, the motion of an object will be affected by the observable properties of that medium in which the object is moving. Thus a number of observable properties are to be measured to identify the state of the system.

A unit is a definite standard by which a dimension can be described. The difference between a dimension and the unit is that a dimension is a measurable property of the system and the unit is the standard element in terms of which a dimension can be explicitly described with specific numerical values.

Every major country of the world has decided to use SI units. In the study of heat transfer the dimensions are: L for length, M for mass, e for temperature, T for time and the corresponding units are: meter for length, kilogram for mass, degree Celsius (^oC) or Kelvin (K) for temperature and second (s) for time. The parameters important in the study of heat transfer are tabulated in Table 1.1 with their basic dimensions and units of measurement.

Table 1.1 Dimensions and units of various parameters

Para	Dimension	Unit
Mass	M	Kilogram, kg
Length	L	meter, m
Time	T	seconds, s
Temperature	θ	Kelvin, K, Celcius ^oC
Velocity	L/T	meter/second,m/s
Density	ML^{-3}	kg/m^3
Force	$ML^{-1}T^{-2}$	Newton, N = 1 kg m/s^2
Pressure	ML^2T^{-2}	N/m^2, Pascal, Pa
Energy, Work	ML^2T^{-3}	N-m, = Joule, J
Power	ML^2T^{-3}	J/s, Watt, W
Absolute Viscosity	$ML^{-1}T^{-1}$	N-s/m^2, Pa-s
Kinematic Viscosity	L^2T^{-1}	m^2/s
Thermal Conductivity	$MLT^{-3}\theta^{-1}$	W/mK, W/m^oC

Heat Transfer Coefficient	$MT^{-3}\Upsilon^{-1}$	W/m^2K, $W/m^{2o}C$
Specific Heat	$L^2T^{-2}\Upsilon^{-1}$	J/kg K, J/kg^oC
Heat Flux	MT^{-3}	W/m^2

1.1.6 Mechanism of Heat Transfer by Conduction

The transfer of heat energy by conduction takes place within the boundaries of a system, or a cross the boundary of t he system into another system placed in direct physical contact with the first, without any appreciable displacement of matter comprising the system, or by the exchange of kinetic energy of motion of the molecules by direct communication, or by drift of electrons in the case of heat conduction in metals. The rate equation which describes this mechanism is given by Fourier Law

$$\dot{Q} = -kAdT/dx$$

Where $\dot{Q}$ = rate of heat flow in X-direction by conduction in J/S or W,

k = thermal conductivity of the material. It quantitatively measures the heat conducting ability and is a physical property of t he material that depends upon the composition of the material, W/mK,

A = cross-sectional area normal to the direction of heat flow, m^2,

dT/dx = temperature gradient at the section, as shown in Fig. 1 I The negative sign IS Included to make the heat transfer rate Q positive in the direction of heat flow (heat flows in the direction of decreasing temperature gradient).

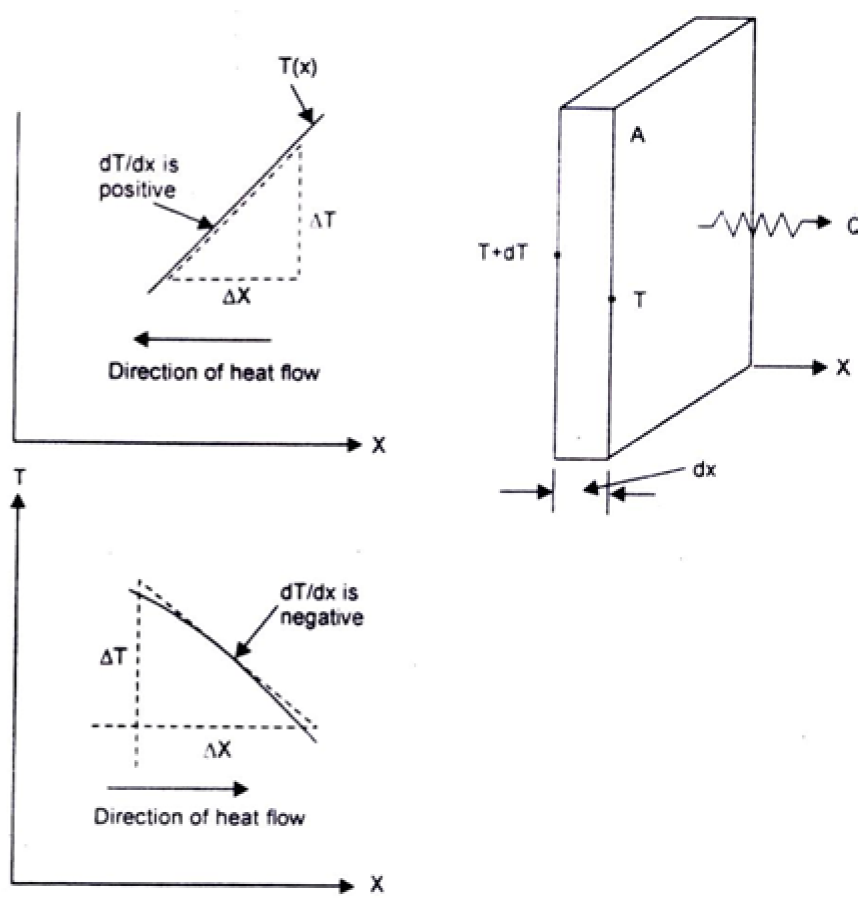


Fig 1.1: Heat flow by conduction

1.1.7 Thermal Conductivity of Materials

Thermal conductivity is a physical property of a substance and In general, it depends upon the temperature, pressure and nature of the substance. Thermal conductivity of materials is usually determined experimentally and a number of methods for this purpose are well known.

Thermal Conductivity of Gases: According to the kinetic theory of gases, the heat transfer by conduction in gases at ordinary pressures and temperatures take place through the transport of the kinetic energy arising from the collision of the gas molecules. Thermal conductivity of gases depends on pressure when very low «2660 Pal or very high (> 2×10^9 Pa). Since the specific heat of gases Increases with temperature, the thermal conductivity Increases with temperature and with decreasing molecular weight.

Thermal Conductivity of Liquids: The molecules of a liquid are more closely spaced and molecular force fields exert a strong influence on the energy exchange In the collision process. The mechanism of heat propagation in liquids can be conceived as transport of energy by way of unstable elastic oscillations. Since the density of liquids decreases with increasing temperature, the thermal conductivity of non-metallic liquids generally decreases with increasing temperature, except for liquids like water and alcohol because their thermal conductivity first Increases with increasing temperature and then decreases.

Thermal Conductivity of Solids (i) Metals and Alloys: The heat transfers in metals arise due to a drift of free electrons (electron gas). This motion of electrons brings about the equalization in temperature at all points of t he metals. Since electrons carry both heat and electrical energy. The thermal conductivity of metals is proportional to its electrical conductivity and both the thermal and electrical conductivity decrease with increasing temperature. In contrast to pure metals, the thermal conductivity of alloys increases with increasing temperature. Heat transfer In metals is also possible through vibration of lattice structure or by elastic sound waves but this mode of heat transfer mechanism is insignificant in comparison with the transport of energy by electron gas. (ii) Nonmetals: Materials having a high volumetric density have a high thermal conductivity but that will depend upon the structure of the material, its porosity and moisture content High volumetric density means less amount of air filling the pores of the materials. The thermal conductivity of damp materials considerably higher than the thermal conductivity of dry material because water has a higher thermal conductivity than air. The thermal conductivity of granular material increases with temperature. (Table 1.2 gives the thermal conductivities of various materials at 0^oC.)

Table 1.2 Thermal conductivity of various materials at 0^oC.

Material	Thermal conductivity (W/m K)	Material	Thermal conductivity (W/m K)
Gases		Solids: Metals	
Hydrogen .	0175	Sliver, pure	410
Helium	0141	Copper, pure	385
A"	0024	AlumllllUm, pure	202
Water vapour (saturated)	00206	Nickel, pure	93
Carbon dioxide	00146	Iron, pure	73

(thermal conductivity of helium and hydrogen are much higher than other gases. because then molecules have small mass and higher mean travel velocity)		Carbon steel, I %C	43
		Lead, pure	35
		Chrome-nickel-steel (18% Cr, 8% Ni)	16.3
		Non-metals	
Liquids		Quartz, parallel to axis	41.6
Mercury	821	Magnesite	4.15
Water*	0.556	Marble	2.08 to 2.94
Ammonia	0.54	Sandstone	1.83
Lubricating 011		Glass, window	0.78
SAE 40	0.147	Maple or Oak	0.17
Freon 12	0.073	Saw dust	0.059
		Glass wool	0.038

* Water has its maximum thermal conductivity (k = 068 W/mK) at about 150°C

1.2 STEADY STATE CONDUCTION ONE DIMENSION

1.2.1 The General Heat Conduction Equation for an Isotropic Solid with Constant Thermal Conductivity

Any physical phenomenon is generally accompanied by a change in space and time of its physical properties. The heat transfer by conduction in solids can only take place when there is a variation of temperature, in both space and time. Let us consider a small volume of a solid element as shown in Fig. 1.2. The dimensions are: Δx, Δy, Δz along the X-, Y-, and Z-coordinates.

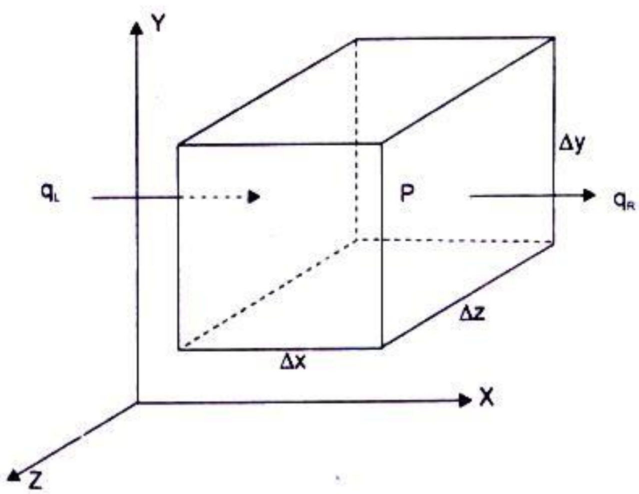


Fig 1.2 Elemental volume in Cartesian coordinates

First we consider heat conduction the X-direction. Let T denote the temperature at the point P (x, y, z) located at the geometric centre of the element. The temperature gradient at the left hand face (x - ~x12) and at the right hand face (x + ϫx/2), using the Taylor's series, can be written as:

$$\partial T/\partial x\,|_L = \partial T/\partial x - \partial^2 T/\partial x^2 . \Delta x/2 \text{ + higher order terms.}$$

$$\partial T/\partial x\,|_R = \partial T/\partial x + \partial^2 T/\partial x^2 . \Delta x/2 + \text{ higher order terms.}$$

The net rate at which heat is conducted out of the element 10 X-direction assuming k as constant and neglecting the higher order terms,

We get $-k\Delta y\Delta z\left[\frac{\partial T}{\partial x}+\frac{\partial^2 T}{\partial x^2}\frac{\Delta x}{2}-\frac{\partial T}{\partial x}+\frac{\partial^2 T}{\partial x^2}\frac{\Delta x}{2}\right] = -k\Delta y\Delta z\Delta x\left(\frac{\partial^2 T}{\partial x^2}\right)$

Similarly for Y- and Z-direction,

We have $-k\Delta x\Delta y\Delta z\ \partial^2 T/\Delta y^2$ and $-k\Delta x\Delta y\Delta z\ \partial^2 T/\Delta z^2$.

If there is heat generation within the element as Q, per unit volume and the internal energy of the element changes with time, by making an energy balance, we write

Heat generated within the element	Heat conducted away from the element	Rate of change of internal energy within with the element

or, $\dot{Q}_v(\Delta x\Delta y\Delta z) + k(\Delta x\Delta y\Delta z)\left(\partial^2 T/\partial x^2 + \partial^2 T/\partial y^2 + \partial^2 T/\partial z^2\right)$

$$= \rho c(\Delta x\Delta y\Delta z)\ \partial T/\partial t$$

Upon simplification, $\partial^2 T/\partial x^2 + \partial^2 T/\partial y^2 + \partial^2 T/\partial z^2 + \dot{Q}_v/k = \frac{\rho c}{k}\partial T/\partial t$

or, $\nabla^2 T + \dot{Q}_v/k = 1/\alpha(\partial T/\partial t)$

Where, $\alpha = k/\rho . c$, is called the thermal diffusivity and is seen to be a physical property of the material of which the solid is composed.

The Eq. (2.la) is the general heat conduction equation for an isotropic solid with a constant thermal conductivity. The equation in cylindrical (radius r, axis Z and longitude Ɵ) coordinates is written as: Fig. 2.I (b),

$$\partial^2 T/\partial r^2 + (1/r)\partial T/\partial r + \left(1/r^2\right)\partial^2 T/\partial \theta^2 + \partial^2 T/\partial z^2 + \dot{Q}_v/k = 1/\alpha\, \partial T/\partial t \qquad (2.1b)$$

And, in spherical polar coordinates Fig. 2.1(c) (radius, Θ longitude, and ϕ colatitudes) is

$$\frac{1}{r^2}\frac{\partial}{\partial r}\left(r^2\frac{\partial T}{\partial r}\right) + \frac{1}{r^2 \sin\theta\, \partial\theta}\left(\sin\theta\frac{\partial T}{\partial \theta}\right) + \frac{1}{r^2\sin^2\theta}\frac{\partial^2 T}{\partial \phi^2} + \frac{\dot{Q}_v}{k} = \frac{1}{\alpha}\frac{\partial T}{\partial t} \qquad (2.1c)$$

Under steady state or stationary condition, the temperature of a body does not vary with time, i.e. $\partial T/\partial t = 0$. And, with no internal generation, the equation (2.1) reduces to

$$\nabla^2 T = 0$$

It should be noted that Fourier law can always be used to compute the rate of heat transfer by conduction from the knowledge of temperature distribution even for unsteady condition and with internal heat generation.

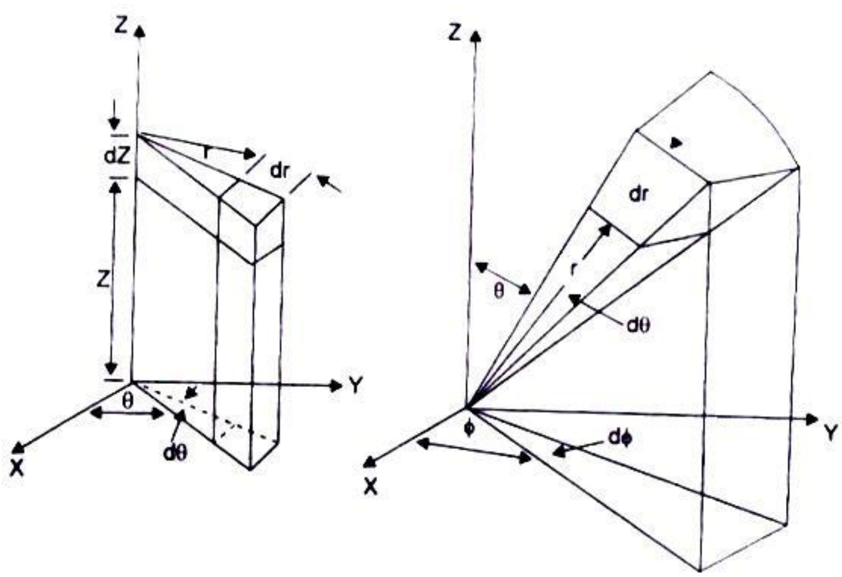


Fig1.3: Elemental volume in cylindrical coordinates and spherical coordinates

. One-Dimensional Heat Flow

The term 'one-dimensional' is applied to heat conduction problem when:

(i) Only one space coordinate is required to describe the temperature distribution within a heat conducting body;

(ii) Edge effects are neglected;

(iii) The flow of heat energy takes place along the coordinate measured normal to the surface.

1. 3 Thermal Diffusivity and its Significance

Thermal diffusivity is a physical property of the material, and is the ratio of the material's ability to transport energy to its capacity to store energy. It is an essential para for transient processes of heat flow and defines the rate of change in temperature. In general, metallic solids have higher α, while nonmetallic, like paraffin, have a lower value of α. Materials having large α respond quickly to changes in their thermal environment, while materials have lowered a respond very slowly, take a longer time to reach a new equilibrium condition.

1.4 TEMPERATURE DISTRIBUTION IN I-D SYSTEMS

1.4.1 Plane Wall

A plane wall is considered to be made out of a constant thermal conductivity material and extends to infinity in the Y- and Z-direction. The wall is assumed to be homogeneous and isotropic, heat flow is one-dimensional, under steady state conditions and losing negligible energy through the edges of the wall under the above mentioned assumptions the Eq. (2.2) reduces to

$d^2T / dx^2 = 0$; the boundary conditions are: at $x = 0, T = T_1$

Integrating the above equation, $x = L, T = T_2$

$T = C_1x + C_2$, where C_1 and C_2 are two constants.

Substituting the boundary conditions, we get $C_2 = T_1$ and $C_1 = (T_2 - T_1)/L$ The temperature distribution in the plane wall is given by

$$T = T_1 - (T_1 - T_2)\, x/L \tag{2.3}$$

Which, is linear and is independent of the material.

Further, the heat flow rate, $\dot{Q}/A = -k\, dT/dx = (T_1 - T_2)\, k/L$, and therefore the temperature distribution can also be written as

$$T - T_1 = \left(\dot{Q}/A\right)\left(x/k\right) \tag{2.4}$$

i.e., "the temperature drop within the wall will increase with greater heat flow rate or when k is small for the same heat flow rate,"

1.4.2 A Cylindrical Shell-Expression for Temperature Distribution

In the cylindrical system, when the temperature is a function of radial distance only and is independent of azimuth angle or axial distance, the differential equation (2.2) would be, (Fig. 1.4)

$$d^2T/dr^2 + (1/r)\, dT/dr = 0$$

with boundary conditions: at $r = r_1$, $T = T_1$ and at $r = r_2$, $T = T_2$.

The differential equation can be written as:

$$\frac{1}{r}\frac{d}{dr}\left(r\, dT/dr\right) = 0, \text{ or, } \frac{d}{dr}\left(r\ dT/dr\right) = 0$$

upon integration, $T = C_1 \ln(r) + C_2$, where C_1 and C_2 are the arbitrary constants.

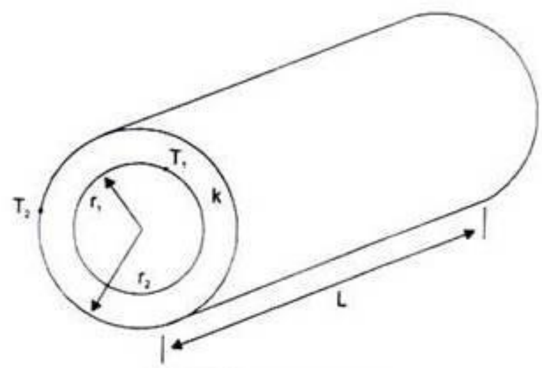


Fig 1.4: A Cylindrical shell

By applying the boundary conditions,

$$C_1 = (T_2 - T_1)/\ln(r_2/r_1)$$

and $$C_2 = T_1 - \ln(r_1).(T_2 - T_1)/\ln(r_2/r_1)$$

The temperature distribution is given by

$$T = T_1 + (T_2 - T_1).\ \ln(r/r_1)/\ln(r_2/r_1) \text{ and}$$

$$\dot{Q}/L = -kA\ dT/dr \quad = 2\pi k(T_1 - T_2)/\ln(r_2/r_1) \qquad (2.5)$$

From Eq (2.5) It can be seen that the temperature varies 10gantJunically through the cylinder wall In contrast with the linear variation in the plane wall.

If we write Eq. (2.5) as $\dot{Q} = kA_m(T_1 - T_2)/(r_2 - r_1)$, where

$$A_m = 2\pi(r_2 - r_1)L/\ln(r_2/r_1) \quad = (A_2 - A_1)/\ln(A_2/A_1)$$

Where, A_2 and A_1 are the outside and inside surface areas respectively. The term A_m is called 'Logarithmic Mean Area' and the expression for the heat flow through a cylindrical wall has the same form as that for a plane wall.

1.4.3 Spherical and Parallelopiped Shells--Expression for Temperature Distribution

Conduction through a spherical shell is also a one-dimensional steady state problem if the interior and exterior surface temperatures are uniform and constant. The Eq. (2.2) in one-dimensional spherical coordinates can be written as

$$(1/r^2)\frac{d}{dT}(r^2 dT/dr) = 0 \text{, with boundary conditions,}$$

at $\quad r = r_1,\ T = T_1;\ \text{at } r = r_2,\ T = T_2$

or, $\quad \frac{d}{dr}(r^2 dT/dr) = 0$

and upon integration, T = –C_1/r + C_2, where c_1 and c_2 are constants. substituting the boundary conditions,

$$C_1 = (T_1 - T_2)r_1 r_2/(r_1 - r_2), \text{ and } C_2 = T_1 + (T_1 - T_2)r_1 r_2/r_1(r_1 - r_2)$$

The temperature distribution m the spherical shell is given by

$$T = T_1 - \left\{\frac{(T_1 - T_2)r_1 r_2}{(r_2 - r_1)}\right\} \times \left\{\frac{(r - r_1)}{r\ r_1}\right\} \qquad (2.6)$$

and the temperature distribution associated with radial conduction through a sphere is represented by a hyperbola. The rate of heat conduction is given by

$$\dot{Q} = 4\pi k(T_1 - T_2)r_1 r_2/(r_2 - r_1) = k(A_1 A_2)^{1/2}(T_1 - T_2)/(r_2 - r_1) \qquad (2.7)$$

Where, $A_1 = 4\pi_1^2$ and $A_2 = 4\pi r_2^2$

If A_l is approximately equal to A_2 i.e., when the shell is very thin,

$$\dot{Q} = kA(T_1 - T_2)/(r_2 - r_1) \text{; and } \dot{Q}/A = (T_1 - T_2)/\Delta r/k$$

which is an expression for a flat slab.

The above equation (2.7) can also be used as an approximation for parallelopiped shells which have a smaller inner cavity surrounded by a thick wall, such as a small furnace surrounded by a large thickness of insulating material, although the h eat flow especially in the corners, cannot be strictly considered one-dimensional. It has been suggested that for $(A_2/A_1) > 2$, the rate of heat flow can be approximated by the above equation by multiplying the geometric mean area $A_m = (A_1 A_2)^{½}$ by a correction factor 0.725.]

1.4.4 Composite Surfaces

There are many practical situations where different materials are placed m layers to form composite surfaces, such as the wall of a building, cylindrical pipes or spherical shells having different layers of insulation. Composite surfaces may involve any number of series and parallel thermal circuits.

1.4.5 Heat Transfer Rate through a Composite Wall

Let us consider a general case of a composite wall as shown m Fig. 1.5 There are 'n' layers of different materials of thicknesses L_1, L_2, etc and having thermal conductivities k_l, k_2, etc. On one side of the composite wall, there is a fluid A at temperature T_A and on the other side of the wall there is a fluid B at temperature T_B. The convective heat transfer coefficients on the two sides of the wall are h_A and h_B respectively. The system is analogous to a series of resistances as shown in the figure.

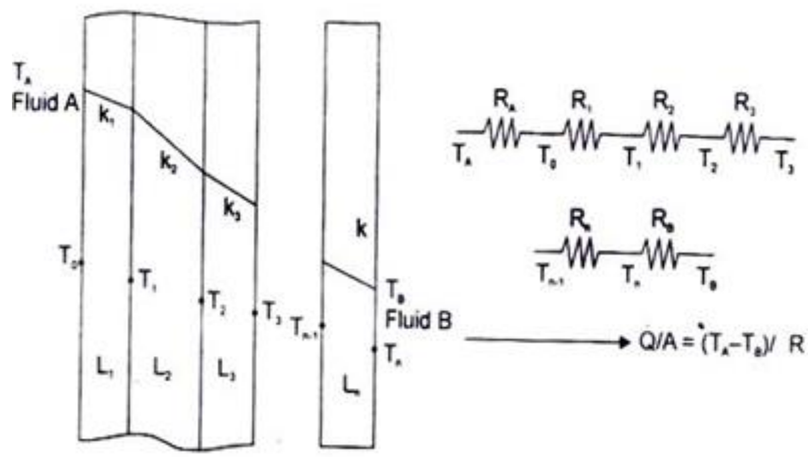


Fig 1.5 Heat transfer through a composite wall

1.4.6 The Equivalent Thermal Conductivity

The process of heat transfer through compos lie and plane walls can be more conveniently compared by introducing the concept of 'equivalent thermal conductivity', k_{eq}. It is defined as:

$$k_{eq} = \left(\sum_{i=1}^{n} L_i\right) \Bigg/ \sum_{i=1}^{n} (L_i / k_i) \qquad (2.8)$$

$$= \frac{\text{Total thickeness of the composite wall}}{\text{Total thermal resistance of the composite wall}}$$

And, its value depends on the thermal and physical properties and the thickness of each constituent of the composite structure.

Example 1.2 A furnace wall consists of 150 mm thick refractory brick (k = 1.6 W/mK) and 150 mm thick insulating fire brick (k = 0.3 W/mK) separated by an au gap (resistance 0 16 K/W). The outside walls covered with a 10 mm thick plaster (k = 0.14 W/mK). The temperature of hot gases is 1250°C and the room temperature is 25°C. The convective heat transfer coefficient for gas side and air side is 45 W/m2K and 20 W/m^2K. Calculate (i) the rate of heat flow per unit area of the wall surface (ii) the temperature at the outside and inside surface of the wall and (iii) the rate of heat flow when the air gap is not there.

Solution: Using the nomenclature of Fig. 2.3, we have per m2 of the area, h_A = 45, and

$R_A = 1/h_A = 1/45 = 0.0222$; $h_B = 20$, and $R_B = 1120 = 0.05$

Resistance of the refractory brick, $R_1 = L_1/k_1 = 0.15/1.6 = 0.0937$

Resistance of the insulating brick, $R_3 = L_3/k_3 = 0.15/0.30 = 0.50$

The resistance of the air gap, $R_2 = 0.16$

Resistance of the plaster, $R_4 = 0.01/0.14 = 0.0714$

Total resistance = 0.8973, m^2K/W

Heat flow rate = $\Sigma T/\Sigma R = (1250-25)/0.8973 = 13662\ W/m^2$

Temperature at the inner surface of the wall

$= T_A - 1366.2 \times 0.0222 = 1222.25$

Temperature at the outer surface of the wall

$= T_B + 1366.2 \times 0.05 = 93.31\ °C$

When the air gap is not there, the total resistance would be

0.8973 - 0.16 = 0.7373

and the heat flow rate = $(1250 - 25)/0/7373 = 1661.46\ W/m^2$

The temperature at the inner surface of the wall

$= 1250 - 1660.46 \times 0.0222 = 1213.12°C$

i.e., when the au gap is not there, the heat flow rate increases but the temperature at the inner surface of the wall decreases.

The overall heat transfer coefficient U with and without the air gap is

$$U = \left(\dot{Q}/A\right) / \Delta T$$

$= 13662 / (1250 - 25) = 1.115\ Wm^2\ °C$

and $1661.46/l225 = 1356\ W/m^{2o}C$

The equivalent thermal conductivity of the system without the air gap

$k_{eq} = (0.15 + 0.15 + 0.01)/ (0.0937 + 0.50 + 0.0714) = 0.466\ W/mK.$

Example 1.2 A brick wall (10 cm thick, k = 0.7 W/m°C) has plaster on one side of the wall (thickness 4 cm, k = 0.48 W/m°C). What thickness of an insulating material (k = 0.065 W m°C) should be added on the other side of the wall such that the heat loss through the wall IS reduced by 80 percent.

Solution: When the insulating material is not there, the resistances are:

$R_1 = L_1/k_1 = 0.1/0.7 = 0.143$

and $R_2 = 0.04/0.48 = 0.0833$

Total resistance = 0.2263

Let the thickness of the insulating material is L_3. The resistance would then be

$L_3/0.065 = 15.385\ L_3$

Since the heat loss is reduced by 80% after the insulation is added.

$$\frac{\dot{Q}\ \text{with insulation}}{\dot{Q}\ \text{without insulation}} = 0.2 = \frac{R\ \text{without insulation}}{R\ \text{with insulation}}$$

or, the resistance with insulation = 0.2263/0.2 = 01.1315

and, 15385 L_3 = I 1315 – 0.2263 = 0.9052

$L_3 = 0.0588$ m = 58.8 mm

Example 1.3 An ice chest IS constructed of styrofoam (k = 0.033 W/mK) having inside dimensions 25 by 40 by 100 cm. The wall thickness is 4 cm. The outside surface of the chest is exposed to air at 25°C with h = 10 W/m^{2}K. If the chest is completely filled with ice, calculate the time for ice to melt completely. The heat of fusion for water is 330 kJ/kg.

Solution: If the heat loss through the comers and edges are ignored, we have three walls of walls through which conduction heat transfer will occur.

(a) 2 walls each having dimensions 25 cm × 40 cm × 4 cm

(b) 2 walls each having dimensions 25 cm × 100 cm × 4 cm

(c) 2 walls each having dimensions 40 cm × 100 cm × 4 cm

The surface area for convection heat transfer (based on outside dimensions)

$2(33 \times 48 + 33 \times 108 + 48 \times 108) \times 10^{-4} = 2.0664\ m^2$.

Resistance due to conduction and convection can be written as

$$2\left(\frac{0.04}{0.033\times0.25\times0.4}+\frac{0.04}{0.033\times0.25\times1}+\frac{0.04}{0.033\times0.4\times1}\right)+\frac{1}{10\times2.0664}$$

$= 40 + 0.0484 = 40.0484$ K/W

$\dot{Q} = \Delta T / \Sigma R = (25 - 0.0) / 40.0484 = 0.624$ W

Inside volume of the container - $0.25 \times 04 \times 1 = 0.1\ m^3$

Mass of Ice stored = $800 \times 0.1 = 80$ kg; taking the density of Ice as 800 kg/m^3. The time required to melt 80 kg of ice is

$$t = \frac{80\times330\times1000}{0.624\times3600\times24} = 490 \text{ days}$$

Example1.4 A composite furnace wall is to be constructed with two layers of materials (k_1 = 2.5 W/m°C and k_2 = 0.25 W/m°C). The convective heat transfer coefficient at the inside and outside surfaces is expected to be 250 $W/m^{2o}C$ and 50 $W/m^{2o}C$ respectively. The temperature of gases and air are 1000 K and 300 K. If the interface temperature is 650 K, Calculate (i) the thickness of the two materials when the total thickness does not exceed 65 cm and (ii) the rate of heat flow. Neglect radiation.

Solution: Let the thickness of one material (k = 2.5 W / mK) is xm, then the thickness of the other material (k = 0.25 W/mK) will be (0.65 –x) m.

For steady state condition, we can write

$$\frac{\dot{Q}}{A} - \frac{1000-650}{\frac{1}{250}+\frac{x}{2.5}} = \frac{1000-300}{\frac{1}{250}+\frac{x}{2.5}+\frac{(0.65-x)}{0.25}+\frac{1}{50}}$$

$$\therefore\ \ 700(0.004+0.4x) = 350\{0.004+0.4x+4(0.65-x)+0.02\}$$

(i) 6x = 3.29 and x= 0.548 m.

and the thickness of the other material = 0.102 m.

(ii) $\dot{Q}/A$ = (350) / (0.004 + 0.4 × 0.548) = 1.568 kW/m^2

Example 1.5 A composite wall consists of three layers of thicknesses 300 rum, 200 mm and 100 mm with thermal conductivities 1.5, 3.5 and is W/mK respectively. The inside surface is exposed to gases at 1200°C with convection heat transfer coefficient as 30W/m^{2}K. The temperature of air on the other side of the wall is 30°C with convective heat transfer coefficient 10 Wm^{2}K. If the temperature at the outside surface of the wall is 180°C, calculate the temperature at other surface of the wall, the rate of heat transfer and the overall heat transfer coefficient.

Solution: The composite wall and its equivalent thermal circuits is shown in the figure.

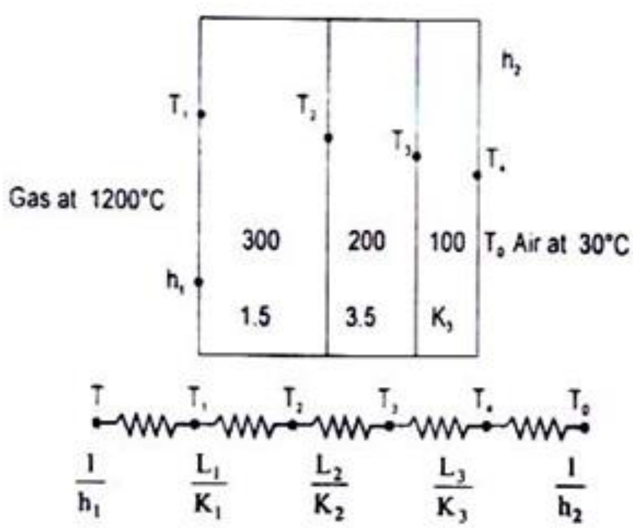


Fig 1.6

The heat energy will flow from hot gases to the cold air through the wall.

From the electric Circuit, we have

$$\dot{Q}/A = h_2(T_4 - T_0) = 10\times(180-30) = 1500\,W/m^2$$

also, $\dot{Q}/A = h_1(1200 - T_1)$

$$T_1 = 1200 - 1500/30 = 1150^oC$$

$$\dot{Q}/A = (T_1 - T_2)/L_1/k_1$$

$$T_2 = T_1 - 1500 \times 0.3/1.5 = 850$$

Similarly, $\dot{Q}/A = (T_2 - T_3)/(L_2/k_2)$

$$T_3 = T_2 - 1500 \times 0.2/3.5 = 764.3^{o}C$$

and $\dot{Q}/A = (T_3 - T_4)/(L_3/k_3)$

$L_3/k_3 = (764.3 - 180)/1500$ and k_3 = 0.256 W/mK

Check:

$$\dot{Q}/A = (1200 - 30)/\Sigma R;$$

Where, $\Sigma R = 1/h_1 + L_1/k_1 + L_2/k_2 + L_3/k_3 + 1/h_2$

$$\Sigma R = 1/30 + 0.3/1.5 + 0.2/3.5 + 0.1/0.256 + 1/10 = 0.75$$

and $\dot{Q}/A = 1170/0.78 = 1500\ W/m^2$

The overall heat transfer coefficient, $U = 1/\Sigma R = 1/0.78 = 1.282\ W/m^2K$

Since the gas temperature is very high, we should consider the effects of radiation also. Assuming the heat transfer coefficient due to radiation = 3.0 W/m^2K the electric circuit would be:

The combined resistance due to convection and radiation would be

$$\frac{1}{R} = \frac{1}{R_1} + \frac{1}{R_2} = \frac{1}{\frac{1}{h_c}} + \frac{1}{\frac{1}{h_r}} = h_c + h_r = 60W/m^{2o}C$$

$$\therefore\ \dot{Q}/A = 1500 = 60(T - T_1) = 60(1200 - T_1)$$

$$\therefore\ T_1 = 1200 - \frac{1500}{60} = 1175^{o}C$$

again, $\therefore\ \dot{Q}/A = (T_1 - T_2)/L_1/k_1 \Rightarrow T_2 = T_1 - 1500 \times 0.3/1.5 = 875^{o}C$

and $\quad T_3 = T_2 - 1500 \times 0.2/3.5 = 789.3^{o}C$

$L_3/k_3 = (789.3 - 180)/1500; \ \therefore k_3 = 0.246 \text{ W/mK}$

$$\Sigma R = \frac{1}{60} + \frac{0.3}{1.5} + \frac{0.2}{1.5} + \frac{0.2}{3.5} + \frac{0.1}{0.246} + \frac{1}{10} = 0.78$$

and $\quad U = 1/\Sigma R = 1.282 \text{ W/m}^2\text{K}$

Example 1.6 A flat roof (12 m x 20 m) of a building has a composite structure It consists of a 15 cm lime-khoa plaster covering (k = 0 17 W/m°C) over a 10 cm cement concrete (k = 0.92 W/m°C). The ambient temperature is 42°C. The outside and inside heat transfer coefficients are 30 W/m2°C and 10 W/m2 0C. The top surface of the roof absorbs 750 W/m2 of solar radiant energy. The temperature of the space may be assumed to be 260 K. Calculate the temperature of the top surface of the roof and the amount of water to be sprinkled uniformly over the roof surface such that the inside temperature is maintained at 18°C.

Solution: The physical system is shown in Fig. 1.7 and it is assumed we have one-dimensional flow, properties are constant and steady state conditions prevail.

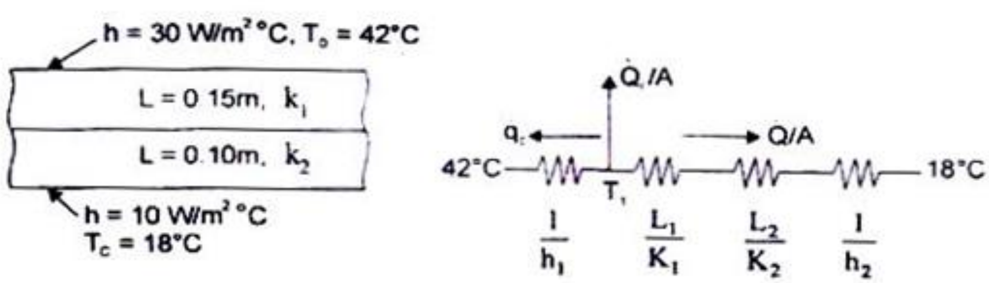


Fig 1.7

Let the temperature of the top surface be T_1°C.

Heat lost by thee top surface by convection to the surroundings is

$$\dot{Q}_c / A = h(\Delta T) = 30 \times (T_1 - 42) = (30T_1 - 1260)$$

Heat energy conducted inside through the roof = $(\Delta T / \Sigma R)$

or, $\frac{\dot{Q}}{A} = \frac{T_1 - 18}{\frac{L_1}{k_1} + \frac{L_2}{k_2} + \frac{1}{h_2}} = (T_1 - 18) / \left(\frac{0.15}{0.17} + \frac{0.1}{0.92} + \frac{1}{10} \right)$ = 0.918 (T_1 – 18)

Assuming that the top surface of the roof behaves like a black body, energy lost by radiation.

$$\dot{Q}_r / A = \sigma \left[(T_1 + 273)^4 - 260^4 \right] \quad = 5.67 \times 10^{-8} (T_1 + 273)^4 - 259.1$$

By making an energy balance on the top surface of the roof,

Energy coming in = Energy going out

750 = (30T, -1260)+ 0.918 (T_1-18) + 5.67 × 10^{-8} (T_1 + 273)^4 - 259.1

or, 2285.624 = 30.918 T_1 + 5.67 × 10^{-8} (T_1 + 273)^4

Solving by trial and error, T_1 = 53.4°C, and the total energy conducted through the roof per hour is

0.918 (53.4 – 18) × (12 × 20) × 3600 = 28077.58 kJ/hr

Assuming the latent heat of vaporization of water as 2430 kJ/kg, the quantity of water to be sprinkled over the surface such that it evaporates and consumes 28077.58 kJ/hr, is

$\dot{M}_w$ = 28077.58/2430 = 11.55 kg/hr.

Example 1.7 An electric hot plate is maintained at a temperature of 350°C and is used to keep a solution boiling at 95°C. The solution is contained in a cast iron vessel (wall thickness 25 mm, k = 50 W/mK) which is enameled inside (thickness 0.8 mm, k = 1.05 Wm/K) the heat transfer coefficient for the boiling solution is 5.5 kW/mK. Calculate (i) the overall heat transfer coefficient and (ii) heat transfer rate.

If the base of the cast iron vessel is not perfectly flat and the resistance of the resulting air film is 35 m2K1kW, calculated the rate of heat transfer per unit area. (Gate'93)

Solution: The physical system is shown in the figure below.

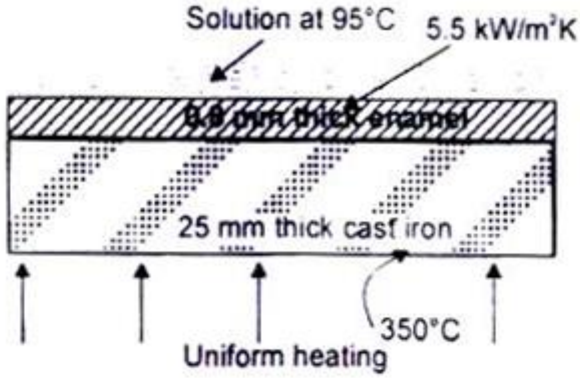


Fig 1.8

Under steady state conditions,

$$\dot{Q}/A = U(\Delta T) = \frac{(\Delta T)}{1/U}$$, where, U is the overall heat transfer coefficient.

$$= \frac{(\Delta T)}{R} = \frac{(\Delta T)}{\frac{L_1}{k_1} + \frac{L_2}{k_2} + \frac{1}{h}}$$

Therefore,

$$1/U = \frac{L_1}{k_1} + \frac{L_2}{k_2} + \frac{1}{h} = \left(\frac{0.025}{50} + \frac{0.0008}{1.05} + \frac{1}{5500}\right) = 0.00144$$

U = 692.65 W/m^{2}K

$\dot{Q}/A = U(\Delta T)$ = 692.65 × (350 – 95) = 176.65 kW/m^2.

With the presence of air film at the base, the total resistance to heat flow would be:

0.00144 + 0.035 = 0.03644 m^{2}K/W

and the rate of heat transfer, $\dot{Q}/A$ = 255/0.03644 = 7 kW/m^2.

(Fig. 1.9 shows a combination of thermal resistance placed in series and parallel for a composite wall having one-dimensional steady state heat transfer. By drawing analogous electric circuits, we can solve such complex problems having both parallel and series thermal resistances.)

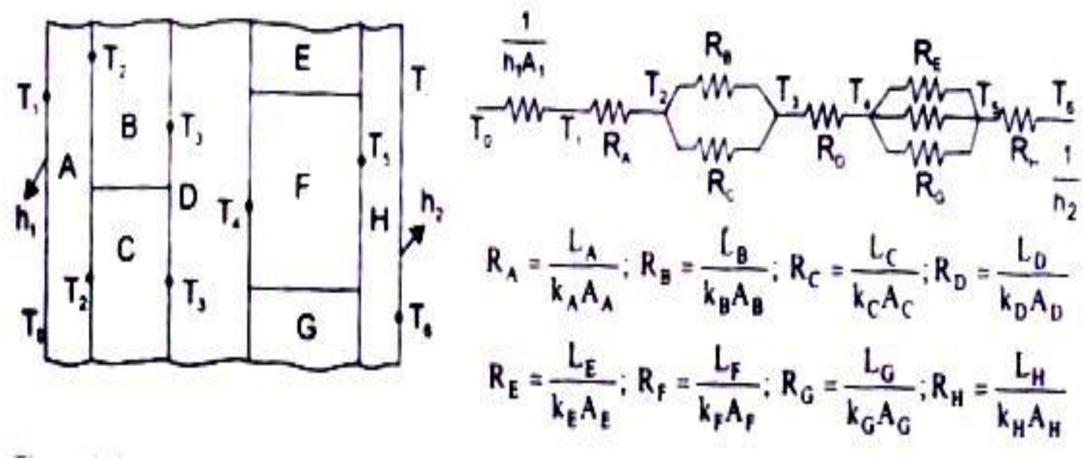


Fig. 1.9 Series and parallel one-dimensional heat transfer through a composite wall with convective heat transfer and its electrical analogous circuit

Example 1.8 A door (2 m x I m) is to be fabricated with 4 cm thick card board (k = 0.2 W /mK) placed between two sheets of fiber glass board (each having a thickness of 40 mm and k = 0.04 W/mK). The fiber glass boards are fastened with 50 steel studs (25 mm dia, k = 40 W/mK). Estimate the percentage of heat transfer flow rate through the studs.

Solution: The thermal circuit with steel studs can be drawn as in Fig. 1.10.

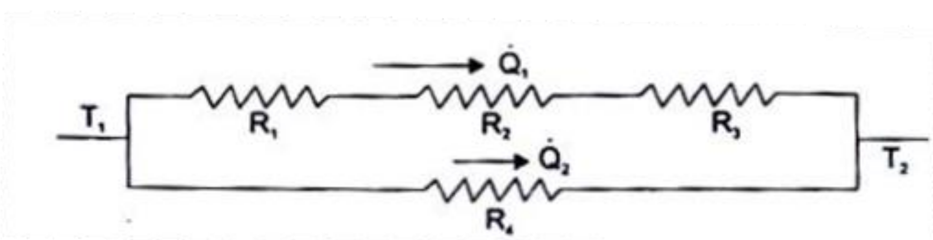


Fig 1.10

The cross-sectional area or the surface area of the door for the heat transfer is $2 m^2$. The cross-sectional area of the steel studs is:

$50 \times □/4\ (0.025)^2 = 0.02455\ m^2$

and the area of the door – area of the steel studs = 2.0 – 0.02455 = 1.97545

R_1, the resistance due to fiber glass board on the outside

= L/kA = 0.04/(0.04 × 1.97545) = 0.506.

R_2, the resistance due to card board = 0.101

R_3, the resistance due to fiber glass board on the inside = 0.506

R_4, the resistance due to steel studs = L/kA = 0.121 (40 × 0.2455) = 0 1222

With reference to Fig 2.9, $\dot{Q}_1 = (T_1 - T_2)/\Sigma R = (T_1 - T_2)/1.113$

and $\quad \dot{Q}_2 = (T_1 - T_2)/0.1222$

Therefore, $\quad \dot{Q}_2 / (\dot{Q}_1 + \dot{Q}_2) = 8.1833/9.0818 = 0.9$

ie, 90 percent of the heat transfer will take place through the studs.

Example 1.9 Find the heat transfer rate per unit depth through the composite wall sketched. Assume one dimensional heat flow.

Solution: The analogous electric circuit has been drawn in the figure.

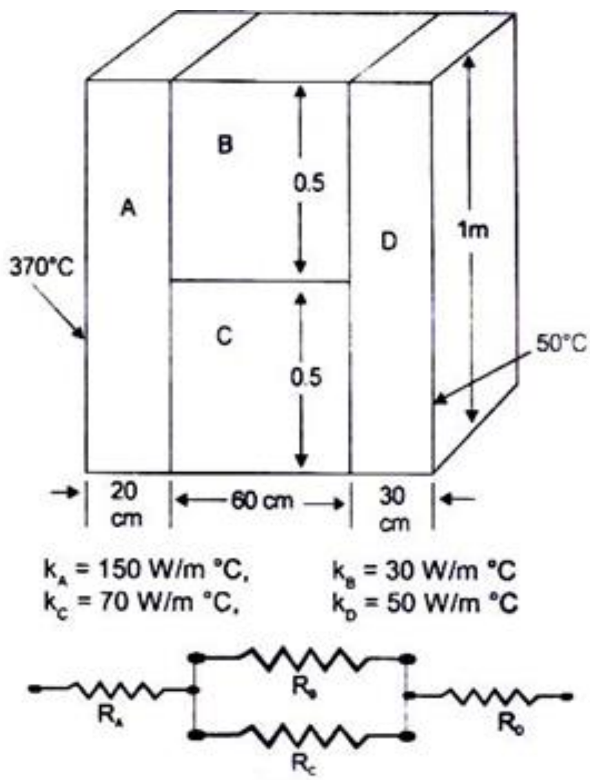


Fig 1.11

R_A = 0.2/150 = 0.00133

$R_B = 0.6/(30 \times 0.5) = 0.04$

$R_C = 0.6/(70 \times 0.5) = 0.017$

$R_D = 0.3/50 = 0.006$

$1/R_B + 1/R_C = 1/R_{BC} = 83.82$

Therefore, $R_{BC} = 1/83.82 = 0.0119$

Total resistance to heat flow = 0.00133 + 0.0119 + 0006 = 0.01923

Rate of heat transfer per unit depth = (370–50)/ 0.01923 = 16.64 kW m.

The Significance of Biot Number

Let us consider steady state conduction through a slab of thickness L and thermal conductivity k. The left hand face of the wall is maintained at T constant temperature T_1 and the right hand face is exposed to ambient air at T_o, with convective heat transfer coefficient h. The analogous electric circuit will have two thermal resistances: $R_1 = L/k$ and $R_2 = l/h$. The drop in temperature across the wall and the air film will be proportional to their resistances, that is, $(L/k)/(1/h) = hL/k$.

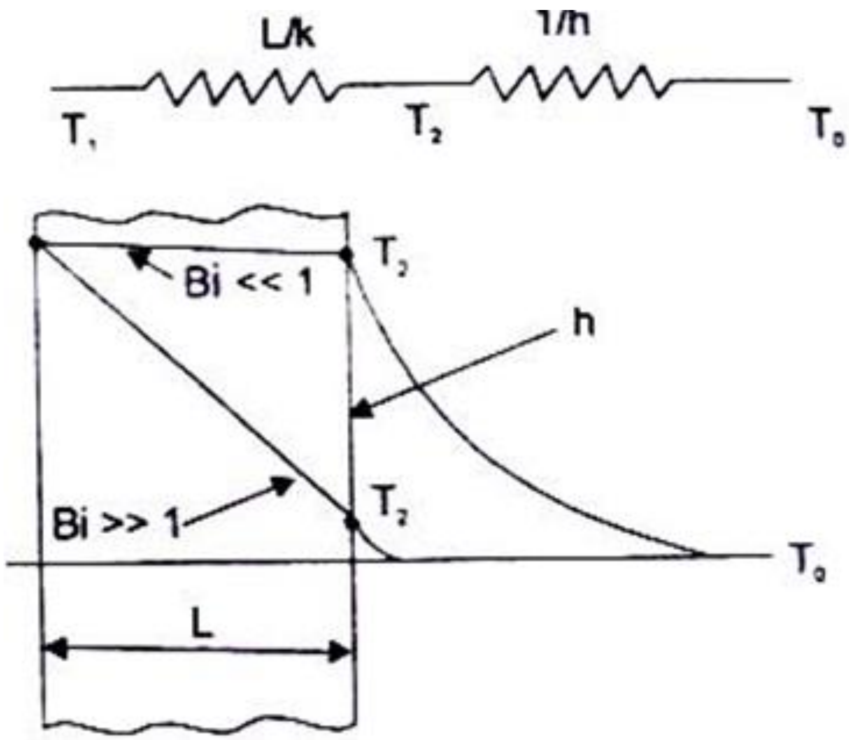


Fig 1.12: Effect of Biot number on temperature profile

This dimensionless number is called ‘Biot Number’ or,

$$B_i = \frac{\text{Conduction resistance}}{\text{Convection resistance}}$$

When Bi >> 1, the temperature drop across the air film would be negligible and the temperature at the right hand face of the wall will be approximately equal to the ambient temperature. Similarly, when Bi «I, the temperature drop across the wall is negligible and the transfer of heat will be controlled by the air film resistance.

1.5. The Concept of Thermal Contact Resistance

Heat flow rate through composite walls are usually analyzed on the assumptions that - (i) there is a perfect contact between adjacent layers, and (ii) the temperature at the interface of the two plane surfaces is the same. However, in real situations, this is not true. No surface, even a so-called 'mirror-finish surface', is perfectly smooth ill a microscopic sense. As such, when two surfaces are placed together, there is not a single plane of contact. The surfaces touch only at limited number of spots, the aggregate of which is only a small fraction of the area of the surface or 'contact area'. The remainder of the space between the surfaces may be filled with air or other fluid. In effect, this introduces a resistance to heat flow at the interface. This resistance IS called 'thermal contact resistance' and causes a temperature drop between the materials at the interfaces as shown In Fig. 2.12. (That is why, Eskimos make their houses having double ice walls separated by a thin layer of air, and in winter, two thin woolen blankets are more comfortable than one woolen blanket having double thickness.)

Fig. 2.12 Temperature profile with and without contact resistance when two solid surfaces are joined together

Example 1.10 A furnace wall consists of an inner layer of fire brick 25 cm thick k = 0.4 W/mK and a layer of ceramic blanket insulation, 10 cm thick k = 0.2 W/mK. The thermal contact resistance between the two walls at the interface is 0.01 m^2K/w. Calculate the temperature drop at the interface if the temperature difference across the wall is 1200K.

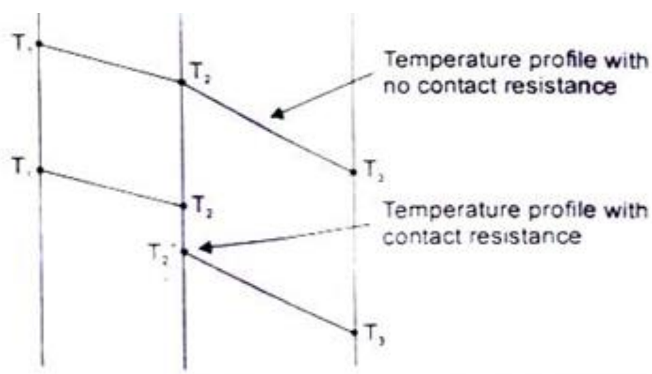


Fig 1.13: temperature profile with and without contact resistance when two solid surfaces are joined together

Solution: The resistance due to inner fire brick = L/k = 0.25/0.4 = 0.625.

The resistance of the ceramic insulation = 0.1/0.2 = 0.5

Total thermal resistance = 0.625 + 0.01 + 0.5 = 1 135

Rate of heat flow, $\dot{Q}/A = \Delta t / R_{th}$ =1200/1 135 =1057.27 W/m^2

Temperature drop at the interface,

$$\Delta T = \left(\dot{Q}/A\right) \times R = 1057.27 \times 0.01 = 10.57\ K$$

Example 1.11 A 20 cm thick slab of aluminum (k = 230 W/mK) is placed in contact with a 15 cm thick stainless steel plate (k = 15 W/mK). Due to roughness, 40 percent of the area is in direct contact and the gap (0.0002 m) is filled with air (k = 0.032 W/mK). The difference in temperature between the two outside surfaces of the plate is 200°C Estimate (i) the heat flow rate, (ii) the contact resistance, and (iii) the drop in temperature at the interface.

Solution: Let us assume that out of 40% area m direct contact, half the surface area is occupied by steel and half is occupied by aluminum.

The physical system and its analogous electric circuits is shown in Fig. 2.13.

$$R_1 = \frac{0.2}{230 \times 1} = 0.00087, \qquad R_2 = \frac{0.0002}{230 \times 0.2} = 4.348 \times 10^{-6}$$

$$R_3 = \frac{0.0002}{0.032 \times 0.6} = 1.04 \times 10^{-2}, \qquad R_4 = \frac{0.0002}{15 \times 0.2} = 6.667 \times 10^{-5}$$

and $R_5 = \frac{0.15}{(15 \times 1)} = 0.01$

Again $1/R_{2,3,4} = 1/R_2 + 1/R_3 + 1/R_4$

$= 2.3 \times 10^5 + 96.15 + 1.5 \times 10^4 = 24.5 \times 10^4$

Therefore, $R_{2,3,4} = 4.08 \times 10^{-6}$

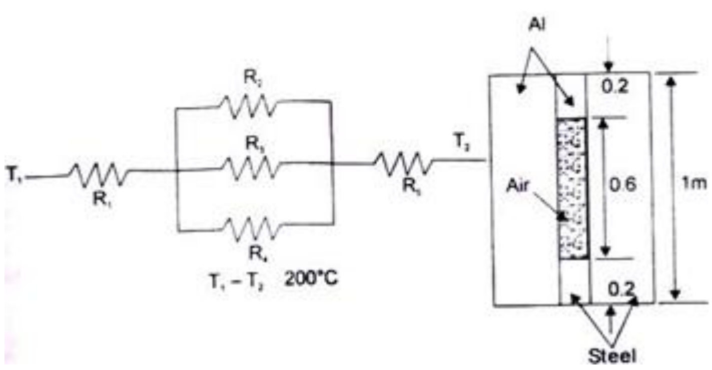


Fig 1.14

Total resistance, $\Sigma R = R_1 + R_{2,3,4} + R_5$

$= 870 \times 10^{-6} + 4.08 \times 10^{-6} + 1000 \times 10^{-6} = 1.0874 \times 10^{-2}$

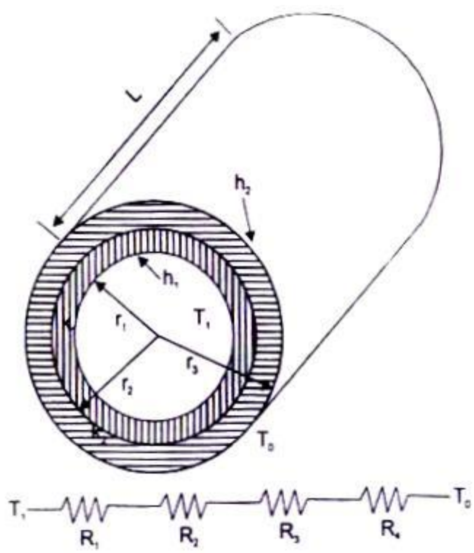
Heat flow rate, $\dot{Q}$ = 200/1.087 × 10^{-2} = 18.392 kW per unit depth of the plate.

Contact resistance, R $R_{2,\,3,\,4} = 4.08 \times 10^{-6}$ mK / W

Drop in temperature at the interface, ΔT = 4.08 × 10^{-6} × 18392 = 0.075°C

1.6. An Expression for the Heat Transfer Rate through a Composite Cylindrical System

Let us consider a composite cylindrical system consisting of two coaxial cylinders, radii r_1, r_2 and r_2 and r_3, thermal conductivities k_l and k_2 the convective heat transfer coefficients at the inside and outside surfaces h_1 and h_2 as shown in the figure. Assuming radial conduction under



steady state conditions we have:

Fig 1.15

$R_1 = 1/h_1A_1 = 1/2\ \pi_1\ Lh_1$

$R_2 = \ln(r_2/r_1) 2\pi Lk_1$

$R_3 = \ln(r_3/r_2) 2\pi Lk_2$

$R_4 = 1/h_2A_2 = 1/2\pi_3 h_2 L$

And $\dot{Q}/2\pi L = (T_1 - T_0)/\Sigma R$

$$= \left(T_1 - T_0\right) / \left[\left(1/h_1 r_1 + \ln\left(r_2/r_1\right)/k_1 + \ln\left(r_3 + r_2\right)/k_2 + 1/h_2 r_3\right)\right]$$

Example 1.12 A steel pipe. Inside dia 100 mm, outside dia 120 mm (k 50 W/mK) IS Insulated with a 40 mm thick high temperature Insulation (k = 0.09 W/mK) and another Insulation 60 mm thick (k = 0.07 W/mK). The ambient temperature IS 25°C. The heat transfer coefficient for the inside and outside surfaces is 550 and 15 W/m^{2}K respectively. The pipe carries steam at 300^{o}C. Calculate (1) the rate of heat loss by steam per unit length of the pipe (11) the temperature of the outside surface

Solution: I he cross-section of the pipe with two layers of insulation is shown 111 Fig. 1.16. with its analogous electrical circuit.

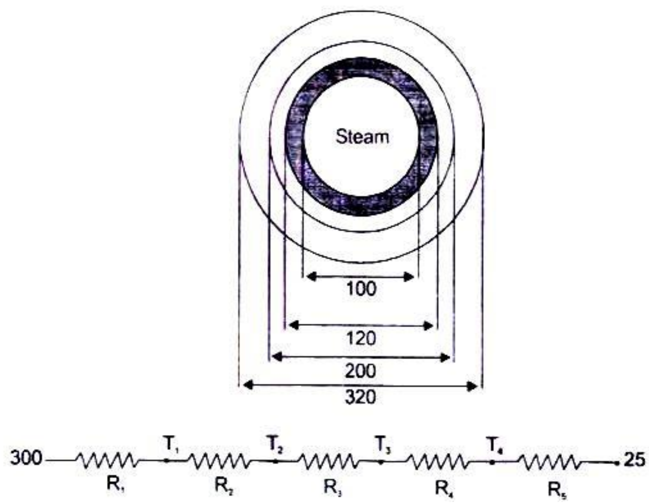


Fig1.16 Cross-section through an insulated cylinder, thermal resistances in series.

For L = 1.0 m. we have

R_1, the resistance of steam film = 1/hA = $1/(500 \times 2 \times 3.14 \times 50 \times 10^{-3}) = 0.00579$

R_2, the resistance of steel pipe = $\ln(r_2/r_l) / 2\ \pi\ k$

$= \ln(60/50)/2\ \pi \times 50 = 0.00058$

R_3, resistance of high temperature Insulation

$\ln(r_3/r_2) / 2\ \pi\ k = \ln(100/60) / 2\ \pi \times 0.09 = 0.903$

$R_4 = 1n(r_4/r_3)/2\ \pi\ k = \ln(160/100)/2\ \pi \times 0.07 = 1.068$

R_5 = resistance of the air film = $1/(15 \times 2\pi \times 160 \times 10^{-3}) = 0.0663$

The total resistance = 2.04367

and $\dot{Q} = \Delta T / \Sigma R = (300 - 25) / 204367 = 134.56$ W per meter length of pipe.

Temperature at the outside surface. $T_4 = 25 + R_5$,

$\dot{Q} = 25 + 134.56 \times 0.0663 = 33.92^o$ C

When the better insulating material (k = 0.07, thickness 60 mm) is placed first on the steel pipe, the new value of R_3 would be

$R_3 = \ln(120/60) / 2\pi \times 0.07 = 1.576$; and the new value of R_4 will be

$R_4 = \ln(160/120)\ 2\pi \times 0.09 = 0.5087$

The total resistance = 2.15737 and Q = 275/2.15737 = 127.47 W per m length (Thus the better insulating material be applied first to reduce the heat loss.) The overall heat transfer coefficient, U, is obtained as $U = \dot{Q} / A\,\Delta T$

The outer surface area = $\pi \times 320 \times 10^{-3} \times 1 = 1.0054$

and U = 134.56/(275 × 1.0054) = 0.487 W/m^2 K.

Example 1.13 A steam pipe 120 mm outside dia and 10m long carries steam at a pressure of 30 bar and 099 dry. Calculate the thickness of a lagging material (k = 0.99 W/mK) provided on the steam pipe such that the temperature at the outside surface of the insulated pipe does not exceed 32°C when the steam flow rate is 1 kg/s and the dryness fraction of steam at the exit is 0.975 and there is no pressure drop.

Solution: The latent heat of vaporization of steam at 30 bar = 1794 kJ/kg.

The loss of heat energy due to condensation of steam = 1794(0.99 – 0.975)

= 26.91 kJ/kg.

Since the steam flow rate is 1 kg/s, the loss of energy = 26.91 kW.

The saturation temperature of steam at 30 bar IS 233.84°C and assuming that the pipe material offers negligible resistance to heat flow, the temperature at the outside surface of the uninsulated steam pipe or at the inner surface of the lagging material is 233.84°C. Assuming one-dimensional radial heat flow through the lagging material, we have

$\dot{Q} = (T_1 - T_2)/[\ln(r_2/r_l)]\ 2\pi Lk$

or, 26.91 × 1000 (W) = (233.84 – 32) × 2 π × 10 × 0.99/1n(r/60)

ln (r/60) = 0.4666

$r_2/60 = \exp(0.4666) = 1.5946$

r_2 = 95.68 mm and the thickness = 35.68 mm

Example 1.14 A Wire, dia 0.5 mm length 30 cm, is laid coaxially in a tube (inside dia 1 cm, outside dia 1.5 cm, k = 20 W/mK). The space between the wire and the inside wall of the tube behaves like a hollow tube and is filled with a gas. Calculate the thermal conductivity of the gas if the current flowing through the wire is 5 amps and voltage across the two ends is 4.5 V, temperature of the wire is 160°C, convective heat transfer coefficient at the outer surface of the tube is 12 W/m^2K and the ambient temperature is 300K.

Solution: Assuming steady state and one-dimensional radial heat flow, we can draw the thermal circuit as shown In Fig. 1 17.

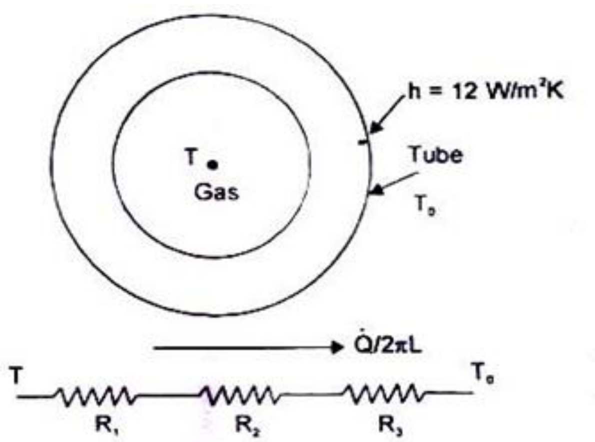


Fig 1.17

The rate of heat transfer through the system,

$\dot{Q}/2\pi L = VI/2\pi L = (4.5 \times 5)/(2 \times 3.142 \times 0.3) = 11.935$ (W/m)

R_1, the resistance due to gas = $\ln(r_2/r_l)$, k = ln(0.01/0.0005)/k = 2.996/k.

R_2, resistance offered by the metallic tube = ln(r_3 / r_2) k

= ln(1.5 /1.0) / 20 = 0.02

R_3, resistance due to fluid film at the outer surface

$l/hr_3 = 1/(l2\times1.5\times I0^{-2}) = 5.556$

and $\dot{Q} / 2\pi L$ = ΣT/Rth = [(273 + 160) – 300]/Rth

Therefore, R = 133/11.935 = 11.l437, and

R_1 = 2.9996/k = 11.1437 – 0.02 – 5.556 = 5.568

or, k = 2.996/5.568 = 0.538 W/mK.

Example 1.15 A steam pipe (inner dia 16 cm, outer dia 20 cm, k = 50 W/mK) is covered with a 4 cm thick insulating material (k = 0.09 W/mK). In order to reduce the heat loss, the thickness of the insulation is Increased to 8mm. Calculate the percentage reduction in heat transfer assuming that the convective heat transfer coefficient at the Inside and outside surfaces are 1150 and 10 W/m^2K and their values remain the same.

Solution: Assuming one-dimensional radial conduction under steady state,

$\dot{Q}$ / 2□L = □T/□R

R_1, resistance due to steam film = 1/hr = 1/(1150 × 0.08) = 0.011

R_2, resistance due to pipe material = ln (r_2/r_1)/k = ln (10/8)/50 = 0.00446

R_3, resistance due to 4 cm thick insulation

= ln(r_3/r_2)/k = ln(14/10)/0.09 = 3.738

R_4, resistance due to air film = 1/hr = 1/(10 × 0.14) = 0.714.

Therefore, $\dot{Q} / 2\pi L = \Delta T$ / (0.011 + 0.00446 + 3.738 + 0.714) = 0.2386ΣT

When the thickness of the insulation is increased to 8 cm, the values of R_3 and R_4 will change.

R_3 = ln(r_3/r_2)/k = ln(18/10)/0.09 = 6.53 ; and

R_4 = 1/hr = 1/(10 × 0.18) = 0.556

Therefore, $\dot{Q}/2\pi L = \Delta T$ / (0.011 + 0.00446 + 6.53 + 0.556)

= ΔT / 7.1 = 0.14084 ΔT

Percentage reduction in heat transfer = $\dfrac{(0.22386 - 0.14084)}{0.22386} = 0.37 = 37\%$

Example 1.16 A small hemispherical oven is built of an inner layer of insulating fire brick 125 mm thick (k = 0.31 W/mK) and an outer covering of 85% magnesia 40 mm thick (k = 0.05 W/mK). The inner surface of the oven is at 1073 K and the heat transfer coefficient for the outer surface is 10 W/m^{2}K, the room temperature is 20^{o}C. Calculate the rate of heat loss through the hemisphere if the inside radius is 0.6 m.

Solution: The resistance of the fire brick

$$= (r_2 - r_1)/2\pi k r_1 r_2 = \frac{0.725 - 0.6}{2\pi \times 0.31 \times 0.6 \times 0.725} = 0.1478$$

The resistance of 85% magnesia

$$= (r_3 - r_2)/2\pi k r_2 r_3 = \frac{0.765 - 0.725}{2\pi \times 0.05 \times 0.725 \times 0.765} = 0.2295$$

The resistance due to fluid film at the outer surface = 1/hA

$$= \frac{1}{10 \times 2\pi \times (0.765 \times 0.765)} = 0.2295$$

The resistance due to fluid film at the outer surface = 1/hA

$$= \frac{1}{10 \times 2\pi \times (0.765 \times 0.765)} = 0.0272$$

Rate of heat flow, $\dot{Q} = \Delta T/\Sigma R = \dfrac{800 - 20}{0.1478 + 0.2295 + 0.272} = 1930\text{W}$

Example 1.17 A cylindrical tank with hemispherical ends is used to store liquid oxygen at –180^{o}C. The dia of the tank is 1.5 m and the total length is 8 m. The tank is covered with a 10 cm thick layer of insulation. Determine the thermal conductivity of the insulating material so that the boil off rate does not exceed 10 kg/hr. The latent heat of vaporization of liquid oxygen is 214 kJ/kg. Assume that the outer surface of insulation is at 27^{o}C and the thermal resistance of the wall of the tank is negligible. (ES-94)

Solution: The maximum amount of heat energy that flows by conduction from outside to inside = Mass of liquid oxygen × Latent heat of vaporization.

= 10 × 214 = 2140 kJ/hr = 2140 × 1000/3600 = 594.44 W

Length of the cylindrical part of the tank = 8 – 2r = 8 – 1.5 = 6.5m

Since, the thermal resistance of the wall does not offer any resistance to heat flow, the temperature at the inside surface of the insulation can be assumed as - 183°C whereas the temperature at the outside surface of the insulation is 27°C.

Heat energy coming in through the cylindrical part, $\dot{Q}_1 = \dfrac{\Delta T}{\dfrac{\ln(r_2/r_1)}{2\pi Lk}}$

or, $\dot{Q}_1 = \dfrac{(27+183)\times 2\pi \times 6.5\,k}{\ln(8.5/7.5)} = 68531.84\ k$

Heat energy coming in through the two hemispherical ends,

$\dot{Q}_2 = 2\times(\Delta T\times 2\pi k\ r_2 r_1)/(r_2 - r_1) = \dfrac{2\times 210\times 2\pi k\times 0.85\times 0.75}{0.10}$ = 16825.4 K

Therefore, 594.44 = (68531.84 + 16825.4) k; or, k = 6.96 × 10^{-3} W/mK.

Example 1.18 A spherical vessel, made out of2.5 mm thick steel plate IS used to store 10m3 of a liquid at 200°C for a thermal storage system. To reduce the heat loss to the surroundings, a 10 cm thick layer of insulation (k = 0.07 W/rnK) is used. If the convective heat transfer coefficient at the outer surface is W/m^{2}K and the ambient temperature is 25°C, calculate the rate of heat loss neglecting the thermal resistance of the steel plate.

If the spherical vessel is replaced by a 2 m dia cylindrical vessel with flat ends, calculate the thickness of insulation required for the same heat loss.

Solution: Volume of the spherical vessel = $10\text{m}^3 = \dfrac{4\pi r^3}{3} \quad \therefore\ r = 1.336\ \text{m}$

Outer radius of the spherical vessel, $r_2 = 1.3364 + 0.025 = 1.361\ \text{m}$

Outermost radius of the spherical vessel after the insulation = 1.461 m.

Since the thermal resistance of the steel plate is negligible, the temperature at the inside surface of the insulation is 200°C.

Thermal resistance of the insulating material = $\left(r_3 - r_2\right)/4\pi\ k\ r_3 r_2$

$$= \frac{0.1}{4\pi \times 0.07 \times 1.461 \times 1.361} = 0.057$$

Thermal resistance of the fluid film at the outermost surface = 1/hA

$$= 1/\left[10 \times 4\pi \times (1.461)^2\right] = 0.00373$$

Rate of heat flow = $\Delta T/\Sigma R = \left(200 - 25\right)/\left(0.057 + 0.00373\right) = 2873.8\ \text{W}$

Volume of the insulating material used = $\left(4/3\right)\pi\left(r_3^3 - r_2^3\right) = 2.5\ \text{m}^3$

Volume of the cylindrical vessel $= 10\ \text{m}^3 = \dfrac{\pi}{4}(d)^2\ L;\ \ \therefore\ L = 10/\pi = 3.183\text{m}$

Outer radius of cylinder without insulation = 1.0 + 0.025 = 1.025 m.

Outermost radius of the cylinder (with insulation) = r_3.

Therefore, the thickness of insulation = r_3 – 1.025 = □

Resistance, the heat flow by the cylindrical element

$$= \frac{\ln\left(r_3/1.025\right)}{2\pi Lk} + 1/hA = \frac{\ln\left(r_3/1.025\right)}{2\pi \times 3.183 \times 0.07} + \frac{1}{10 \times 2\pi \times r_3 \times 3.183}$$

= 0.714 ln (r_3 / 1.025) + 0.005/r_3

Resistance to heat flow through sides of the cylinder

$$= 2\delta / kA + 1/hA = \frac{2\left(r_3 - 1.025\right)}{0.07\times\pi\times1} + \frac{1}{10\times2\times\pi}$$

$$= 9.09\left(r_3 - 1.025\right) + 0.0159$$

For the same heat loss, $\Delta T / \Sigma R$ would be equal in both cases, therefore,

$$\frac{1}{0.06073} = \frac{1}{0.714\ \ln\left(r_3/1.025\right) + 0.005/r_3} + \frac{1}{9.09\left(r_3 - 1.025\right) + 0.0159}$$

Solving by trial and error, (r – 1.025j) = □ = 9.2 cm.

and the volume of the insulating material required = 2.692 m^3.

1.7 Unsteady State Conduction Heat Transfer

1.7.1 Transient State Systems-Defined

The process of heat transfer by conduction where the temperature varies with time and with space coordinates is called 'unsteady or transient'. All transient state systems may be broadly classified into two categories:

(a) Non-periodic Heat Flow System - the temperature at any point within the system changes as a non-linear function of time.

(b) Periodic Heat Flow System - the temperature within the system undergoes periodic changes which may be regular or irregular but definitely cyclic.

There are numerous problems where changes in conditions result in transient temperature distributions and they are quite significant. Such conditions are encountered in - manufacture of ceramics, bricks, glass and heat flow to boiler tubes, metal forming, heat treatment, etc.

1.7.2. Biot and Fourier Modulus-Definition and Significance

Let us consider an initially heated long cylinder (L >> R) placed in a moving stream of fluid at $T_\infty < T_s$, as shown In Fig. 3.1(a). The convective heat transfer coefficient at the surface is h, where,

Q = hA ($T_s - T_\infty$)

This energy must be conducted to the surface, and therefore,

$Q = -kA(dT / dr)_{r = R}$

or, $h\,(T_s - T_\infty) = -k\,(dT/dr)_{r=R} \approx -k\,(T_c - T_s)/R$

Where, T_c is the temperature at the axis of the cylinder

By rearranging, $(T_s - T_c) / (T_s - T_\infty)\ h/Rk$ (3.1)

The term, hR/k, IS called the 'BlOT MODULUS'. It is a dimensionless number and is the ratio of internal heat flow resistance to external heat flow resistance and plays a fundamental role in transient conduction problems involving surface convection effects. I t provides a measure 0 f the temperature drop in the solid relative to the temperature difference between the surface and the fluid.

For Bi << 1, it is reasonable to assume a uniform temperature distribution across a solid at any time during a transient process.

Founer Modulus - It is also a dimensionless number and is define as

$Fo = \alpha t/L^2$ (3.2)

Where, L is the characteristic length of the body, a is the thermal diffusivity, and t is the time

The Fourier modulus measures the magnitude of the rate of conduction relative to the change in temperature, i.e., the unsteady effect. If Fo << 1, the change in temperature will be experienced by a region very close to the surface.

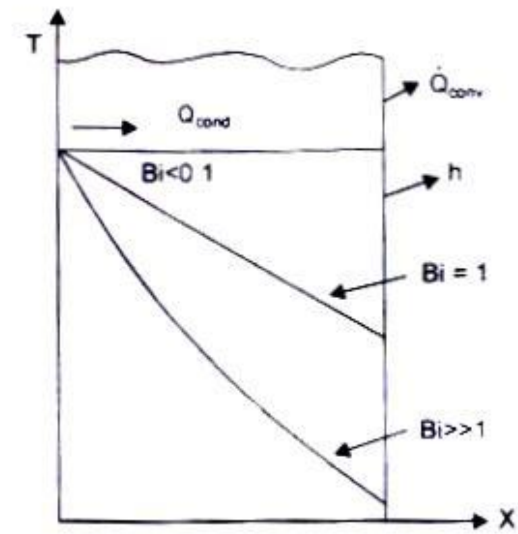


Fig. 1.18 Effect of Biot Modulus on steady state temperature distribution in a plane wall with surface convection.

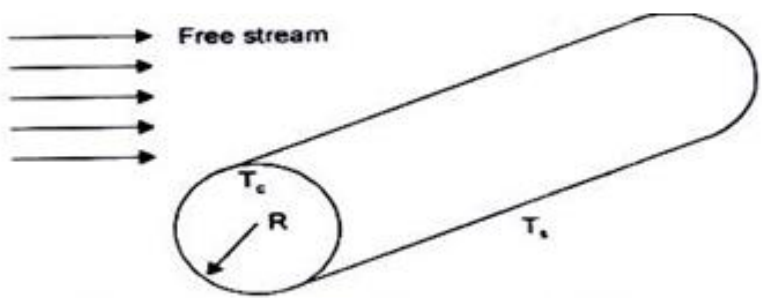


Fig. 1.18 (a) Nomenclature for Biot Modulus

1.7.3. Lumped Capacity System-Necessary Physical Assumptions

We know that a temperature gradient must exist in a material if heat energy is to be conducted into or out of the body. When Bi < 0.1, it is assumed that the internal thermal resistance of the body is very small in comparison with the external resistance and the transfer of heat energy is primarily controlled by the convective heat transfer at the surface. That is, the temperature within the body is approximately uniform. This idealized assumption is possible, if

(a) The physical size of the body is very small,

(b) The thermal conductivity of the material is very large, and

(c) The convective heat transfer coefficient at the surface is very small and there is a large temperature difference across the fluid layer at the interface.

7.4. An Expression for Evaluating the Temperature Variation in a Solid Using Lumped Capacity Analysis

Let us consider a small metallic object which has been suddenly immersed in a fluid during a heat treatment operation. By applying the first law of

Heat flowing out of the body = Decrease in the internal thermal energy of

during a time dt the body during that time dt

or, $hA_s\,(T - T_\infty)\,dt = -\,pCVdT$

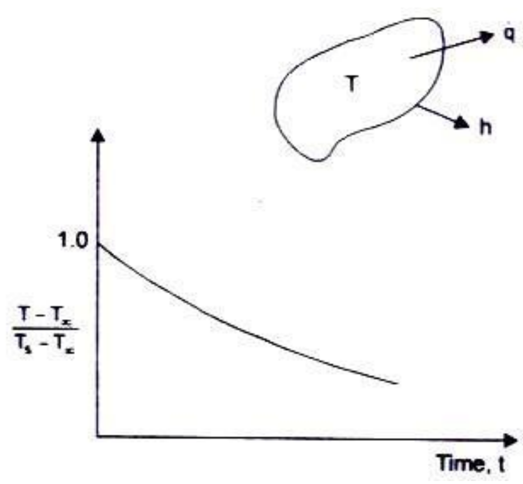
Where A_s is the surface area of the body, V is the volume of the body and C is the specific heat capacity.

or, $(hA/\rho CV)\,dt = -\,dT/(T - T_\infty)$

with the initial condition being: at t = 0, $T = T_s$

The solution is: $(T - T_\infty)/(T_s - T_\infty) = \exp(-hA/\rho CV)t$ (3.3)

Fig. 3.2 depicts the cooling of a body (temperature distribution ~ time) using lumped thermal capacity system. The temperature history is seen to be an exponential decay.



We can express

$Bi \times Fo = (hL/k) \times (\alpha t/L^2) = (hL/k)\,(k/\rho C)\,(t/L^2) = (hA/\rho CV)\,t,$

Where, V / A is the characteristic length L.

And, the solution describing the temperature variation of the object with respect to time is given by

$$(T - T_\infty)/(T_s - T_\infty) = \exp(-Bi \cdot Fo) \qquad (3.4)$$

Example 1.19 Steel balls 10 mm in dia (k = 48 W/mK), (C = 600 J/kgK) are cooled in air at temperature 35°C from an initial temperature of 750°C. Calculate the time required for the temperature to drop to 150°C when h = 25 W/m2K and density p = 7800 kg/m3.

Solution: Characteristic length, $L = VIA = 4/3\,\pi r^3/4\,\pi r^2 = r/3 = 5 \times 10^{-3}/3$m

$Bi = hL/k = 25 \times 5 \times 10\text{-}^3/(3 \times 48) = 8.68 \times 10^{-4} << 0.1$,

Since the internal resistance is negligible, we make use of lumped capacity analysis: Eq. (3.4),

$(T - T_\infty)/(T_s - T_\infty) = \exp(-Bi\ Fo)$; (150 35) / (750 35) = 0.16084

$\therefore$ $Bi \times Fo = 1827$; $Fo = 1.827/(8.68 \times 10^{-4})\ 2.1 \times 10^3$

or, $\alpha t/L^2 = k/(\rho CL^2)\ t = 2100$ and t = 568 = 0.158 hour

We can also compute the change in the internal energy of the object as:

$$U_0 - U_t = -\int_0^1 \rho CV dT = \int_0^1 \rho CV(T_s - T_\infty)(-hA/\rho CV)\exp t(-hAt/\rho CV)dt$$

$$= -\rho CV(T_s - T_\infty)\left[\exp(-hAt/\rho CV) - 1\right] \qquad (3.5)$$

$= -7800 \times 600 \times (4/3)\ \pi(5 \times 10^{-3})^3\ (750\text{-}35)\ (0.16084 - 1)$

$= 1.47 \times 10^3$ J = 1.47 kJ.

If we allow the time't' to go to infinity, we would have a situation that corresponds to steady state in the new environment. The change in internal energy will be $U_0 - U_\infty = [\rho CV(T_s - T_\infty)\exp(-\infty) - 1] = [\rho CV(T_s - T_\infty]$.

We can also compute the instantaneous heal transfer rate at any time.

or. $Q = -\rho VCdT/dt = -\rho VCd/dt\,[T_\infty + (T_s - T_\infty)\exp(-hAt/\rho CV)]$

$= hA(T_s - T_\infty)[\exp(-hAt/\rho CV))$ and for t = 60s,

$Q = 25 \times 4 \times 3.142\ (5 \times 10^{-3})^2(750\ 35)\ [\exp(-25 \times 3 \times 60/5 \times 10^{-3} \times 7800 \times 600)]$

= 4.63 W.

Example 1.20 A cylindrical steel ingot (dia 10 cm. length 30 cm, k = 40 W mK. ρ = 7600 kg/m^3, C = 600 J/kgK) is to be heated in a furnace from 50°C to 850°C. The temperature inside the furnace is 1300°C and the surface heat transfer coefficient is 100 W/m^{2}K. Calculate the time required.

Solution: Characteristic length. $L = V/A = \pi r^2 L/2\pi r(r+L) = rL/2(r+L)$

$= 5 \times 10^{-2} \times 30 \times 10^{-2}/2\ (2\ (5+30) \times 10^{-2})$

$= 2.143 \times 10^{-2}$ m.

$Bi = hL/k = 100 \times 2.143 \times 10^{-2}/40 = 0.0536 << 0.1$

$Fo = \alpha t/L^2 = (k/\rho C) \times (t/L^2)$

$= 40 \times t/\ (7600 \times 600 \times [2.143 \times 10^{-2})^2] = 191 \times 10^{-2}\ t$

and $(T - T_\infty)/(T_s - T_\infty) = \exp(-Bi\ .Fo)$

or, $(850 - 1300) / (50 - 1300) = 0.36 = \exp(-Bi\ Fo)$

$\therefore$ Bi Fo= 102

and Fo = 19.06 and $t = 19.06/(1.91 \times 10^{-2}) = 16.63$ min

(The length of the ingot is 30 cm and it must be removed from the furnace after a period of 16.63 min. therefore, the speed of the ingot would be $0.3/16.63 = 1.8 \times 10^{-2}$ m/min.)

Example 1.21 A block of aluminum (2cm × 3cm × 4cm, k = 180 W/mK, $\alpha = 10^{-4}$m^2/s) initially at 300°C is cooled in air at 30°C. Calculate the temperature of the block after 3 min. Take h = 50W/ m^{2}K.

Solution: Characteristic length, $L = [2 \times 3 \times 4 /2(2 \times 3 + 2 \times 4 + 3 \times 4)] \times 10^{-2}$

$= 4.6 \times 10^{-3}$m

$Bi = hL/k = 50 \times 4.6 \times 10^{-3}/180 = 1.278 \times 10^{-3} << 0.1$

$Fo = \alpha t/L^2 = 10^{-4} \times 180 / (4.6 \times 10^{-3})^2 = 850$

$\exp(-Bi\ Fo) = \exp(-1.278 \times 10^{-3} \times'850) = 0.337$

$(T - T_\infty)\ (T_s - T_\infty) - (T - 30)/(300 - 30) = 0.337$

$\therefore$ T= 121.1°C.

Example 1.22 A copper wire 1 mm in dia initially at 150°C is suddenly dipped into water at 35°C. Calculate the time required to cool to a temperature of 90°C if h = 100 W/ m^{2}K. What would be the time required if h = 40 W/m^{2}K. (for copper; k = 370 W/mK, ρ = 8800 kg/m^3. C = 381 J/kgK.

Solution: The characteristic length for a long cylindrical object can be approximated as r/2. As such,

$Bi = hL/k = 100 \times 0.5 \times 10^{-3} / (2 \times 370) = 6.76 \times 10^{-5} << 0.1$

$Fo = \alpha t/L^2 = (k/\rho C) \times (t/L^2)$

$= [370t/ (8800 \times 381 \times (0.25 \times 10^{-3})^2] = 1760t$

$\exp(-Bi\ Fo) = (T - T_\infty)(T_s - T_\infty)$

$= (90 - 35)/(150 - 35) = 0.478$

$Bi\ Fo = 0.738 = 6.76 \times 10^{-5} \times 1760\ t;$ $\therefore$ t = 6.2s

when h = 40 W/ m^{2}K, $Bi = 2.7 \times 10^{-5}$ and $2.7 \times 10^{-5} \times 1760\ t = 0.738$;

or, t = 15.53s.

Example 1.23 A metallic rod (mass 0.1 kg, C = 350 J/kgK, dia 12.5 mm, surface area 40cm^2) is initially at 100°C. It is cooled in air at 25°C. If the temperature drops to 40°C in 100 seconds, estimate the surface heat transfer coefficient.

Solution: $hA/\rho CV = hA/mC = h \times 40 \times 10^{-4} / (0.1 \times 350) = 1.143 \times 10^{-4} h$

and, $hAt/\rho CV = 1.143 \times 10^{-4}\ h \times 100 = 1.143 \times 10^{-2} h$

$(T - T_\infty)/ (T_s - T_\infty) = (40 - 25) / (100 - 25) = 0.2$

$\therefore$ $\exp(-1.143 \times 10^{-2} h) = 0.2$ or, $1.143 \times 10^{-2} h = 1.6094$, and h = 140W/m^{2}K.

SCHOOL OF MECHANICAL ENGINEERING

DEPARTMENT OF AERONAUTICAL ENGINEERING

UNIT – II – Heat Transfer Techniques For Aerospace Applications – SAE1306

UNIT – 2

CONVECTION

2.1. Convection Heat Transfer-Requirements

The heat transfer by convection requires a solid-fluid interface, a temperature difference between the solid surface and the surrounding fluid and a motion of the fluid. The process of heat transfer by convection would occur when there is a movement of macro-particles of the fluid in space from a region of higher temperature to lower temperature.

2.2. Convection Heat Transfer Mechanism

Let us imagine a heated solid surface, say a plane wall at a temperature T_w placed in an atmosphere at temperature T_∞, Fig. 2.1 Since all real fluids are viscous, the fluid particles adjacent to the solid surface will stick to the surface. The fluid particle at A, which is at a lower temperature, will receive heat energy from the plate by conduction. The internal energy of the particle would Increase and when the particle moves away from the solid surface (wall or plate) and collides with another fluid particle at B which is at the ambient temperature, it will transfer a part of its stored energy to B. And, the temperature of the fluid particle at B would increase. This way the heat energy is transferred from the heated plate to the surrounding fluid. Therefore the process of heat transfer by convection involves a combined action of heat conduction, energy storage and transfer of energy by mixing motion of fluid particles.

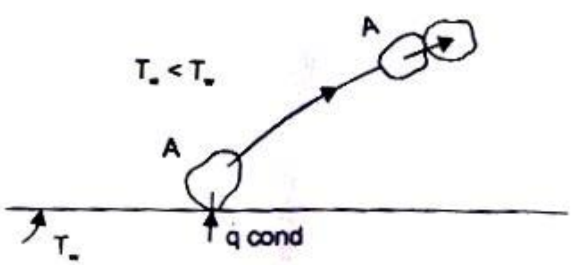


Fig. 2.1 Principle of heat transfer by convection

2.3. Free and Forced Convection

When the mixing motion of the fluid particles is the result of the density difference caused by a temperature gradient, the process of heat transfer is called natural or free convection. When the mixing motion is created by an artificial means (by some external agent), the process of heat transfer is called forced convection Since the effectiveness of heat transfer by convection depends largely on the mixing motion of the fluid particles, it is essential to have a knowledge of the characteristics of fluid flow.

2.4. Basic Difference between Laminar and Turbulent Flow

In laminar or streamline flow, the fluid particles move in layers such that each fluid p article follows a smooth and continuous path. There is no macroscopic mixing of fluid particles between successive layers, and the order is maintained even when there is a turn around a comer or an obstacle is to be crossed. If a lime dependent fluctuating motion is observed indirections which are parallel and transverse to the main flow, i.e., there is a random macroscopic mixing of fluid particles across successive layers of fluid flow, the motion of the fluid is called' turbulent flow'. The path of a fluid particle would then be zigzag and irregular, but on a statistical basis, the overall motion of the macro particles would be regular and predictable.

2.5. Formation of a Boundary Layer

When a fluid flow, over a surface, irrespective of whether the flow is laminar or turbulent, the fluid particles adjacent to the solid surface will always stick to it and their velocity at the solid surface will be zero, because of the viscosity of the fluid. Due to the shearing action of one fluid layer over the adjacent layer moving at the faster rate, there would be a velocity gradient in a direction normal to the flow.

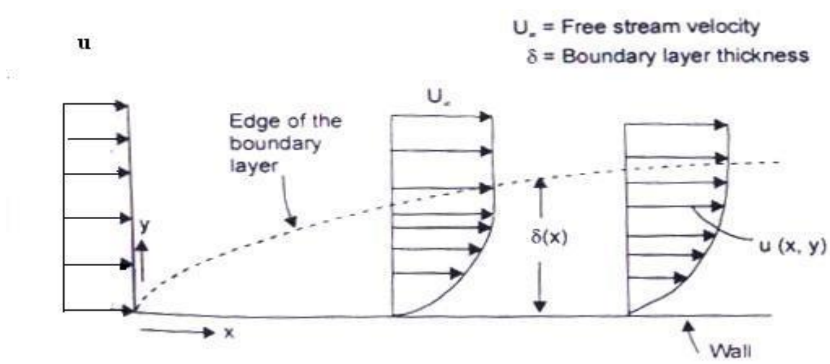


Fig 2.2: sketch of a boundary layer on a wall

Let us consider a two-dimensional flow of a real fluid about a solid (slender in cross-section) as shown in Fig. 2.2. Detailed investigations have revealed that the velocity of the fluid particles at the surface of the solid is zero. The transition from zero velocity at the surface of the solid to the free stream velocity at some distance away from the solid surface in the V-direction (normal to the direction of flow) takes place in a very thin layer called 'momentum or hydrodynamic boundary layer'. The flow field can thus be divided in two regions:

(i) A very thin layer in t he vicinity 0 f t he body w here a velocity gradient normal to the direction of flow exists, the velocity gradient du/dy being large. In this thin region, even a very small Viscosity μ of the fluid exerts a substantial Influence and the shearing stress $\tau = \mu\, du/dy$ may assume large values. The thickness of the boundary layer is very small and decreases with decreasing viscosity.

(ii) In the remaining region, no such large velocity gradients exist and the Influence of viscosity is unimportant. The flow can be considered frictionless and potential.

2.6. Thermal Boundary Layer

Since the heat transfer by convection involves the motion of fluid particles, we must superimpose the temperature field on the physical motion of fluid and the two fields are bound to interact. It is intuitively evident that the temperature distribution around a hot body in a fluid stream will often have the same character as the velocity distribution in the boundary layer flow. When a heated solid body IS placed in a fluid stream, the temperature of the fluid stream will also vary within a thin layer in the immediate neighborhood of the solid body. The variation in temperature of the fluid stream also takes place in a thin layer in the neighborhood of the body and is termed 'thermal boundary layer'. Fig. 2.3 shows the temperature profiles inside a thermal boundary layer.

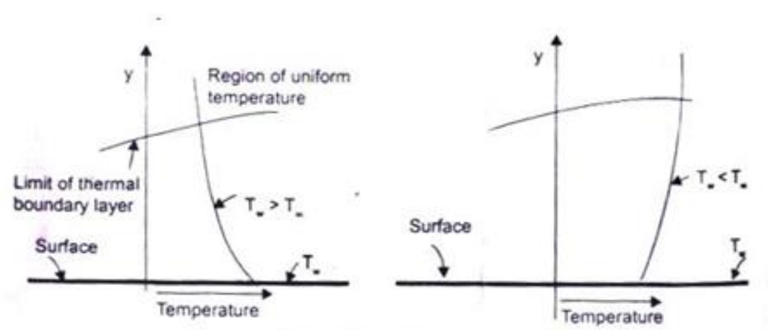


Fig2.3: The thermal boundary layer

2.7. Dimensionless Parameters and their Significance

The following dimensionless parameters are significant in evaluating the convection heat transfer coefficient:

(a) *The Nusselt Number (Nu)*-It is a dimensionless quantity defined as hL/ k, where h = convective heat transfer coefficient, L is the characteristic length and k is the thermal conductivity of the fluid The Nusselt number could be interpreted physically as the ratio of the temperature gradient in the fluid immediately in contact with the surface to a reference temperature gradient (T_s - T_∞) /L. The convective heat transfer coefficient can easily be obtained if the Nusselt number, the thermal conductivity of the fluid in that temperature range and the characteristic dimension of the object is known.

Let us consider a hot flat plate (temperature T_w) placed in a free stream (temperature T_∞ < T_w). The temperature distribution is shown ill Fig. 2.4. Newton's Law of Cooling says that the rate of heat transfer per unit area by convection is given by

$$\dot{Q}/A = h\left(T_w - T_\infty\right)$$

$$\frac{\dot{Q}}{A} = h(T_w - T_\infty)$$

$$= k\frac{T_w - T_\infty}{\delta_t}$$

$$h = \frac{k}{\delta_t}$$

$$Nu = \frac{hL}{k} = \frac{L}{\delta_t}$$

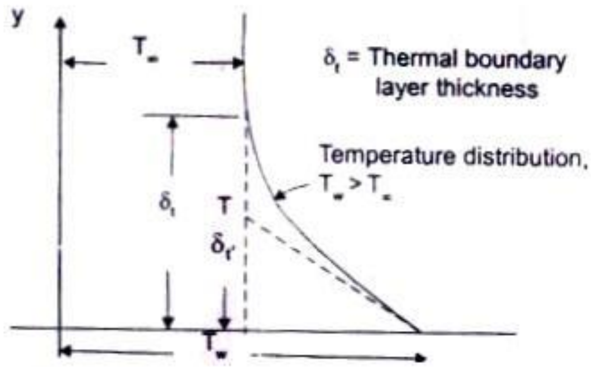


Fig. 2.4 Temperature distribution in a boundary layer: Nusselt modulus

The heat transfer by convection involves conduction and mixing motion of fluid particles. At the solid fluid interface (y = 0), the heat flows by conduction only, and is given by

$$\frac{\dot{Q}}{A} = -k\left(\frac{dT}{dy}\right)_{Y=0} \qquad \therefore h = \frac{\left(-k\,{}^{dT}\!/_{dy}\right)_{y=0}}{\left(T_w - T_\infty\right)}$$

Since the magnitude of the temperature gradient in the fluid will remain the same, irrespective of the reference temperature, we can write dT = d(T - T_w) and by introducing a characteristic length dimension L to indicate the geometry of the object from which the heat flows, we get

$$\frac{hL}{k} = \frac{\left({}^{dT}\!/_{dy}\right)_{y=0}}{\left(T_w - T_\infty\right)/L}$$, and in dimensionless form,

$$= \left(\frac{d(T_w - T)/(T_w - T_\infty)}{d\left(y/L\right)}\right)_{y=0}$$

(b) *The Grashof Number (Gr)*-In natural or free convection heat transfer, die motion of fluid particles is created due to buoyancy effects. The driving force for fluid motion is the body force arising from the temperature gradient. If a body with a constant wall temperature T_w is exposed to a qui scent ambient fluid at T_∞, the force per unit volume can be written as $\rho g \beta\left(t_w - T_\infty\right)$ where ρ = mass density of the fluid, β = volume coefficient of expansion and g is the acceleration due to gravity.

The ratio of inertia force × Buoyancy force/(viscous force)^2 can be written as

$$\mathrm{Gr} = \frac{\left(\rho V^2 L^2\right) \times \rho g \beta \left(T_w - T_\infty\right) L^3}{\left(\mu V L\right)^2}$$

$$= \frac{\rho^2 g \beta \left(T_w - T_\infty\right) L^3}{\mu^2} = g \beta L^3 \left(T_w - T_\infty\right) / \nu^2$$

The magnitude of Grashof number indicates whether the flow is laminar or turbulent. If the Grashof number is greater than 10^9, the flow is turbulent and for Grashof number less than 10^8, the flow is laminar. For $10^8 < \mathrm{Gr} < 10^9$, It is the transition range.

(c) *The Prandtl Number (Pr)* - It is a dimensionless parameter defined as

$\mathrm{Pr} = \mu C_p / k = \nu / \alpha$

Where μ is the dynamic viscosity of the fluid, v = kinematic viscosity and α = thermal diffusivity.

This number assumes significance when both momentum and energy are propagated through the system. It is a physical parameter depending upon the properties of the medium It is a measure of the relative magnitudes of momentum and thermal diffusion in the fluid: That is, for Pr = I, the r ate of diffusion of momentum and energy are equal which means that t he calculated temperature and velocity fields will be Similar, the thickness of the momentum and thermal boundary layers will be equal. For Pr <<I (in case of liquid metals), the thickness of the thermal boundary layer will be much more than the thickness of the momentum boundary layer and vice versa. The product of Grashof and Prandtl number is called Rayleigh number. Or, Ra = Gr × Pr.

2.8. Evaluation of Convective Heat Transfer Coefficient

The convective heat transfer coefficient in free or natural convection can be evaluated by two methods:

(a) Dimensional Analysis combined with experimental investigations

(b) Analytical solution of momentum and energy equations 10 the boundary layer.

Dimensional Analysis and Its Limitations

Since the evaluation of convective heat transfer coefficient is quite complex, it is based on a combination of physical analysis and experimental studies. Experimental observations become necessary to study the influence of pertinent variables on the physical phenomena.

Dimensional analysis is a mathematical technique used in reducing the number of experiments to a minimum by determining an empirical relation connecting the relevant variables and in grouping the variables together in terms of dimensionless numbers. And, the method can only be applied after the pertinent variables controlling t he phenomenon are Identified and expressed In terms of the primary dimensions. (Table 1.1)

In natural convection heat transfer, the pertinent variables are: h, ρ, k, μ, C_p, L, (ΔT), β and g. Buckingham π's method provides a systematic technique for arranging the variables in dimensionless numbers. It states that the number of dimensionless groups, π's, required to describe a phenomenon involving 'n' variables is equal to the number of variables minus the number of primary dimensions 'm' in the problem.

In SI system of units, the number of primary dimensions are 4 and the number of variables for free convection heat transfer phenomenon are 9 and therefore, we should expect (9 - 4) = 5 dimensionless numbers. Since the dimension of the coefficient of volume expansion, β, is θ^{-1}, one dimensionless number is obviously $\beta(\Delta T)$. The remaining variables are written in a functional form:

$$\phi\left(h, \rho, k, \mu, C_p, L, g\right) = 0.$$

Since the number of primary dimensions is 4, we arbitrarily choose 4 independent variables as primary variables such that all the four dimensions are represented. The selected primary variables are: ρ,g, k. L Thus the dimensionless group,

$$\pi_1 = \rho^a g^b k^c L^d h = \left(ML^{-3}\right)^a \left(LT^{-2}\right)^b . \left(MLT^{-3}\theta^{-1}\right) = M^0 L^0 T^0 \theta^0$$

Equating the powers of M, L, T, θ on both sides, we have

$$\left.\begin{array}{l} M : a + c + 1 = 0 \text{ \} Upon solving them,} \\ L : -3a + b + c + d = 0 \\ T : -2b - 3c - 3 = 0 \\ \theta : -c - 1 = 0 \end{array}\right\} \text{Up on solving them,}$$

c = 1, b = a = 0 and d = 1.

and π_1 = hL/k, the Nusselt number.

The other dimensionless number

$\pi_2 = p^a g^b k^c L^d C_p = (ML^{-3})^a (LT^{-2})^b (MLT^{-3}\theta^{-1})^c (L)^d (MT^{-1}\theta^{-1}) = M^0 L^0 T^0 \theta^0$ Equating the powers of M,L,T and θ and upon solving, we get

$\pi_3 = \mu^2 / \rho^2 g L^3$

By combining π_2 and π_3, we write $\pi_4 = \left[\pi_2 \times \pi_3\right]^{1/2}$

$$= \left[\rho^2 g L^3 C_p^2 / k^2 \times \mu^2 / g L^3\right]^{1/2} = \frac{\mu C_p}{k} \text{ (the Prandtl number)}$$

By combining π_3 with $(\beta \Delta T)$, we have $\pi_5 = (\beta \Delta T) * \frac{1}{\pi_3}$

$$= \beta(\Delta T) \times \frac{\rho^2 g L^3}{\mu^2} = g\beta(\Delta T) L^3 / \nu^2 \text{ (the Grashof number)}$$

Therefore, the functional relationship is expressed as:

$$\phi(Nu, Pr, Gr) = 0; Or, Nu = \phi_1(Gr\,Pr) = Const \times (Gr \times Pr)^m \qquad (2.1)$$

and values of the constant and 'm' are determined experimentally.

Table 2.1 gives the values of constants for use with Eq. (2.1) for isothermal surfaces.

Table 2.1 Constants for use with Eq. 2.1 for Isothermal Surfaces

Geometry	$G_{r\,f}\ p_{r\,f}$	*C*	*m*
Vertical planes and cylinders	10^4 - 10^9	0.59	1/4
	10^9 - 10^{13}	0.021	2/5
	10^9 - 10^{13}	0.10	1/3
Horizontal cylinders	0 - 10^{-5}	0.4	0
	10^4 - 10^9	0.53	1/4
	10^9 - 10^{12}	0.13	1/3
	10^{10} - 10^{-2}	0.675	0.058
	10^{-2} - 10^2	1.02	0.148
	10^2 - 10^4	0.85	0.188
	10^4 - 10^7	0.48	1/4
	10^7 - 10^{12}	0.125	1/3
Upper surface of heated plates or lower surface of cooled plates	8×10^6 - 10^{11}	0.15	1/3
- do -	2×10^4 - 8×10^6	0.54	1/4
Lower surface of heated plates or upper surface of cooled plates	10^5 - 10^{11}	0.27	1/4
Vertical cylinder height = diameter characteristic length = diameter	10^4 - 10^6	0.775	0.21
Irregular solids, characteristic length = distance the fluid particle travels in boundary layer	10^4 - 10^9	0.52	1/4

__Analytical Solution-Flow over a Heated Vertical Plate in Air__

Let us consider a heated vertical plate in air, shown in Fig. 2.5. The plate is maintained at uniform temperature T_w .The coordinates are chosen in such a way that x - is in the stream wise direction and y - is in the transverse direction. There will be a thin layer of fluid adjacent to the hot surface of the vertical plate within

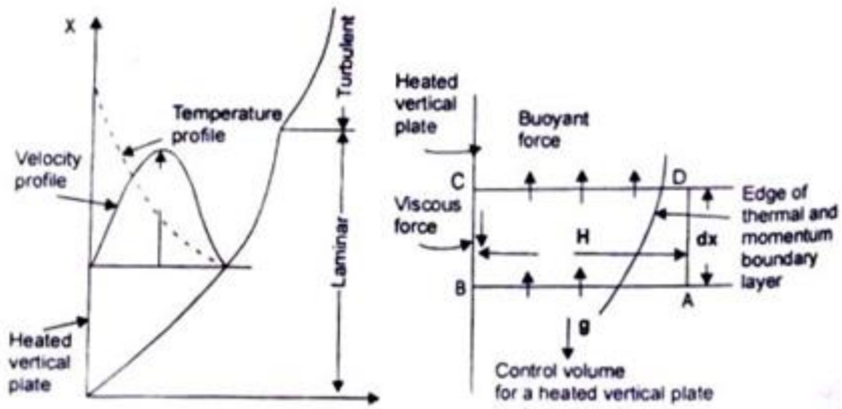


Fig. 2.5 Boundary layer on a heated vertical plate

Which the variations in velocity and temperature would remain confined. The relative thickness of the momentum and the thermal boundary layer strongly depends upon the Prandtl number. Since in natural convection heat transfer, the motion of the fluid particles is caused by the temperature difference between the temperatures of the wall and the ambient fluid, the thickness of the two boundary layers are expected to be equal. When the temperature of the vertical plate is less than the fluid temperature, the boundary layer will form from top to bottom but the mathematical analysis will remain the same.

The boundary layer will remain laminar upto a certain length of the plate ($Gr < 10^8$) and beyond which it will become turbulent ($Gr > 10^9$). In order to obtain the analytical solution, the integral approach, suggested by von-Karman is preferred.

We choose a control volume ABCD, having a height H, length dx and unit thickness normal to the plane of paper, as shown in Fig. 25. We have:

(b) Conservation of Mass:

Mass of fluid entering through face AB = $\dot{m}_{AB} = \int_0^H \rho u dy$

Mass of fluid leaving face CD = $\dot{m}_{CD} = \int_0^H \rho u dy + \frac{d}{dx}\left[\int_0^H \rho u dy\right]dx$

$\therefore$ Mass of fluid entering the face DA = $\frac{d}{dx}\left[\int_0^H \rho u dy\right]dx$

(ii) Conservation of Momentum:

Momentum entering face AB = $\int_0^H \rho u^2 dy$

Momentum leaving face CD = $\int_0^H \rho u^2 dy + \frac{d}{dx}\left[\int_0^H \rho u^2 dy\right]dx$

$\therefore$ Net efflux of momentum in the + x-direction = $\frac{d}{dx}\left[\int_0^H \rho u^2 dy\right]dx$

The external forces acting on the control volume are:

(a) Viscous force = $\mu \left.\frac{du}{dy}\right|_{y=0} dx$ acting in the –ve x-direction

(b) Buoyant force approximated as $\left[\int_0^H \rho g \beta (T - T_\infty) dy\right]dx$

From Newton's law, the equation of motion can be written as:

$$\frac{d}{dx}\left[\int_0^\delta \rho u^2 dy\right] = -\mu \left.\frac{du}{dy}\right|_{y=0} + \int_0^\delta \rho g \beta (T - T_\infty) dy \qquad (2.2)$$

Because the value of the integrand between δ and H would be zero.

(iii) Conservation of Energy:

$\dot{Q}_{AB}$, convection + $\dot{Q}_{AD}$, convection + $\dot{Q}_{BC}$, conduction = $\dot{Q}_{CD}$ convection

or, $\int_0^H \rho u C T dy + C T_\infty \left[\frac{d}{dx}\int_0^H \rho u dy\right]dx - k \left.\frac{dT}{dy}\right|_{y=0} dx$

$= \int_0^H \rho u C T dy + \frac{d}{dx}\left[\int_0^H \rho u T C dy\right]dx$

or $\frac{d}{dx}\int_0^{\delta}\rho u\left(T_{\infty}-T\right)dy\,\frac{k}{\rho C}\frac{dT}{dy}\bigg|_{y=0}=\alpha\frac{dT}{dy}\bigg|_{y=0}$ (2.3)

The boundary conditions are:

or,

(2.3)

Velocity profile	Temperature profile
$u = 0$ at $y = 0$	$T = T_w$ at $y = 0$
$u = 0$ at $y = \delta$	$T = T_{\infty}$ at $y = \delta_1 \equiv \delta$
$du/dy = 0$ at $y = \delta$	$dT/dy \equiv 0$ at $y = \delta_1 \equiv \delta$

Since the equations (2.2) and (2.3) are coupled equations, it is essential that the functional form of both the velocity and temperature distribution are known in order to arrive at a solution.

The functional relationships for velocity and temperature profiles which satisfy the above boundary conditions are assumed of the form:

$$\frac{u}{u_*}=\frac{y}{\delta}\left(1-\frac{y}{\delta}\right)^2 \qquad (2.4)$$

Where u_* is a fictitious velocity which is a function of x; and

$$\frac{\left(T-T_{\infty}\right)}{\left(T_w-T_{\infty}\right)}=\left(1-\frac{y}{\delta}\right)^2 \qquad (2.5)$$

After the Eqs. (5.4) and (5.5) are inserted in Eqs. (5.2) and (5.3) and the operations are performed (details of the solution are given in Chapman, A.J. Heat Transfer, Macmillan Company, New York), we get the expression for boundary layer thickness as:

$$\delta / x = 3.93\,Pr^{-0.5}\left(0.952+Pr\right)^{0.25}Gr_x^{-0.25}$$

Where Gr, is the local Grashof number = $g\beta x^3\left(T_w-T_{\infty}\right)/\nu^2$

The heat transfer coefficient can be evaluated from:

$$\dot{q}_w = -k\frac{dT}{dy}\bigg|_{y=0} = h\left(T_w - T_\infty\right)$$

Using Eq. (5.5) which gives the temperature distribution, we have

$h = 2k/\delta$ or, $hx/k = Nu_x = 2x/\delta$

The non-dimensional equation for the heat transfer coefficient is

$$Nu_x = 0.508\ Pr^{0.5}\ (0.952 + Pr)^{-0.25}\ Gr_x^{0.25} \qquad (2.7)$$

The average heat transfer coefficient, $\bar{h} = \frac{1}{L}\int_0^L h_x dx = 4/3h_{x=L}$

$$Nu_L = 0.677\ Pr^{0.5}\ (0.952 + Pr)^{-0.25}\ Gr^{0.25} \qquad (2.8)$$

Limitations of Analytical Solution: Except for the analytical solution for flow over a flat plate, experimental measurements are required to evaluate the heat transfer coefficient. Since in free convection systems, the velocity at the surface of the wall and at the edge of the boundary layer is zero and its magnitude within the boundary layer is so small. It is very difficult to measure them. Therefore, velocity measurements require hydrogen-bubble technique or sensitive hot wire anemometers. The temperature field measurement is obtained by interferometer.

Expression for 'h' for a Heated Vertical Cylinder in Air

The characteristic length used in evaluating the Nusselt number and Grashof number for vertical surfaces is the height of the surface. If the boundary layer thickness is not to large compared with the diameter of the cylinder, the convective heat transfer coefficient can be evaluated by the equation used for vertical plane surfaces. That is, when $D/L \geq 25/\left(Gr_L\right)^{0.25}$

Example 2.1 A large vertical flat plate 3 m high and 2 m wide is maintained at 75°C and is exposed to atmosphere at 25°C. Calculate the rate of heat transfer.

Solution: The physical properties of air are evaluated at the mean temperature. i.e. T = (75 + 25)/2 = 50°C

From the data book, the values are:

$\rho = 1.088$ kg/m^3; $C_p = 1.00$ kJ/kg.K;

$\mu = 1.96 \times 10^{-5}$ Pa-s k = 0.028 W/mK.

Pr = $\mu C_p/k = 1.96 \times 10^{-5} \times 1.0 \times 10^3 /0.028 = 0.7$

$$\beta = \frac{1}{T} = \frac{1}{323}$$

Gr = $\rho^2 g\beta(\Delta T)L^3/\mu^2$

$$= \frac{(1.088)^2 \times 9.81 \times 1 \times (3)^3 \times 50}{323 \times (1.96 \times 10^{-5})^2}$$

$= 12.62 \times 10^{10}$

Gr.Pr = 8.834×10^{10}

Since Gr.Pr lies between 10^9 and 10^{13}

We have from Table 2.1

$$\text{Nu} = \frac{hL}{k} = 0.1(\text{Gr.Pr})^{1/3} = 441.64$$

$\therefore$ h = 441.64 × 0.028/3 = 4.122 W/m^{2}K

$$\dot{Q} = hA(\Delta T) = 4.122 \times 6 \times 50 = 1236.6\text{W}$$

We can also compute the boundary layer thickness at x = 3m

$$\delta = \frac{2x}{Nu_x} = \frac{2 \times 3}{441.64} = 1.4 \text{ cm}$$

Example 2.2 A vertical flat plate at 90°C. 0.6 m long and 0.3 m wide, rests in air at 30°C. Estimate the rate of heat transfer from the plate. If the plate is immersed in water at 30°C. Calculate the rate of heat transfer

Solution: (a) *Plate in Air*: Properties of air at mean temperature 60°C

Pr = 0.7, k = 0.02864 W/ mK, v = 19.036×10^{-6} m^2/s

Gr = $9.81 \times (90 - 30)(0.6)^3/ [333\ (19.036 \times 10^{-6})^2]$

$= 1.054 \times 10^9$; Gr × Pr $1.054 \times 10^9 \times 0.7 = 7.37 \times 10^8 < 10^9$

From Table 5.1: for Gr × Pr $< 10^9$, Nu = 0.59 (Gr. Pr)$^{1/4}$

$\therefore$ h = 0.02864 × 0.59 $(7.37 \times 10^8)^{1/4}/0.6 = 4.64$ W/m^{2}K

The boundary layer thickness, $\delta = 2\ k/h = 2 \times 0.02864/4.64 = 1.23$ cm

and $\dot{Q} = hA(\Delta T) = 4.64 \times (2 \times 0.6 \times 0.3) \times 60 = 100$ W.

Using Eq (2.8). Nu = $0.677\ (0.7)^{0.5}\ (0.952 + 0.7)^{0.25}\ (1.054 \times 10^9)^{0.25}$,

Which gives h = 4.297 W/m^{2}K and heat transfer rate, $\dot{Q}$ 92.81 W

Churchill and Chu have demonstrated that the following relations fit very well with experimental data for all Prandtl numbers.

For $Ra_L < 10^9$, $Nu = 0.68 + (0.67\ Ra_L^{0.25})/\ [1 + (0.492/Pr)^{9/16}]^{4/9)}$ (5.9)

$Ra_L > 10^9$, $Nu = 0.825 + (0.387\ Ra_L^{1/6})/[1 + (0.492/Pr)^{9/16}]^{8/27}$ (5.10)

Using Eq (5.9): Nu = $0.68 + [0.67(7.37 \times 10^8)^{0.25}]\ /\ [1 + (0.492/0.7)^{9/16}]^{4/9}$

= 58.277 and h = 4.07 W /m^{2}k; $\dot{Q}$ = 87.9 W

(b) Plate in Water: Properties of water from the Table

Pr = 3.01, $\rho^2 g \beta C_p/\mu k = 6.48 \times 10^{10}$;

Gr.Pr = $\rho^2 g \beta C_p L^3(\Delta T)/\ \mu k = 6.48 \times 10^{10} \times (0.6)^3 \times 60 = 8.4 \times 10^{11}$

Using Eq (5.10): Nu = $0.825 + [0.387\ (8.4 \times 10^{11})^{1/6}]/\ [1 + (0.492/3.01)^{9/16})]^{8/27} = 89.48$ which gives h = 97.533 and Q = 2.107 kW.

2.9. Modified Grashof Number

When a surface is being heated by an external source like solar radiation incident on a wall, a surface heated by an electric heater or a wall near a furnace, there is a uniform heat flux distribution along the surface. The wall surface will not be an isothermal one. Extensive experiments have been performed by many research workers for free convection from vertical and inclined surfaces to water under constant heat flux conditions. Since the temperature difference (ΔT) is not known beforehand, the Grashof number is modified by multiplying it by

Nusselt number. That is,

$$Gr_x^* = Gr_x . Nu_x = (g\ \beta\ \Delta T / \nu^2) \times (hx/k) = g\ \beta\ x^4\ q/k\,\nu^2 \qquad (2.11)$$

Where q is the wall heat flux in Wm^2. ($q = h\,(\Delta T)$)

It has been observed that the boundary layer remains lam mar when the modified Rayleigh number, $Ra^* = Gr_x^{*}$'. Pr is less than 3×10^{12} and fully turbulent flow appears for $Ra^* > 10^{14}$. The local heat transfer coefficient can be calculated from:

$$q \text{ constant and } 10^5 < Gr_x^* < 10^{11}: Nu_x = 0.60\,(Gr_x^* . Pr)^{0.2} \qquad (2.12)$$

$$q \text{ Constant and } 2 \times 10^{13} < Gr_x^* < 10^{16} : Nu_x = 0.17\,(Gr_x^* . Pr)^{0.25} \qquad (2.13)$$

Although these results are based on experiments for water, they are applicable to air as well. The physical properties are to be evaluated at the local film temperature.

Example 2.3 Solar radiation of intensity 700W/m' is incident on a vertical wall, 3 m high and 3 m wide. Assuming that the wall does not transfer energy to the inside surface and all the incident energy is lost by free convection to the ambient air at 30oe, calculate the average temperature of the wall

Solution: Since the surface temperature of the wall is not known, we assume a value for $h = 7\ W/m^2\,K$.

$\Delta T = \dot{q}\ / h = 700/7 = 100^oC$ and the film temperature = $(30 + 130)/2 = 80^oC$

The properties of air at 273 +80 = 353 are: $\beta = 1/353$, Pr = 0.697

$k = 0.03\ W/mK$, $\nu = 20.76 \times 10^{-6}\ m^2/s$.

Modified Grashof number, $Gr_x^* = 9.81.\ (1/353) \cdot (3)^4 \times 700/[0.03 \times (20.76 \times 10^{-6})^2] = 1.15 \times 10^{14}$

From Eq. (2.13), $h = (k/x)\,(0.17)\,(Gr_x^*\ Pr)^{0.25}$

$= (0.03/3)\,(0.17)\,(1.15 \times 10^{14} \times 0.697)^{1/4}$

$= 5.087\ W/m^2K$, a different value from the assumed value.

Second Trial: $\Delta T = \dot{q}\ / h = 700/5.087 = 137.66$ and film temperature

= 98.8°C

The properties of air at (273 + 98.8) °C are: β =1/372, k = 0.0318 W/mK

Pr = 0.693, v = 23.3 × 10^{-6} m^2/s

$Gr_x^* = 9.81.\ (1/372)\cdot (3)^4 \times 700/\ [0.318(23.3 \times 10^{-6})^2] = 8.6 \times 10^{13}$

Using Eq (2.13), h = (k/x) (0.17) (Gr_x^* Pr)$^{1/4}$ = 5.015 W/m^2k, an acceptable value. In turbulent heat transfer by convection, Eq. (5.13) tells us that the local heat transfer coefficient h_x does not vary with x and therefore, the average and local heat transfer coefficients are the same.

2.10 Laminar Flow Forced Convection Heat Transfer

2.10.1Forced Convection Heat Transfer Principles

The mechanism of heat transfer by convection requires mixing of one portion of fluid with another portion due to gross movement of the mass of the fluid. The transfer of heat energy from one fluid particle or a molecule to another one is still by conduction but the energy is transported from one point in space to another by the displacement of fluid.

When the motion of fluid is created by the imposition of external forces in the form of pressure differences, the process of heat transfer is called 'forced convection'. And, the motion of fluid particles may be either laminar or turbulent and that depends upon the relative magnitude of inertia and viscous forces, determined by the dimensionless parameter Reynolds number. In free convection, the velocity of fluid particle is very small in comparison with the velocity of fluid particles in forced convection, whether laminar or turbulent. In forced convection heat transfer, $Gr/Re^2 << 1$, in free convection heat transfer, $GrRe^2 >> 1$ and we have combined free and forced convection when $Gr/Re^2 \approx 1$.

2.10.2. Methods for Determining Heat Transfer Coefficient

The convective heat transfer coefficient in forced flow can be evaluated by: (a) Dimensional Analysis combined with experiments;

(b) Reynolds Analogy – an analogy between heat and momentum transfer; (c) Analytical Methods – exact and approximate analyses of boundary layer equations.

2.10.3. Method of Dimensional Analysis

As pointed out in Chapter 5, dimensional analysis does not yield equations which can be solved. It simply combines the pertinent variables into non-dimensional numbers which facilitate the interpretation and extend the range of application of experimental data. The relevant variables for forced convection heat transfer phenomenon whether laminar or turbulent, are

(b) The properties of the fluid – density p, specific heat capacity C_p, dynamic or absolute viscosity μ, thermal conductivity k.

(ii) The properties of flow – flow velocity Y, and the characteristic dimension of the system L.

As such, the convective heat transfer coefficient, h, is written as h = f (ρ, V, L, μ, Cp, k) = 0 (5.14)

Since there are seven variables and four primary dimensions, we would expect three dimensionless numbers. As before, we choose four independent or core variables as ρ ,V, L, k, and calculate the dimensionless numbers by applying Buckingham π's method:

$$\pi_1 = \rho^a V^b L^c K^d h = \left(ML^{-3}\right)^a \left(LT^{-1}\right)^b \left(L\right)^c \left(MLT^{-3}\theta^{-1}\right)^d \left(MT^{-3}\theta^{-1}\right)$$

$$= M^o L^o T^o \theta^o$$

Equating the powers of M, L, T and θ on both sides, we get

M : a + d + 1 = O

L : - 3a + b + c + d = 0

T : - b – 3d – 3 = 0 By solving them, we have

θ : - d – 1= 0. D = -1, a = 0, b = 0, c = 1.

Therefore, π_1 = hL/k is the Nusselt number.

$$\pi_2 = \rho^a V^b L^c K^d \mu = \left(ML^{-3}\right)^a \left(LT^{-1}\right)^b \left(L\right)^c \left(MLT^{-3}\theta^{-1}\right)^d \left(ML^{-1}T^{-1}\right)$$

$$= M^o L^o T^o \theta^o$$

Equating the powers of M, L, T and on both sides, we get

M : a + d +1 = 0

L : - 3a + b + c + d = 1 = 0

T : - b – 3d – 1 = 0

θ : - d = 0.

By solving them, d = 0, b = - 1, a = - 1, c = - 1

and $\pi_2 = \mu / \rho VL; \text{or}, \pi_3 = \dfrac{1}{\pi_2} = \dfrac{\rho VL}{\mu}$

(Reynolds number is a flow parameter of greatest significance. It is the ratio of inertia forces to viscous forces and is of prime importance to ascertain the conditions under which a flow is laminar or turbulent. It also compares one flow with another provided the corresponding length and velocities are comparable in two flows. There would be a similarity in flow between two flows when the Reynolds numbers are equal and the geometrical similarities are taken into consideration.)

$$\pi_4 = \rho^a V^b L^c k^d C_p = \left(ML^{-3}\right)^a \left(LT^{-1}\right)^b (L)^c \left(MLT^{-3}\theta^{-1}\right)^d \left(L^2T^{-2}\theta^{-1}\right)$$

$M^o L^o T^o \theta^o$

Equating the powers of M, L, T, on both Sides, we get

M : a + d = 0; L : - 3a + b + c + d + 2 = 0

T : - b – 3d – 2 = 0; θ : - d – 1 = 0

By solving them,

d =- 1,a = l, b = l, c = l,

$$\pi_4 = \frac{\rho VL}{k} C_p; \quad \pi_5 = \pi_4 \times \pi_2$$

$$= \frac{\rho VL}{k} C_p \times \frac{\mu}{\rho VL} = \frac{\mu C_p}{k}$$

$\therefore$ π_5 is Prandtl number.

Therefore, the functional relationship is expressed as:

Nu = f (Re, Pr); or Nu = C Re^m Pr^n (5.15)

Where the values of c, m and n are determined experimentally.

2.10.4. Principles of Reynolds Analogy

Reynolds was the first person to observe that there exists a similarity between the exchange of momentum and the exchange of heat energy in laminar motion and for that reason it has been termed 'Reynolds analogy'. Let us consider the motion of a fluid where the fluid is flowing over a plane wall. The X-coordinate is measured parallel to the surface and the V-coordinate is measured normal to it. Since all fluids are real and viscous, there would be a thin layer, called momentum boundary layer, in the vicinity of the wall where a velocity gradient normal to the direction of flow exists. When the temperature of the surface of the wall is different than the temperature of the fluid stream, there would also be a thin layer, called thermal boundary layer, where there is a variation in temperature normal to the direction of flow. Fig. 2.6 depicts the velocity distribution and temperature profile for the laminar motion of the fluid flowing past a plane wall.

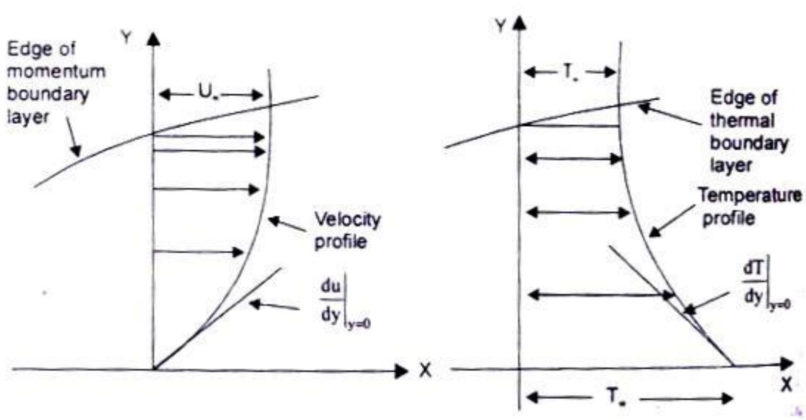


Fig. 2.6 velocity distribution and temperature profile for laminar motion of the fluid over a plane surface

In a two-dimensional flow, the shearing stress is given by $\tau_w = \mu \left.\frac{du}{dy}\right|_{y=0}$

and the rate of heat transfer per unit area is given by $\frac{\dot{Q}}{A} = \frac{\tau_w k}{\mu} \frac{dT}{du}$

For Pr = $\mu C_p/k = 1$, we have $k/\mu = C_p$ and therefore, we can write after separating the variables,

$$\frac{\dot{Q}}{A\tau_w C_p} du = -dT \qquad (5.16)$$

Assuming that Q and τ_w are constant at any station x, we integrate equation (5.16) between the limits: u = 0 when T = T_w, and u = U_∞ when T = T_∞, and we get,

$$\dot{Q}/\left(A\tau_w C_p\right) \times U_\infty = \left(T_w - T_\infty\right)$$

Since by definition, $\dot{Q}/A = h_x\left(T_w - T_\infty\right)$, and $\tau_w = C_{fx} \times \rho U_\infty{}^2/2$,

Where C_{fx}, is the skin friction coefficient at the station x. We have

$$C_{fx}/2 = h_x/\left(C_p \rho U_\infty\right) \qquad (5.17)$$

Since $h_x/C_p\rho U_\infty = \left(h_x.x/k\right) \times \left(\mu/\rho \times U_\infty\right) \times \left(k/\mu.C_p\right) = Nu_x/\left(Re.Pr\right)$,

$$Nu_x/Re.Pr = C_{fx}/2 = \text{Stan tonnumer}, St. \qquad (5.18)$$

Equation (5.18) is satisfactory for gases in which Pr is approximately equal to unity. Colburn has shown that Eq. (5.18) can also be used for fluids having Prandtl numbers ranging from 0.6 to about 50 if it is modified in accordance with experirnental results.

$$\text{Or, } \frac{Nu_x}{Re_x Pr}.Pr^{2/3} = St_x Pr^{2/3} = C_{fx}/2 \qquad (5.19)$$

Eq. (5.19) expresses the relation between fluid friction and heat transfer for laminar flow over a plane wall. The heat transfer coefficient could thus be determined by making measurements of the frictional drag on a plate under conditions in which no heat transfer is involved.

Example 2.4 Glycerine at 35°C flows over a 30cm by 3Ocm flat plate at a velocity of 1.25 m/s. The drag force is measured as 9.8 N (both Side of the plate). Calculate the heat transfer for such a flow system.

Solution: From tables, the properties of glycerine at 35°C are:

ρ = 1256 kg/m^3, C_p = 2.5 kJ/kgK, μ = 0.28 kg/m-s, k = 0.286 W/mK, Pr = 2.4 Re= $\rho VL/\mu$ = 1256 × 1.25 × 0.30/0.28 = 1682.14, a laminar flow.*

Average shear stress on one side of the plate = drag force/area

= 9.8/(2 × 0.3 × 0.3) = 54.4

and shear stress = $C_f \rho U^2/2$

$\therefore$ The average skin friction coefficient, Cr/ 2 = $\dfrac{\tau}{\rho U^2}$

= 54.4/(1256 ×1.25 × 1.25) = 0.0277

From Reynolds analogy, $C_f/2 = St.\ Pr^{2/3}$

or, $h = \rho\ C_p\ U \times C_f/2 \times Pr^{-2/3} = \dfrac{1256 \times 2.5 \times 1.25 \times 0.0277}{(2.45)^{0.667}}$ =59.8 kW/m^{2}K.

2.10.5. Analytical Evaluation of ‘h’ for Laminar Flow over a Flat Plat – Assumptions

As pointed out earlier, when the motion of the fluid is caused by the imposition of external forces, such as pressure differences, and the fluid flows over a solid surface, at a temperature different from the temperature of the fluid, the mechanism of heat transfer is called ‘forced convection’. Therefore, any analytical approach to determine the convective heat transfer coefficient would require the temperature distribution in the flow field surrounding the body. That is, the theoretical analysis would require the use of the equation of motion of the viscous fluid flowing over the body along with the application of the principles of conservation of mass and energy in order to relate the heat energy that is convected away by the fluid from the solid surface.

For the sake of simplicity, we will consider the motion of the fluid in 2 space dimension, and a steady flow. Further, the fluid properties like viscosity, density, specific heat, etc are constant in the flow field, the viscous shear forces m the Y –direction is negligible and there are no variations in pressure also in the Y –direction.

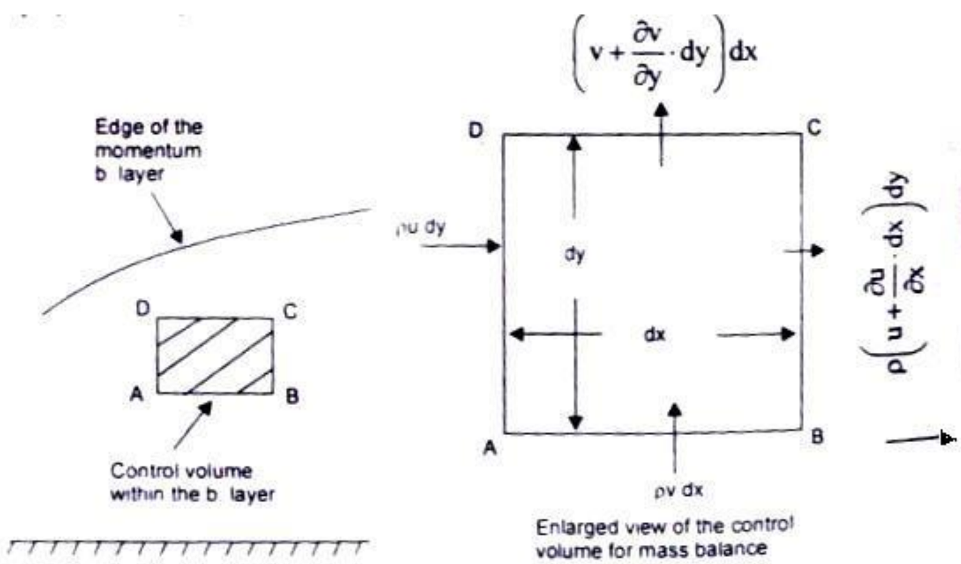
2.10.6. Derivation of the Equation of Continuity–Conservation of Mass

We choose a control volume within the laminar boundary layer as shown in Fig. 6.2. The mass will enter the control volume from the left and bottom face and will leave the control volume from the right and top face. As such, for unit depth in the Z-direction,

$$\dot{m}_{AD} = \rho u dy \; ; \quad \dot{m}_{BC} = \rho\left(u + \frac{\partial u}{dx}.dx\right)dy;$$

$$\dot{m}_{AB} = \rho v dx \; ; \quad \dot{m}_{CD} = \rho\left(v + \frac{\partial u}{dy}.dy\right)dx;$$

For steady flow conditions, the net efflux of mass from the control volume is zero, therefore,



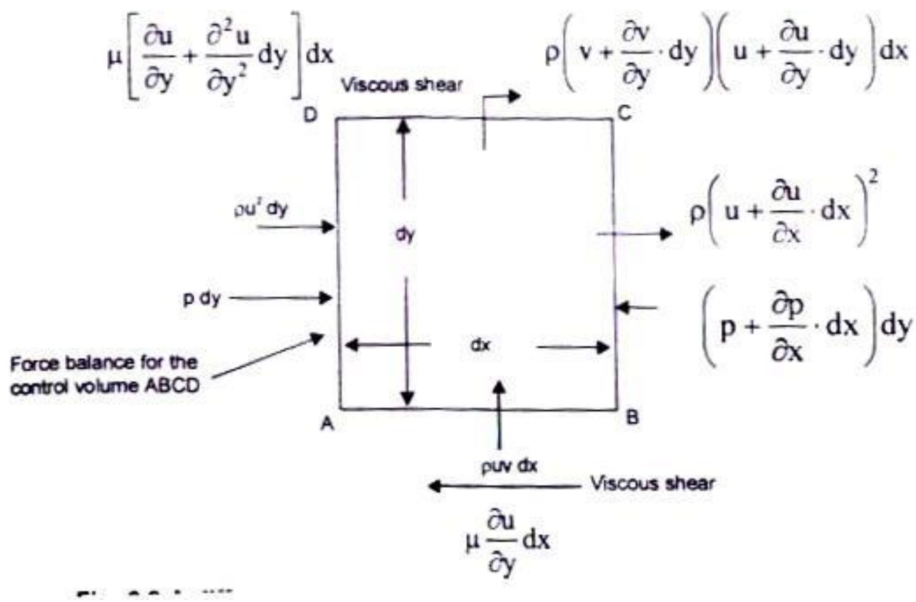


Fig. 2.7 a differential control volume within the boundary layer for laminar flow over a plane wall

$$\rho udy + \rho xdx = \rho udy + \rho\frac{\partial u}{\partial x}dxdy + \rho vdx + \rho\frac{\partial v}{\partial x}.dxdy$$

or, $\partial u/\partial x + \partial v + \partial y = 0,$ the equation of continuity. (2.20)

Concept of Critical Thickness of Insulation

The addition of insulation at the outside surface of small pipes may not reduce the rate of heat transfer. When an insulation is added on the outer surface of a bare pipe, its outer radius, r_0 increases and this increases the thermal resistance due to conduction logarithmically whereas t he thermal resistance to heat flow due to fluid film on the outer surface decreases linearly with increasing radius, r_0. Since the total thermal resistance is proportional to the sum of these two resistances, the rate of heat flow may not decrease as insulation is added to the bare pipe.

Fig. 2.7 shows a plot of heat loss against the insulation radius for two different cases. For small pipes or wires, the radius r_l may be less than re and in that case, addition of insulation to the bare pipe will increase the heat loss until the critical radius is reached. Further addition of insulation will decrease the heat loss rate from this peak value. The insulation thickness ($r^* - r_l$) must be added to reduce the heat loss below the uninsulated rate. If the outer pipe radius r_l is greater than the critical radius re any insulation added will decrease the heat loss.

2.10.7 Expression for Critical Thickness of Insulation for a Cylindrical Pipe

Let us consider a pipe, outer radius r_I as shown in Fig. 2.18. An insulation is added such that the outermost radius is r a variable and the insulation thickness is $(r - r_I)$. We assume that the thermal conductivity, k, for the insulating material is very small in comparison with the thermal conductivity of the pipe material and as such the temperature T_1, at the inside surface of the insulation is constant. It is further assumed that both h and k are constant. The rate of heat flow, per unit length of pipe, through the insulation is then,

$\dot{Q}/L = 2\pi(T_1 - T_\infty)/(\ln(r/r_1)/k + 1/hr)$, where T_∞ is the ambient temperature.

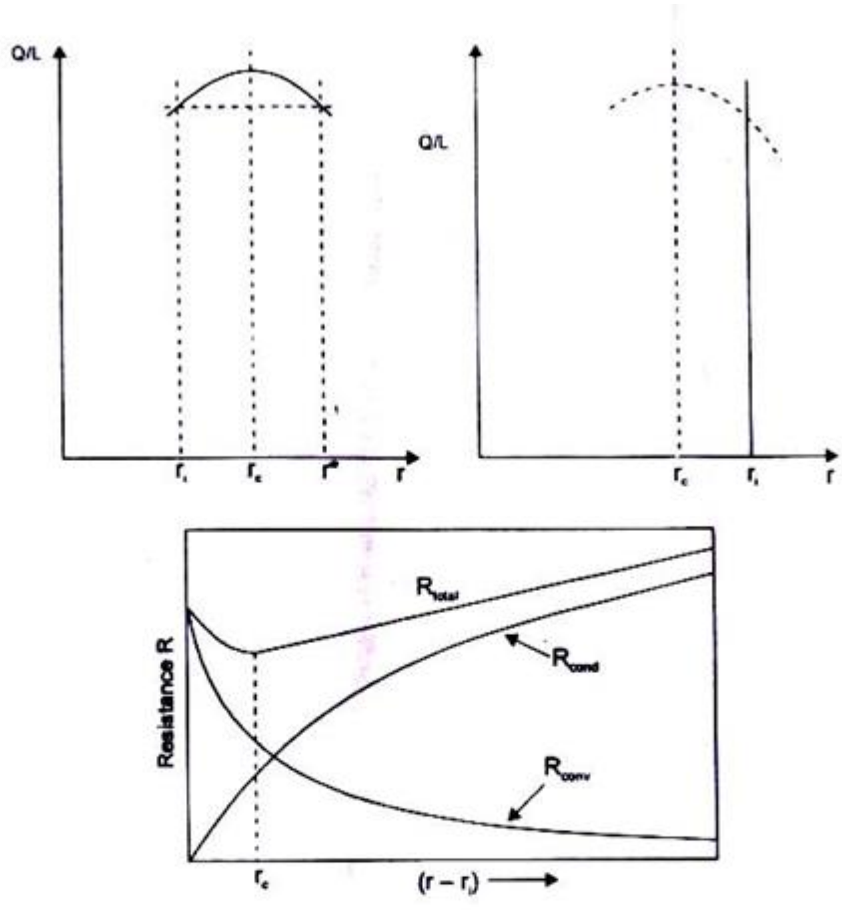


Fig 2.8 Critical thickness for pipe insulation

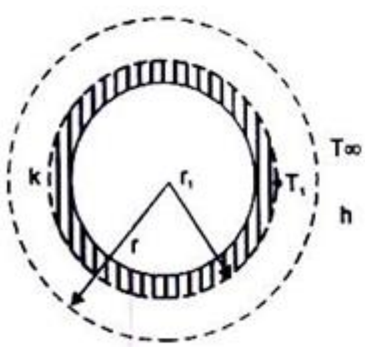


Fig 2.9 critical thickness of insulation for a pipe

An optimum value of the heat loss is found by setting $\frac{d\left(\dot{Q}/L\right)}{dr}=0$.

or, $\frac{d\left(\dot{Q}/L\right)}{dr}=0=-\frac{2\pi\left(T_1-T_\infty\right)\left(1/kr-1/hr^2\right)}{\left(\ln\left(r/r_1\right)/k+1/hr^2\right)}$

or, $(1/kr)-\left(1/hr^2\right)=0$ and $r = r_c = k/h$ (2.21)

where r_c denote the 'critical radius' and depends only on thermal quantities k and h.

If we evaluate the second derivative of (Q/L) at $r = r_c$, we get

$$\left.\frac{d^2(Q/L)}{dr^2}\right|_{r=r_c}=-2\pi\left(T_1-T_\infty\right)\left[\frac{\frac{k}{hr}+\text{In}\left(\frac{r}{r_1}\right)\left(\frac{2k}{hr}-1\right)-2\left(1-\frac{k}{hr}\right)^2}{\frac{1}{kr}\left(\frac{k}{h}+r\,\text{In}\left(\frac{r}{r_1}\right)\right)}\right]_{r=r_c}$$

$$= -\left[2\pi\left(T_1-T_\infty\right)h^2/k\right]/\left[1+1\text{n } r_c/r_1\right]^2$$

Which is always a negative quantity. Thus, the optimum radius, $r_c = k/h$ will always give a maximum heal loss and not a minimum.

2.10.8. An Expression for the Critical Thickness of Insulation for a Spherical Shell

Let us consider a spherical shell having an outer radius r_1 and the temperature at that surface T_1 . Insulation is added such that the outermost radius of the shell is r, a variable. The thermal conductivity of the insulating material, k, and the convective heat transfer coefficient at the outer surface, h, and the ambient temperature T_∞ is constant. The rate of heat transfer through the insulation on the spherical shell is given by

$$\dot{Q} = \frac{(T_1 - T_\infty)}{(r - r_1)/4\pi k\, r\, r_1 + 1/h\, 4\pi r^2}$$

$$\frac{d\dot{Q}}{dr} = 0 = \frac{4\pi(T_1 - T_\infty)\left(1/kr^2 - 2/hr^3\right)}{\left[(r - r_1)/k\, r\, r_1 + 1/hr^2\right]^2}$$

which gives, $1/Kr^2 - 2/hr^3 = 0$;

or $\quad r = r_c = 2\ k/h \qquad (2.22)$

2.10.9 Heat and Mass Transfer

Example 2.5 Hot gases at 175°C flow through a metal pipe (outer diameter 8 cm). The convective heat transfer coefficient at the outside surface of the insulation (k = 0.18 W /mK) IS 2.6 W m1K and the ambient temperature IS 25°C. Calculate the insulation thickness such that the heat loss is less than the uninsulated case.

Solution: (a) Pipe without Insulation

Neglecting the thermal resistance of the pipe wall and due to the inside convective heat transfer coefficient, the temperature of the pipe surface would be 175°C.

$\dot{Q}/L = h \times 2\pi r\ (T_1\text{-}T_\infty) = 2.6 \times 2 \times 3.14 \times .04\{175\text{-}25) = 98$ W/m (b) Pipe Insulated. Outermost Radius, r*

$$\dot{Q}/L = 98 = (T_1\text{-}T_\infty) / \left(\frac{1n(r^*/4)}{2\pi \times 0.18} + \frac{100}{2.6 \times 2\pi \times r^*}\right)$$

or $\frac{150}{98}$ = 08841n (r*/4)+6.12/r*; which gives r* = 13.5 cm.

Therefore, the insulation thickness must be more than 9.5 cm.

(Since the critical thickness of insulation is r_c = k/h = 0.18/2.6 = 6.92 cm, and is greater than the radius of the bare pipe, the required insulation thickness must give a radius greater than the critical radius.)

If the outer radius of the pipe was more than the critical radius, any addition of insulating material will reduce the rate of heat transfer. Let us assume that the outer radius of the pipe is 7 cm ($r > r_c$)

$\dot{Q}$/L, without insulation = hA (ΔT) = 2.6 × 2 ×3.142 × 0.07× (175-25)

= 171.55 W/m

By adding 4 cm thick insulation, outermost radius = 7.0 + 4.0 = 11.0 cm.

and $\dot{Q}$/L= (175 - 25)/ $\left[\frac{\ln(11/7)}{2\pi\times 0.18}+\frac{1}{2.6\pi\times 2\times 0.11}\right]$=133.58W/m.

Reduction in heat loss = $\frac{171.55-133.58}{171.55}$ = 0.22 or 22%.

Example 2.6 An electric conductor 1.5 mm in diameter at a surface temperature of 80°C is being cooled in air at 25°C. The convective heat transfer coefficient from the conductor surface is 16W/m^{2}K. Calculate the surface temperature of the conductor when it is covered with a layer of rubber insulation (2 mm thick, k = 0.15 W /mK) assuming that the conductor carries the same current and the convective heat transfer coefficient is also the same. Also calculate the increase in the current carrying capacity of the conductor when the surface temperature of the conductor remains at 80°C.

Solution: When there is no insulation,

$\dot{Q}$/ L = hA (ΔT) = 16 × 2 × 3.142 × 0.75 × 10^{-3} = 4.147 W/m

When the insulation is provided, the outermost radius = 0.75 + 2 = 2.75 mm

$\dot{Q}$/ L = 4.147 = (T_1-25) / $\left(\frac{\ln 2.75/0.75}{2\pi\times 0.15}+\frac{1000}{16\times 2\pi\times 2.75}\right)$

or T_1 = 45.71°C

i.e., the temperature at the outer surface of the wire decreases because the insulation adds a resistance.

The critical radius of insulation, r c = k/h = 0.15/16 = 9.375 mm

i.e., when an insulation of thickness (9.375 - 0.75) = 8.625 mm is added, the heat transfer rate would be the maximum and the conductor can carry more current. The heat transfer rate with outermost radius equal to r_c = 9.375 mm

$$\dot{Q}/L = (80 - 25) / \left(\frac{1n\,9.375/0.75}{2\pi \times 0.15} + \frac{1000}{16 \times 2\pi \times 9.375} \right) = 14.7 \text{ W/m}$$

The rate of heat transfer is proportional to $(\text{current})^2$, the new current I_2 would be:

$I_2/I_1 = (14.7 / 4.147)^{1/2} = 1.883$

or, the current carrying capacity can be increased 1.883 times. But the maximum current capacity of wire would be limited by the permissible temperature at the centre of the wire.

The surface temperature of the conductor when the outermost radius with insulation is equal to the critical radius, is given by

$$\dot{Q}/L = 4.147 = (T-25/ \left(\frac{1n\,9.375/0.75}{2 \times 3.142 \times 0.15} + \frac{1000}{16 \times 2 \times 3.142 \times 9.375} \right)$$

or $\quad T = 40.83°C.$

SATHYABAMA

INSTITUTE OF SCIENCE AND TECHNOLOGY

(DEEMED TO BE UNIVERSITY)

Accredited "A" Grade by NAAC | 12B Status by UGC | Approved by AICTE

www.sathyabama.ac.in

SCHOOL OF MECHANICAL ENGINEERING

DEPARTMENT OF AERONAUTICAL ENGINEERING

UNIT – III – Heat Transfer Techniques For Aerospace Applications – SAE1306

UNIT III

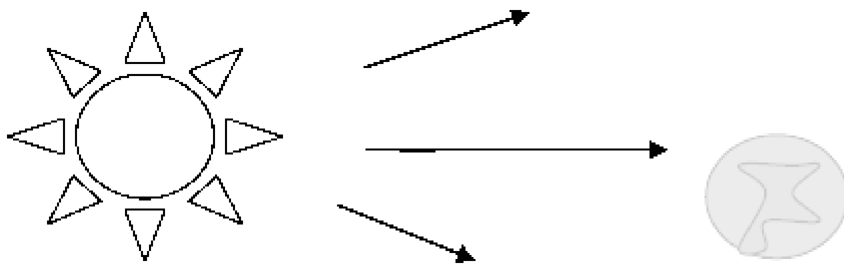
RADIATION HEAT TRANSFER

3.1RADIATION

Definition:

Radiation is the energy transfer across a system boundary due to a ΔT, by the mechanism of photon emission or electromagnetic wave emission.

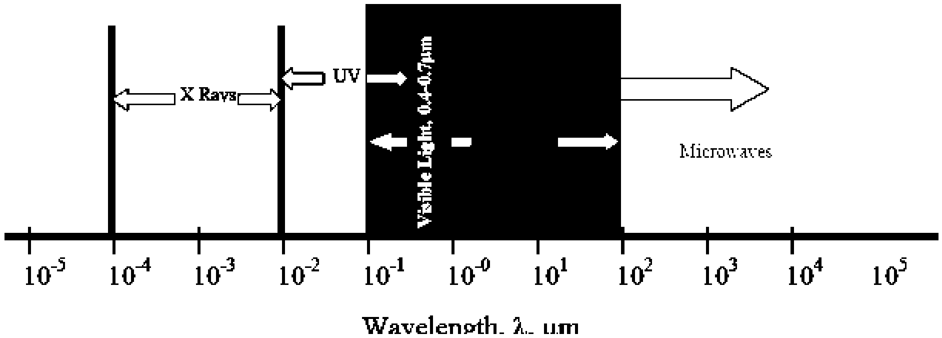
Because the mechanism of transmission is photon emission, unlike conduction and convection, there need be no intermediate matter to enable transmission.



The significance of this is that radiation will be the only mechanism for heat transfer whenever a vacuum is present.

3.2 Electromagnetic Phenomena.

We are well acquainted with a wide range of electromagnetic phenomena in modern life. These phenomena are sometimes thought of as wave phenomena and are, consequently, often described in terms of electromagnetic wave length, λ. Examples are given in terms of the wave distribution shown below:



One aspect of electromagnetic radiation is that the related topics are more closely associated with optics and electronics than with those normally found in mechanical engineering courses. Nevertheless, these are widely encountered topics and the student is familiar with them through every day life experiences.

From a viewpoint of previously studied topics students, particularly those with a background in mechanical or chemical engineering will find the subject of Radiation Heat Transfer a little unusual. The physics background differs fundamentally from that found in the areas of continuum mechanics. Much of the related material is found in courses more closely identified with quantum physics or electrical engineering, i.e. Fields and Waves. At this point, it is important for us to recognize that since the subject arises from a different area of physics, it will be important that we study these concepts with extra care.

3.3Stefan-Boltzman Law

Both Stefan and Boltzman were physicists; any student taking a course in quantum physics will become well acquainted with Boltzman's work as he made a number of important contributions to the field. Both were contemporaries of Einstein so we see that the subject is of fairly recent vintage. (Recall that the basic equation for convection heat transfer is attributed to Newton)

$$E_b = \sigma \cdot T_{abs}^{\ 4}$$

where: E_b = Emissive Power, the gross energy emitted from an ideal surface per unit area, time.

σ = Stefan Boltzman constant, $5.67 \cdot 10^{-8}$ W/m$^2 \cdot$K^4

T_{abs} = Absolute temperature of the emitting surface, K.

Take particular note of the fact that absolute temperatures are used in Radiation. It is suggested, as a matter of good practice, to convert all temperatures to the absolute scale as an initial step in all radiation problems.

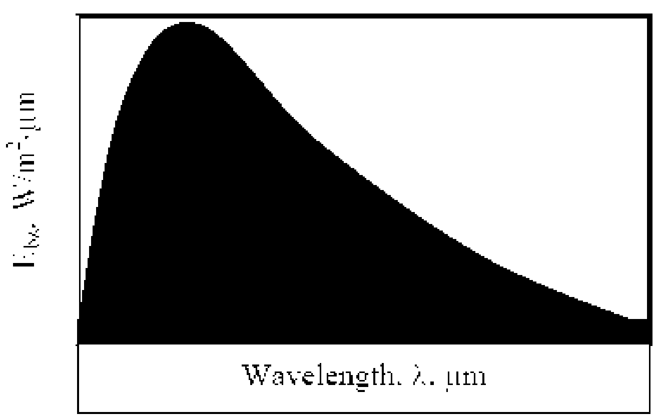
You will notice that the equation does not include any heat flux term, q". Instead we have a term the emissive power. The relationship between these terms is as follows. Consider two infinite plane surfaces, both facing one another. Both surfaces are ideal surfaces. One surface is found to be at temperature, T1, the other at temperature, T2. Since both temperatures are at temperatures above absolute zero, both will radiate energy as described by the Stefan-Boltzman law. The heat flux will be the net radiant flow as given by:

$$q'' = E_{b1} - E_{b2} = \sigma \cdot T_1^4 - \sigma \cdot T_2^4$$

3.4Plank's Law

While the Stefan-Boltzman law is useful for studying overall energy emissions, it does not allow us to treat those interactions, which deal specifically with wavelength, λ. This problem was overcome by another of the modern physicists, Max Plank, who developed a relationship for wave-based emissions.

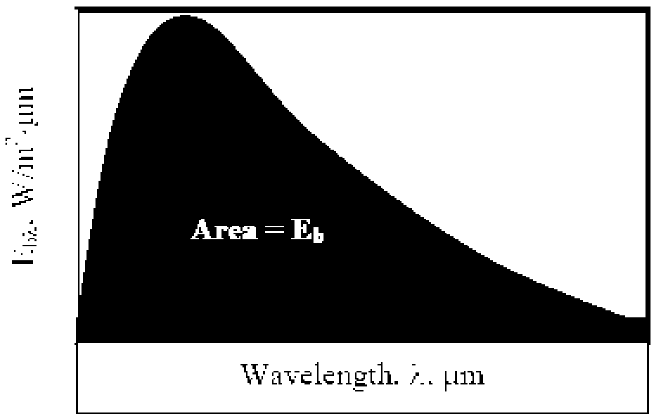
$$E_{b\lambda} = f(\lambda)$$



We haven't yet defined the Monochromatic Emissive Power, $E_{b\lambda}$. An implicit definition is provided by the following equation:

$$E_b = \int_0^\infty E_{b\lambda} \cdot d\lambda$$

We may view this equation graphically as follows:



A definition of monochromatic Emissive Power would be obtained by differentiating the integral equation:

$$E_{b\lambda} \equiv \frac{dE_b}{d\lambda}$$

The actual form of Plank's law is:

$$E_{b\lambda} = \frac{C_1}{\lambda^5 \cdot \left[e^{C_2/\lambda \cdot T} - 1\right]}$$

$C_1 = 2\cdot\pi\cdot h\cdot c_o^2 = 3.742\cdot10^8\ W\cdot\mu m^4/m^2$
$C_2 = h\cdot c_o/k = 1.439\cdot10^4\ \mu m\cdot K$

Where: h, c_o, k are all parameters from quantum physics. We need not worry about their precise definition here.

This equation may be solved at any T, λ to give the value of the monochromatic emissivity at that condition. Alternatively, the function may be substituted into the integral $E_b = \int_0^\infty E_{b\lambda} \cdot d\lambda$ to find the Emissive power for any temperature. While performing this integral by hand is difficult, students may readily evaluate the integral through one of several computer programs, i.e. MathCad, Maple, Mathmatica, etc.

$$E_b = \int_0^\infty E_{b\lambda} \cdot d\lambda = \sigma \cdot T^4$$

3.5 Emission over Specific Wave Length Bands

Consider the problem of designing a tanning machine. As a part of the machine, we will need to design a very powerful incandescent light source. We may wish to know how much energy is being emitted over the

Ultraviolet band (10^{-4} to 0.4 μm), known to be particularly dangerous.

$$E_b(0.0001 \rightarrow 0.4) = \int_{0.001 \cdot \mu m}^{0.4 \cdot \mu m} E_{b\lambda} \cdot d\lambda$$

With a computer available, evaluation of this integral is rather trivial. Alternatively, the text books provide a table of integrals. The format used is as follows:

$$\frac{E_b(0.001 \rightarrow 0.4)}{E_b} = \frac{\int_{0.001 \cdot \mu m}^{0.4 \cdot \mu m} E_{b\lambda} \cdot d\lambda}{\int_0^{\infty} E_{b\lambda} \cdot d\lambda} = \frac{\int_{0}^{0.4 \cdot \mu m} E_{b\lambda} \cdot d\lambda}{\int_0^{\infty} E_{b\lambda} \cdot d\lambda} - \frac{\int_{0.}^{0.0001 \cdot \mu m} E_{b\lambda} \cdot d\lambda}{\int_0^{\infty} E_{b\lambda} \cdot d\lambda} = F(0 \rightarrow 0.4) - F(0 \rightarrow 0.0001)$$

Referring to such tables, we see the last two functions listed in the second column. In the first column is a parameter, λ·T. This is found by taking the product of the absolute temperature of the emitting surface, T, and the upper limit wave length, λ. In our example, suppose that the incandescent bulb is designed to operate at a temperature of 2000K. Reading from the table:

λ.,

λ., μm	T, K	λ·T, μm·K	F(0→λ)
0.0001	2000	0.2	0
0.4	2000	600	0.000014
F(0.4→0.0001μm) = F(0→0.4μm)- F(0→0.0001μm)			**0.000014**

This is the fraction of the total energy emitted which falls within the IR band. To find the absolute energy emitted multiply this value times the total energy emitted:

$$E_{bIR} = F(0.4 \rightarrow 0.0001\mu m) \cdot \sigma \cdot T^4 = 0.000014 \cdot 5.67 \cdot 10^{-8} \cdot 2000^4 = \mathbf{12.7\ W/m^2}$$

3.6 Solar Radiation

The magnitude of the energy leaving the Sun varies with time and is closely associated with such factors as solar flares and sunspots. Nevertheless, we often choose to work with an average value. The energy leaving the sun is emitted outward in all directions so that at any particular distance from the Sun we may imagine the energy being dispersed over an imaginary spherical area. Because this area increases with the distance squared, the solar flux also decreases with the distance squared. At the average distance between Earth and Sun this heat flux is 1353 W/m2, so that the average heat flux on any object in Earth orbit is found as:

$$G_{s,o} = S_c \cdot f \cdot \cos\theta$$

Where S_c = Solar Constant, 1353 W/m^2

f = correction factor for eccentricity in Earth Orbit, ($0.97<f<1.03$)

θ = Angle of surface from normal to Sun.

Because of reflection and absorption in the Earth's atmosphere, this number is significantly reduced at ground level. Nevertheless, this value gives us some opportunity to estimate the potential for using solar energy, such as in photovoltaic cells.

Some Definitions

In the previous section we introduced the Stefan-Boltzman Equation to describe radiation from an ideal surface.

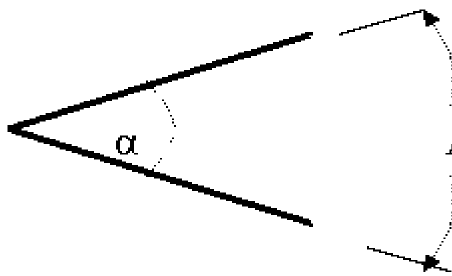
$$E_b = \sigma \cdot T_{abs}^{\ 4}$$

This equation provides a method of determining the total energy leaving a surface, but gives no indication of the direction in which it travels. As we continue our study, we will want to be able to calculate how heat is distributed among various objects.

For this purpose, we will introduce the radiation intensity, I, defined as the energy emitted per unit area, per unit time, per unit solid angle. Before writing an equation for this new property, we will need to define some of the terms we will be using.

3.7 Angles and Arc Length

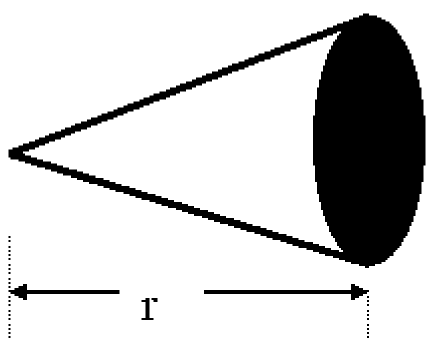
We are well accustomed to thinking of an angle as a two dimensional object. It may be used to find an arc length:



$$L = r \cdot \alpha$$

Solid Angle

We generalize the idea of an angle and an arc length to three dimensions and define a solid angle, Ω, which like the standard angle has no dimensions. The solid angle, when multiplied by the radius squared will have dimensions of length squared, or area, and will have the magnitude of the encompassed area.

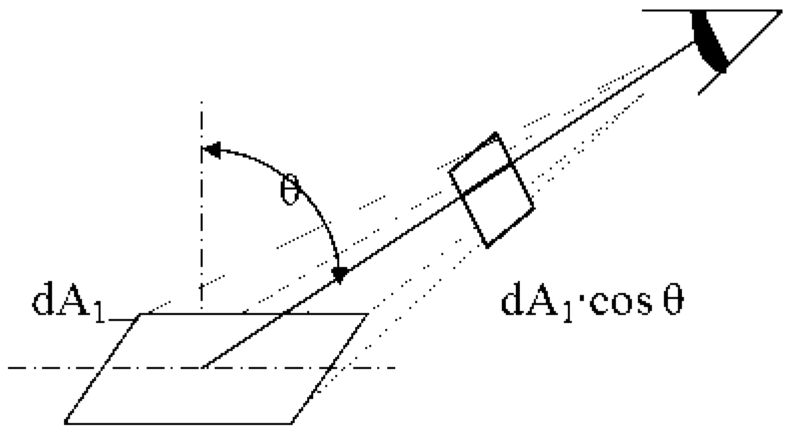


$$A = r^2 \cdot d\Omega$$

3.8 Projected Area

The area, dA1, as seen from the prospective of a viewer, situated at an angle θ from the normal to the surface, will appear somewhat smaller, as cos θ·dA1. This smaller area is termed the projected area.

$$A_{projected} = \cos\theta \cdot A_{normal}$$



3.9 Intensity

The ideal intensity, Ib, may now be defined as the energy emitted from an ideal body, per unit projected area, per unit time, per unit solid angle.

$$I = \frac{dq}{\cos\theta \cdot dA_1 \cdot d\Omega}$$

3.10 Spherical Geometry

Since any surface will emit radiation outward in all directions above the surface, the spherical coordinate system provides a convenient tool for analysis. The three basic coordinates shown are R, φ, and θ, representing the radial, azimuthal and zenith directions.

In general dA1 will correspond to the emitting surface or the source. The surface dA2 will correspond to the receiving surface or the target. Note that the area proscribed on the hemisphere, dA2, may be written as:

$$dA_2 = [(R \cdot \sin\theta) \cdot d\varphi] \cdot [R \cdot d\theta]$$

or, more simply as:

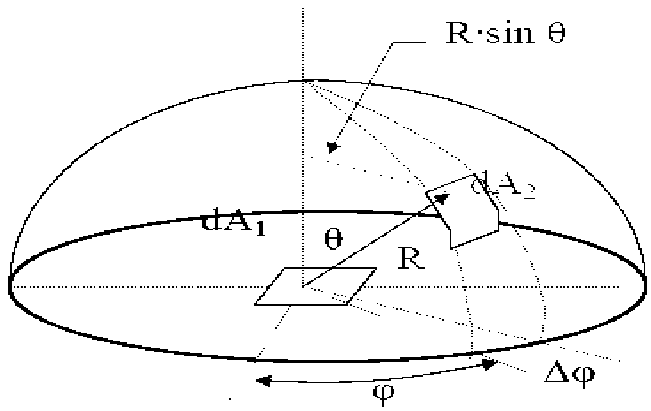
$$dA_2 = R^2 \cdot \sin\theta \cdot d\varphi \cdot d\theta]$$

Recalling the definition of the solid angle,

$dA = R^2 \cdot d\Omega$

we find that:

$d\Omega = R^2 \sin\theta \cdot d\theta \cdot d\varphi$



3.11 Real Surfaces

Thus far we have spoken of ideal surfaces, i.e. those that emit energy according to the Stefan-Boltzman law:

$$E_b = \sigma \cdot T_{abs}^{\ 4}$$

Real surfaces have emissive powers, E, which are somewhat less than that obtained theoretically by Boltzman. To account for this reduction, we introduce the emissivity, ε.

$$\varepsilon \equiv \frac{E}{E_b}$$

so that the emissive power from any real surface is given by:

$$E = \varepsilon \cdot \sigma \cdot T_{abs}^{\ 4}$$

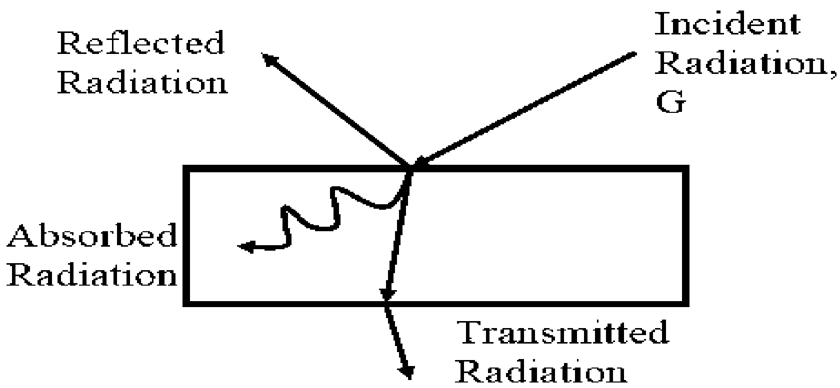
Receiving Properties

Targets receive radiation in one of three ways; they absorption, reflection or transmission. To account for these characteristics, we introduce three additional properties:

• Absorptivity, α, the fraction of incident radiation absorbed.

• Reflectivity, ρ, the fraction of incident radiation reflected.

• Transmissivity, τ, the fraction of incident radiation transmitted.



We see, from Conservation of Energy, that:

$\alpha + \rho + \tau = 1$

In this course, we will deal with only opaque surfaces, $\tau = 0$, so that:

$\alpha + \rho = 1$ Opaque Surfaces

3.12 Relationship Between Absorptivity,α, and Emissivity,ε

Consider two flat, infinite planes, surface A and surface B, both emitting radiation toward one another. Surface B is assumed to be an ideal emitter, i.e. $\varepsilon_B = 1.0$. Surface A will emit radiation according to the Stefan-Boltzman law as:

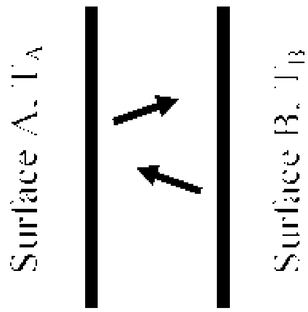
$$E_A = \varepsilon_A \cdot \sigma \cdot T_A^4$$

and will receive radiation as:

$$G_A = \alpha_A \cdot \sigma \cdot T_B^4$$

The net heat flow from surface A will be:

$$q'' = \varepsilon_A \cdot \sigma \cdot T_A^4 - \alpha_A \cdot \sigma \cdot T_B^4$$



Now suppose that the two surfaces are at exactly the same temperature. The heat flow must be zero according to the 2nd law. If follows then that:

$\alpha_A = \varepsilon_A$

Because of this close relation between emissivity, ε, and absorptivity, α, only one property is normally measured and this value may be used alternatively for either property.

Let's not lose sight of the fact that, as thermodynamic properties of the material, α and ε may depend on temperature. In general, this will be the case as radiative properties will depend on wavelength, λ. The wave length of radiation will, in turn, depend on the temperature of the source of radiation. The emissivity, ε, of surface A will depend on the material of which surface A is composed, i.e. aluminum, brass, steel, etc. and on the temperature of surface A. The absorptivity, α, of surface A will depend on the material of which surface A is composed, i.e. aluminum, brass, steel, etc. and on the temperature of surface B.

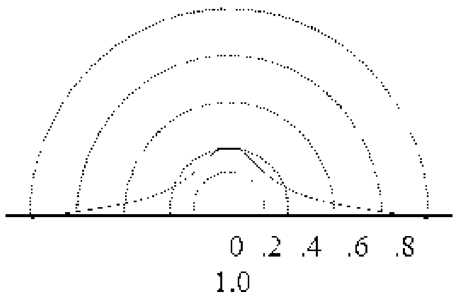
In the design of solar collectors, engineers have long sought a material which would absorb all solar radiation, ($\alpha = 1$, Tsun ~ 5600K) but would not re-radiate energy as it came to temperature ($\varepsilon << 1$, $T_{collector}$~ 400K). NASA developed an anodized chrome, commonly called "black chrome" as a result of this research.

3.13 Black Surfaces

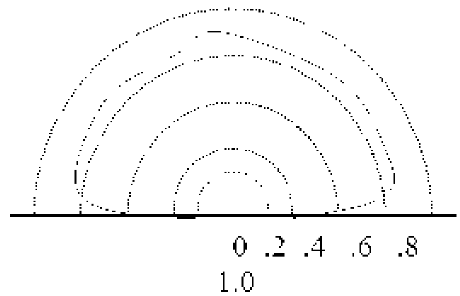
Within the visual band of radiation, any material, which absorbs all visible light, appears as black. Extending this concept to the much broader thermal band, we speak of surfaces with $\alpha = 1$ as also being "black" or "thermally black". It follows that for such a surface, $\varepsilon = 1$ and the surface will behave as an ideal emitter. The terms ideal surface and black surface are used interchangeably.

3.14 Lambert's Cosine Law:

A surface is said to obey Lambert's cosine law if the intensity, I, is uniform in all directions. This is an idealization of real surfaces as seen by the emissivity at different zenith angles:



Dependence of Emissivity on
Zenith Angle. Typical Metal.



Dependence of Emissivity on
Zenith Angle. Typical Non-Metal.

The sketches shown are intended to show is that metals typically have a very low emissivity, ε, which also remain nearly constant, expect at very high zenith angles, θ. Conversely, non-metals will have a relatively high emissivity, ε, except at very high zenith angles. Treating the emissivity as a constant over all angles is

Generally a good approximation and greatly simplifies engineering calculations.

3.15 Relationship between Emissive Power and Intensity

By definition of the two terms, emissive power for an ideal surface, Eb, and intensity for an ideal surface, I_b

$$E_b = \int_{hemisphere} I_b \cdot \cos\theta \cdot d\Omega$$

Replacing the solid angle by its equivalent in spherical angles:

$$E_b = \int_0^{2\cdot\pi} \int_0^{\pi/2} I_b \cdot \cos\theta \cdot \sin\theta \cdot d\theta \cdot d\varphi$$

Integrate once, holding I_b constant:

$$E_b = 2\cdot\pi\cdot I_b \cdot \int_0^{\pi/2} \cos\theta \cdot \sin\theta \cdot d\theta$$

Integrate a second time. (Note that the derivative of sin θ is cos θ·dθ.)

$$E_b = 2 \cdot \pi \cdot I_b \cdot \frac{\sin^2 \theta}{2} \Big|_0^{\pi/2} = \pi \cdot I_b$$

$$E_b = \pi \cdot I_b$$

3.16 Radiation Exchange

During the previous lecture we introduced the intensity, I, to describe radiation within a particular solid angle.

$$I = \frac{dq}{\cos\theta \cdot dA_1 \cdot d\Omega}$$

This will now be used to determine the fraction of radiation leaving a given surface and striking a second surface.

Rearranging the above equation to express the heat radiated:

$$dq = I \cdot \cos\theta \cdot dA_1 \cdot d\Omega$$

Next we will project the receiving surface onto the hemisphere surrounding the source. First find the projected area of surface dA_2, $dA_2 \cdot \cos\theta_2$. (θ_2 is the angle between the normal to surface 2 and the position vector, R.) Then find the solid angle, Ω, which encompasses this area.

Substituting into the heat flow equation above:

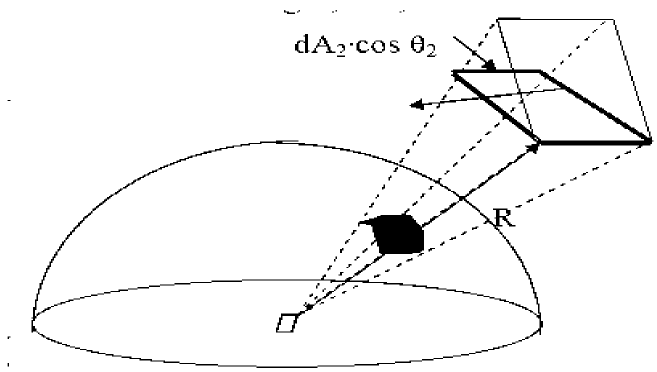
$$dq = \frac{I \cdot \cos\theta_1 \cdot dA_1 \cdot \cos\theta_2 dA_2}{R^2}$$

To obtain the entire heat transferred from a finite area, dA1, to a finite area, dA, we integrate over both surfaces:

$$q_{1\to 2} = \int_{A_2}\int_{A_1} \frac{I \cdot \cos\theta_1 \cdot dA_1 \cdot \cos\theta_2 dA_2}{R^2}$$

To express the total energy emitted from surface 1, we recall the relation between emissive power, E, and intensity, I.

$q_{emitted} = E_1 \cdot A_1 = \pi \cdot I_1 \cdot A_1$



3.17 View Factors-Integral Method

Define the view factor, F_{1-2}, as the fraction of energy emitted from surface 1, which directly strikes surface 2.

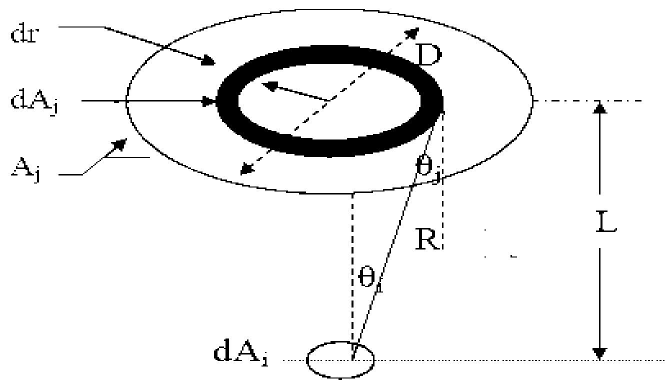
$$F_{1\to 2} = \frac{q_{1\to 2}}{q_{emitted}} = \frac{\int_{A_2}\int_{A_1} \frac{I \cdot \cos\theta_1 \cdot dA_1 \cdot \cos\theta_2 dA_2}{R^2}}{\pi \cdot I \cdot A_1}$$

after algebraic simplification this becomes:

$$F_{1\to 2} = \frac{1}{A_1} \cdot \int_{A_2}\int_{A_1} \frac{\cos\theta_1 \cdot \cos\theta_2 \cdot dA_1 \cdot dA_2}{\pi \cdot R^2}$$

Example 3.1 Consider a diffuse circular disk of diameter D and area Aj and a plane diffuse surface of area A

i << Aj. The surfaces are parallel, and Ai is located at a distance L from the center of Aj. Obtain an expression for the view factor F_{ij}



The view factor may be obtained from:

$$F_{1\to 2} = \frac{1}{A_1}\cdot\int_{A_2}\int_{A_1}\frac{\cos\theta_1\cdot\cos\theta_2\cdot dA_1\cdot dA_2}{\pi\cdot R^2}$$

Since dA_i is a differential area

$$F_{1\to 2} = \int_{A_1}\frac{\cos\theta_1\cdot\cos\theta_2\cdot dA_1}{\pi\cdot R^2}$$

Substituting for the cosines and the differential area:

$$F_{1\to 2} = \int_{A_1}\frac{\left(L/R\right)^2\cdot 2\pi\cdot r\cdot dr}{\pi\cdot R^2}$$

After simplifying:

$$F_{1\to 2} = \int_{A_1}\frac{L^2\cdot 2\cdot r\cdot dr}{R^4}$$

Let $\rho^2 \equiv L^2 + r^2 = R^2$. Then $2\cdot\rho\cdot d\rho = 2\cdot r\cdot dr$.

$$F_{1\to 2} = \int_{A_1}\frac{L^2\cdot 2\cdot\rho\cdot d\rho}{\rho^4}$$

After integrating,

$$F_{1\to 2} = -2 \cdot L^2 \cdot \frac{\rho^{-2}}{2}\Bigg|_{A_2} = -L^2 \cdot \left[\frac{1}{L^2 + \rho^2}\right]_0^{D/2}$$

Substituting the upper & lower limits

$$F_{1\to 2} = -L^2 \cdot \left[\frac{4}{4 \cdot L^2 + D^2} - \frac{1}{L^2}\right]_0^{D/2} = \frac{D^2}{4 \cdot L^2 + D^2}$$

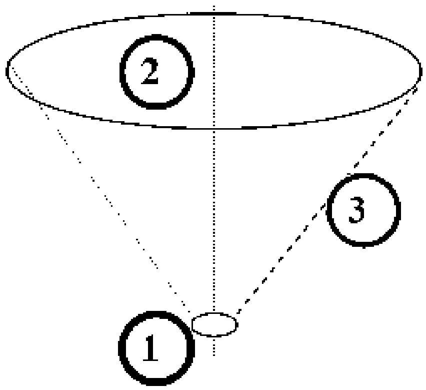
This is but one example of how the view factor may be evaluated using the integral method. The approach used here is conceptually quite straight forward; evaluating the integrals and algebraically simplifying the resulting equations can be quite lengthy.

Enclosures

In order that we might apply conservation of energy to the radiation process, we must account for all energy leaving a surface. We imagine that the surrounding surfaces act as an enclosure about the heat source which receives all emitted energy. Should there be an opening in this enclosure through which energy might be lost, we place an imaginary surface across this opening to intercept this portion of the emitted energy. For an N surfaced enclosure, we can then see that:

$\sum_{j=1}^{N} F_{i,j} = 1$ This relationship is known as "Conservation Rule".

Example: Consider the previous problem of a small disk radiating to a larger disk placed directly above at a distance L.



The view factor was shown to be given by the relationship:

$$F_{1\to2} = \frac{D^2}{4\cdot L^2 + D^2}$$

Here, in order to provide an enclosure, we will define an imaginary surface 3, a truncated cone intersecting circles 1 and 2.

From our conservation rule we have:

$$\sum_{j=1}^{N} F_{i,j} = F_{1,1} + F_{1,2} + F_{1,3}$$

Since surface 1 is not convex F1,1 = 0. Then:

$$F_{1\to3} = 1 - \frac{D^2}{4\cdot L^2 + D^2}$$

3.18 Reciprocity

We may write the view factor from surface i to surface j as:

$$A_i \cdot F_{i\to j} = \int_{A_j}\int_{A_i} \frac{\cos\theta_i \cdot \cos\theta_j \cdot dA_i \cdot dA_j}{\pi \cdot R^2}$$

Similarly, between surfaces j and i:

$$A_j \cdot F_{j \to i} = \int_{A_j} \int_{A_i} \frac{\cos\theta_j \cdot \cos\theta_i \cdot dA_j \cdot dA_i}{\pi \cdot R^2}$$

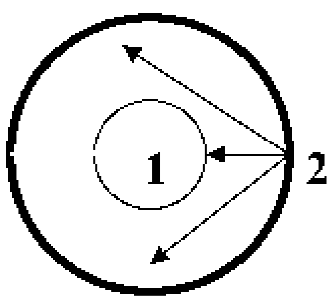
Comparing the integrals we see that they are identical so that:

$$A_i \cdot F_{i \to j} = A_j \cdot F_{j \to i}$$

This relation is known as “Reciprocity”.

Example:4.2 Consider two concentric spheres shown to the right. All radiation leaving the outside of surface 1 will strike surface 2. Part of the radiant energy leaving the inside surface of object 2 will strike surface 1, part will return to surface 2. To find the fraction of energy leaving surface 2 which strikes surface 1, we apply reciprocity:

$$A_2 \cdot F_{2,1} = A_1 \cdot F_{1,2} \Rightarrow F_{2,1} = \frac{A_1}{A_2} \cdot F_{1,2} = \frac{A_1}{A_2} = \frac{D_1}{D_2}$$



3.19 Associative Rule

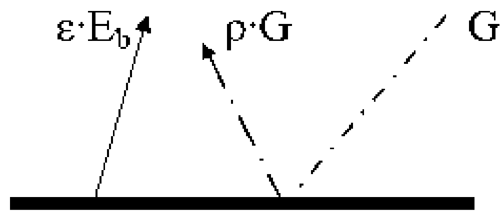
Consider the set of surfaces shown to the right: Clearly, from conservation of energy, the fraction of energy leaving surface i and striking the combined surface j+k will equal the fraction of energy emitted from i and striking j plus the fraction leaving surface i and striking k.

$$F_{i \Rightarrow (j+k)} = F_{i \Rightarrow j} + F_{i \Rightarrow k}$$

3.20 Radiosity

We have developed the concept of intensity, I, which let to the concept of the view factor. We have discussed various methods of finding view factors. There remains one additional concept to introduce before we can consider the solution of radiation problems.

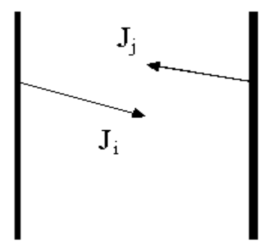
$$J \equiv \varepsilon \cdot E_b + \rho \cdot G$$



Radiosity, J, is defined as the total energy leaving a surface per unit area and per unit time. This may initially sound much like the definition of emissive power, but the sketch below will help to clarify the concept.

3.21 Net Exchange Between Surfaces

Consider the two surfaces shown. Radiation will travel from surface i to surface j and will also travel from j to i.



$$q_{i\to j} = J_i \cdot A_i \cdot F_{i\to j}$$

likewise,

$$q_{j\to i} = J_j \cdot A_j \cdot F_{j\to j}$$

The net heat transfer is then:

$$q_{j\to i\ (net)} = J_i \cdot A_i \cdot F_{i\to j} - J_j \cdot A_j \cdot F_{j\to j}$$

From reciprocity we note that $F_{1\to 2} \cdot A_1 = F_{2\to 1} \cdot A_2$ so that

$$q_{j\to i\ (net)} = J_i \cdot A_i \cdot F_{i\to j} - J_j \cdot A_i \cdot F_{i\to j} = A_i \cdot F_{i\to j} \cdot (J_i - J_j)$$

3.22 Net Energy Leaving a Surface

The net energy leaving a surface will be the difference between the energy leaving a surface and the energy received by a surface:

$$q_{1\rightarrow} = [\varepsilon \cdot E_b - \alpha \cdot G] \cdot A_1$$

Combine this relationship with the definition of Radiosity to eliminate G.

$$J \equiv \varepsilon \cdot E_b + \rho \cdot G \Rightarrow G = [J - \varepsilon \cdot E_b]/\rho$$

$$q_{1\rightarrow} = \{\varepsilon \cdot E_b - \alpha \cdot [J - \varepsilon \cdot E_b]/\rho\} \cdot A_1$$

Assume opaque surfaces so that $\alpha + \rho = 1 \Rightarrow \rho = 1 - \alpha$, and substitute for ρ.

$$q_{1\rightarrow} = \{\varepsilon \cdot E_b - \alpha \cdot [J - \varepsilon \cdot E_b]/(1 - \alpha)\} \cdot A_1$$

Put the equation over a common denominator:

$$q_{1\rightarrow} = \left[\frac{(1-\alpha) \cdot \varepsilon \cdot E_b - \alpha \cdot J + \alpha \cdot \varepsilon \cdot E_b}{1-\alpha}\right] \cdot A_1 = \left[\frac{\varepsilon \cdot E_b - \alpha \cdot J}{1-\alpha}\right] \cdot A_1$$

If we assume that $\alpha = \varepsilon$ then the equation reduces to:

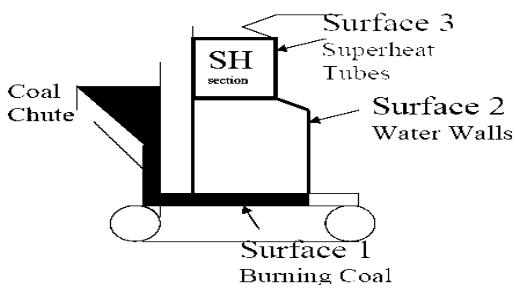
$$q_{1\rightarrow} = \left[\frac{\varepsilon \cdot E_b - \varepsilon \cdot J}{1-\varepsilon}\right] \cdot A_1 = \left[\frac{\varepsilon \cdot A_1}{1-\varepsilon}\right] \cdot (E_b - J)$$

3.23 Electrical Analogy for Radiation

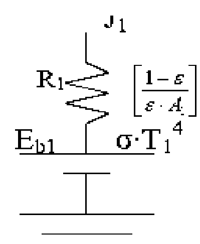
We may develop an electrical analogy for radiation, similar to that produced for conduction. The two analogies should not be mixed: they have different dimensions on the potential differences, resistance and current flows.

	Equivalent Current	Equivalent Resistance	Potential Difference
Ohms Law	I	R	ΔV
Net Energy Leaving Surface	$q_{1\rightarrow}$	$\left[\dfrac{1-\varepsilon}{\varepsilon \cdot A}\right]$	E_b - J
Net Exchange Between Surfaces	$q_{i\rightarrow j}$	$\dfrac{1}{A_1 \cdot F_{1\rightarrow 2}}$	$J_1 - J_2$

Example 4.3: Consider a grate fed boiler. Coal is fed at the bottom, moves across the grate as it burns and radiates to the walls and top of the furnace. The walls are cooled by flowing water through tubes placed inside of the walls. Saturated water is introduced at the bottom of the walls and leaves at the top at a quality of about 70%. After the vapor is separated from the water, it is circulated through the superheat tubes at the top of the boiler. Since the steam is undergoing a sensible heat addition, its temperature will rise. It is common practice to subdivide the super-heater tubes into sections, each having nearly uniform temperature. In our case we will use only one superheat section using an average temperature for the entire region.

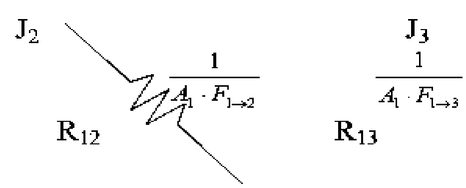


Energy will leave the coal bed, Surface 1, as described by the equation for the net energy leaving a surface. We draw the equivalent electrical network as seen to the right:

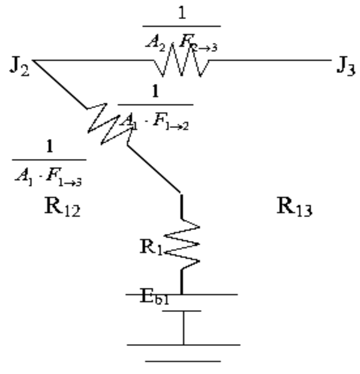


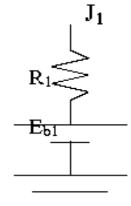
The heat leaving from the surface of the coal may proceed to either the water walls or to the super-heater section. That part of the circuit is represented by a potential difference between Radiosity:

It should be noted that surfaces 2 and 3

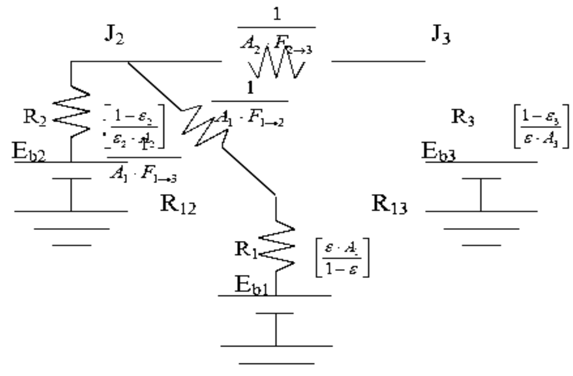


will also radiate to one another.





It remains to evaluate the net heat flow leaving (entering) nodes 2 and 3.



I

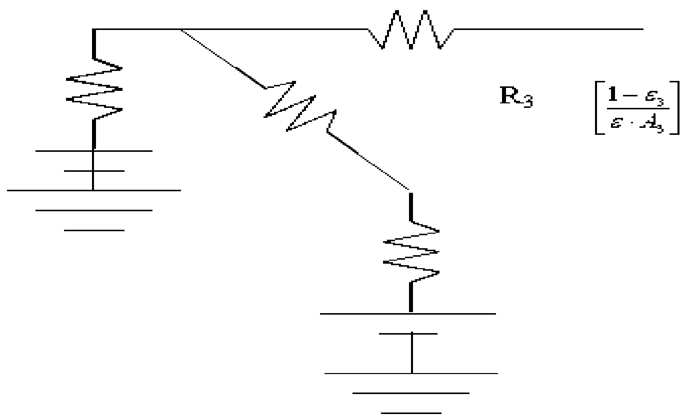
Alternate Procedure for Developing Networks

- Count the number of surfaces. (A surface must be at a "uniform" temperature and have uniform properties, i.e. ε, α, ρ.)
- Draw a radiosity node for each surface.
- Connect the Radiosity nodes using view factor resistances, $1/A_i \cdot F_{i\to j}$.
- Connect each Radiosity node to a grounded battery, through a surface resistance, $\left[1-\varepsilon / \varepsilon \cdot A\right]$.

This procedure should lead to exactly the same circuit as we obtain previously.

3.24 Simplifications to the Electrical Network

• Insulated surfaces. In steady state heat transfer, a surface cannot receive net energy if it is insulated. Because the energy cannot be stored by a surface in steady state, all energy must be re-radiated back into the enclosure. *Insulated surfaces are often termed as re-radiating surfaces*.



Electrically cannot flow through a battery if it is not grounded.

Surface 3 is not grounded so that the battery and surface resistance serve no purpose and are removed from the drawing.

• Black surfaces: A black, or ideal surface, will have no surface resistance:

$$\left[\frac{1-\varepsilon}{\varepsilon \cdot A}\right] = \left[\frac{1-1}{1 \cdot A}\right] = 0$$

In this case the nodal Radiosity and emissive power will be equal.

This result gives some insight into the physical meaning of a black surface. Ideal surfaces radiate at the maximum possible level. Non-black surfaces will have a reduced potential, somewhat like a battery with a corroded terminal. They therefore have a reduced potential to cause heat/current flow.

• Large surfaces: Surfaces having a large surface area will behave as black surfaces, irrespective of the actual surface properties:

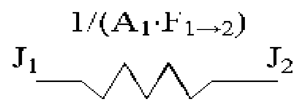
$$\left[\frac{1-\varepsilon}{\varepsilon \cdot A}\right] = \left[\frac{1-\varepsilon}{\varepsilon \cdot \infty}\right] = 0$$

Physically, this corresponds to the characteristic of large surfaces that as they reflect energy, there is very little chance that energy will strike the smaller surfaces; most of the energy is reflected back to another part of the same large surface. After several partial absorptions most of the energy received is absorbed.

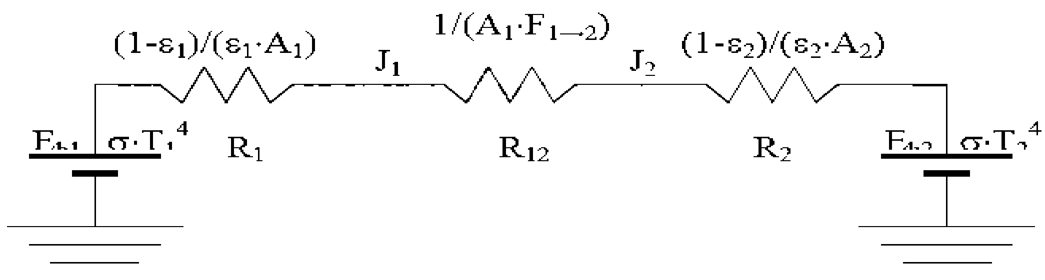
3.25 Solution of Analogous Electrical Circuits.

• Large Enclosures

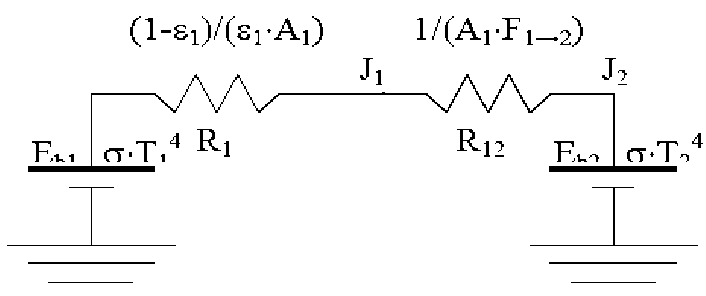
Consider the case of an object, 1, placed inside a large enclosure, 2. The system will consist of two objects, so we proceed to construct a circuit with two radiosity nodes



Now we ground both Radiosity nodes through a surface resistance.



Since A_2 is large, $R_2 = 0$. The view factor, $F_{1\to 2} = 1$



Sum the series resistances:

$$R_{Series} = (1-\varepsilon_1)/(\varepsilon_1\cdot A_1) + 1/A_1 = 1/(\varepsilon_1\cdot A_1)$$

Ohm's law:

$$i = \Delta V/R$$

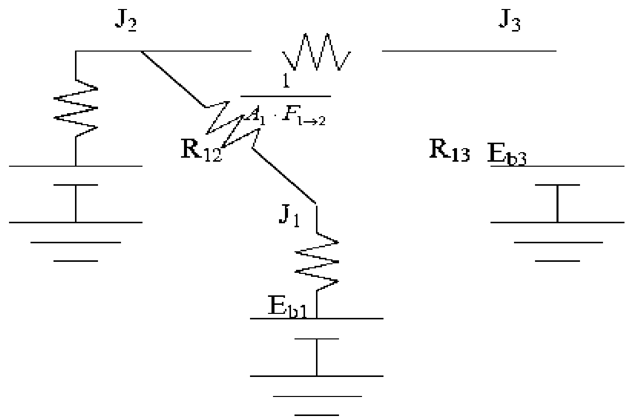
or by analogy:

$$q = \Delta E_b/R_{Series} = \varepsilon_1\cdot A_1\cdot\sigma\cdot(T_1^4 - T_2^4)$$

You may recall this result from Thermo I, where it was introduced to solve this type of radiation problem.

- Networks with Multiple Potentials

Systems with 3 or more grounded potentials will require a slightly different solution, but one which students have previously encountered in the Circuits course.



The procedure will be to apply Kirchoff's law to each of the Radiosity junctions.

$$\sum_{i=1}^{3} q_i = 0$$

In this example there are three junctions, so we will obtain three equations. This will allow us to solve for three unknowns.

Radiation problems will generally be presented on one of two ways:

1. The surface net heat flow is given and the surface temperature is to be found.

2. The surface temperature is given and the net heat flow is to be found.

Returning for a moment to the coal grate furnace, let us assume that we know (a) the total heat being produced by the coal bed, (b) the temperatures of the water walls and (c) the temperature of the super heater sections.

Apply Kirchoff's law about node 1, for the coal bed:

$$q_1 + q_{2\to 1} + q_{3\to 1} = q_1 + \frac{J_2 - J_1}{R_{12}} + \frac{J_3 - J_1}{R_{13}} = 0$$

Similarly, for node 2:

$$q_2 + q_{1\to 2} + q_{3\to 2} = \frac{E_{b2} - J_2}{R_2} + \frac{J_1 - J_2}{R_{12}} + \frac{J_3 - J_2}{R_{23}} = 0$$

(Note how node 1, with a specified heat input, is handled differently than node 2, with a specified temperature.

And for node 3:

$$q_3 + q_{1\to 3} + q_{2\to 3} = \frac{E_{b3} - J_3}{R_3} + \frac{J_1 - J_3}{R_{13}} + \frac{J_2 - J_3}{R_{23}} = 0$$

The three equations must be solved simultaneously. Since they are each linear in J, matrix methods may be used:

$$\begin{bmatrix} -\frac{1}{R_{12}} - \frac{1}{R_{13}} & \frac{1}{R_{12}} & \frac{1}{R_{13}} \\ \frac{1}{R_{12}} & -\frac{1}{R_2} - \frac{1}{R_{12}} - \frac{1}{R_{13}} & \frac{1}{R_{23}} \\ \frac{1}{R_{13}} & \frac{1}{R_{23}} & -\frac{1}{R_3} - \frac{1}{R_{13}} - \frac{1}{R_{23}} \end{bmatrix} \cdot \begin{bmatrix} J_1 \\ J_2 \\ J_3 \end{bmatrix} = \begin{bmatrix} -q_1 \\ -\frac{E_{b2}}{R_2} \\ -\frac{E_{b3}}{R_3} \end{bmatrix}$$

The matrix may be solved for the individual Radiosity. Once these are known, we return to the electrical analogy to find the temperature of surface 1, and the heat flows to surfaces 2 and 3.

Surface 1: Find the coal bed temperature, given the heat flow:

$$q_1 = \frac{E_{b1} - J_1}{R_1} = \frac{\sigma \cdot T_1^4 - J_1}{R_1} \Rightarrow T_1 = \left[\frac{q_1 \cdot R_1 + J_1}{\sigma}\right]^{0.25}$$

Surface 2: Find the water wall heat input, given the water wall temperature:

$$q_2 = \frac{E_{b2} - J_2}{R_2} = \frac{\sigma \cdot T_2^4 - J_2}{R_2}$$

Surface 3: (Similar to surface 2) Find the water wall heat input, given the water wall temperature:

$$q_3 = \frac{E_{b3} - J_3}{R_3} = \frac{\sigma \cdot T_3^4 - J_3}{R_3}$$

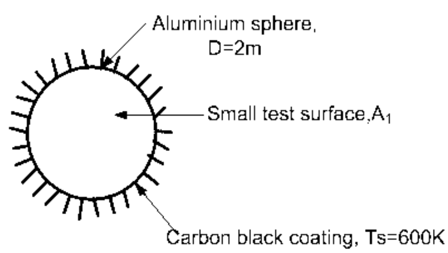
Module 9: Worked out problems

1. A spherical aluminum shell of inside diameter D=2m is evacuated and is used as a radiation test chamber. If the inner surface is coated with carbon black and maintained at 600K, what is the irradiation on a small test surface placed in the chamber? If the inner surface were not coated and maintained at 600K, what would the irradiation test?

Known: Evacuated, aluminum shell of inside diameter D=2m, serving as a radiation test chamber.

Find: Irradiation on a small test object when the inner surface is lined with carbon black and maintained at 600K.what effect will surface coating have?

Schematic:



Assumptions: (1) Sphere walls are isothermal, (2) Test surface area is small compared to the enclosure surface.

Analysis: It follows from the discussion that this isothermal sphere is an enclosure behaving as a black body. For such a condition, the irradiation on a small surface within the enclosure is equal to the black body emissive power at the temperature of the enclosure. That is

$$G_1 = E_b(T_s) = \sigma T_s^4$$

$$G_1 = 5.67 \times 10^{-8} W/m^2.K(600K)^4 = 7348 W/m^2$$

The irradiation is independent of the nature of the enclosure surface coating properties.

Comments: (1) The irradiation depends only upon the enclosure surface temperature and is independent of the enclosure surface properties.

(2) Note that the test surface area must be small compared to the enclosure surface area. This allows for inter-reflections to occur such that the radiation field, within the enclosure will be uniform (diffuse) or isotropic.

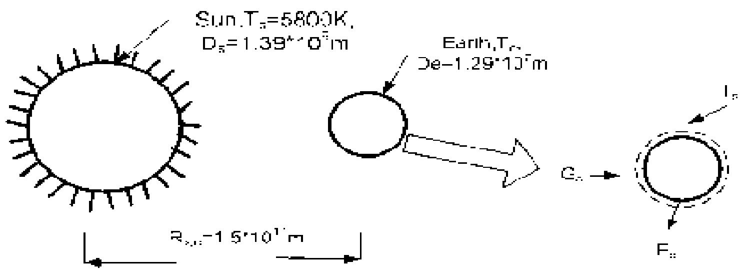
(3) The irradiation level would be the same if the enclosure were not evacuated since; in general, air would be a non-participating medium.

2 Assuming the earth's surface is black, estimate its temperature if the sun has an equivalently blackbody temperature of 5800K.The diameters of the sun and earth are 1.39*109 and 1.29*107m, respectively, and the distance between the sun and earth is 1.5*1011m.

Known: sun has an equivalently blackbody temperature of 5800K. Diameters of the sun and earth as well as separation distances are prescribed.

Find: Temperature of the earth assuming the earth is black.

Schematic:



Assumptions: (1) Sun and earth emit black bodies, (2) No attenuation of solar irradiation enroute to earth, and (3) Earth atmosphere has no effect on earth energy balance.

Analysis: performing an energy balance on the earth

$$\dot{E}_{in} - \dot{E}_{out} = 0$$

$$A_{e,p}.G_S = A_{e,s}.E_b(T_e)$$

$$(\pi D_e^2 / 4)G_S = \pi D_e^2 \sigma T_e^4$$

$$T_e = \left(G_S / 4\sigma\right)^{1/4}$$

Where $A_{s,p}$ and $A_{e,s}$ are the projected area and total surface area of the earth, respectively. To determine the irradiation GS at the earth's surface, perform an energy bounded by the spherical surface shown in sketch

$$\dot{E}_{in} - \dot{E}_{out} = 0$$

$$\pi D_S^2 .\sigma T_S^4 = 4\pi [R_{S,e} - D_e / 2]^2 G_S$$

$$\pi (1.39 \times 10^9 m)^2 \times 5.67 \times 10^{-8} W / m^2 .K (5800 K)^4 =$$

$$4\pi [1.5 \times 10^{11} - 1.29 \times 10^7 / 2]^2 m^2 \times G_S$$

$$G_S = 1377.5 W / m^2$$

Substituting numerical values, find

$$T_e = (1377.5 W / m^2 / 4 \times 5.67 \times 10^{-8} W / m^2 .K^4)^{1/4} = 279 K$$

Comments:

(1) The average earth's temperature is greater than 279 K since the effect of the atmosphere is to reduce the heat loss by radiation.

(2) Note carefully the different areas used in the earth energy balance. Emission occurs from the total spherical area, while solar irradiation is absorbed by the projected spherical area.

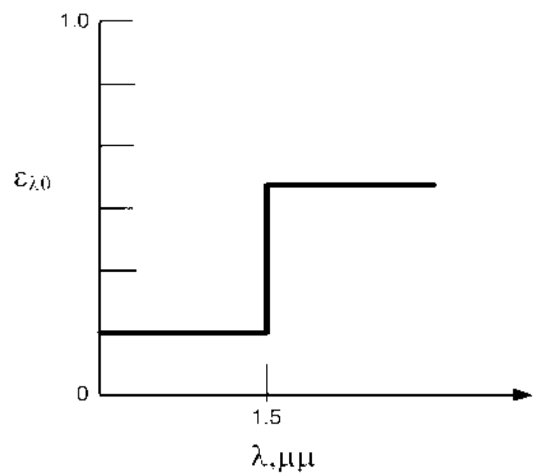
3 The spectral, directional emissivity of a diffuse material at 2000K has the following distribution.

Determine the total, hemispherical emissivity at 2000K.Determine the emissive power over the spherical range 0.8 t0 2.5 μm and for the directions 0≤θ≤30°.

Known: Spectral, directional emissivity of a diffuse material at 2000K.

Find: (1) The total, hemispherical emissivity, (b) emissive power over the spherical range 0.8 t0 2.5 μm and for the directions $0 \le \theta \le 30°$.

Schematic:



Assumptions: (1) Surface is diffuse emitter.

Analysis: (a) Since the surface is diffuse,ελ,θ is independent of direction; from Eq. $\varepsilon_{\lambda,\theta} = \varepsilon_\lambda$

$$\varepsilon(T) = \int_0^\infty \varepsilon_\lambda(\lambda) E_{\lambda,b}(\lambda, T)\, d\lambda / E_b(T)$$

$$\varepsilon(T) = \int_0^{1.5} \varepsilon_1) E_{\lambda,b}(\lambda, 2000)\, d\lambda / E_b) + \int_0^{1.5} \varepsilon_2 E_{\lambda,b}(\lambda, 2000)\, d\lambda / E_b$$

Written now in terms of $F_{(0\to\lambda)}$, with $F_{(0\to1.5)}$ =0.2732 at λT=1.5*2000=3000μm.K, find

$$\varepsilon(2000K) = \varepsilon_1 F_{(0\to1.5)} + \varepsilon_2\left[1 - F_{(0\to1.5)}\right] = 0.2 \times 0.2732 + 0.8[1 - 0.2732] = 0.636$$

(b) For the prescribed spectral and geometric limits,

$$\Delta E = \int_{0.8}^{2.5} \int_0^{2\pi} \int_0^{\pi/6} \varepsilon_{\lambda,\theta} I_{\lambda,b}(\lambda, T) \cos\theta \sin\theta\, d\theta\, d\phi\, d\lambda$$

where $I_{\lambda,e}(\lambda,\theta,\phi)=\varepsilon_{\lambda,\theta}I_{\lambda,b}(\lambda,T)$. Since the surface is diffuse, $\varepsilon_{\lambda,\theta}=\varepsilon_{\lambda}$, and nothing $I_{\lambda,b}$ is independent of direction and equal to $E_{\lambda,b}/\pi$, we can write

$$\Delta E=\left\{\int_0^{2\pi}\int_0^{\pi/6}\cos\theta\sin\theta\,d\theta\,d\phi\right\}\frac{E_b(T)}{\pi}\frac{\int_{0.8}^{1.5}\varepsilon_1 E_{\lambda,b}(\lambda,T)d\lambda}{E_b(T)}+\frac{\int_{1.5}^{2.5}\varepsilon_2 E_{\lambda,b}(\lambda,T)d\lambda}{E_b(T)}$$

Or in terms $F_{(0\to\lambda)}$ values,

$$\Delta E=\left\{\phi\Big|_0^{2\pi}\times\frac{\sin^2\theta}{2}\Big|_0^{\pi/6}\right\}\frac{\sigma T^4}{\pi}\left\{\varepsilon_1[F_{(0\to1.5)}-F_{(0\to0.8)}]\right\}+\varepsilon_2[F_{(0\to2.5)}-F_{(0\to1.5)}]\}$$

From table $\lambda T=0.8\times2000=1600\mu m.K$ $\qquad F_{(0\to0.8)}=0.0197$

$\lambda T=2.5\times2000=5000\mu m.K$ $\qquad F_{(0\to2.5)}=0.6337$

$$\Delta E=2\pi\times\frac{\sin^2\pi/6}{2}\frac{5.67\times10^{-8}2000^4}{\pi}\frac{W}{m^2}\{0.2[0.2732-0.0197]+[0.80.6337-0.2732]\}$$

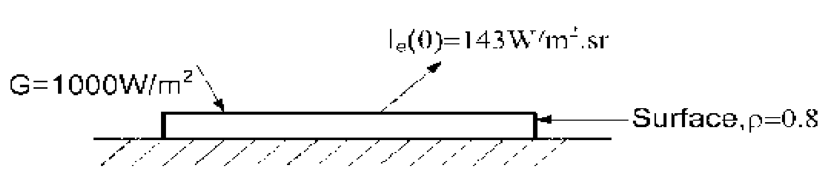
$$\Delta E=0.25\times(5.67\times10^{-8}\times2000^4W/m^2\times0.339=76.89W/m^2$$

4. A diffusely emitting surface is exposed to a radiant source causing the irradiation on the surface to be 1000W/m^2.The intensity for emission is 143W/m^2.sr and the reflectivity of the surface is 0.8.Determine the emissive power ,E(W/m^2),and radiosity ,J(W/m^2),for the surface. What is the net heat flux to the surface by the radiation mode?

Known: A diffusely emitting surface with an intensity due to emission of Is=143W/m^2.sr and a reflectance ρ=0.8 is subjected to irradiation=1000W/m^2.

Find: (a) emissive power of the surface, E (W/m^2), (b) radiosity, J (W/m^2), for the surface, (c) net heat flux to the surface.

Schematic:



Assumptions: (1) surface emits in a diffuse manner.

Analysis: (a) For a diffusely emitting surface, $I_s(\theta)=I_e$ is a constant independent of direction. The emissive power is

$$E = \pi I_e = \pi sr \times 143\,W/m^2.sr = 449\,W/m^2$$

Note that π has units of steradians (sr).

(b) The radiosity is defined as the radiant flux leaving the surface by emission and reflection,

$$J = E + \rho G = 449W/m^2 + 0.8 \times 1000W/m^2 = 1249W/m^2$$

(c) The net radiative heat flux to the surface is determined from a radiation balance on the surface.

$$q''_{net} = q''_{rad,in} - q''_{rad,out}$$

$$q''_{net} = G - J = 1000W/m^2 - 1249W/m^2 = -249W/m^2$$

Comments: No matter how the surface is irradiated, the intensity of the reflected flux will be independent of direction, if the surface reflects diffusely.

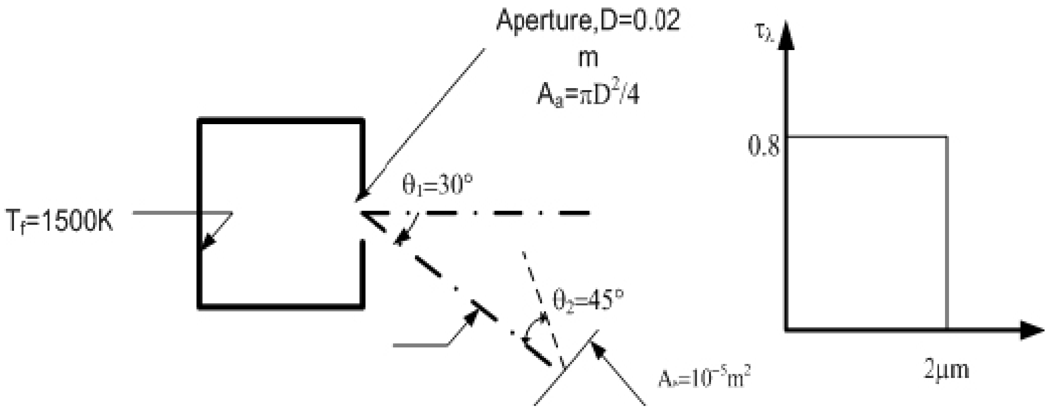
5. Radiation leaves the furnace of inside surface temperature 1500K through an aperture 20mm in diameter. A portion of the radiation is intercepted by a detector that is 1m from the aperture, as a surface area $10^{-5}m^2$, and is oriented as shown.

If the aperture is open, what is the rate at which radiation leaving the furnace is intercepted by the detector? If the aperture is covered with a diffuse, semitransparent material of spectral transmissivity $\tau\lambda=0.8$ for $\lambda\leq 2\mu m$ and $\tau\lambda=0$ for $\lambda>2\mu m$, what is the rate at which radiation leaving the furnace is intercepted by the detector?

Known: Furnace wall temperature and aperture diameter. Distance of detector from aperture and orientation of detector relative to aperture.

Find: Rate at which radiation leaving the furnace is intercepted by the detector, (b) effect of aperture window of prescribed spectral transmissivity on the radiation interception rate.

Schematic:



Assumptions:

(1) Radiation emerging from aperture has characteristics of emission from a black body, (2) Cover material is diffuse, (3) Aperture and detector surface may be approximated as infinitesimally small.

Analysis: (a) the heat rate leaving the furnace aperture and intercepted by the detector is

$q = I_e A_s \cos\theta\, w_{a-a}$ Heat and Mass Transfer

$$I_e = \frac{E_b(T_f)}{\pi} = \frac{\sigma T_f^4}{\pi} = \frac{5.67\times10^{-8}(1500)^4}{\pi} = 9.14\times10^4 W/m^2.sr$$

$$w'_{s-a} = \frac{A''}{r^2} = \frac{A_s.\cos\theta^2}{r^2} = \frac{10^{-5}m^2\cos45°}{(1m)^2} = 0.70710^{-5}.sr$$

Hence

$$q = 9.14\times10^4 W/m^2.sr[\pi(0.02)m^2/4]\cos30°\times0.707\times10^{-5}sr = 1.76\times10^{-4}W$$

(b) With the window, the heat rate is

$$q = \tau(I_e A_a \cos\theta_1 w_{a-a})$$

where τ is the transmissivity of the window to radiation emitted by the furnace wall.

$$\tau = \frac{\int_0^\infty \tau_\lambda G_\lambda d_\lambda}{\int_0^\infty G_\lambda d_\lambda} = \frac{\int_0^\infty \tau_\lambda E_{\lambda,b}(T_f) d\lambda}{\int_0^\infty E_{\lambda,b} d\lambda} = 0.8\int_0^2 (E_{\lambda,b} / E_b) d\lambda = 0.8F_{(0\rightarrow 2\mu m)}$$

with $\lambda T = 2\mu m \times 1500K = 3000\mu m.K$, from table $F(0 \rightarrow 2\mu m) = 0.273$.

hence with $0.273 \times 0.8 = 0.218$, find

$$q = 0.218 \times 1.76 \times 10^{-4}\,W = 0.384 \times 10^{-4}\,W$$

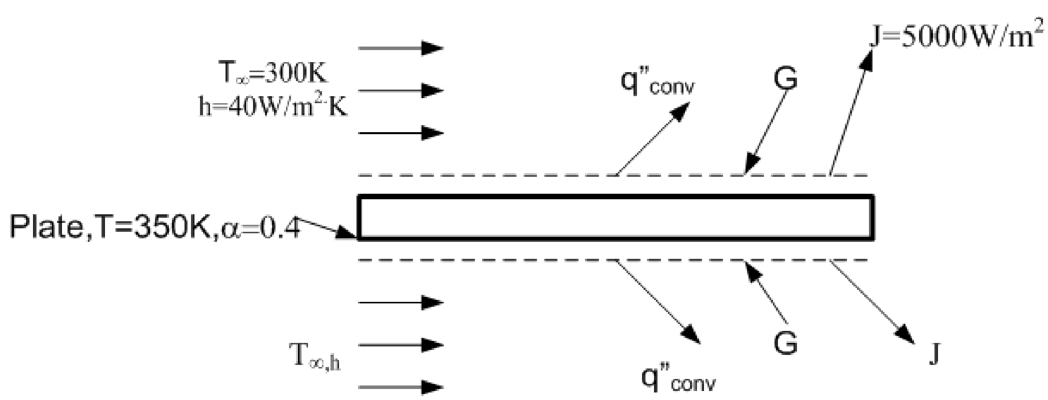
6.A horizontal semitransparent plate is uniformly irradiated from above and below, while air at T=300K flows over the top and bottom surfaces. providing a uniform convection heat transfer coefficient of h=40W/m2.K.the total, hemispherical absorptivity of the plate to the irradiation is 0.40.Under steady-state conditions measurements made with radiation detector above the top surface indicate a radiosity(which includes transmission, as well as reflection and emission) of J=5000W/m2,while the plate is at uniform temperature of T=350K

Determine the irradiation G and the total hemispherical emissivity of the plate. Is the plate gray for the prescribed conditions?

Known: Temperature, absorptivity, transmissivity, radiosity and convection conditions for a semi-transparent plate.

Find: Plate irradiation and total hemispherical emissivity.

Schematic:



Assumptions: From an energy balance on the plate

E_{in}- E_{out}

2G=2q”$_{conv}$+2J

Solving for the irradiation and substituting numerical values,

$G=40W/m^2.K\ (350\text{-}300)\ K+5000W/m^2=7000W/m^2$

From the definition of J

$$J = E + \rho G + \tau G = E + (1-\alpha)G$$

Solving for the emissivity and substituting numerical values,

$$\varepsilon = \frac{J-(1-\alpha)G}{\sigma T^4} = \frac{(5000\,W/m^2) - 0.6(7000\,W/m^2)}{5.67\times10^{-8}\,W/m^2.K^4\,(350\,K)^4} = 0.94$$

Hence

$$\alpha \neq \varepsilon$$

And the surface is not gray for the prescribed conditions.

Comments: The emissivity may also be determined by expressing the plate energy balance as

$$2\alpha G = 2q''_{conv}\,2E$$

$$hence$$

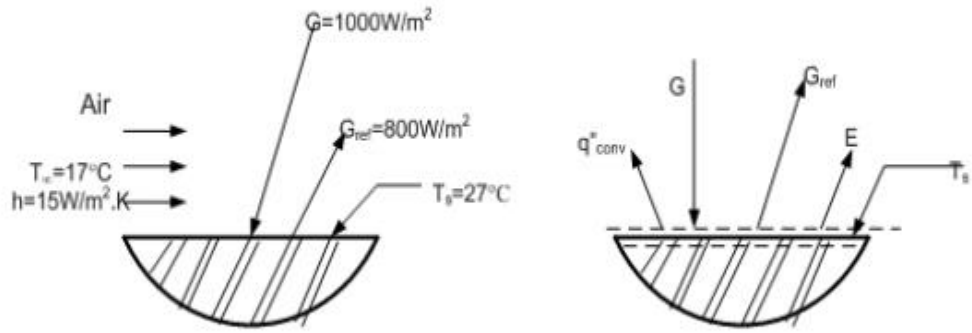
$$\varepsilon\sigma T^4 = \alpha G - h(T-T_\infty)$$
$$\varepsilon = \frac{0.4(7000W/m^2) - 40W/m^2.K(50K)}{5.67\times10^{-8}W/m^2.K4(350K)^4} = 0.94$$

7 An opaque, gray surface at 27°C is exposed to irradiation of 1000W/m2, and 800W/m2 is reflected. Air at 17°C flows over the surface and the heat transfer convection coefficient is 15W/m2.K.Determine the net heat flux from the surface.

Known: Opaque, gray surface at 27°C with prescribed irradiation, reflected flux and convection process.

Find: Net heat flux from the surface.

Schematic:



Assumptions:

1) Surface is opaque and gray,

2) Surface is diffuse,

3) Effects of surroundings are included in specified irradiation.

Analysis: From an energy balance on the surface, the net heat flux from the surface is

$$q''_{net} = \dot{E}''_{out} - E''_{in}$$

$$q''_{net} = q''_{conv} + E + G_{ref} - G = h(T_s - T_\infty) + \varepsilon\sigma T_s^4 + G_{ref} - G$$

$$\varepsilon = \alpha = 1 - \rho = 1 - (G_{ref} / G) = 1 - (800/1000) = 1 - 0.8 = 0.2$$

where $\rho = G_{ref}/G$**. the net heat flux from the surface**

$$q''_{net} = 15W/m^2.K(27-17)K + 0.2 \times 5.67 \times 10^{-8} W/m^2.K^4 (27+273)^4 K^4 + 800W/m^2 - 1000W/m^2$$

$$q''_{net} = (150 + 91.9 + 800 - 1000)W/m^2 = 42W/m^2$$

Comments: (1) For this situation, the radiosity is

$$J = G_{ref} + E = (800 + 91.9)W/m^2 = 892W/m^2$$

The energy balance can be written involving the radiosity (radiation leaving the surface) and the irradiation (radiation to the surface).

$$q''_{net,out} = J - G + q''_{conv} = (892 - 1000 + 150) W/m^2 = 42 W/m^2$$

Note the need to assume the surface is diffuse, gray and opaque in order that Eq (2) is applicable.

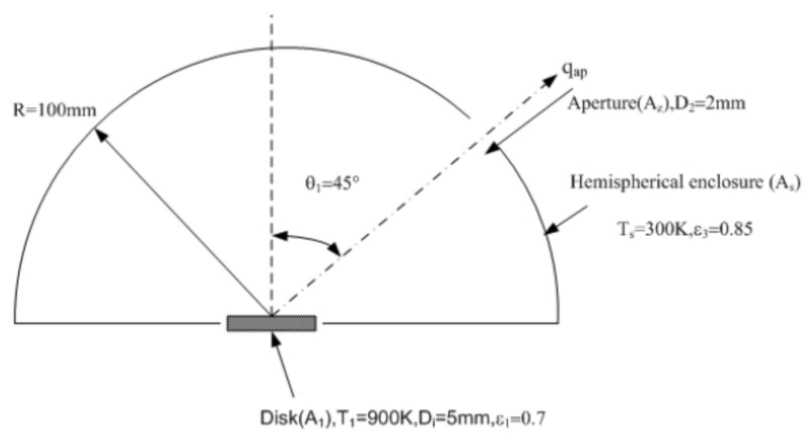
8. A small disk 5 mm in diameter is positioned at the center of an isothermal, hemispherical enclosure. The disk is diffuse and gray with an emissivity of 0.7 and is maintained at 900 K. The hemispherical enclosure, maintained at 300 K, has a radius of 100 mm and an emissivity of 0.85.

Calculate the radiant power leaving an aperture of diameter 2 mm located on the enclosure as shown.

Known: Small disk positioned at center of an isothermal, hemispherical enclosure with a small aperture.

Find: radiant power [μW] leaving the aperture.

Schematic:



Assumptions: (1)Disk is diffuse-gray,(2) Enclosure is isothermal and has area much larger than disk,(3) Aperture area is very small compared to enclosure area, (4) Areas of disk and aperture are small compared to radius squared of the enclosure.

Analysis: the radiant power leaving the aperture is due to radiation leaving the disk and to irradiation on the aperture from the enclosure. That is

$$q_{ap} = q_{1\to 2} + G_2 . A_2$$

The radiation leaving the disk can be written in terms of the radiosity of the disk. For the diffuse disk

$$q_{1\to 2} = \frac{1}{\pi} J_1 . A_1 \cos\theta_1 . w_{2-1}$$

and with $\varepsilon = \alpha$ for the gray behavior, the radiosity is

$$\mathbf{J_1} = \varepsilon_1 E_b(T_1) + \rho G_1 = \varepsilon_1 \sigma T_1^4 + (1-\varepsilon_1)\sigma T_3^4$$

Where the irradiatin G_1 is the emissive power of the black enclosure, $E_b(T_3)$;

$G_1 = G_2 = E_b(T_3)$. The solid angle $\omega_{2\text{-}1}$ follows

$$\omega_{2-1} = A_2 / R^2$$

Combining equations. (2), (3) and (4) into eq.(1) with $G2=\sigma T^4{}_3$, the radiant power is

$$q_{ap} = \frac{1}{\pi}\sigma[\varepsilon_1 T_1^4 + (1-\varepsilon_1)T_3^4]A_1 \cos\theta_1 . \frac{A_2}{R^2} + A_2\sigma T_3^4$$

$$q_{ap} = \frac{1}{\pi} 5.67\times 10^{-8} W/m^2.K^4 [0.7(900K)^4 + (1-0.7)(300K)^4 \frac{\pi}{4}(0.005m)^2 \cos 45° \times$$

$$\frac{\pi/4(0.002m)^2}{(0.100m)^2} + \frac{\pi}{4}(0.002m)25.67\times 10^{-8} W/m^2.K^4(300K)^4$$

$$q_{ap} = (36.2 + 0.19 + 1443)\mu W = 1479\mu W$$

SCHOOL OF MECHANICAL ENGINEERING

DEPARTMENT OF AERONAUTICAL ENGINEERING

UNIT – IV – Heat Transfer Techniques For Aerospace Applications – SAE1306

UNIT IV

HEAT EXCHANGERS

4.1 Heat Exchangers: Regenerators and Recuperators

A heat exchanger is equipment where heat energy is transferred from a hot fluid to a colder fluid. The transfer of heat energy between the two fluids could be carried out (i) either by direct mixing of the two fluids and the mixed fluids leave at an intermediate temperature determined from the principles of conservation of energy, (ii) or by transmission through a wall separating the two fluids. The former types are called direct contact heat exchangers such as water cooling towers and jet condensers. The latter types are called regenerators, recuperator surface exchangers.

In a regenerator, hot and cold fluids alternately flow over a surface which provides alternately a sink and source for heat flow. Fig. 4.1 (a) shows a cylinder containing a matrix that rotates in such a way that it passes alternately through cold and hot gas streams which are sealed from each other. Fig. 4.2 (b) shows a stationary matrix regenerator ill which hot and cold gases flow through them alternately.

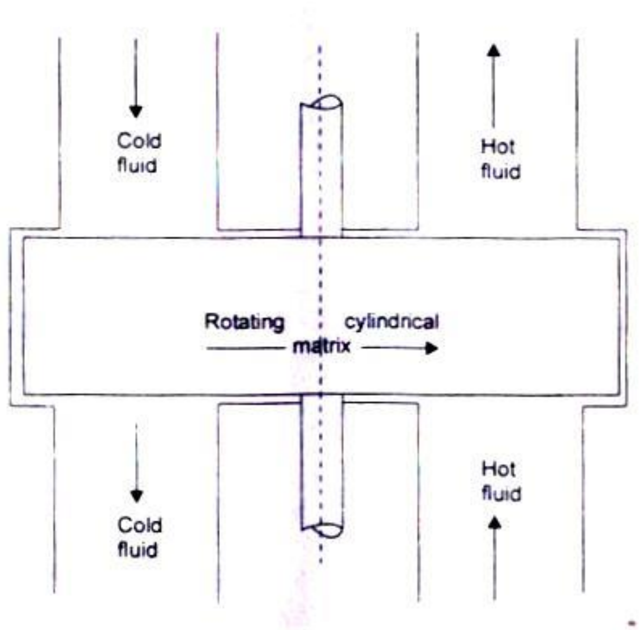


Fig. 4.1 (a) Rotating matrix regenerator

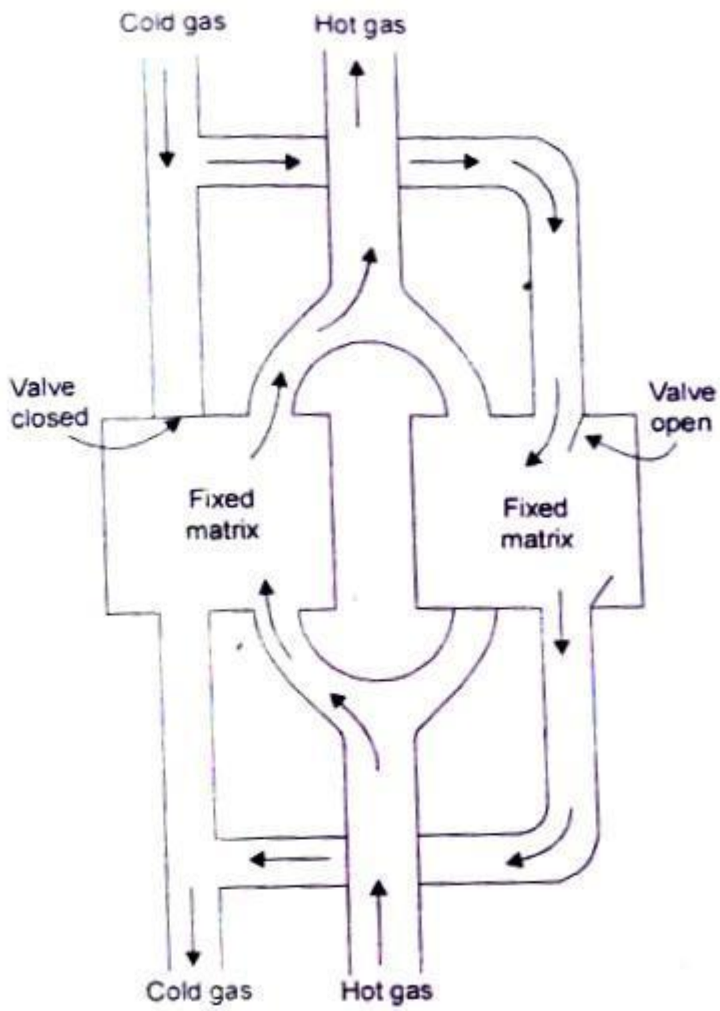


Fig. 4.2 (b) Stationary matrix regenerator

In a recuperator, hot and cold fluids flow continuously following the same path. The heat transfer process consists of convection between the fluid and the separating wall, conduction through the wall and convection between the wall and the other fluid. Most common heat exchangers are of recuperative type having a Wide variety of geometries:

4.2. Classification of Heat Exchangers

Heat exchangers are generally classified according to the relative directions of hot and cold fluids:

(a) Parallel Flow – the hot and cold fluids flow in the same direction. Fig 4.3 depicts such a heat exchanger where one fluid (say hot) flows through the pipe and the other fluid (cold) flows through the annulus.

(b) Counter Flow – the two fluids flow through the pipe but in opposite directions. A common type of such a heat exchanger is shown in Fig. 4.4. By comparing the temperature distribution of the two types of heat exchanger

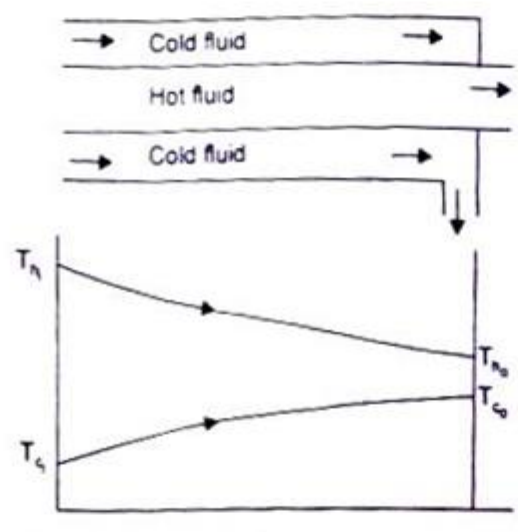


Fig 4.3 Parallel flow heat exchanger with Temperature distribution

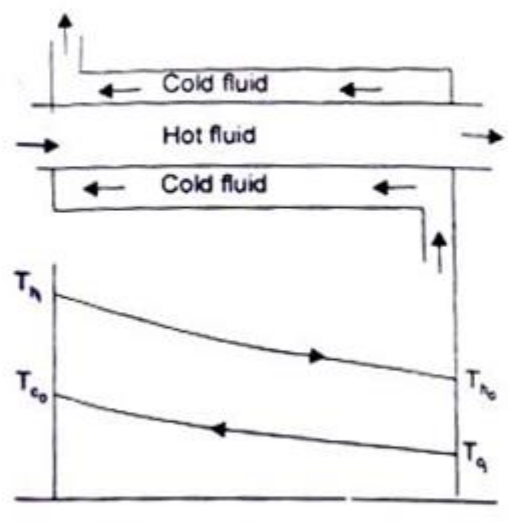


Fig 4.4 Counter-flow heat exchanger with temperature distribution

We find that the temperature difference between the two fluids is more uniform in counter flow than in the parallel flow. Counter flow exchangers give the maximum heat transfer rate and are the most favoured devices for heating or cooling of fluids.

When the two fluids flow through the heat exchanger only once, it is called one-shell-pass and one-tube-pass . If the fluid flowing through the tube makes one pass through half of the tube, reverses its direction of flow, and makes a second pass through the remaining half of the tube, it is called 'one-shell-pass, two-tube-pass' heat exchanger, fig 4.5. Many other possible flow arrangements exist and are being used. Fig. 4.6 depicts a 'two-shell-pass, four-tube-pass' exchanger.

(c) Cross-flow - A cross-flow heat exchanger has the two fluid streams flowing at right angles to each other. Fig. 4.7 illustrates such an arrangement an automobile radiator is a good example of cross-flow exchanger. These exchangers are 'mixed' or 'unmixed' depending upon the mixing or not mixing of either fluid in the direction transverse to the direction of the flow stream and the analysis of this type of heat exchanger is extremely complex because of the variation in the temperature of the fluid in and normal to the direction of flow.

(d) Condenser and Evaporator - In a condenser, the condensing fluid temperature remains almost constant throughout the exchanger and temperature of the colder fluid gradually increases from the inlet to the exit, Fig. (a). In an evaporator, the temperature of the hot fluid gradually decreases from the inlet to the outlet whereas the temperature of the colder fluid remains the same during the evaporation process, Fig. (b). since the temperature of one of the fluids can be treated as constant, it is immaterial whether the exchanger is parallel flow or counter flow.

(e) Compact Heat Exchangers - these devices have close arrays of finned tubes or plates and are typically used when at least one of the fluids is a gas. The tubes are either flat or circular as shown in Fig. 4.8 and the fins may be flat or circular. Such heat exchangers are used to achieve a very large (≥ 700 m^2/mJ) heat transfer surface area per unit volume. Flow passages are typically small and the flow is usually laminar.

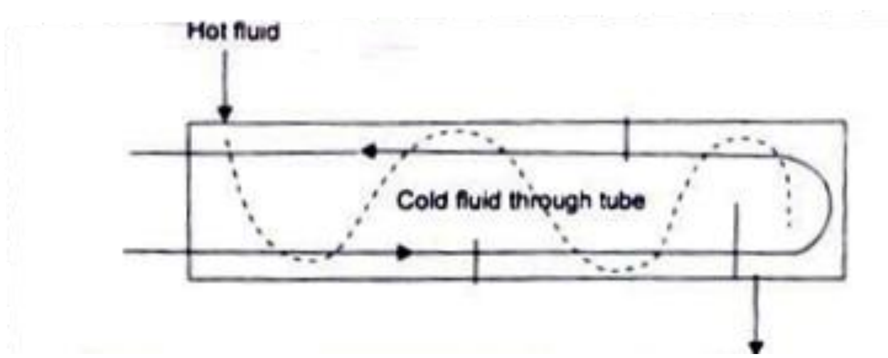


Fig 4.5: multi pass exchanger one shell pass, two tube pass

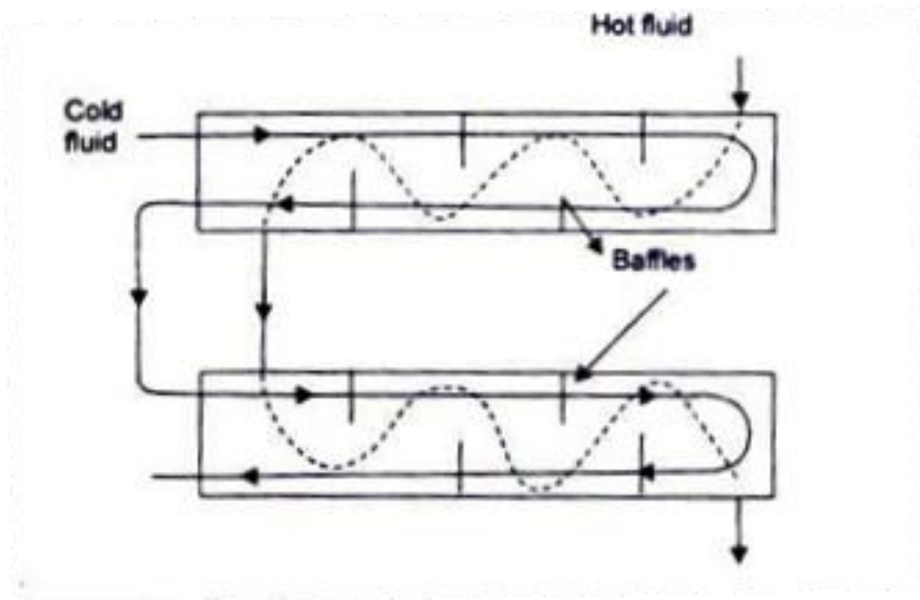


Fig 4.6: Two shell passes, four-tube passes heat exchanger (baffles increases the convection coefficient of the shell side fluid by inducing turbulence and a cross flow velocity component)

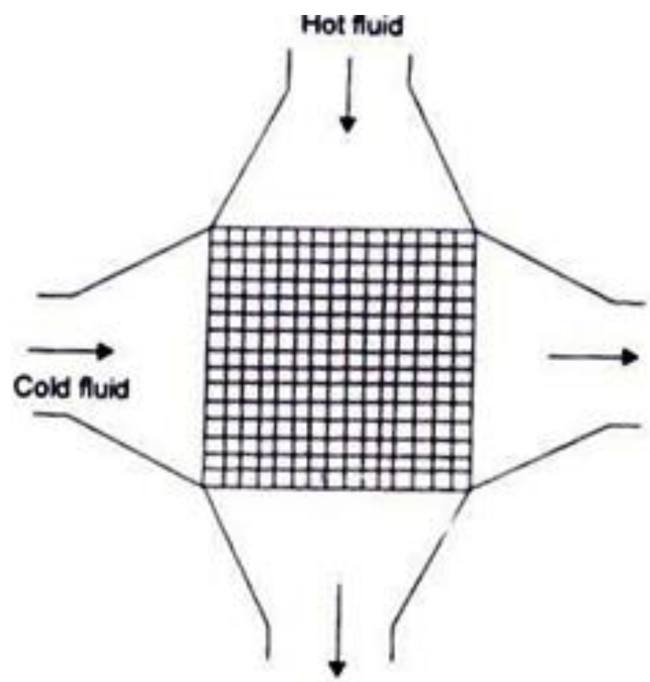
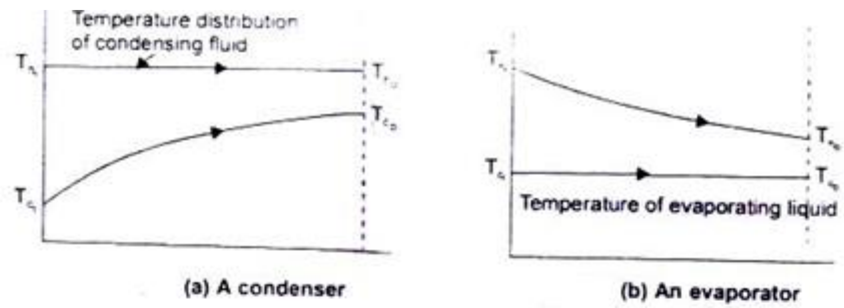


Fig 4.7: A cross-flow exchanger



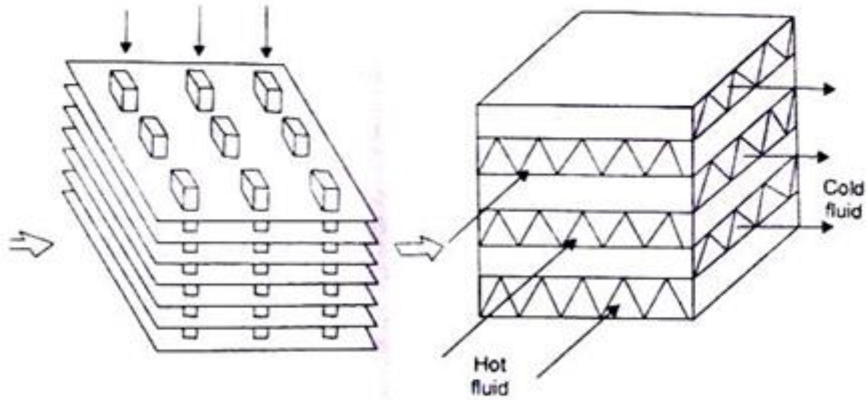


Fig. 4.8 Compact heat exchangers: (a) flat tubes, continuous plate fins, (b) plate fin (single pass)

4.3. Expression for Log Mean Temperature Difference - Its Characteristics

Fig. 4.9 represents a typical temperature distribution which is obtained in heat exchangers. The rate of heat transfer through any short section of heat exchanger tube of surface area dA is: $dQ = U\,dA(T_h - T_c) = U\,dA\,\Delta T$. For a parallel flow heat exchanger, the hot fluid cools and the cold fluid is heated in the direction of increasing area. therefore, we may write

$d\dot{Q} = -\dot{m}_h\ c_h\ dT_h = \dot{m}_c\ c_c\ dT_c$ and $d\dot{Q} = -\dot{C}_h\,dT_h = \dot{C}_c dT_c$ where $\dot{C} = \dot{m} \times c$, and is called the 'heat capacity rate.'

$$\text{Thus,}\quad d(\Delta T) = d(T_h - T_c) = dT_h - dT_c = -(1/C_h + 1/C_c)\,d\dot{Q} \qquad (4.1)$$

For a counter flow heat exchanger, the temperature of both hot and cold fluid decreases in the direction of increasing area, hence

$$d\dot{Q} = -\dot{m}_h\ c_h\ dT_h = -\dot{m}_c\ c_c\,dT_c,\ \text{and}\ d\dot{Q} = -C_h\,dT_h = -C_c dT_c$$

or, $d(\Delta T) = dT_h - dT_c = (1/C_h - 1/C_c)d\dot{Q}$ (4.2)

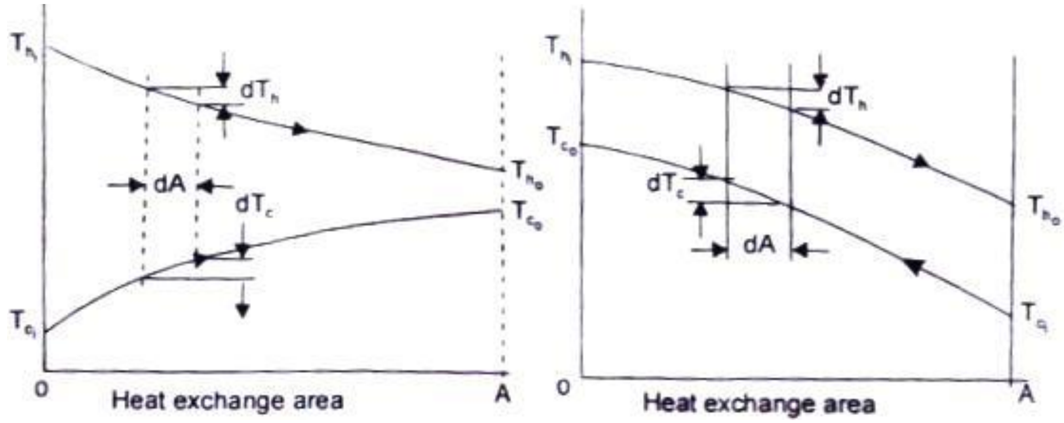


Fig. 4.9 Parallel flow and Counter flow heat exchangers and the temperature distribution with length

Integrating equations (4.1) and (4.2) between the inlet and outlet. and assuming that the specific heats are constant, we get

$$-(1/C_h \pm 1/C_c)\dot{Q} = \Delta T_o - \Delta T_i \quad (4.3)$$

The positive sign refers to parallel flow exchanger, and the negative sign to the counter flow type. Also, substituting for dQ in equations (4.1) and (4.2) we get

$$-(1/C_h \pm 1/C_c)UdA = d(\Delta T)/\Delta T \quad (4.4)$$

Upon integration between inlet i and outlet 0 and assuming U as a constant,

We have $-(1/C_h \pm 1/C_c)U\ A = \ln\left(\Delta T_0/\Delta T_i\right)$

By dividing (4.3) by (4.4), we get

$$\dot{Q} = UA\left[(\Delta T_o - \Delta T_i)/\ln(\Delta T_o/\Delta T_i)\right] \quad (4.5)$$

Thus the mean temperature difference is written as

Log Mean Temperature Difference,

$$\text{LMTD} = (\Delta T_0 - \Delta T_i)/\ln(\Delta T_o/\Delta T_i) \quad (4.6)$$

(The assumption that U is constant along the heat exchanger is never strictly true but it may be a good approximation if at least one of the fluids is a gas. For a gas, the physical properties do not vary appreciably over moderate range of temperature and the resistance of the gas film is considerably higher than that of the metal wall or the liquid film, and the value of the gas film resistance effectively determines the value of the overall heat transfer coefficient U.)

It is evident from Fig.4.9 that for parallel flow exchangers, the final temperature of fluids lies between the initial values of each fluid whereas m counter flow exchanger, the temperature of the colder fluid at exit is higher than the temperature of the hot fluid at exit. Therefore, a counter flow exchanger provides a greater temperature range, and the LMTD for a counter flow exchanger will be higher than for a given rate of mass flow of the two fluids and for given temperature changes, a counter flow exchanger will require less surface area.

4.4. Special Operating Conditions for Heat Exchangers

(i) Fig. 4.9 shows temperature distributions for a heat exchanger (condenser) where the hot fluid has a much larger heat capacity rate, $\dot{C}_h = m_h c_h$ than that of cold fluid, $\dot{C}_c = \dot{m}_c c_c$ and therefore, the temperature of the hot fluid remains almost constant throughout the exchanger and the temperature of the cold fluid increases. The LMTD, in this case is not affected by whether the exchanger is a parallel flow or counter flow.

(ii) Fig. 4.9 shows the temperature distribution for an evaporator. Here the cold fluid expenses a change in phase and remains at a nearly uniform temperature$\left(\dot{C}_c \to \infty\right)$. The same effect would be achieved without phase change if $\dot{C}_c >> \dot{C}_h$, and the LMTD will remain the same for both parallel flow and counter flow exchangers.

(iii) In a counter flow exchanger, when the heat capacity rate of the fluids are equal, $\dot{C}_c = \dot{C}_h$, the temperature difference is the same all along the length of the tube. And in that case, LMTD should be replaced by $\Delta T_a = \Delta T_b$, and the temperature profiles of the two fluids along Its length would be parallel straight lines.

(Since $d\dot{Q} = -\dot{C}_c dT_c = -\dot{C}_h \; dT_h$; $dT_c = -d\dot{Q}/\dot{C}_c$, and $dT_h = -d\dot{Q}/\dot{C}_h$

and, $dT_c - dT_h = d\theta = -dQ\left(1/\dot{C}_c - 1/\dot{C}_h\right) = 0$ (because $\dot{C}_c = \dot{C}_h$)

Or, dθ = 0, the temperature profiles of the two fluids along Its length would be parallel straight lines.)

4.5. LMTD for Cross-flow Heat Exchangers

LMTD given by Eq (4.6) is strictly applicable to either parallel flow or counter flow exchangers. When we have multipass parallel flow or counter flow or cross flow exchangers, LMTD is first calculated for single pass counter flow exchanger and the mean temperature difference is obtained by multiplying the LMTD with a correction factor F which takes care of the actual flow arrangement of the exchanger. Or,

$$\dot{Q} = U\,A\,F\,(LMTD) \qquad (4.7)$$

The correction factor F for different flow arrangements are obtained from charts given in Fig. 4.10 (a, b, c, d).

4.6. Fouling Factors in Heat Exchangers

Heat exchanger walls are usually made of single materials. Sometimes the walls are bimetallic (steel with aluminum cladding) or coated with a plastic as a protection against corrosion, because, during normal operation surfaces are subjected to fouling by fluid impurities, rust formation or other reactions between the fluid and the wall material. The deposition of a film or scale on the surface greatly increases the resistance to heat transfer between the hot and cold fluids. And, a scale coefficient of heat transfer h, is defined as:

$R_s = 1/h_s A,\ ^oC/W$ or K/W

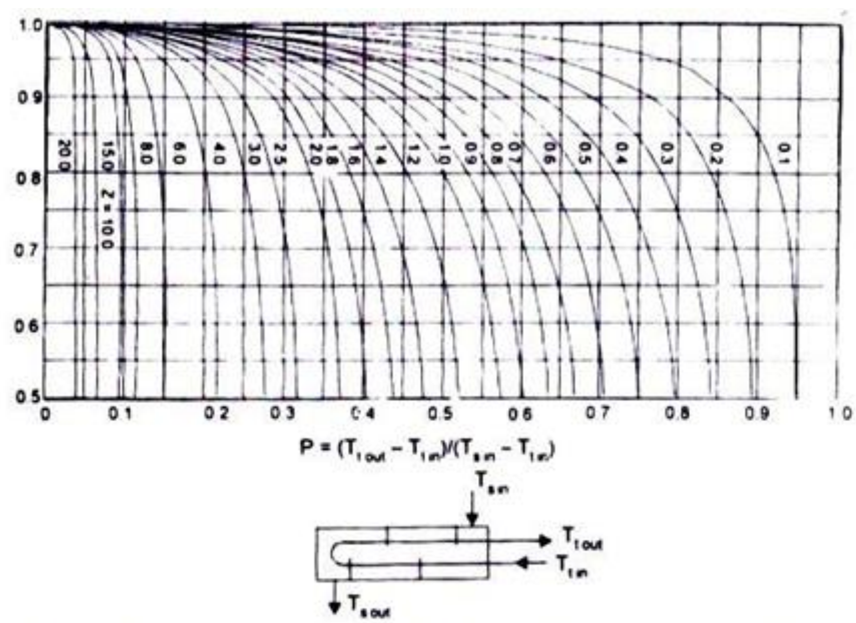


Fig 4.10(a) correctio factor to counter flow LMTD for heat exchanger with one shell pass andtwo, or a muliple of two,tube passes

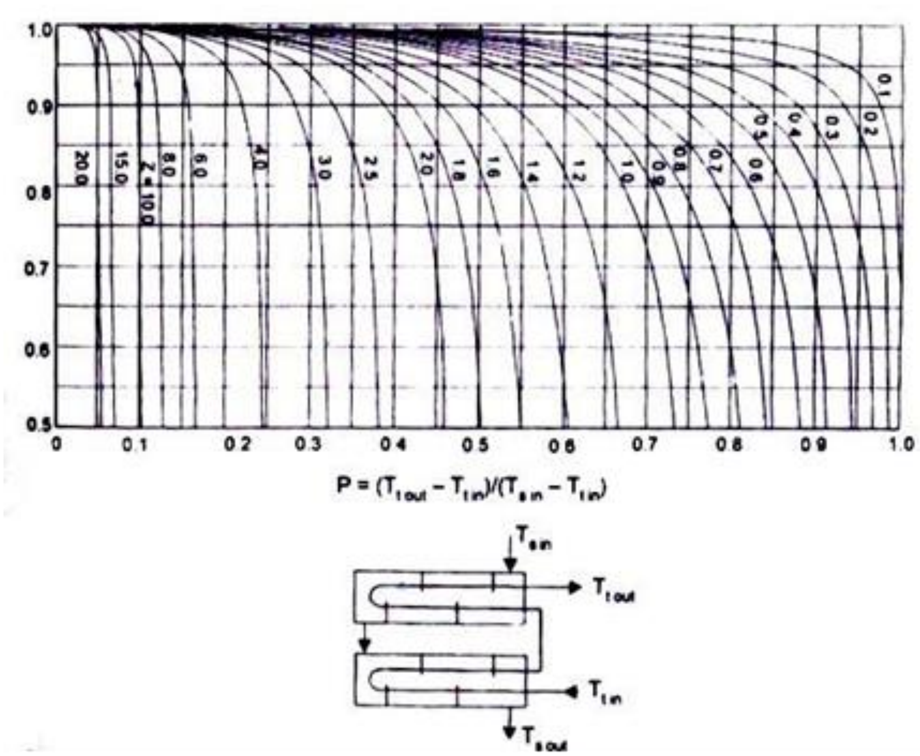


Fig 4.10 (b) Correction factor to counter flow LMTD for heat exchanger with two shell passes and a multiple of two tube passes

where A is the area of the surface before scaling began and l/h_s, is called ‘Fouling Factor'. Its value depends upon the operating temperature, fluid velocity, and length of service of the heat exchanger. Table 4.1 gives the magnitude of l/h, recommended for inclusion in the overall heat transfer coefficient for calculating the required surface area of the exchanger

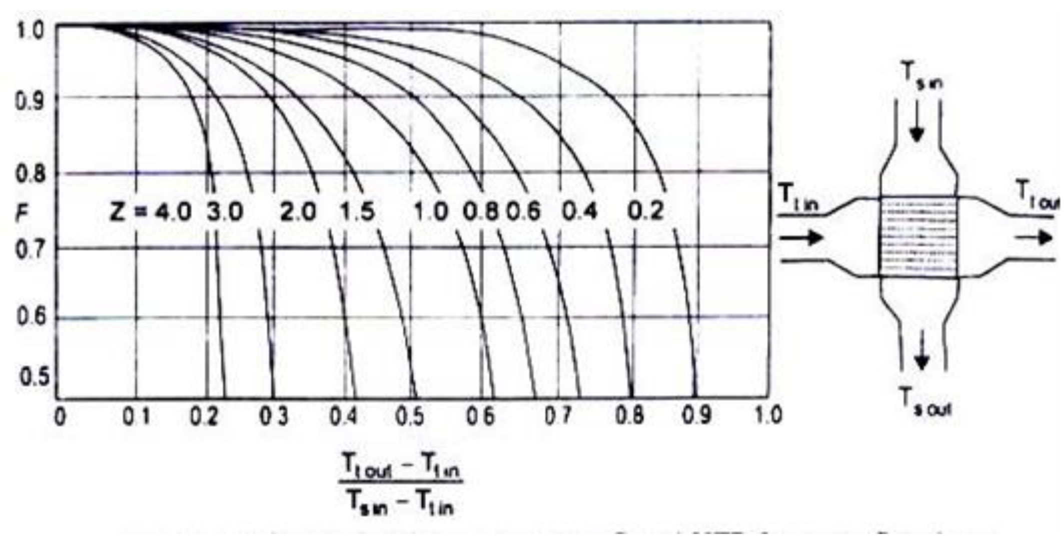


Fig4.10(c) Correction factor to counter flow LMTD for cross flow heat exchangers, fluid on shell side mixed, other fluid unmixed one tube pass..

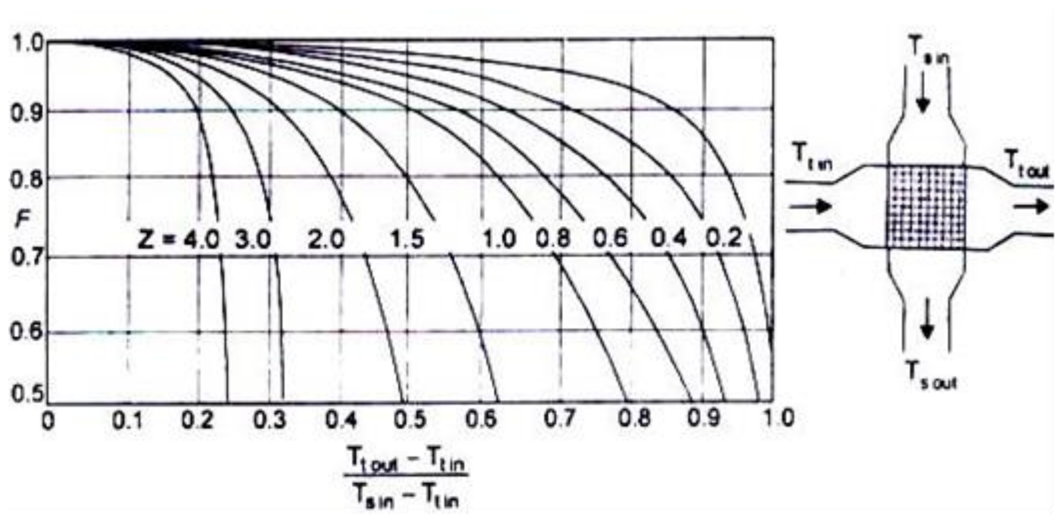


Fig. 4.10 (d) Correction factor to counter flow LMTD for cross flow heat exchangers, both fluids unmixed, one tube pass..

Table 4.1 Representative fouling factors ($1/h_s$)

Type of fluid	Fouling factor	Type of fluid	Fouling Factor
Sea water below 50°C	000009 m'K/W	Refrigerating liquid	0.0002 m'K/W

above 50°C	0.002		
Treated feed water	0.0002	Industrial air	0.0004
Fuel oil	0.0009	Steam, non-oil-bearing	0.00009
Quenching oil	0.0007	Alcohol vapours	0.00009

However, fouling factors must be obtained experimentally by determining the values of U for both clean and dirty conditions in the heat exchanger.

4.7 The Overall Heat Transfer Coefficient

The determination of the overall heat transfer coefficient is an essential, and often the most uncertain, part of any heat exchanger analysis. We have seen that if the two fluids are separated by a plane composite wall the overall heat transfer coefficient is given by:

$$1/U = (1/h_i) + (L_1/k_1) + (L_2/k_2) + (1/h_o) \quad (4.8)$$

If the two fluids are separated by a cylindrical tube (inner radius r_i, outer radius r_0), the overall heat transfer coefficient is obtained as:

$$1/U_i = (1 + h_i) + (r_i/k)\ln(r_o/r_i) + (r_i/r_o)(1/h_o) \quad (4.9)$$

where h_i, and h_o are the convective heat transfer coefficients at the inside and outside surfaces and V, is the overall heat transfer coefficient based on the inside surface area. Similarly, for the outer surface area, we have:

$$1/U_o = (1/h_o) + (r_o/k)\ln(r_o + r_i) + (r_o + r_i)(1/h_i) \quad (4.10)$$

and U_iA_i will be equal to U_oA_o; or, $U_ir_i = U_or_o$.

The effect of scale formation on the inside and outside surfaces of the tubes of a heat exchanger would be to introduce two additional thermal resistances to the heat flow path. If h_{si} and h_{so} are the two heat transfer coefficients due to scale formation on the inside and outside surface of the inner pipe, the rate of heat transfer is given by

$$Q = (T_i - T_o)/\left[(1/h_iA_i) + 1/h_{si}A_i + \ln(r_o + r_i)/2\pi Lk + 1/h_{so}A_o + (1/h_oA_o)\right] \quad (4.11)$$

where T_i, and T_o are the temperature of the fluid at the inside and outside of the tube. Thus, the overall heat transfer coeffiCIent based on the inside and outside surface area of the tube would be:

$$1/U_i = 1/h_i + 1/h_{si} + (r_i/k)\ln(r_o/r_i) + (r_i/r_o)(1/h_{so}) + (r_i/r_o)(1/h_o); \qquad (4.12)$$

and

$$1/U_o = (r_o/r_i)(1/h_i) + (r_o/r_i)(1/h_{si}) + \ln(r_o/r_i)(r_0/k) + 1/h_{so} + 1/h_o$$

Example 4.1 In a parallel flow heat exchanger water flows through the inner pipe and is heated from 25°C to 75°C. Oil flowing through the annulus is cooled from 210°C to 110°C. It is desired to cool the oil to a lower temperature by increasing the length of the tube. Estimate the minimum temperature to which the oil can be cooled.

Solution: By making an energy balance, heat received by water must be equal to 4he heat given out by oil.

$$\dot{m}_w c_w (75 - 25) = \dot{m}_o c_o (210 - 110); \dot{C}_w / \dot{C}_o = 100/50 = 2.0$$

In a parallel flow heat exchanger, the minimum temperature to which oil can be cooled will be equal to the maximum temperature to which water can be heated,

Fig. 10.2: $(T_{ho} = T_{co})$

therefore, C_w (T –25) = C_o (210 – T);

(T – 25)/(210 – T) = 1/2 = 0.5; or, T = 260/3 = 86.67°C.

or the same capacity rates the oil can be cooled to 25°C (equal to the water inlet temperature) in a counter-flow arrangement.

Example 4.2 Water at the rate of 1.5 kg/s IS heated from 30°C to 70°C by an oil (specific heat 1.95 kJ/kg C). Oil enters the exchanger at 120°C and leaves the exchanger at 80°C. If the overall heat transfer coefficient remains constant at 350 W /m^2°C, calculate the heat exchange area for (i) parallel-flow, (ii) counter-flow, and (iii) cross-flow arrangement.

Solution: Energy absorbed by water,

$$\dot{Q} = \dot{m}_w c_w (\Delta T) = 1.5 \times 4.182 \times 40 = 250.92 \text{ kW}$$

(i) Parallel flow: Fig. 10.9; $\Delta T_a = 120 - 30 = 90$; $\Delta T_b = 80 - 70 = 10$

LMTD = (90 – 10)/ln(90/10) = 36.4;

Area = Q/U (LMTD) = 250920 /(350 × 36.4) = 19.69 m^2.

(ii) Counter flow: Fig 10.9; ΔT_a. = 120 – 70 = 50, ΔT_b = 80 – 30 = 50

Since ΔT_a. = ΔT_b,. LMTD should be replaced by ΔT = 50

Area A = $\dot{Q}/U(\Delta T) = 250920/(350 \times 50) = 14.33\ m^2$

(iii) Cross flow: assuming both fluids unmixed - Fig. 10.10d

using the nomenclature of the figure and assuming that water flows through the tubes and oil flows through the shell,

$$P = (T_{to} - T_{ti})/(T_{si} - T_{ti}) = (70 - 30)/(120 - 30) = 0.444$$

$$Z = (T_{si} - T_{so})/(T_{to} - T_{ti}) = (120 - 80)/(70 - 30) = 1.0$$

and the correction factor, F = 0.93

$\dot{Q} = UAF(\Delta T)$; or Area A = 250920/(350 × 0.93 × 50) = 15.41 m^2.

Example 4.3 0.5 kg/s of exhaust gases flowing through a heat exchanger are cooled from 400°C to 120°C by water initially at 25°C. The specific heat capacities of exhaust gases and water are 1.15 and 4.19 kJ/kgK respectively, and the overall heat transfer coefficient from gases to water is 150 W/m^2K. If the cooling water flow rate is 0.7 kg/s, calculate the surface area when (i) parallel-flow (ii) cross-flow with exhaust gases flowing through tubes and water is mixed in the shell.

Solution: The heat given out by the exhaust gases is equal to the heat gained by water.

or, 0.5 × 1.15 × (400 – 120) = 0.7 × 4.19 × (T – 25)

Therefore, the temperature of water at exit, T = 79.89°C

For paralle1-flow: $\Delta T_a = 400 - 25 = 375; \quad \Delta T_b = 120 - 79.89 = 40.11$

LMID = (375 – 40.11)/ln(375/40.11) = 149.82

$\dot{Q}$ = 0.5 × 1.15 × 280 = 161000 W;

Therefore Area A = 161000/(150 × 149.82) = 7.164 m^2

For cross-flow: $\dot{Q}$ = U A F (LMTD);

and LMTD is calculated for counter-flow system.

$$\Delta T_a = (400 - 79.89) = 320.11; \ \Delta T_b = 120 - 25 = 95$$

$$\text{LMTD} = (320.11 - 95)/\ln(320.11/95) = 185.3$$

Using the nomenclature of Fig 10.10c,

$$P = (120 - 400)/(25 - 400) = 0.747$$

$$Z = (25 - 79.89)/(120 - 400) = 0.196 \qquad \therefore \ F = 0.92$$

and the area $A = 161000/(150 \times 0.92 \times 185.3) = 6.296 \text{ m}^2$

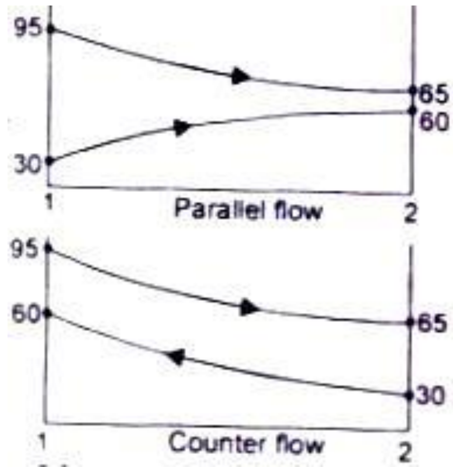
Example 4.4 In a certain double pipe heat exchanger hot water flows at a rate of 5000 kg/h and gets cooled from 95°C to 65°C. At the same time 5000 kg/h of cooling water enters the heat exchanger. The overall heat transfer coefficient is 2270 W/m^2K. Calculate the heat transfer area and the efficiency assuming two streams are in (i) parallel flow (ii) counter flow. Take C_p for water as 4.2 kJ/kgK, cooling water inlet temperature 30°C.

Solution: By making an energy balance:

Heat lost by hot water = 5000 ×4.2 ×(95 –65)

= heat gained by cold water = 5000 ×4.2 ×(T –30) 30

T = 60^oC



(i) Parallel now

$$\theta_1 = (95 - 30) = 65$$

$$\theta_2 = (65 - 60) = 5$$

$$\text{LMTD} = (65 - 5)/\ln(65/5) = 23.4$$

$$\text{Area, } A = \dot{Q}/(U \times \text{LMTD}) = \frac{500 \times 4.2 \times 10^3 \times 30}{3600 \times 2270 \times 23.4} = 3.295 \text{ m}^2$$

(ii) Counter flow: $\theta_1 = (95 - 60) = 35$

$\theta_2 = (65 - 30) = 35$

LMTD = ΔT = 35

Area A = 500 × 4200 × 30/(3600 × 2270 × 35) = 2.2 m^2

$\in$, Efficiency.= Actual heat transferred/Maximum heat that could be transferred. Therefore, for parallel flow, $\in$ = (95 – 65)/(95 – 60) = 0.857

For counter flow, $\in$ = (95 - 65)/(95 – 30) = 0.461.

Example 4.5 The flow rates of hot and cold water streams running through a double pipe heat exchanger (inside and outside diameter of the tube 80 mm and 100 mm) are 2 kg/s and 4 kg/so The hot fluid enters at 75°C and comes out at 45°C. The cold fluid enters at 20°C. If the convective heat transfer at the inside and outside surface of the tube is 150 and 180 W /m^{2}K, thermal conductivity of the tube material 40 W/mK, calculate the area of the heat exchanger assuming counter flow.

Solution: Let T is the temperature of the cold water at outlet.

By making an energy balance, $\dot{Q} = \dot{m}_h c_h (T_{h1} - T_{h2}) = \dot{m}_c c_c (T_{c2} - T_{c1})$

since $c_h = c_c$, 4.2 kJ / kgK; $2 \times (75 - 45) = 4 \times (T - 20)$; $T = 35^o C$

and $\dot{Q} = 252$ kW

for counter flow: $\theta_1 = (75 - 35) = 40$; $\theta_2 = (45 - 20) = 25$

LMTD = (40 – 25) / ln (40/25) = 31.91

overall heat transfer coefficient based in the inside surface of tube

$$1/U = (1/h_i) + (r_i / k)\ln(r_o / r_i) + (r_o r_i)(1/h_o)$$

$$= 1/150 + (0.04/40)\ln(50/40) + (50/40)(1/180) = 0.0138$$

and U = 72.28

area $A = \dot{Q}/(U \times LMTD) = 252 \times 10^3 / (72.28 \times 31.91) = 109.26\ m^2$

Example 4.6 Water flows through a copper tube (k = 350 W/mK, inner and outer diameter 2.0 cm and 2.5 cm respectively) of a double pipe heat exchanger. Oil flows through the annulus between this pipe and steel pipe. The convective heat transfer coefficient on the inside and outside of the copper tube are 5000 and 1500 W /m^2 K. The fouling factors on the water and oil sides are 0.0022 and 0.00092 K W^{-1}. Calculate the overall heat transfer coefficient with and without the fouling factor.

Solution: The scales formed on the inside and outside surface of the copper tube introduces two additional resistances in the heat flow path. Resistance due to inside convective heat transfer coefficient

$$1/h_i A_i = 1/5000\ A_i$$

Resistance due to scale formation on the inside = $1/h_sA_i$ = 0.0022

Resistance due to conduction through the tube wall = $\ln(r_o / r_i) / 2\pi Lk$

$= \ln(2.5 / 2.0) / 2\pi \times L \times 350 = 1.014 \times 10^{-4} / L$

Resistance due to convective heat transfer on the outside

$$1/h_oA_o = 1/1500A_o$$

Resistance due to scale formation on the outside = $1/h_sA_o = 0.00092$

Since, $Q = \Delta T \sum R = U_iA_1(\Delta T) = \Delta T / (1/U_iA_i)$; we have

(a) With fouling factor:-

Overall heat transfer coefficient based on the inside pipe surface

$$U_i = 1/\left(1/5000 + \pi \times 0.02(0.0022 + 0.00092) + 0.02\pi \times 1.014 \times 10^{-4} + 8.33 \times 10^{-4}\right)$$

= 809.47 W/m^{2}K per metre length of pipe

(b) Without fouling factor

$$U_i = 1/\left(1/5000 + 0.02\pi \times 1.014 \times 10^{-4} + 8.33 \times 10^{-4}\right)$$

= 962.12 W/m^{2}K per m of pipe length.

The heat transfer rate will reduce by (962.12 – 809.47)1962.12 = 15.9 percent when fouling factor is considered.

Example 4.7 In a surface condenser, dry and saturated steam at 50^{o}C enters at the rate of 1 kg/s The circulating water enters the tube, (25 mm inside diameter, 28 mm outside diameter, k = 300 W/mK) at a velocity of 2 m/s. If the convective heat transfer coefficient on the outside surface of the tube is 5500 W/m^{2}K, the inlet and outlet temperatures of water are 25^{o}C and 35^{o}C respectively, calculate the required surface area.

Solution: For calculating the convective heat transfer coefficient on the inside surface of the tube, we calculate the Reynolds number on the basis of properties of water at the mean temperature of 30^{o}C. The properties are:

μ = 0.001 Pa-s, ρ = 1000 kg/m^3, k = 0.6 W/mK, h_{fg} at 50^{o}C = 2375 kJ/kg

Re = $\rho VD/\mu$ = $10^3 \times 2 \times 0.025/0.001$ = 50,000, a turbulent flow. Pr = 7.0.

The heat transfer coefficient at the inside surface can be calculated by:

Nu = 0.023 Re$^{0.8}$ 8 Pr$^{0.3}$ = 0.023 (50000)$^{0.8}$ (7)$^{0.3}$ = 236.828

and h_i = 236.828 × 0.6/0.025 = 5684 W/m^{2}K.

The overall heat transfer coefficient based on the outer diameter,

U = 1/(0.028/(0.025 × 5684) + 1/5500 + 0.014 ln(28/25)/300)

= 2603.14 W/m^{2}K

ΔT_a. = (50 - 25) = 25; ΔT_b = (50 - 35) = 15;

LMTD = (25 -15)/ln(25/15) = 19.576.

Assuming one shell pass and one tube pass, Q = UA (LMTD)

or A = 2375 × 10^3/(2603.14 x 19.576) = 46.6 m^2

Mass of Circulating water = Q/(c_p ΔT) = 2375/(4.182 × 10) = 56.79 kg/s

also, m_w = ρ× area × V × n, where n is the number of tubes.

n = 56.79 × 4/(2 × Л × 0.025 × 0.025 × 1000) = 58 tubes

Surface area, 46.6 = n × Л × d × L

and L = 46.6/(58 × Л × 0.025) = 10.23 m.

Hence more than one pass should be used.

Example 4.8 A heat exchanger is used to heat water from 20°C to 50°C when thin walled water tubes (inner diameter 25 mm, length IS m) are laid beneath a hot spring water pond, temperature 75°C. Water flows through the tubes with a velocity of 1 m/s. Estimate the required overall heat transfer coefficient and the convective heat transfer coefficient at the outer surface of the tube.

Solution: Water flow rate, $\dot{m}$ = $\rho \times V \times A$ = 10^3 × 1 × (Л/4) (0.025)^2

= 0.49 kg/s

Heat transferred to water, Q = $\dot{m}\,c\,(\Delta T)$ = 0.49 × 4200 × 30 = 61740 W.

Since the temperature of the water in the hot spring is constant,

$\theta_1 = (75 - 20) = 55;\ \ \theta_2 = (75 - 50) = 25\,;$

LMTD = (55 – 25) / ln(55/25) = 38

Overall heat transfer coefficient, U = Q/ (A × LMTD)

= 61740/(38 × Л × 0.025 × 15) = 1378.94 W/m^{2}K.

The properties of water at the mean temperature (20 + 50)/2 = 35°C are:

$\mu = 0.001\ \mathrm{Pa-s},\ \ k = 0.6\ \mathrm{W/mk}$ and Pr = 7.0

Reynolds number, $\mathrm{Re} = \rho V d / \mu = 1000 \times 1.0 \times 0.25 / 0.001 = 25000$, turbulent flow.

Nu = 0.023 (Re)$^{0.8}$ (Pr)$^{0.33}$ = 0.023 (25000)$^{0.8}$ × (7)$^{0.33}$ = 144.2

and h_i = 144.2 × k/d = 144.2 × 0.6/0.025 = 3460.8 W/m^{2}K

Neglecting the resistance of the thin tube wall,

1/U = 1/h_i + 1/h_o; $\therefore\ 1/h_o = 1/1378.94 = 1/3460.8$

or, $h_o = 2292.3\ \mathrm{W/m^2K}$

Example 4.9 A hot fluid at 200°C enters a heat exchanger at a mass rate of 10000 kg/h. Its specific heat is 2000 J/kg K. It is to be cooled by another fluid entering at 25°C with a mass flow rate 2500 k g/h and specific heat 400 J/kgK. The overall heat transfer coefficient based on outside area of 20 m2 is 250 W/m^{2}K. Find the exit temperature of the hot fluid when the fluids are in parallel flow.

Solution: From Eq(10.3a), $-U\ dA\left(1/C_h + 1/C_c\right) = d\left(\Delta T\right)/\Delta T$

Upon integration,

$$-U\,A\left(1/C_h + 1/C_c\right) = \ln\left(\Delta T\right)\Big|_1^2 \ = \ln\left(T_{h_0} - T_{c_0}\right)/\left(T_{h_i} - T_{c_i}\right)$$

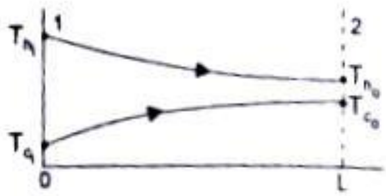
The values are: U = 250 W/m^{2}K

$A = 20\ m^2$

$l/C_h = 3600/(10000 \times 2000) = 1.8 \times 10^{-4}$

$1/C_c = 3600/(2500 \times 400) = 3.6 \times 10^{-3}$

$-UA(1/C_h + l/C_c) = -250 \times 20\ (1.8 \times 10^{-4} + 3.6 \times 10^{-3}) = -18.9$



; or, $T_{h_0} = T_{c_0}$

By making an energy balance,

$$10000 \times 2000\left(200 - T_{h_0}\right) = 2500 \times 400\left(T_{c_0} - 25\right)$$

$$= 2500 \times 400\left(T_{h_0} - 25\right) \text{ and } 21\ T_{h_0} = 20 \times 200 + 25$$

or, $T_{h_0} = 191.67^{o}C$

Example 4.10 Cold water at the rate of 4 kg/s is heated from 30°C to 50°C in a shell and tube heat exchanger with hot water entering at 95°C at a rate of 2 kg/s. The hot water flows through the shell. The cold water flows through tubes 2 cm inner diameter, velocity of flow 0.38 m/s. Calculate the number 0 f tube passes, the number 0 f tubes per pass if the maximum length of the tube is limited to 2.0 m and the overall heat transfer coefficient is 1420 W/m^2K.

Solution: Let T be the temperature of the hot water at exit. By making an energy balance: $4c\left(50 - 30\right) = 2c\left(95 - T\right)$; $\therefore T = 55^{o}C$

For a counter-flow arrangement:

$$\Delta T_a = \left(95 - 50\right) = 45, \quad \Delta T_b = \left(55 - 30\right) = 25,$$

$\therefore$ $\text{LMTD} = (45-25)/\ln(45/25) = 34$; $Q = mC(\Delta T) = 4 \times 4.182 \times 20 = 334.56$ kW

Since the cold water is flowing through the tubes, the number of tubes, n is given by

$\dot{m} = n \times \rho \times$ Area × velocity; the cross-sectional area 3.142 × 10^{-2} m^2

4 = n × 1000 × 3.142 × 10^{-4} × 0.38; n = 33.5, or 34 (say)

Assuming one shell and two tube pass, we use Fig. 10.9(a).

$P(50-30)/(95-30) = 0.3$; $Z = (95-55)/(50-30) = 2.0$

Therefore, the correction factor, F = 0.88

Q = UAF LMTD; 34560 = 1420 × A × 0.88 × 34; or A = 7.875 m^2.

For 2 tube pass, the surface area of 34 tubes per pass = 2 L Л d 34

L = 1.843 m

Thus we will have 1 shell pass, 2 tube; 34 tubes of 1.843 m in length.

Example 4.11 A double pipe heat exchanger is used to cool compressed air (pressure A bar, volume flow rate 5 m^3/min at I bar and 15°C) from 160°C to 35°C. Air flows with a velocity of 5 m/s through thin walled tubes, 2 cm inner diameter. Cooling water flows through the annulus and its temperature rises from 25°C to 40°C. The convective heat transfer coefficient at the inside and outside tube surfaces are 125 W/m^2K and 2000 W/m^2K respectively. Calculate (i) mass of water flowing through the exchanger, and (ii) number of tubes and length of each tube.

Solution: Air is cooled from 1600C to 35°C while water is heated from 25°C to 400C and therefore this must be a counter flow arrangement.

Temperature difference at section 1 : $\left(T_{h_i} - T_{c_0}\right) = (160-40) = 120$

Temperature difference at section 2 : $\left(T_{h_0} - T_{c_i}\right) = (35-25) = 10$

LMTD = (120 –10)/ln 120/10) = 44.27

Mass of air flowing, $\dot{m}$ = ρ × Volume flow rate = $\left(10^5/287 \times 288\right)(5/60) = 0.1$ kg/s

Heat given out by air = Heat taken in by water,

$\therefore\ 0.1 \times 1.005 \times (160 - 35) = \dot{m}_w \times 4.182 \times (40 - 25)$; Or $\dot{m}_w = 0.20\ kg/s$

Density of air flowing through the tube, ρ = p/RT. The mean temperature of air flowing through the tube is (160 + 35)/2 = 97.5 ^oC = 370.5K

$\rho = 4 \times 10^5/(287 \times 370.5) = 3.76\ kg/m^3$. If n is the number of tubes, from the conservation of mass, $\dot{m} = \rho AV$; 0.1 = 3.76 × (Л/4) $(0.02)^2$ × 5 × n

n = 16.9 ≡ 17 tubes; $\dot{Q}$ = UA (LMTD)

U = 1/(1/2000 + 1/125) = 117.65, Area for heat transfer A = ЛDLn

Q = UA(LMTD); 0.1 × 1005 × 125 = 117.65 × 3.142 × 0.02 × L × 17 × 44.27 and L = 2.26 m.

Example 4.12 A refrigerant (mass rate of flow 0.5 kg/s, S = 907 J/kgK k = 0.07 W/mK, μ = 3.45 × 10^{-4} Pa-s) at –20°C flows through the annulus (inside diameter 3 c m) of a double pipe counter flow heat exchanger used to cool water (mass flow rate 0.05 kg/so k = 0.68 W/mK, μ = 2.83 × 10^{-4} Pa-s) at 98°C flowing through a thin walled copper tube of 2 cm inner diameter. If the length of the tube is 3m, estimate (i) the overall heat transfer coefficient, and (ii) the temperature of the fluid streams at exit.

Solution: Mass rate of flow, $\dot{m} = \rho AV = \rho(\pi/4)D^2V$;

$\rho VD = 4\dot{m}/\pi D$ and, Reynolds number, $Re = \rho VD/\mu = 4\dot{m}/\pi D\mu$

Water is flowing through the tube of diameter 2 cm,

$\therefore Re = 4 \times 0.05/\left(3.142 \times 0.02 \times 2.83 \times 10^{-4}\right) = 1.12 \times 10^4$, turbulent flow.

$$Nu = 0.023\ Re^{0.8}(Pr)^{0.33} = 0.023\left(1.12 \times 10^4\right)^{0.8}(1.8)^{0.33}$$

= 48.45; and $h_i = Nu \times k/D = 48.45 \times 0.68/0.02 = 1647.3\ W/m^2K$

Refrigerant is flowing through the annulus. The hydraulic diameter is

D_o - D_i, and the Reynolds number would be, $Re = 4m/\mu\pi(D_0 + D_i)$

$$Re = 4\times0.5/\left(3.45\times10^{-4}\times3.142\times(0.02+0.03)\right)$$ = 3.69×10^4, a turbulent flow.

$$Nu = 0.023(Re)^{0.8}(Pr)^{0.33},$$

where $Pr = \mu\ c/k = 3.45\times10^{-4}\times907/0.07 = 4.47$

$$= 0.023\left(3.69\times10^4\right)^{0.8}(4.47)^{0.33} = 169.8$$

$$\therefore\ h_o = nu\times k/(D_o - D_i) = 169.8\times0.07/0.01 = 1188.6\ W/m^2K$$

and, the overall heat transfer coefficient, U = 1/(1/1647.3 + 1/1188.6)

= 690.43 W/m^2K

For a counter flow heat exchanger, from Eq. (10.4), we have,

$$(1/C_c - 1/C_h)UA = \ln(\Delta T_0/\Delta T_i) = \ln\left[\left(T_{h_0} - T_{c_i}\right)/\left(T_{h_i} - T_{c_0}\right)\right]$$

$$C_c = 0.5\times907 = 453.5;\ C_h = 0.05\times4182 = 209.1$$

$$1/C_c - 1/C_h UA = (1/453.5 - 1/209.1)\times690.43\times3.142\times0.02\times3 = -0.335$$

$$\therefore\ \left(T_{h_0} - T_{c_i}\right)/\left(T_{h_i} - T_{c_0}\right) = \exp(-0.335) = 0.715$$

or, $\left(T_{h_0} + 20\right)/\left(98 - T_{c_0}\right) = 0.715$; By making an energy balance,

$$453.5\left(T_{c_0} + 20\right) = 209.1\left(98 - T_{h_0}\right)$$

which gives $T_{c_0} = 3.12^oC; T_{h_0} = 47.8^oC$

4.8. Heat Exchangers Effectiveness - Useful Parameters

In the design of heat exchangers, the efficiency of the heat transfer process is very important. The method suggested by Nusselt and developed by Kays and London is now being extensively used. The effectiveness of a heat exchanger is defined as the ratio of the actual heat transferred to the maximum possible heat transfer.

Let $\dot{m}_h$ and $\dot{m}_c$ be the mass flow rates of the hot and cold fluids, c_h and c_c be the respective specific heat capacities and the terminal temperatures be Th and T h for the hot fluid at inlet and outlet, T_{h_i} and T_{h_0} for the cold fluid at inlet and outlet. By making an energy balance and assuming that there is no loss of energy to the surroundings, we write

$$\dot{Q} = \dot{m}_h \; c_h \left(T_{h_i} - T_{h_0}\right) = \dot{C}_h \left(T_{h_i} - T_{c_0}\right), \text{ and}$$

$$= \dot{m}_c c_c \left(T_{c_0} - T_{c_i}\right) = \dot{C}_c \left(T_{c_0} - T_{c_i}\right) \tag{4.13}$$

From Eq. (4.13), it can be seen that the fluid with smaller thermal capacity, C, has the greater temperature change. Further, the maximum temperature change of any fluid would be $\left(T_{h_i} - T_{c_i}\right)$ and this Ideal temperature change can be obtained with the fluid which has the minimum heat capacity rate. Thus,

$$\text{Effectiveness, } \in = \dot{Q} / C_{min} \left(T_{h_i} - T_{c_i}\right) \tag{4.14}$$

Or, the effectiveness compares the actual heat transfer rate to the maximum heat transfer rate whose only limit is the second law of thermodynamics. An useful parameter which also measures the efficiency of the heat exchanger is the 'Number of Transfer Units', NTU, defined as

NTU = Temperature change of one fluid/LMTD.

Thus, for the hot fluid: NTU = $\left(T_{h_i} - T_{h_0}\right) / \text{LMTD}$, and

for the cold fluid: $\text{NTU} = \left(T_{c_0} - T_{c_i}\right) / \text{LMTD}$

Since $\dot{Q} = UA\left(\text{LMTD}\right) = C_h \left(T_{h_i} - T_{h_0}\right) = \dot{C}_c \left(T_{c_0} - T_{c_i}\right)$

we have $\text{NTU}_h = UA / C_h$ and $\text{NTU}_c = UA / C_c$

The heat exchanger would be more effective when the NTU is greater, and therefore,

$$NTU = AU/C_{min} \tag{4.15}$$

An another useful parameter in the design of heat exchangers is the ratio the minimum to the maximum thermal capacity, i.e., $R = C_{min}/C_{max}$,

where R may vary between 1(when both fluids have the same thermal capacity) and 0 (one of the fluids has infinite thermal capacity, e.g., a condensing vapour or a boiling liquid).

4.9. Effectiveness - NTU Relations

For any heat exchanger, we can write: $\in = f\left(\mathrm{NTU}, C_{min}/C_{max}\right)$. In order to determine a specific form of the effectiveness-NTU relation, let us consider a parallel flow heat exchanger for which $C_{min} = C_h$. From the definition of effectiveness (equation 3.24), we get

$$\varepsilon = \left(T_{h_i} - T_{h_0}\right)/\left(T_{h_i} - T_{c_i}\right)$$

and, $C_{min}/C_{max} = C_h/C_c = \left(T_{c_0} - T_{c_i}\right)/\left(T_{h_i} - T_{h_0}\right)$ for a parallel flow heat exchanger, from Equation 3.24,

$$\ln\left(T_{h_0} - T_{c_0}\right)/\left(T_{h_i} - T_{c_i}\right) = -UA\left(1/C_h + 1/C_c\right) = \frac{-UA}{C_{min}}\left(1 + C_{min}/C_{max}\right)$$

or, $\left(T_{h_0} - T_{c_0}\right)/\left(T_{h_i} - T_{c_i}\right) = \exp\left[-\mathrm{NTU}\left(1 + C_{min}/C_{max}\right)\right]$

But, $\left(T_{h_0} - T_{c_0}\right)/\left(T_{h_i} - T_{c_i}\right) = \left(T_{h_0} - T_{h_i} + T_{h_i} - T_{c_0}\right)/\left(T_{h_i} - T_{c_i}\right)$

$$= \left[\left(T_{h_0} - T_{h_i}\right) + \left(T_{h_i} - T_{c_i}\right) - \left\{R\left(T_{h_i} - T_{h_0}\right)\right\}\right]/\left(T_{h_i} - T_{c_i}\right)$$

$$= \in + 1 - R \in = 1 - \in\left(1 + R\right)$$

Therefore, $\in = \left[1 - \exp\left\{-\mathrm{NTU}(1+R)\right\}\right]/\left(1+R\right)$

$$\mathrm{NTU} = -\ln\left[1 - \in\left(1+R\right)\right]/\left(1+R\right)$$

Similarly, for a counter flow exchanger, $\varepsilon = \dfrac{\left[1 - \exp\left\{-\mathrm{NTU}(1-R)\right\}\right]}{\left[1 - R\exp\left\{-\mathrm{NTU}(1-R)\right\}\right]}$;

and, $\text{NTU} = \left[1/\left(\text{R}-1\right)\right] \ln \left[\left(\in -1\right)/\left(\in \text{R}-1\right)\right]$

Heat Exchanger Effectiveness Relation

Flow arrangement	relationship
Concentric tube	
Parallel flow	$\in = \dfrac{1-\exp\left[-\text{N}\left(1+\text{R}\right)\right]}{\left(1+\text{R}\right)}$; $\text{R} = \text{C}_{\min}/\text{C}_{\max}$
Counter flow	$\in = \dfrac{1-\exp\left[-\text{N}\left(1-\text{R}\right)\right]}{1-\text{R}\ \exp\left[-\text{N}\left(1-\text{R}\right)\right]}$; $\text{R} < 1$
	$\in = \text{N}/\left(1+\text{N}\right)$ for $\text{R} = 1$
Cross flow (single pass)	
Both fluids unmixed	$\in = 1-\exp\left[\left(1/\text{R}\right)\left(\text{N}\right)^{0.22}\left\{\exp\left(-\text{R}\left(\text{N}\right)^{0.78}\right)-1\right\}\right]$
C_{max} mixed , C_{min} unmixed	$\in = \left(1/\text{R}\right)\left[1-\exp\left\{-\text{R}\left(1-\exp\left(-\text{N}\right)\right)\right\}\right]$
C_{min} mixed, C_{max} unmixed	$\in = 1-\exp\left[-\text{R}^{-1}\left\{1-\exp\left(-\text{RN}\right)\right\}\right]$
All exchangers (R = 0)	$\in = 1-\exp\left(-\text{N}\right)$

Kays and London have presented graphs of effectiveness against NTU for Various values of R applicable to different heat exchanger arrangements, Fig. 4.11 to Fig. (4.15).

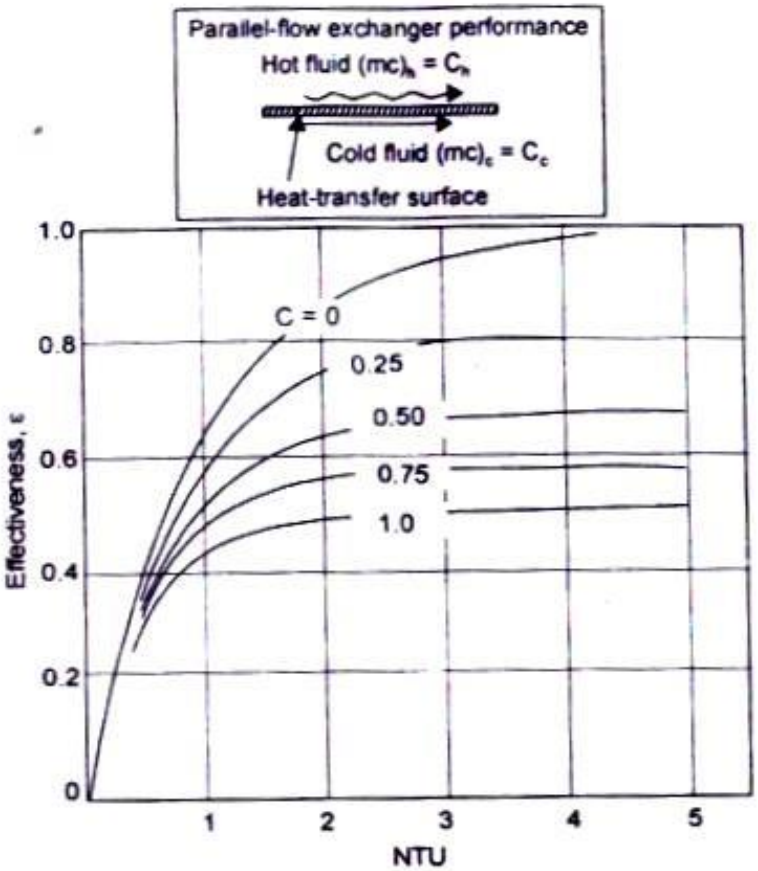


Fig 4.11 Heat exchanger effectiveness for parallel flow

Example 4.13 A single pass shell and tube counter flow heat exchanger uses exhaust gases on the shell side to heat a liquid flowing through the tubes (inside diameter 10 mm, outside diameter 12.5 mm, length of the tube 4 m). Specific heat capacity of gas 1.05 kJ/kgK, specific heat capacity of liquid 1.5 kJ/kgK, density of liquid 600 kg/m^3, heat transfer coefficient on the shell side and on the tube sides are: 260 and 590 W/m^{2}K respectively. The gases enter the exchanger at 675 K at a mass flow rate of 40 kg/s and the liquid enters at 375 K at a mass flow rate of 3 kg/s. If the velocity of liquid is not to exceed 1 m/s, calculate (i) the required number of tubes, (ii) the effectiveness of the heat exchanger, and (iii) the exit temperature of the liquid. Neglect the thermal resistance of the tube wall.

Solution: Volume flow rate of the liquid = 3/600 = 0.005 m/s. For a velocity of 1 m/s through the tube, the cross-sectional area of the tubes will be 0.005 m^2. Therefore, the number of tubes would be

$$n(0.005 \times 4)/(3.142 \times 0.01)^2 = 63.65 = 64 \text{ tubes}$$

The overall heat transfer coefficient based on the outside surface area of the tubes, after neglecting the thermal resistance of the tube wall, is

$$U = 1/(1/h_0 + r_o/r_i h_i) = 1/[1/260 + 12.5/(10 \times 590)] = 167.65 \, W/m^2K$$

$$C_{max} = 40 \times 1.05 = 42; \; C_{min} = 3 \times 1.5 = 4.5; \; R = 4.5/42 = 0.107$$

$$NTU = AU/C_{min} = 3.142 \times 0.0125 \times 4 \times 64 \times 167.65/(4.5 \times 1000) = 0.374$$

From Fig. 10.12, for R = 0.107, and NTU = 0.3 74, E = 0.35 approximately Therefore, $0.35 = (T_{c_0} - 375)/(675 - 375)$ or $T_{c_0} = 207^o C$

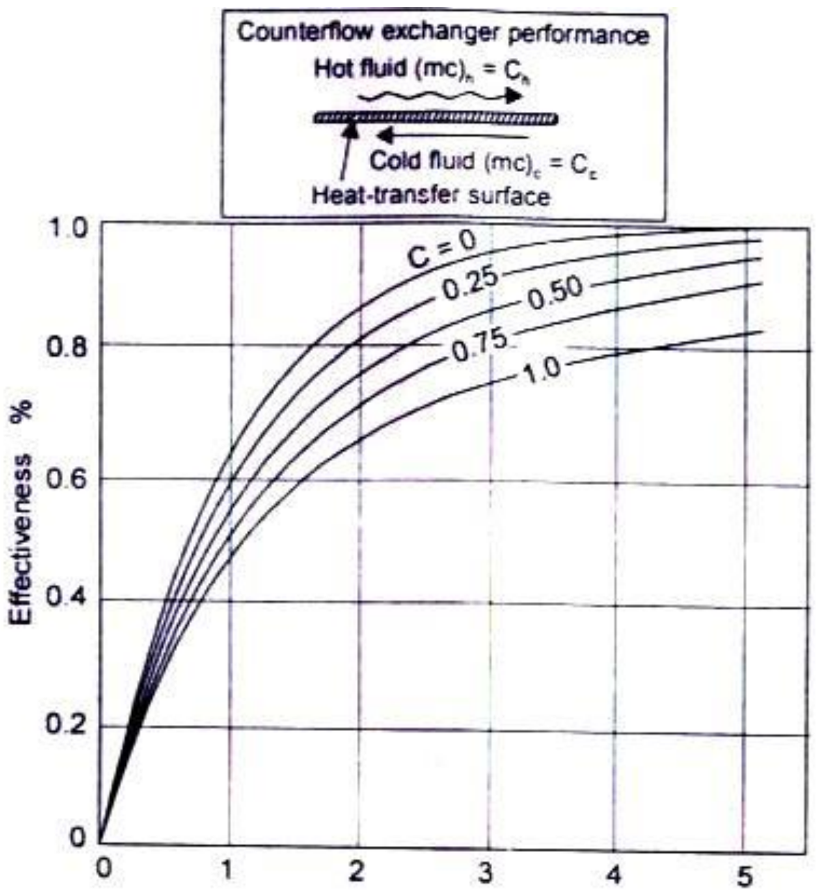


Fig 4.12 Heat exchanger effectiveness for counter flow

Example 4.14 Air at 25°C, mass flow rate 20 kg/min, flows over a cross-flow heat exchanger and cools water from 85°C to 50°C. The water flow rate is 5 kg/mm. If the overall heat transfer coefficient IS 80 W/m^{2}K and air is the mixed fluid, calculate the exchanger effectiveness and the surface area.

Solution: Let the specific heat capacity of air and water be 1.005 and 4.182 kJ/ kgK. By making an energy balance:

$$\dot{m}_c \times c_c \times \left(T_{c_0} - T_{c_i}\right) = \dot{m}_h \times c_h \times \left(T_{h_i} - T_{h_0}\right)$$

or, 5 × 4182 × (85 – 50) = 20 × 1005 × $\left(T_{c_0} - 25\right)$

i.e., the air will come out at 61.4 ^{o}C.

Heat capacity rates for water and air are:

$$C_w = 4182 \times 5/60 = 348.5; \quad C_a = 1005 \times 20/60 = 335$$

$$R = C_{min}/C_{max} = 335/348.5 = 0.96$$

The effectiveness on the basis of minimum heat capacity rate is

$$\in = \left(61.4 - 25\right)/\left(85 - 25\right) = 0.6$$

From Fig. 10.13, for R = 0.96 and $\in$ = 06, NTU = 2.5

Since NTU = AU/C_{min}; A = 2.5 × 335/80 = 10.47 m^2

Since all the four terminal temperatures are easily obtained, we can also use the LMTD approach. Assuming a simple counter flow heat exchanger,

LMTD = (25 – 23.6)/ ln (25/23.6) = 24.3

The correction factor for using a cross-flow heat exchanger with one fluid mixed and the other unmixed, F = 0.55

$\dot{Q}$ = U A F (LMTD)

Therefore, A = 348.5 × 35/(80 × 0.55 × 24.3) = 11.4 m^2

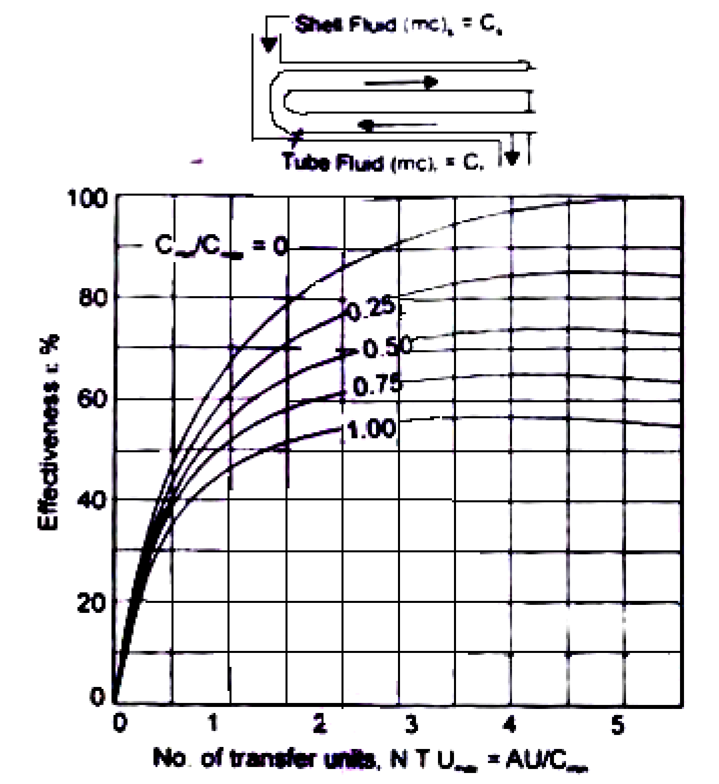
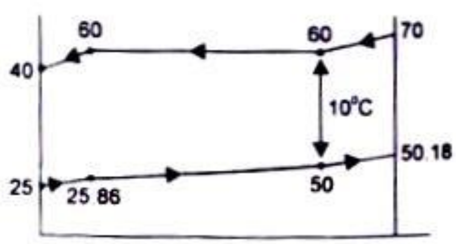


Fig. 4.13 Heat exchanger effectiveness for shell and tube heat exchanger with one shell pass and two, or a multiple of two, tube passes

Example 4.15 Steam at 20 kPa and 70°C enters a counter flow shell and tube exchanger and comes out as subcooled liquid at 40°C. Cooling water enters the condenser at 25°C and the temperature difference at the pinch point is 10°C. Calculate the (i) amount of water to be circulated per kg of steam condensed, and (ii) required surface area if the overall heat transfer coefficient is 5000 W/m^{2}K and is constant.

Solution: The temperature profile of the condensing steam and water is shown in the 40 accompanying sketch.



The saturation temperature corresponding to 20 kPa is 60°C and as such the temperature of the cooling water at the pinch pint is 50°C. The condensing unit may be considered as a combination of three sections:

(i) desuperheater - the superheated steam is condensed to saturated steam from 70°C to 60°C.

(ii) the condenser - saturated steam is condensed into saturated liquid.

(iii) subcooler - saturated liquid at 60°C is cooled to 40°C.

Assuming that the specific heat capacity of superheated steam is 1.8 kJ/kgK, heat given out in the desuperheater section is 1.8 × (70 - 60) = 18000 J/kg. Heat given out in the condenser section = 2358600 J/kg (= hfg)

Heat given out in the subcooler = 4182 × (60 - 40) = 83640 J/kg

By making an energy balance, for subcooler and condenser section, we have

$\dot{m}_w \times 4182 \times (50 - 25) = (83640 + 2358600)$;

∴. Mass of water circulated, $\dot{m}_w$ = 23.36 kg/kg steam condensed.

The temperature of water at exit

= 25 + (83640 + 2358600 + 18000)/ (23.36 × 4182) = 50.18 °C

LMTD for desuperheater section

= [(70 - 50.18) - (60 - 50))/ln(20.18/1 0) = 14.5

LMTD for condenser section = [(60 - 50) - (60 - 25.86))/ln(1 0/34.14)

= 19.66

LMTD for subcooler section = [(34.14 - 15)/1n (34.14/15) = 23.27

Since U is constant through out,

Surface area for subcooler section = 83640/(5000 × 23.27) = 0.7188 m^2

Surface area for condenser section = 2358600/(5000 × 19.66) = 23.9939 m^2

Surface area for desuperheater section = 18000/(5000 × 14.5) = 0.2483 m^2

$\therefore$. Total surface area = 24.96 m^2 and average temperature difference = 19.71°C.

Example 4.16 In an economiser (a cross flow heat exchanger, both fluids unmixed) water, mass flow rate 10 kg/s, enters at 175°C. The flue gas mass flow rate 8 kg/s, specific heat 1.1 kJ/kgK, enters at 350°C. Estimate the temperature of the flue gas and water at exit, if U = 500 W/m^2K, and the surface area 20 m^2 What would be the exit temperature if the mass flow rate of flue gas is (i) doubled, and (ii) halved.

Solution: The heat capacity rate of water = 4182 × 10 = 41820 W /K

The heat capacity rate of flue gas = 1100 × 8 = 8800 W/K

C_{min}/C_{max} = 8800/41820 = 0.21

$\text{NTU} = \text{AU}/\text{C}_{\text{min}} = 500 \times 20/8800 = 1.136$

From for NTU = 1.136 and $C_{min}/C_{max} = 0.21$, $\in = 0.62$

Therefore, 0.62 = (350 - T)/(350 - 175) and T = 241.5°C

The temperature of water at exit, T_w = 175 + 8800 × (350 - 241.5)/41820

= 197.83°C

When the mass flow rate of the flue gas is doubled. C_{gas} = 17600 W/K

$C_{min}/C_{max} = 0.42$, $\text{NTU} = \text{AU}/C_{min} = 0.568$

$\in$ = 0.39 = (350 - T)/(350 - 175);

T = 281. 75°C, an increase of 40°C

and T_w = 175 + 28.72 = 203.72°C, an increase of about 6°C.

When the mass flow rate of the flue gas is halved, C_{min} = 4400 W/K

$C_{min}/C_{max} = 0.105$, NTU = 2.272, and from the figure, $\in$ = 0.83, an increase and T_g = 204.75 and T_w = 190.3°C

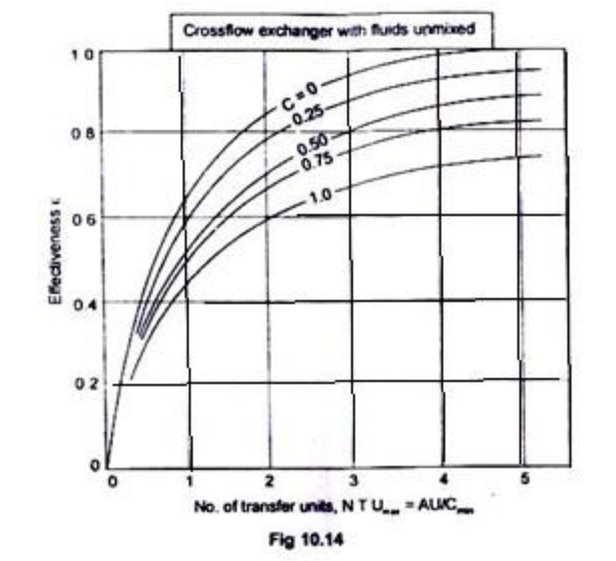


Fig 4.14. Cross flow exchanger with fluids un mixed

Example 4.17 In a tubular condenser, steam at 30 kPa and 0.95 dry condenses on the external surfaces of tubes. Cooling water flowing through the tubes has mass flow rate 5 kg/s, inlet temperature 25°C, exit temperature 40°C. Assuming no subcooling of the condensate, estimate the rate of condensation of steam, the effectiveness of the condenser and the NTU.

Solution: Since there is no subcooling of the condensate, the steam will lose its latent heat of condensation = 0.95 × h_{fg} = 0.95 × 2336100 = 2.22 × 10^6 J/kg. At pressure, 30kPa, saturation temperature is 69.124°C

Steam condensation rate × 2.22 × 10^6 = Heat gained by water

= 5 × 4182 × (40 – 25) = 313650 J

Therefore, m, = 313650/2.22 × 106 = 847.7 kg/hour.

When the temperature of the evaporating or condensing fluid remains constant, the value of LMTD is the same whether the system is having a parallel flow or counter flow arrangement, therefore,

LMTD = [(69.124 - 25) - (69.124-40)]/ln(44.124/29.124) = 36.1

Q = UA(LMTD)

Therefore, UA = 5 × 4182 × (40 - 25)/36.1 = 8688.36 W/K

$NTU = UA/C_{min} = 8688.36/(5 \times 4182) = 0.4155$

Effectiveness= Actual temp. difference; Maximum possible temp. difference

= (40 - 25)/(69.124 - 25) = 34%.

Example 4.18 A single shell 2 tube pass steam condenser IS used to cool steam entering at 50°C and releasing 2000 MW of heat energy. The cooling water, mass flow rate 3 × 104 kg/s, enters the condenser at 25°C. The condenser has 30,000 thin walled tube of 30 mm diameter. If the overall heat transfer coefficient is 4000 Wim^{2}K, estimate the (I) rise m temperature of the cooling water, and (II) length of the tube per pass.

Solution: By making an energy balance:

Heat released by steam = heat taken in by cooling water,

or, $2000 \times 10^6 = 3 \times 10^4 \times 4182 \times (\Delta T)$; $\Delta T = 15.94°C$.

Since in a condenser, heat capacity rate of condensing steam is usually very large m comparison with the heat capacity rate of cooling water, the effectiveness

$$\in = \left(T_{c_0} - T_{c_i}\right) / \left(T_{h_i} - T_{c_i}\right) = 15.94/(50-25) = 0.6376$$

$$\text{And, for } C_{min}/C_{max} = 0, \ \in = 1 - \exp(-NTU)$$

$$\therefore \ \exp(-NTU) = 1.0 - 0.6376 = 0.3624$$

$$\text{And, } NTU = 1.015 = AU/C_{min} = (2 \times 3.142 \times 0.03 \times L \times 30000) \times 4000/(1.25 \times 108)$$

L = 5.546 m

4.10. Heat Exchanger Design-Important Factors

A comprehensive design of a heat exchanger involves the consideration of the thermal, mechanical and manufacturing aspect. The choice of a particular design for a given duty depends on either the selection of an existing design or the development of a new design. Before selecting an existing design, the analysis of his performance must be made to see whether the required performance would be obtained within acceptable limits.

In the development of a new design, the following factors are important:

(a) Fluid Temperature - the temperature of the two fluid streams are either specified for a given inlet temperature, or the designer has to fix the outlet temperature based on flow rates and heat transfer considerations. Once the terminal temperatures are defined, the effectiveness of the heat exchanger would give an indication of the type of flow path-parallel or counter or cross-flow.

(b) Flow Rates - The maximum velocity (without causing excessive pressure drops, erosion, noise and vibration, etc.) in the case of liquids is restricted to 8 m1s and m case of gases below 30 m/s. With this restriction, the flow rates of the two fluid streams lead to the selection of flow passage cross-sectional area required for each of the two fluid steams.

(c) Tube Sizes and Layout - Tube sizes, thickness, lengths and pitches have strong influence on heat transfer calculations and therefore, these are chosen with great care. The sizes of tubes vary from 1/4" O.D. to 2" O.D.; the more commonly used sizes are: 5/8", 3/4" and 1" O.D. The sizes have to be decided after making a compromise between higher heat transfer from smaller tube sizes and the easy clean ability of larger tubes. The tube thickness will depend pressure, corrosion and cost. Tube pitches are to be decided on the basis of heat transfer calculations and difficulty in cleaning. Fig. 4.16 shows several arrangements for tubes in bundles. The two standard types of pitches are the square and the triangle. the usual number 0 f tube passes in a given shell ranges from one to eight. In multipass designs, even numbers of passes are generally used because they are Simpler to design.

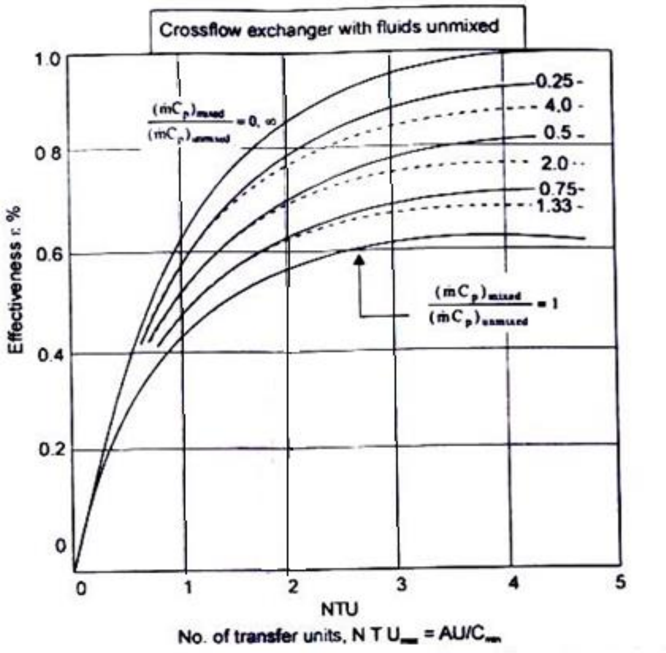


Fig 4.15: Heat exchanger effectiveness for crossflowwith one fluid mixed and the other unmixed

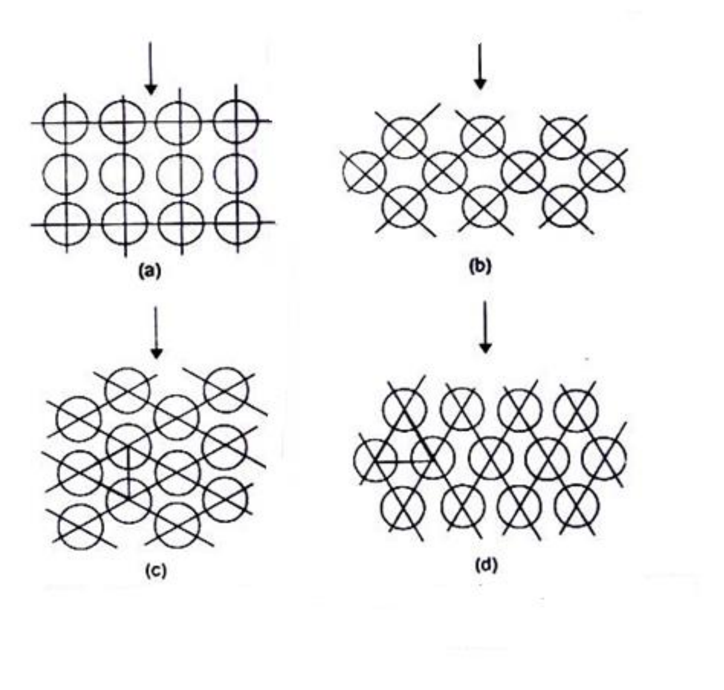


Fig 4.16 Several arrangements of tubes in bundles : (a) I line arrangement with square pitch, (b) staggered arrangement with triangular pitches (c) and(d) stagged arrangement with triangular pitches

Fig 4.17 shows three types of transverse baffles used to increase velocity on the shell side. The choice of baffle spacing and baffle cut is a variable and the optimum ratio of baffle cuts and spacing cannot be specified because of many uncertainties and insufficient data.

(d)Dirt F actor and Fouling - the accumulation of dirt or deposits affects significantly the r ate of heat transfer and the pressure drop. P roper allowance for the fouling factor and dirt factor should receive the greatest attention design because they cannot be avoided. A heat exchanger requires frequent cleaning. Mechanical cleaning will require removal of the tube bundle for cleaning. Chemical cleaning will require the use of non-corrosive materials for the tubes.

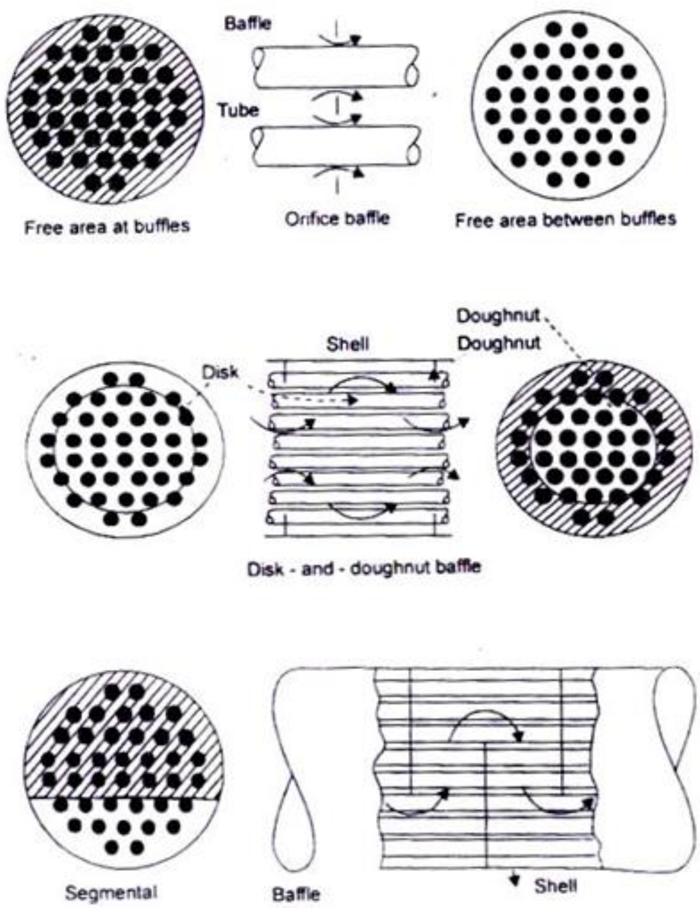


Fig. 4.17 Three types of transverse baffles

(e) Size and Installation - In designing a heat exchanger, It is necessary that the constraints on length, height, width, volume and weight is known at the outset. Safety regulations should also be kept in mind when handling fluids under pressure or toxic and explosive fluids.

(f) Mechanical Design Consideration - While designing, operating temperatures, pressures, the differential thermal expansion and the accompanying thermal stresses require attention.

And, above all, the cost of materials, manufacture and maintenance cannot be Ignored.

Example 4.19 In a counter flow concentric tube heat exchanger cooling water, mass flow rate 0.2 kg/s, enters at 30°C through a tube inner diameter 25rnnl. The oil flowing through the annulus, mass flow rate 0.1 kg/s, diameter 45 rum, has temperature at inlet 100°C. Calculate the length of the tube if the oil comes out at 60°C. The properties of oil and water are:

Oil: C_p – 2131 J/kgK, μ = 3.25 × 10^{-2} Pa-s, k = 0.138 W/mK,

Water; C_p = 4178 J/kg K, μ = 725 × 10^{-6} Pa-s,

k = 0.625 W/mK, Pr = 4.85

Solution: By making an energy balance: Heat given out by oil = heat taken in by water.

$$0.1\times2131\times(100-60)=0.2\times4187\times\left(T_{c_0}-30\right)$$

$$T_{c_0}=40.2^{o}C$$

$$\text{LMTD} = \left[\left(T_{h_i}-T_{c_0}\right)-\left(T_{h_0}-T_{c_i}\right)\right]/\ln\left[\left(T_{h_i}-T_{c_0}\right)/\left(T_{h_0}-T_{c_0}\right)\right]$$

$$=\left[(100-40.2)-(60-30)\right]/\ln(59.8/30)=43.2^{o}C$$

Since water is flowing through the tube,

$$Re=4\dot{m}/\pi D\mu=\frac{4\times0.2}{3.142\times0.025\times725\times10^{-6}}$$ = 14050, a turbulent flow.

Nu = 0.023 $Re^{0.8}$ $Pr^{0.4}$, fluid being heated.

= 0.023 $(14050)^{0.8}$ $(4.85)^{0.4}$ = 90; ∴ h_i = 90 × 0.625/0.025 = 2250 W/m^2K

The oil is flowing through the annulus for which the hydraulic diameter is:

(0.045 - 0.025) = 0.02 m

$$Re = 4\dot{m}/\pi\left(D_o + D_i\right)\mu = 4 \times 0.1/\left(3.142 \times 0.07 \times 3.25 \times 10^{-2}\right) = 56.0$$

Laminar flow.

Assuming Uniform temperature along the Inner surface of the annulus and a perfectly insulated outer surface.

Nu = 5.6, by interpolation (chapter 6)

h_o = 5.6 × 0.138/0.02 = 38.6 W/m^2K.

The overall heat transfer coefficient after neglecting the tube wall resistance,

U = 1/(1/2250 + 1/38.6) = 38 W/m^2K

$\dot{Q} = UA\left(LMTD\right)$, where where A = $\pi D_i \times L$

L = (0.1 × 2131 × 40)/(38 × 3.142 × 0.025 × 43.2) = 66.1 m requires more than one pass.

Example 4.20 A double pipe heat exchanger has an effectiveness of 0.5 for the counter flow arrangement and the thermal capacity of one fluid is twice that of the other fluid. Calculate the effectiveness of the heat exchanger if the direction of flow of one of the fluids is reversed with the same mass flow rates as before.

Solution: For a counter flow arrangement and R = 0.5, $\in$ = 0.5

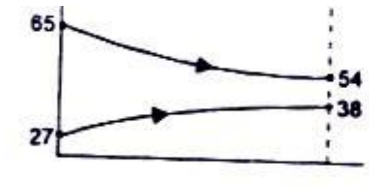
$$NTU = \left[1/\left(R-1\right)\right]\ln\left(\in R - 1\right) = -2.0\ \ln\left(0.5/0.75\right) = 0.811$$

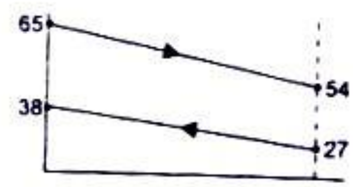
For parallel flow, $\in = \left[1 - \exp\left\{-NTU(1+R)\right\}\right]/\left(1+R\right)$

$$= \left[1 - \exp\left(-0.811 \times 1.5\right)\right]/1.5 = 0.469$$

Example 4.21 Oil is cooled in a cooler from 65°C to 54°C by circulating water through the cooler. The cooling load is 200 kW and water enters the cooler at 27°C. If the overall heat transfer coefficient, based on the outer surface area of the tube is 740 W/m^2K and the temperature rise of cooling water is 11°C, calculate the mass flow rate of water, the effectiveness and the heat transfer area required for a single pass In a parallel flow and in a counter flow arrangement.

Solution: Cooling load = 200 kW = mass of water × sp. heat × temp. rise Mass of water = 200/(4.2 × 11) = 4.329 kg/s





(i) Parallel flow:

From the temperature profile:

LMTD = (38 – 16)/ln(38/16) = 25.434 Q = U A (LMTD);

Area A = 200 × 10^3/(740 × 25.434) = 10.626 m^2

Effectiveness, $\in$ = (38 - 27)/(54 - 27) = 0.407.

(ii) Counter flow:

From the temperature profile:

LMTD = mean temperature difference = 27°C

Area A = 200 × 10^3/(740 × 27) = 10 m^2

Effectiveness, E = (38 - 27)/(65 - 27) = 0.289.

Example 4.22 Oil (mass flow rate 1.5 kg/s C_p = 2 kJ/kgK) is cooled in a single pass shell and tube heat exchanger from 65 to 42°C. Water (mass flow rate 1 kg/s, C_p = 4.2 kJ/kgK) has an inlet temperature of 28°C. If the overall heat transfer coefficient is 700 W/m^2K, calculate heat transfer area for a counter flow arrangement using $\in$ - NTU method.

Solution: Heat capacity rate of 011; 1.5 × 2.0 = 3 kW/K

Heat capacity rate of water = 1 × 4.2; 4.2 kW/K

$C_{min} = 3.0\ kW/K$ and $R = C_{min}/C_{max} = 3/4.2 = 0.714$

For a counter flow arrangement, $NTU = \left[1/(R-1)\right]\ln\left[(\in-1)/(\in R-1)\right]$

Effectiveness, $\in = (65-42)/(65-28) = 0.6216$

and NTU = 1.346 = $A\,U/C_{min}$; A = 1.346 × 3000 / 700 = 5.77 m^2

By making an energy balance, we can compute the water temperature at outlet.

or 3.0 × (65 - 42); 4.2 × (T - 28), T; 44.428

LMTD for a counter flow arrangement:

LMTD; (20.572 - 14)/ln (20.572/14) = 17.076

Area, A = $\dot{Q}/U\times(LMTD) = 3\times10^3\times(65-42)/(700\times17.076) = 5.77\ m^2$

Example 4.23 A fluid (mass flow rate 1000 kg/min, sp. heat capacity 3.6 kJ/kgK) enters a heat exchanger at 700 C. Another fluid (mass flow rate 1200 kg/mm, sp. heal capacity 4.2 kJ/kgK) enters al 100 C. If the overall heat transfer coefficient is 420 W/m^2K and the surface area is 100m2, calculate the outlet temperatures of both fluids for both counter flow and parallel flow arrangements.

Solution: Heat capacity rate for the hot fluid

$$1000\times3.6\times10^3\times60 = 60\times10^3\,W/K$$

Heat capacity rate for the cold fluid = 1200 × 4.2 × 10^3/60 = 84 × 10^3 W/K

R = C_{min}/C_{max} = 60/84; 0.714, NTU = U A/C_{min} = 420 × 100/60000 = 0.7

(i) For counter flow heat exchanger:

$$\in = \left[1-\exp\left\{-N(1-R)\right\}\right]/\left[1-\text{Re}\,\text{xp}\left\{-N(1-R)\right\}\right]$$

$$\left[1-\exp\left\{-0.7(1-0.714)\right\}\right]/1\left[1-0.714\ \exp\left\{-0.7(1-0.714)\right\}\right] = 0.4367$$

Since heat capacity rate of the hot fluid IS lower,

$$\in = \left(700 - T_{h_0}\right) / \left(700 - 100\right)$$

and $T_{h_0} = 700 - 0.4367 \times 600 = 438^{o}C$

By making an energy balance, $60 \times 10^3 \left(700 - 438\right) = 84 \times 10^3 \left(T_{c_0} - 100\right)$

or, $T_{c_0} = 60 \times 262 / 84 + 100 = 87.14^{o}C$

(ii) For parallel flow heat exchanger

$$\in = \left[1 - \exp\left\{-N\left(1+R\right)\right\}\right] / \left(1+R\right) = \left[1 - \exp\left\{0.7\left(1+0.714\right)\right\}\right] / \left(1.714\right)$$

$\in = 0.4077$, a lower value

and $\left(T_{h_i} - T_{h_0}\right) / \left(T_{h_i} - T_{c_0}\right) = 0.4077 = \left(700 - T_{h_0}\right) / \left(700 - T_{c_0}\right)$

By making an energy balance: $60 \times 10^3 \times \left(700 - T_{h_0}\right) = 84 \times 10^3 \times \left(T_{c_0} - 100\right)$

or, $\left(700 - T_{c_0}\right) = \left(700 - T_{h_0}\right) / 0.4077$

and $84 \times \left(T_{c_0} - 100\right) / 60 = \left(1.4T_{c_0} - 140\right)$

Therefore, $T_{c_0} = 237.5^{o}C$

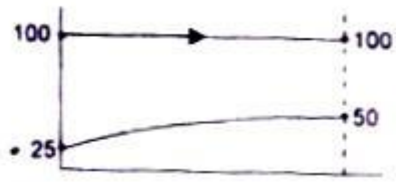
and $T_{h_0} = 511.4^{o}C$

Example 4.24 Steam enters the surface condenser at 100°C and water enters at 25°C with a temperature rise of 25°C. Calculate the effectiveness and the NTU for the condenser. If the water temperature at inlet changes to 35 C, estimate the temperature rise for water.

Solution: Effectiveness, $\in$ = 25/(100 25) = 0.33

For R = 0, $\in$ = 1 - exp (- N)

or, N = -ln(1 - $\in$) = 0.405



Since other parameters remain the same,

$$25/(100 - 25) = \Delta T/(100 - 35)$$

and $\Delta T = 21.66$; or, $T_{c_0} = 35 + 21.66 = 56.66°C$.

4.11. Increasing the Heat Transfer Coefficient

For a heat exchanger, the heat load is equal to Q = UA (LMTD). The effectiveness of the heat exchanger can be increased either by increasing the surface area for heat transfer or by increasing the heat transfer coefficient. Effectiveness versus NTU(AU/C_{min}) curves, Fig. , reveal that by increasing the surface area beyond a certain limit (the knee of the curves), there is no appreciable improvement in the performance of the exchangers. Therefore, different methods have been employed to increase the heat transfer coefficient by increasing turbulence, improved mixing, flow swirl or by the use of extended surfaces. The heat transfer enhancement techniques is gaining industrial importance because it is possible to reduce the heat transfer surface area required for a given application and that leads to a reduction in the size of the exchanger and its cost, to increase the heating load on the exchanger and to reduce temperature differences.

The 'different techniques used for increasing the overall conductance U are: (a) Extended Surfaces - these are probably the most common heat transfer enhancement methods. The analysis of extended surfaces has been discussed in previous Chapter. Compact heat exchangers use extended surfaces to give the required heat transfer surface area in a small volume. Extended surfaces are very effective when applied in gas side heat transfer. Extended surfaces find their application in single phase natural and forced convection pool boiling and condensation.

(b) Rough Surfaces - the inner surfaces of a smooth tube is artificially roughened to promote early transition to turbulent flow or to promote mixing between bulk flow and the various sub-layer in fully developed turbulent flow. This method is primarily used in single phase forced convection and condensation.

(c) Swirl Flow Devices - twisted strips are inserted into the flow channel to impart a rotational motion about an axis parallel to the direction of bulk flow. The heat transfer coefficient increases due to increased flow velocity, secondary flows generated by swirl, or increased flow path length in the flow channel. This technique is used in flow boiling and single phase forced flow.

(d) Treated Surfaces - these are used mainly in pool boiling and condensation.

Treated surfaces promote nucleate boiling by providing bubble nucleation sites. The rate of condensation increases by promoting the formation of droplets, instead of a liquid film on the condensing surface. This can be accomplished by coating the surface with a material that makes the surface non-wetting.

All of these techniques lead to an increase in pumping work (increased frictional losses) and any practical application requires the economic benefit of increased overall conductance. That is, a complete analysis should be made to determine the increased first cost because of these techniques, increased heat exchanger heat transfer performance, the effect on operating costs (especially a substantial increase in pumping power) and maintenance costs.

4.12. Fin Efficiency and Fin Effectiveness

Fins or extended surfaces increase the heat transfer area and consequently, the amount of heat transfer is increased. The temperature at the root or base of the fin is the highest and the temperature along the length of the fm goes on decreasing Thus, the fin would dissipate the maximum amount of heat energy if the temperature all along the length remains equal to the temperature at the root. Thus, the fin efficiency is defined as:

η_{fin} = (actual heat transferred) / (heat which would be transferred if the entire fin area were at the root temperature)

In some cases, the performance of the extended surfaces is evaluated by comparing the heat transferred with the fin to the heat transferred without the fin. This ratio is called 'fin effectiveness' E and it should be greater than 1, if the rate of heat transfer has to be increased with the use of fins.

For a very long fin, effectiveness $E = \dot{Q}_{\text{with fin}} / \dot{Q}_{\text{without fin}}$

$= (hpkA)^{1/2}\theta_0 \ /hA\,\theta_0 \ = \ (kp/hA)^{1/2}$

And η_{fin} = $(hpkA)^{1/2}\theta_0 \ (hpL\,\theta_0) = (hpkA)^{1/2} / (hpL)$

$$\frac{E}{\eta_{fin}} = \frac{(kp/hA)^{1/2}}{(hpkA)^{1/2}} \times hpL = \frac{pL}{A} = \frac{\text{Surface area of fin}}{\text{Cross-sectional area of the fin}}$$

i.e., effectiveness increases by increasing the length of the fin but it will decrease the fin efficiency.

Expressions for Fin Efficiency for Fins of Uniform Cross-section

1. Very long fins: $(hpkA)^{1/2}(T_0 - T_\infty)/\left[hpL(T_0 - T_\infty)\right] = 1/mL$

2 For fins having insulated tips:

$$\frac{(hpkA)^{1/2}(T_0 - T_\infty)\tan h(mL)}{hpL(T_0 - T_\infty)} = \frac{\tan h\,(mL)}{mL}$$

Example 4.25 The total efficiency for a finned surface may be defined as the ratio of the total heat transfer of the combined area of the surface and fins to the heat which would be transferred if this total area were maintained at the root temperature T_0 . Show that this efficiency can be calculated from

$\eta_t = 1 - A_f / A(1 - \eta_t)$ where η_t = total effltiency, A_f = surface area of all fins, A = total heat transfer area, η_f = fin efficiency

Solution: Fin efficiency,

$$\eta_f = \frac{\text{Actual heat transfered}}{\text{Heat that would be transferred if the entire fin were at the root temperature}}$$

or, $\eta_f = \dfrac{\text{Actual heat transfer}}{hA_f(T_0 - T_\infty)}$

$\therefore$ Actual heat transfer from finned surface = $\eta_f hA_f(T_0 - T_\infty)$

Actual heat transfer from un finned surface which are at the root temperature: $h(A-A_f)$

$(T_0 - T_\infty)$

Actual total heat transfer = $h(A - A_f)(T_0 - T_\infty) + \eta_f h A_f (T_0 - T_\infty)$

By the definition of total efficiency,

$$\eta_t = \left[h(A - A_f)(T_0 - T_\infty) + \eta_f h A_f (T_0 - T_\infty)\right] / \left[hA(T_0 - T_\infty)\right]$$

$$= \frac{(A - A_f) + \eta_f h A_f}{A} = 1 - A_f / A + \eta_f A_f / A$$

$$= 1 - (A_f / A) + (1 - \eta_f).$$

4.13. Extended Surfaces do not always Increase the Heat Transfer Rate

The installation of fins on a heat transferring surface increases the heat transfer area but it is not necessary that the rate of heat transfer would increase. For long fins, the rate of heat loss from the fin is given by $(hpkA)^{1/2}\theta_0 = kA(hp/kA)^{1/2}\theta_0 = kAm\,\theta_0$. When h/ mk = 1, Q = $hA\,\theta_0$ which is equal to the heat loss from the primary surface with no extended surface. Thus, when h = mk, an extended surface will not increase the heat transfer rate from the primary surface whatever be the length of the extended surface.

For h/mk > 1, Q < $hA\,\theta_0$ and hence adding a secondary surface reduces the heat transfer, and the added surface will act as an insulation. For h/mk < 1, Q > $hA\,\theta_0$, and the extended surface will increase the heat transfer, Fig. 4.18. Further, h/mk= $(h^2.kA/k^2hp)^{1/2}$ = $(hA/kP)^{1/2}$, i.e. when h/mk < 1, the heat transfer would be more effective when h/k is low for a given geometry.

4.14. An Expression for Temperature Distribution for an Annular Fin of Uniform Thickness

In order to increase the rate of heat transfer from cylinders of air-cooled engines and in certain type of heat exchangers, annular fins of uniform cross-section are employed. Fig. 3.21 shows such a fin with its nomenclature.

In the analysis of such fins, it is assumed that:

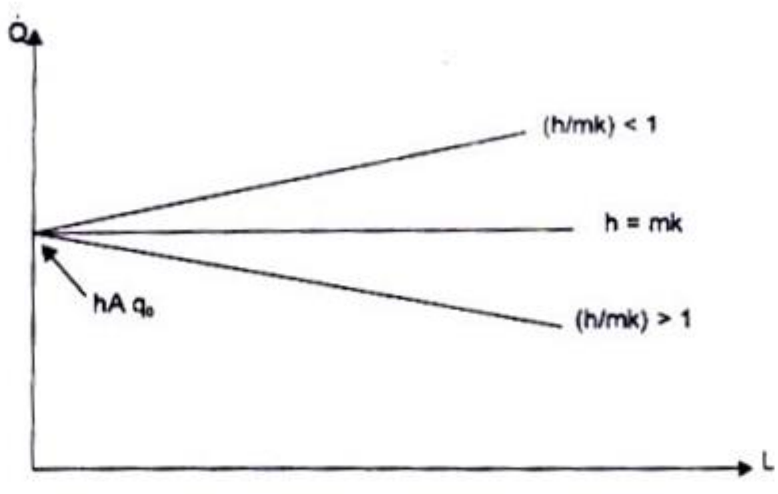
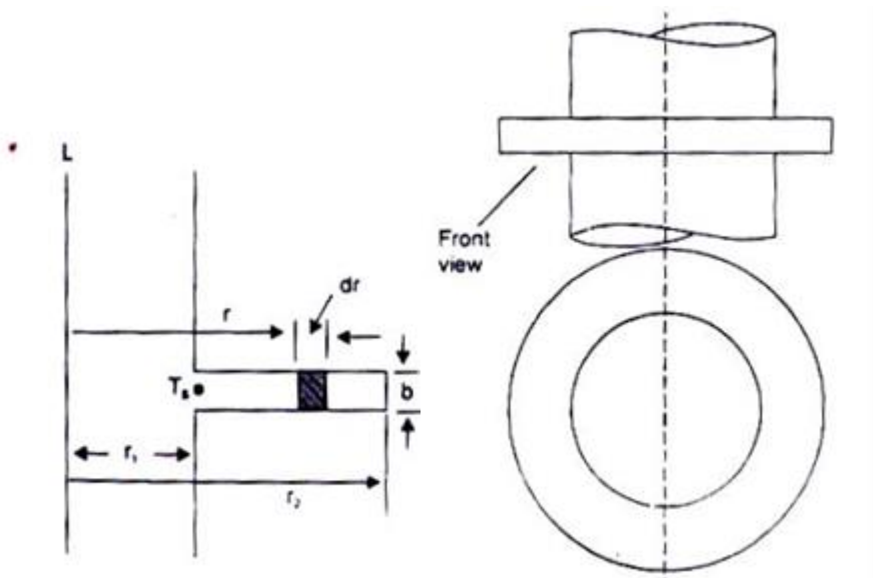


Fig 4.18 Extended surface will increase the heat transfer

(For increasing the heat transfer rate by fins, we should have (i) higher value of thermal conductivity, (ii) a lower value of h, fins are therefore generally placed on the gas side, (iii) perimeter/cross-sectional area should be high and this requires thin fins.)

(i) The thickness b is much smaller than the radial length (r_2 - r_1) so that one-dimensional radial conduction of heat is valid;

(ii) Steady state condition prevails.



Annular fin of uniform thickness Top view of annular fin

We choose an annular element of radius r and radial thickness dr. The cross-sectional area for radial heat conduction at radius r is $2\pi rb$ and at radius r + dr is 2π (r + dr) b. The surface area for convective heat transfer for the annulus is $2(2\pi.r.dr)$. Thus, by making an energy balance,

$$-k2\pi rb\frac{dT}{dr} = -k2\pi\left(r+dr\right)b\left(\frac{dT}{dr}+\frac{d^2T}{dr^2}dr\right)+h\times 4\pi r.dr\left(T-T_\infty\right)$$

or, $d^2T / dr^2 + (1/r)dT/dr - 2h/kb(T - T_\infty) = 0$

Let, $\theta = (T - T_\infty)$ the above equation reduces to

$d^2\theta / dr^2 + (l/r)\, d\theta/dr - (2h\ kb)\ \theta = 0$

The equation is recognized as Bessel's equation of zero order and the solution is $\theta = C_1 I_0(nr) + C_2 K_0(nr)$, where $n=(2h/kb)^{1/2}$, I_0 is the modified Bessel function, 1st kind and K_0 is the modified Bessel function, 2nd kind, zero order, The constants C_1 and C_2 are evaluated by applying the two boundary conditions:

at $r = r_l$, $T = T_s$ and $\theta = T_s - T_\infty$

at $r = r_2$, $dT / dr = 0$ because $b << (r_2 - r_l)$

By applying the boundary conditions, the temperature distribution is given by

$$\frac{\theta}{\theta_0} = \frac{I_0\left(nr\right)K_1(nr_2)+K_0\left(nr\right)I_1\left(nr_2\right)}{I_0\left(nr_1\right)K_1(nr_2)+K_0\left(nr_1\right)I_1\left(nr_2\right)} \qquad (4.16)$$

I_1 (nr) and K_1 (nr) are Bessel functions or order one.

And the rate of heat transfer is given by:

$$Q = 2\pi knb\,\theta_0 r_1\frac{K_1\left(nr_1\right)I_1(nr_2)-I_1\left(nr_1\right)K_1\left(nr_2\right)}{K_0\left(nr_1\right)I_1(nr_2)+I_0\left(nr_1\right)K_1\left(nr_2\right)} \qquad (4.17)$$

The efficiency of circumferential fins is also obtained from curves for efficiencies

$\left(\text{along Y-axis}\right) \sim \left(r_2+\frac{b}{2}-r_1\right)^{\frac{3}{2}}\left(\frac{2h}{Kb}\left(r_2-r_1\right)\right)^{\frac{1}{2}}$ for different values of $\left(r_2+\frac{b}{2}\right)/r_1$.

SATHYABAMA
INSTITUTE OF SCIENCE AND TECHNOLOGY
(DEEMED TO BE UNIVERSITY)
Accredited "A" Grade by NAAC | 12B Status by UGC | Approved by AICTE
www.sathyabama.ac.in

SCHOOL OF MECHANICAL ENGINEERING

DEPARTMENT OF AERONAUTICAL ENGINEERING

UNIT – V – Heat Transfer Techniques For Aerospace Applications – SAE1306

UNIT V

HIGH SPEED FLOW HEAT TRANSFER

5.1 High-Speed Heat Transfer for a Flat Plate

When the free-stream velocity is very high, as in high-speed aircraft, these dissipation effects must be considered. We begin our analysis by considering the adiabatic case, i.e., a perfectly insulated wall. In this case the wall temperature may be considerably higher than the free-stream temperature even though no heat transfer takes place. This high temperature results from two situations: (1) the increase in temperature of the fluid as it is brought to rest at the plate surface while the kinetic energy of the flow is converted to internal thermal energy, and (2) the heating effect due to viscous dissipation. Consider the first situation. The kinetic energy of the gas is converted to thermal energy as the gas is brought to rest, and this process is described by the steady-flow energy equation for an adiabatic process:

$$i_0 = i_\infty + \frac{1}{2g_c}u_\infty^2$$ **……………………5.1**

where i_o is the stagnation enthalpy of the gas. This equation may be written in terms of temperature as:

$$c_p(T_0 - T_\infty) = \frac{1}{2g_c}u_\infty^2$$

where T_o is the stagnation temperature and T_∞ is the static free-stream temperature. Expressed in terms of the free-stream, it is

$$\frac{T_0}{T_\infty} = 1 + \frac{\gamma - 1}{2}M_\infty^2$$ **…………………….5.2**

where M_∞ is the Mach number, defined as $M_\infty = u_\infty/a$, and a is the acoustic velocity, which for an ideal gas may be calculated with,

$$a = \sqrt{\gamma g_c R T}$$ **…………………………..5.3**

where R is the gas constant for the particular gas

In the actual case of a boundary-layer flow problem, the fluid is not brought to rest reversibly because the viscous action is basically an irreversible process in a thermodynamic sense. In addition, not all the free-stream kinetic energy is converted to thermal energy part is lost as heat, and part is dissipated in the form of viscous work. To take into account the irreversibilities in the boundary-layer flow system, a recovery factor is defined by,

$$r = \frac{T_{aw} - T_\infty}{T_0 - T_\infty} \quad \textbf{……………………….5.4}$$

where T_{aw} is the actual adiabatic wall temperature and T_∞ is the static temperature of the free stream. The recovery factor may be determined experimentally, or, for some flow systems, analytical calculations may be made. The boundary-layer energy equation,

$$u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y} = \alpha\frac{\partial^2 T}{\partial y^2} + \frac{\mu}{\rho c_p}\left(\frac{\partial u}{\partial y}\right)^2 \quad \textbf{……5.5}$$

has been solved for the high-speed-flow situation, taking into account the viscous-heating term. Although the complete solution is somewhat tedious, the final results are remarkably simple. For our purposes we present only the results and indicate how they may be applied. An excellent synopsis of the high-speed heat-transfer problem is given in a report by Eckert. The essential result of the high-speed heat-transfer analysis is that heat-transfer rates may generally be calculated with the same relations used for low-speed incompressible flow when the average heat-transfer coefficient is redefined with the relation,

$$q = \bar{h}A(T_w - T_{aw}) \quad \textbf{………………5.6}$$

Notice that the difference between the adiabatic wall temperature and the actual wall temperature is used in the definition so that the expression will yield a value of zero heat flow when the wall is at the adiabatic wall temperature. For gases with Prandtl numbers near unity, the following relations for the recovery factor have been derived:

Laminarflow:

$$r = \mathrm{Pr}^{1/2} \quad \textbf{………………………..5.7}$$

Turbulent flow:

$$r = \mathrm{Pr}^{1/3} \quad \textbf{…………………..……5.8}$$

These recovery factors may be used in conjunction with Equation (5.4) to obtain the adiabatic wall temperature. In high-velocity boundary layers substantial temperature gradients may occur, and there will be correspondingly large property variations across the boundary layer. The constant-property heat-transfer equations may still be used if the properties are introduced at a reference temperature T* as recommended by Eckert:

$$T^* = T_\infty + 0.50(T_w - T_\infty) + 0.22(T_{aw} - T_\infty)$$ **……………5.9**

The analogy between heat transfer and fluid friction may also be used when the friction coefficient is known. Summarizing the relations used for high-speed heat transfer calculations:

Laminar boundary layer (Rex < 5 x 10^5):

$$St_x^* \, Pr^{*2/3} = 0.332 \, Re_x^{*-1/2}$$ **………………….5.10**

Turbulent boundary layer (5 x 10^5 < Rex < 10^7) :

$$St_x^* \, Pr^{*2/3} = 0.0296 \, Re_x^{*-1/5}$$ **……………………5.11**

Turbulent boundary layer (10^7 < Rex < 10^9):

$$St_x^* \, Pr^{*2/3} = 0.185(\log Re_x^*)^{-2.584}$$ **………………….5.12**

The superscript * in the above equations indicates that the properties are evaluated at the reference temperature given by Equation (5.9).

To obtain an average heat-transfer coefficient, the above expressions must be integrated over the length of the plate. If the Reynolds number falls in a range such that Equation (5-12) must be used, the integration cannot be expressed in closed form, and a numerical integration must be performed. Care must be taken in performing the integration for the high-speed heat-transfer problem since the reference temperature is different for the laminar and turbulent portions of the boundary layer. This results from the different value of the recovery factor used for laminar and turbulent flow as given by Equations (5.7) and (5.8). When very high flow velocities are encountered, the adiabatic wall temperature may come so high that dissociation of the gas will take place and there will be a very wide variation of the properties in the boundary layer. Eckert recommends that these problems be treated on the basis of a heat-transfer coefficient defined in terms of enthalpy difference:

$$q = h_i A(i_w - i_{aw})$$ **…………………….5.13**

The enthalpy recovery factor is then defined as

$$r_i = \frac{i_{aw} - i_\infty}{i_0 - i_\infty}$$ **………………………..5.14**

Where i_{aw} is the enthalpy at the adiabatic wall conditions. The same relations as before are used to calculate the recovery factor and heat-transfer except that all properties are evaluated at a reference enthalpy i* given by

$$i^* = i_\infty + 0.5(i_w - i_\infty) + 0.22(i_{aw} - i_\infty)$$ **……………..5.15**

The Stanton number is redefined as

$$\mathrm{St}_i = \frac{h_i}{\rho u_\infty}$$ **……………….……………5.16**

This Stanton number is then used in Equation (5.10), (5.11), or (5.12) to calculate the heat-transfer coefficient. When calculating the enthalpies for use in the above relations, the total enthalpy must be used; that is chemical energy of dissociation as well as internal thermal energy must be included. The reference-enthalpy method has proved successful for calculating high-speed heat-transfer with an accuracy of better than 10 percent.

Problems 5.1

A flat plate 70 cm long and 1.0 m wide is placed in a wind tunnel where the flow conditions are M =3, p = 1/3 atm, and T =-40°C. How much cooling must be used to maintain the plate temperature at 35°C?

Solution:

We must consider the laminar and turbulent portions of the boundary layer separately because the recovery factors, and hence the adiabatic wall temperatures, used to establish the heat flow will be different for each flow regime. It turns out that the difference is rather small in this problem, but we shall follow a procedure that would be used if the difference were appreciable, so that the general method of solution may be indicated. The free-stream acoustic velocity is calculated from

$$a = \sqrt{\gamma g_c R T_\infty}$$

$$= [(1.4)\,(1.0)\,(287)\,(233)]^{1/2} = 306 \text{ m/s}$$

so that the free-stream velocity is

$$U_\infty = (3)\,(306) = 918 \text{ m/s}$$

The maximum Reynolds number is estimated by making a computation based on properties evaluated at free-stream conditions:

$$\rho_\infty = \frac{(1.0132 \times 10^5)(\frac{1}{20})}{(287)(233)} = 0.0758 \text{ kg/m}^3 \quad [4.73 \times 10^{-3} \text{ lb}_m/\text{ft}^3]$$

$$\mu_\infty = 1.434 \times 10^{-5} \text{ kg/m} \cdot \text{s} \quad [0.0347 \text{ lb}_m/\text{h} \cdot \text{ft}]$$

$$\mathrm{Re}_{L,\infty} = \frac{(0.0758)(918)(0.70)}{1.434 \times 10^{-5}} = 3.395 \times 10^6$$

Thus we conclude that both laminar and turbulent-boundary-layer heat transfer must be considered. We first determine the reference temperatures for the two regimes and then evaluate properties at these temperatures.

Laminar portion

$$T_0 = T_\infty \left(1 + \frac{\gamma - 1}{2} M_\infty^2\right)$$

$$= (233)\,[1 + (0.2)\,(3)^2] = 652 \text{ K}$$

Assuming a Prandtl number of about 0.7, we have

$$r = \mathrm{Pr}^{1/2} = (0.7)^{1/2} = 0.837$$

$$r = \frac{T_{aw} - T_\infty}{T_0 - T_\infty} = \frac{T_{aw} - 233}{652 - 233}$$

and

$$T_{aw} = 584 \text{ K} = 311°\text{C} \; [592°\text{F}].$$

Then the reference temperature from Equation (5.8) is

$$T^* = 233 + (0.5)(308 - 233) + (0.22)(584 - 233) = 347.8 \text{ K}$$

Checking the Prandtl number at this temperature, we have

$$\mathrm{Pr}^* = 0.697$$

so that the calculation is valid. If there were an appreciable difference between the value of Pr* and the value used to determine the recovery factor, the calculation would have to be repeated until agreement was reached. The other properties to be used in the laminar heat-transfer analysis are,

$$\rho^* = \frac{(1.0132 \times 10^5)(1/20)}{(287)(347.8)} = 0.0508 \text{ kg/m}^3$$

$$\mu^* = 2.07 \times 10^{-5} \text{ kg/m} \cdot \text{s}$$

$$k^* = 0.03 \text{ W/m} \cdot °\text{C} \quad [0.0173 \text{ Btu/h} \cdot \text{ft} \cdot °\text{F}]$$

$$c_p^* = 1.009 \text{ kJ/kg} \cdot °\text{C}$$

Turbulent portion

Assuming Pr = 0.7 gives

$$r = \mathrm{Pr}^{1/3} = 0.888 = \frac{T_{aw} - T_\infty}{T_0 - T_\infty} = \frac{T_{aw} - 233}{652 - 233}$$

T_{aw} = 605 K = 332°C

T* = 233 + (0.5) (308 - 233) + (0.22) (605 - 233) = 352.3 K

Pr*=0.695

The agreement between Pr* and the assumed value is sufficiently close. The other properties to be used in the turbulent heat-transfer analysis are

$$\rho^* = \frac{(1.0132 \times 10^5)(1/20)}{(287)(352.3)} = 0.0501\ \mathrm{kg/m^3}$$

$$\mu^* = 2.09 \times 10^{-5}\ \mathrm{kg/m \cdot s}$$

$$k^* = 0.0302\ \mathrm{W/m \cdot {}^\circ C}$$

$$c_p^* = 1.009\ \mathrm{kJ/kg \cdot {}^\circ C}$$

Laminar heat transfer

We assume,

$$\mathrm{Re}^*_{crit} = 5 \times 10^5 = \frac{\rho^* u_\infty x_c}{\mu^*}$$

$$x_c = \frac{(5 \times 10^5)(2.07 \times 10^{-5})}{(0.0508)(918)} = 0.222\ \mathrm{m}$$

$$\overline{\mathrm{Nu}^*} = \frac{\bar{h} x_c}{k^*} = 0.664\,(\mathrm{Re}^*_{crit})^{1/2}\ \mathrm{Pr}^{*1/3}$$

$$= (0.664)(5 \times 10^5)^{1/2}(0.697)^{1/3} = 416.3$$

$$\bar{h} = \frac{(416.3)(0.03)}{0.222} = 56.25\ \mathrm{W/m^2 \cdot {}^\circ C} \quad [9.91\ \mathrm{Btu/h \cdot ft^2 \cdot {}^\circ F}]$$

This is the average heat-transfer coefficient for the laminar portion of the boundary layer, and the heat transfer is calculated from

$$q = \bar{h} A (T_w - T_{aw})$$

$$= (56.26)\ (0.222)\ (308 - 584)$$

$$= -3445\mathrm{W}$$

so that 3445 W of cooling is required in the laminar region of the plate per meter of depth in the z direction

Turbulent heat transfer

To determine the turbulent heat transfer we must obtain an expression for the local heat-transfer coefficient from,

$$\mathrm{St}^*_x\ \mathrm{Pr}^{*2/3} = 0.0296\ \mathrm{Re}_x^{*-1/5}$$

and then integrate from X = 0.222 m to x = 0.7 m to determine the total heat transfer;

$$h_x = \mathrm{Pr}^{*-2/3} \rho^* u_\infty c_p (0.0296) \left(\frac{\rho^* u_\infty x}{\mu^*} \right)^{-1/5}$$

Inserting the numerical values for the properties gives

$$h_x = 94.34 x^{-1/5}$$

The average heat-transfer coefficient in the turbulent region is determined from

$$\overline{h} = \frac{\int_{0.222}^{0.7} h_x \, dx}{\int_{0.222}^{0.7} dx} = 111.46 \ \mathrm{W/m^2 \cdot {}^\circ C}$$

Using this value we may calculate the heat transfer in the turbulent region of the flat plate:

$$q = \overline{h} A (T_w - T_{aw})$$

= (111.46) (0.7 - 0.222) (308 - 605)

= -15,823 W

The total amount of cooling required is the sum of the heat transfers for the laminar and turbulent portions:

Total cooling =3445 + 15,823 =19,268 W

These calculations assume unit depth of 1 m in the z direction.

5.2 Basic Aspects of Gas Turbine Heat Transfer

The use of gas turbines for power generation and electricity production in both single cycle and combined cycle plant operation is extensive and will continue to globally grow into the future. Due to its high power density and ability to convert gaseous and liquid fuel into mechanical work with very high thermodynamic efficiencies, significant efforts continue today to further increase both the power output and thermodynamic efficiencies of the gas turbine. In particular, the aero thermal design of gas turbine components has progressed at a rapid pace in the last decade with all gas turbine manufacturers, in order to achieve higher thermodynamic efficiencies. This has been achieved by using higher turbine inlet temperatures and pressures, advanced turbine aerodynamics and efficient cooling systems of turbine airofoils, and advanced high temperature alloys, metallic coatings, and ceramic thermal barrier coatings. In this chapter, issues related to the thermal design of gas turbine blades are highlighted and several heat transfer technologies are examined, such as convective cooling, impingement cooling, film cooling, and application of thermal barrier coatings. Typical methods for validating the thermal designs of gas turbine airofoils are also outlined.

The aerothermal design of advanced gas turbines has progressed significantly in the last decade, primarily due to the requirement of increased turbine efficiencies and power. Performance increases are driven not only for reducing the consumption of fuel and the subsequent cost benefits, but also to reduce the emissions of CO2 , which is a primary component for the increased global warming. Over the last decade, major gas turbine performance enhancements have been achieved by the use of higher turbine inlet temperatures and pressures, design of advanced turbine aerodynamics, through reductions in turbine cooling and leakage air, and via the introduction of new high temperature alloys, metallic antioxidation coatings, and thermal barrier coatings. In today's energy market, there is wide range of gas turbines ranging from 1 to 500 MW and can operate with low and high calorific fuels. Figure 1 shows the GT26 heavy duty gas turbine.

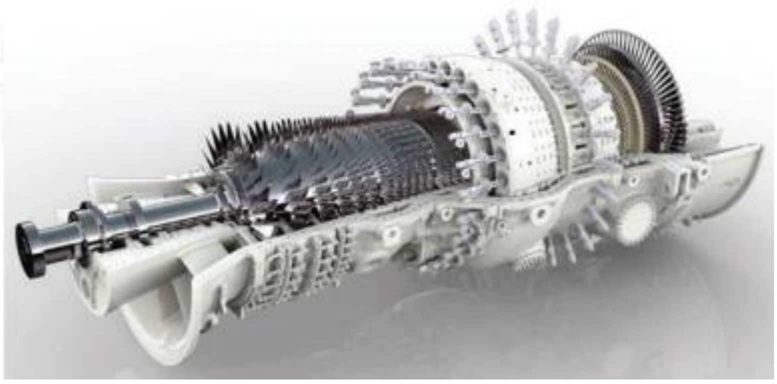


Fig. 5. 1. Heavy duty gas turbine, GT26.

The operation of a gas turbine, which essentially consists of four major components; compressor, combustor, turbine, and the exhaust diffuser, is governed by the Brayton thermodynamic cycle. For simple power generation applications, a generator is normally coupled to the gas turbine, whereby the mechanical work generated by the turbine is converted to useful electrical energy. In today's energy market, most gas turbine-based power plants are operated in combined cycle operation mode. Figure 5.2 shows a typical component layout of a combined cycle plant, whereby the gas turbine plant is coupled to a steam turbine plant via the heat recovery steam generator. Thermodynamically, the gas turbine operates in a Brayton cycle, whereas the steam turbine operates in a Rankine cycle. Due to this combination, Figure 5.2 highlights that combined cycle efficiencies are significantly higher than that of a gas turbine in simple operation. There are many variations of the gas turbine combined cycle plant and the interdependency of the component efficiencies and plant operating conditions.

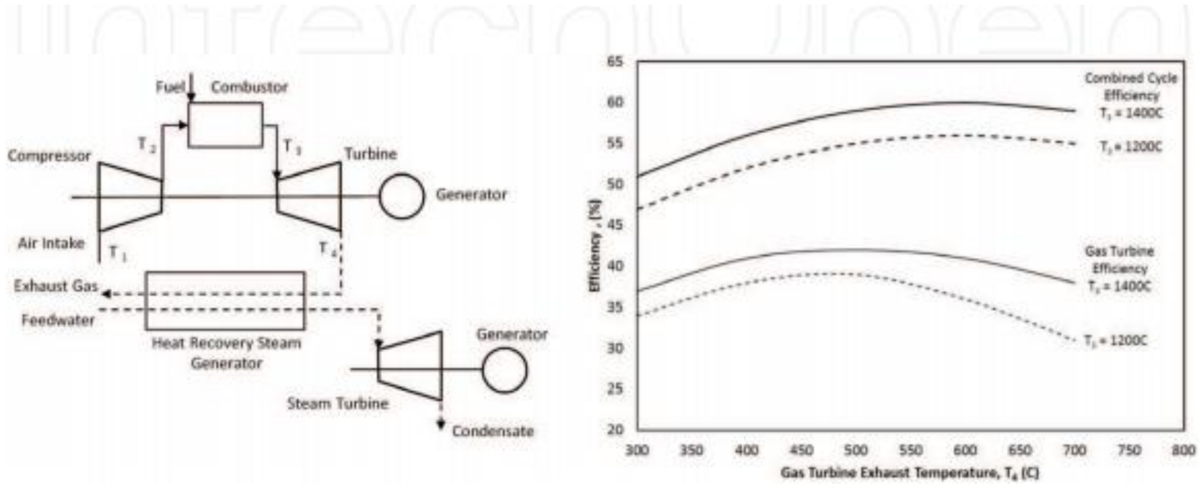


Fig. 5.2. Basic combined cycle plant arrangement

The historical progress in increased combined cycle plant efficiencies was highlighted in Figure 5.3, which predicts a continuous growth in cycle efficiencies approaching 65% over the next decades. Figure 5.3 also shows that a major part of the current growth in combined cycle efficiencies is attributed to improvements in gas turbine thermodynamic efficiencies, particularly with the H and J class gas turbines. A key driver for the latter has been the increased turbine inlet temperatures, and as shown in Figure 5.4, this has also resulted in the development of high-grade alloy, coatings, and very efficient aerofoil cooling designs which can maintain the blade metal temperatures and structural integrity for long continuous operating periods. In this chapter, issues related to the thermal design of gas turbine blades are examined and the various cooling technologies are outlined. In addition, typical methods for validating the thermal designs of gas turbine aerofoils are also outlined.

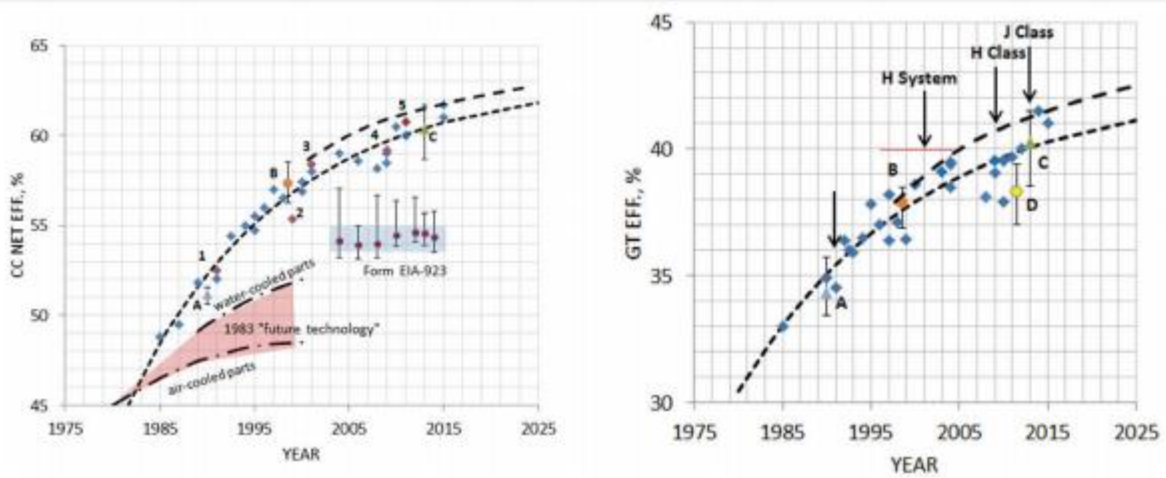


Fig. 5.3 Performance evolution of combined cycle and single cycle gas turbine.

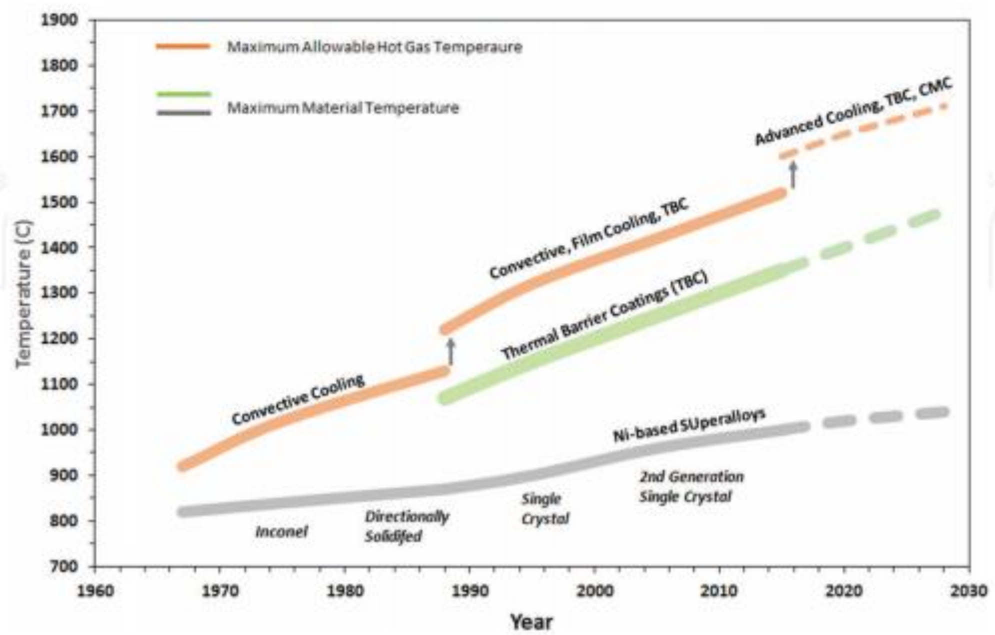


Fig. 5.4. Evolution of gas turbine hot gas temperatures, materials, and cooling technology

5.2.1 Design consideration of cooled turbine blades

In the design of air cooled gas turbine blades, there are several different factors related to the integration of a turbine blade thermal design into the overall gas turbine. Some of the key factors which influence the overall design of the turbine blade include,

- Overall gas turbine performance (power output and efficiency) and airofoil component lifetime requirements.
- Variation of ambient conditions, start-up load gradients, and shut-down conditions.
- Turbine aerodynamics, external heat loads to airofoils and turbine inlet temperatures.
- Hot gas temperature, pressure, and velocity profiles from the combustor chamber, and the expansion characteristics of the hot gas within the turbine.
- Choice of coolant from the compressor bleeds and the supply conditions over the entire operating envelope of the gas turbine.
- Geometrical clearances and gaps.
- Blade material and its properties at elevated temperatures.
- Manufacturing capability of the blade internal cooling core, machining of film cooling holes, application of thermal barrier coatings, and overall manufacturing costs.
- Maintenance methods and reconditioning of the turbine blades.

In Figure 5.5, some of the above parameters are highlighted, such as the impact of the coolant extracted from the different stages of the compressors. The front stage of the turbine will normally use the coolant extracted with the highest pressures, while the middle and rear turbine stages progressively use coolant extracted with lower pressures and temperatures. For the rear stage aerofoils, the cooling systems are normally low pressure drops systems and do not have features such as film cooling and impingement cooling.

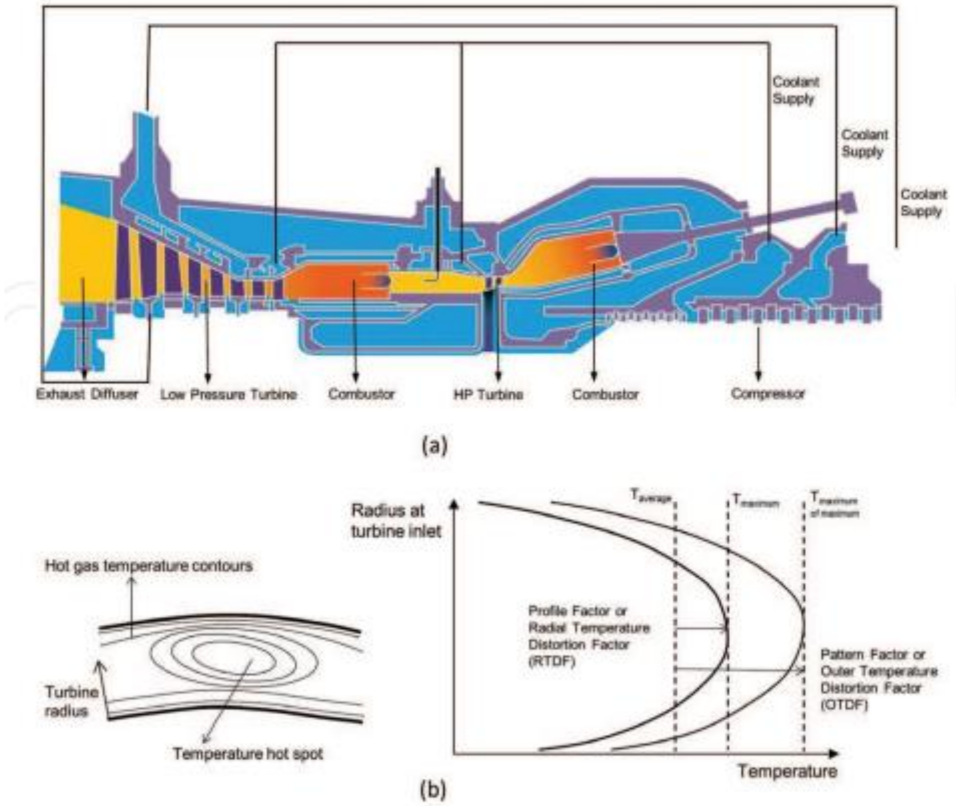


Fig. 5.5. Major design factors influencing the gas turbine overall aerothermal design, (a) coolant supply system, (b) combustor hot gas temperature profiles.

The front stage aerofoils however do have cooling systems with film cooling and impingement, as they are generally fed with the high pressure compressor end air. In addition to the impact of the air flow system, another major interface parameter for designing the airofoil cooling system is the combustor hot gas temperature and its distribution. Figure 5.5 shows a schematic of the hot gas distribution at the turbine inlet, which is generally nonuniform, and dependant on the upstream combustor and burner design. As Figure 5.5 shows, typically there is a radial distribution of the hot gas temperature, which is commonly referred to as the profile factor or the radial temperature distortion factor (RTDF). In addition to this, there is also a circumferential temperature distribution which is referred to as the pattern factor or the outer temperature distortion factor (OTDF). In the thermal design of gas turbine airofoils, blade tips, and endwalls, these radial and circumferential temperature distributions are always considered in the design process, and are normally based on in-situ engine measurements and high fidelity CFD predictions.

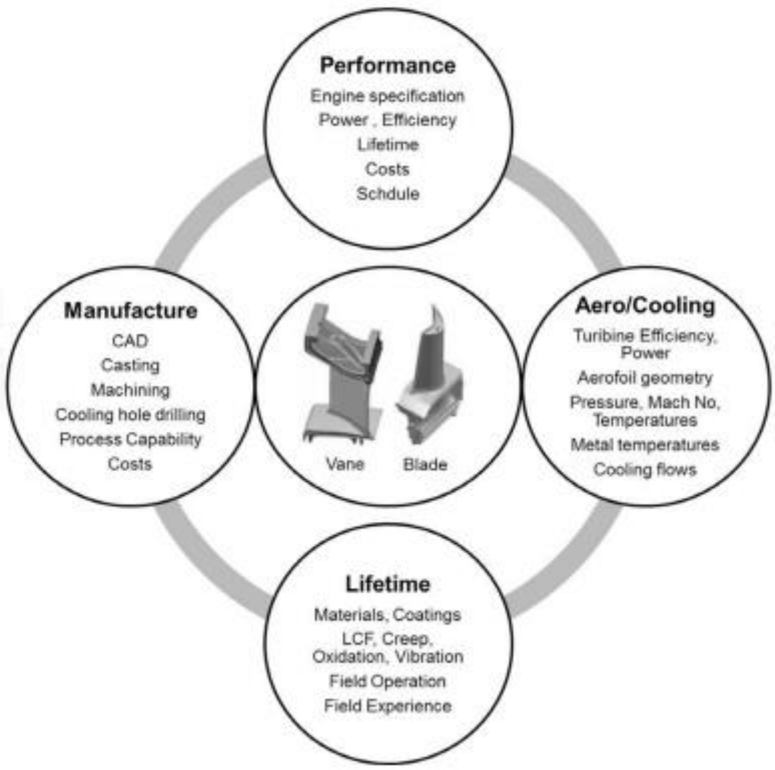


Fig. 5. 6. Major design interfaces for overall aerofoil designs

At the airofoil component level design, Fig. 5.6 shows an overview of several other interface considerations which needs to be accounted for in the overall optimization of the airofoil thermal design. The major design drivers for an optimized airofoil design include engine performance targets, aerothermal targets, component lifetime and mechanical integrity targets, and the manufacturing and cost constraints. Within these global requirements, Fig. 5.6 also highlights that are also many subtargets, such as manufacturing capability and field experience.

5.2.2 Turbine blade thermal analysis

Global thermal assessments

Due to the large number of operating and geometrical parameters that influence the heat transfer mechanism in gas turbine blades, simplified zero-dimensional relationships and design charts are often utilized. This allows for assessing the impact of various operational and geometrical parameters on a given blade cooling system. For such zero-dimensional analysis, it is important to have the detailed 2D and 3D thermal analysis results of the specific turbine blade or vane, and which has effectively been proven for meeting the design performance and lifetime in field gas turbines. This is commonly referred to as the reference blade from which new designs and concepts can be developed with a sufficient degree of confidence.

The thermal analysis is based on a simplified conjugate heat transfer analysis of flow in a cooling passage of a turbine blade as shown in Fig. 5.7 and assumes that the airofoil; (a) metal temperature is the average surface temperature at the airofoil midspan, (b) is exposed to the maximum hot gas temperature profile at the blade inlet, and (c) the coolant enters at the blade root and exits at the blade trailing edge. Then by performing a simple energy balance, it can be observed that, Heat transferred from the hot gas to the airofoil = heat gained by the airofoil = heat gained by the coolant in the airofoil.

$$Q = h_g S_g L (T_f - T_m) = h_c S_c L (T_m - T_c) = m_c C_{pc} (T_{co} - T_{ci}) \quad \textbf{…………..5.17}$$

where Q is the total heat transferred to the airofoil, h g and hc are respectively the hot gas and coolant heat transfer coefficients, L is the airofoil height, S_g and S_c are respectively the total airofoil perimeter on the gas and coolant sides, T_f is the average film cooling temperature, T_m is the average airofoil metal temperature, and T_{ci} and T_{co} are the coolant inlet and out temperatures. T_c is the average of the coolant inlet and outlet temperatures. For film-cooled airofoils, a film cooling effectiveness is additionally defined, which essentially modifies the driving hot gas temperature, T_g , with a film temperature, which is defined by;

Film Cooling Effectiveness,

$$\eta_f = \frac{T_g - T_f}{T_g - T_{co}} \quad \textbf{…………..5.18}$$

After rearranging the above equations, the following relationships can be derived;

Cooling Effectiveness,

$$\varepsilon = \frac{T_g - T_m}{T_g - T_{ci}} \quad \textbf{…………..5.19}$$

Mass flow function,

$$m^* = \frac{m_c \ C_{pc}}{h_g \ S_g \ L} \quad \textbf{…………..5.20}$$

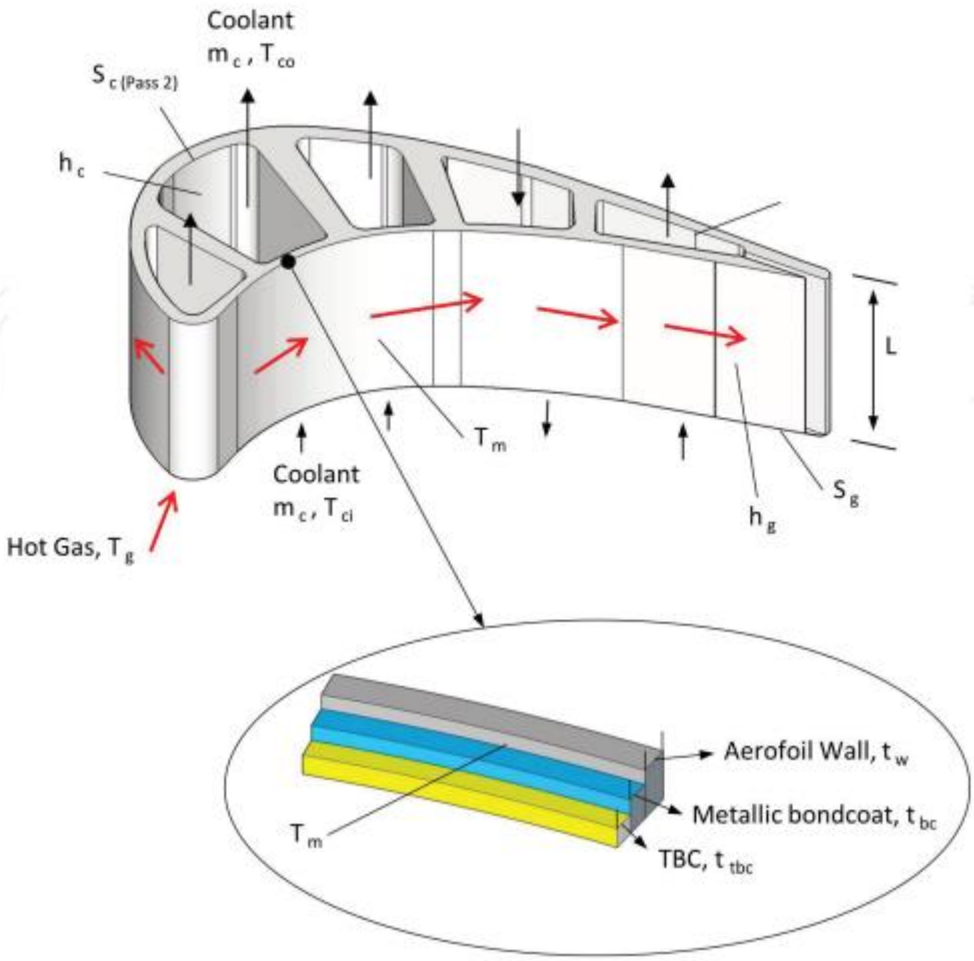


Fig.5. 7. Thermal design parameters of a gas turbine airofoil.

Cooling Efficiency,

$$\eta = \frac{T_{co} - T_{ci}}{T_m - T_{ci}} \quad \text{…………..5.21}$$

By further combining for the effectiveness, massflow function and efficiency, the following practical engineering formulations can be derived.

Overall effectiveness,

$$\varepsilon = \frac{m^* \eta}{1 + m^* \eta} \quad \text{…………..5.22}$$

Overall efficiency,

$$\eta = 1 - \exp\left[-\frac{A}{m^*}\right] \quad \text{Where } A = \frac{h_c \ . \ S_c}{h_g \ . \ S_g} \quad \text{…………..5.23}$$

To represent thermal barrier coatings (TBC) and the airofoil wall thickness, the hot gas and coolant heat transfer coefficients in the above equations are replaced by effective heat transfer coefficients, i.e.,

$$h_{g,eff} = \frac{h_g}{1 + Bi_{tbc}} \quad \text{.............5.24}$$

where the TBC Biot number,

$$Bi_{tbc} = \frac{h_g t_{tbc}}{k_{tbc}}$$

$$h_{c,eff} = \frac{h_c}{1 + Bi_w} \quad \text{.............5.25}$$

where wall Biot number,

$$Bi_w = \frac{h_c t_w}{k_w}$$

Where, t_{tbc} and k_{tbc} are the thermal barrier coating thickness and thermal conductivity, respectively. Similarly, t_w and k_w are the metal wall thickness and thermal conductivity. From the above relationship, it can be observed that for the extreme cases, when ε = 0, $T_m = T_{hg}$, the airofoil metal temperature equal the gas temperature, and when ε = 1, $T_m = T_c$ and the airofoil metal temperature equals the coolant temperature. For most gas turbine blades ranging from the rear to the front turbine stages, the effectiveness values are respectively in the range of 0.1–0.7.

Fig. 5.8 shows the relationship between the cooling effectiveness, mass flow function and the cooling efficiency. Here, the cooling efficiency and the mass flow function parameters are plotted on the horizontal and vertical axis respectively, which makes it easier to compare the cooling efficiencies of different turbines.

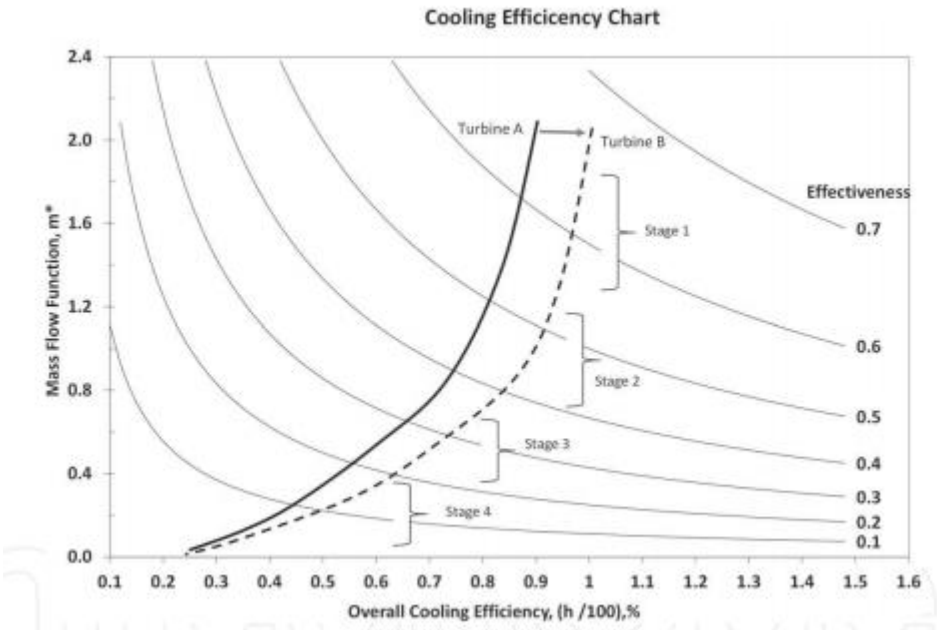


Fig. 5.8. Heat transfer performance chart for gas turbine blades.

From Fig.5.8, it is clear the front stages of the gas turbine, which are exposed to the highest hot gas temperatures, will generally have the highest cooling effectiveness and efficiency levels as their cooling designs will include film cooling, thermal barrier coatings, impingement cooling, turbulator convective cooling, and advanced alloys. The rear stages which are generally exposed to the lowest hot gas temperatures are generally convectively cooled, consume the least amount of cooling air, and are represented by the lowest effectiveness and efficiency values.

5.2.3 Detailed Aerothermal designs

During the detailed design phases, the design of cooled turbine airofoils is normally done using design systems which incorporate the effect of all three-dimensional geometrical and aerothermal effects. There is extensive use of computational fluid dynamics, as part of the overall turbine design process and the thermal analyses are based on conjugate heat transfer-based model. Fig. 5.9(a) shows a typical example of a gas turbine blade conjugate heat transfer model, where both the internal coolant flows in the internal cooling passages and the external heat loads on the airofoil hot gas surfaces are directly simulated. Fig. 5.9(b) shows a typical example of the predicted metal temperature on a turbine vane based on a conjugate heat transfer model and compared to measured metal temperatures from a test engine.

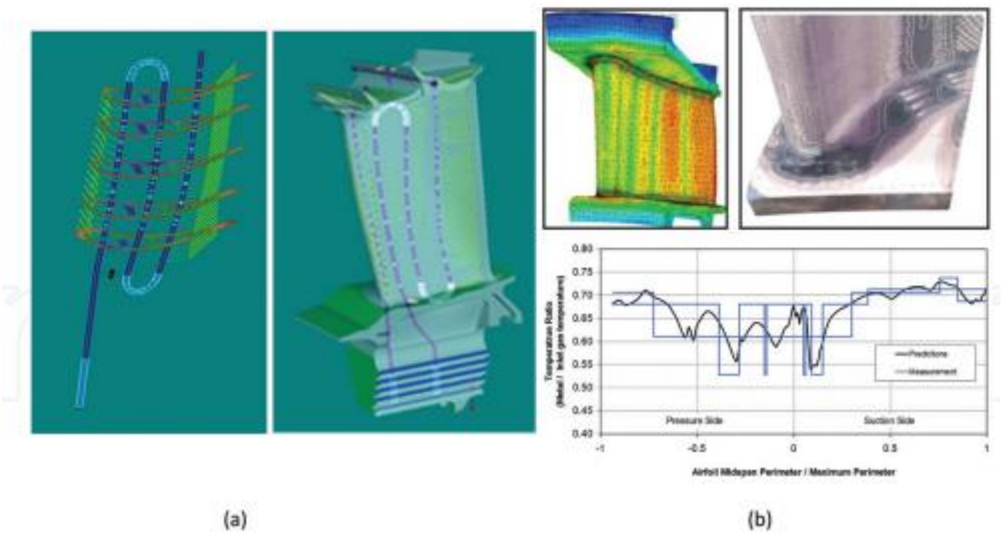


Fig. 5.9. Detailed airofoil aerothermal design using (a) conjugate thermal modelling [9], and (b) 3D thermal modelling and comparisons with measured engine

5.2.4. External heat transfer of cooled turbine airofoils

The aerodynamics of the gas path flows through the static turbine vanes and rotating blades consist of a range of flow phenomena and flow structures such as accelerating sonic and transonic flows, unsteady flows, separated flows, secondary flows, overtip leakage flows, and interacting flows between the main gas path flows and coolant and leakage flows. To enhance the turbine aerodynamic efficiency and manage the external heat loads, significant research efforts have been made over the past decade to minimize the energy losses which are associated with the latter flow phenomenon. Similarly, there has been a significant research in understanding and minimizing the external heat transfer on the turbine airofoils and, endwalls, which is essentially defined by the gas path aerodynamics, thermodynamics, turbine geometrical annulus, and the geometrical profiles of the airofoils.

Airofoil external heat loads

The aerodynamic development of the boundary layer on the turbine static and rotating airofoils is highly nonuniform, and it largely determines the absolute levels of the external heat transfer coefficient to which it will be exposed. Other factors that significantly influence the airofoil heat transfer include the mainstream turbulence, profile curvature, streamwise pressure gradients, surface roughness, upstream wakes, and film cooling. Fig 5.10 shows the Mach number measured on a turbine vane and blade, and highlights; (a) the strong Mach

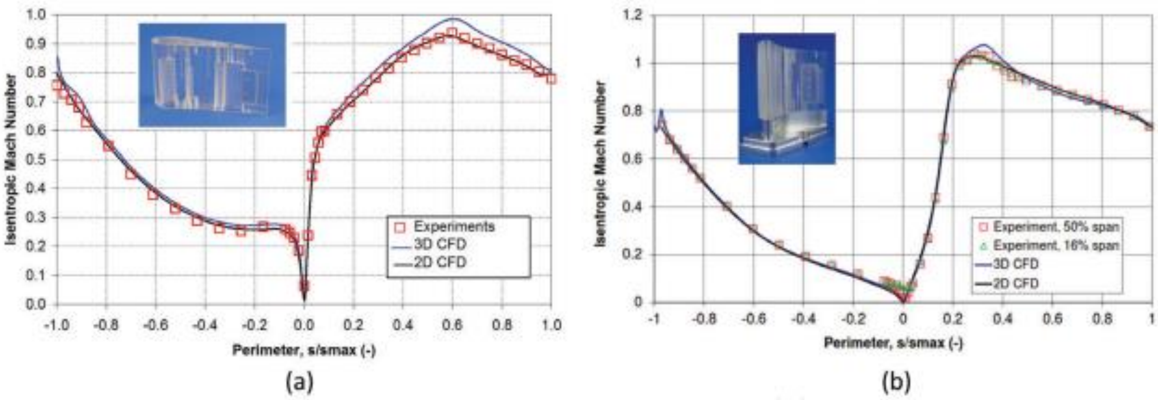


Fig. 5.10. Aerodynamic measurements and predictions on a 1st stage (a) vane and (b) blade

number variations near the leading edge stagnation point, (b) accelerating flow on the pressure and suction sides immediately downstream of the leading edge, (c) region of transitional boundary layer, (d) regions of accelerating turbulent flows on the pressure side, and (e) regions of peak Mach numbers on the suction side followed by decelerating flows towards the trailing edge. It is this variation in the profile Mach number which largely determines the vane and blade external heat transfer coefficients.

The detailed distribution of the heat transfer coefficients on the turbine vane and blade of a high pressure turbine was measured for a range of operating conditions. Fig.5.11 shows the distribution of the measured and predicted Stanton numbers, (St = Nu/ Re.Pr) at 50% airofoil span and at Re/L = 3.1 e6, and clearly highlights the differences in the heat transfer distribution between the vane and the blade. This is largely due to the different profile shapes, leading edge diameters, Mach number distributions, and the overall pressure ratio across the vane and blade. Fig. 5.11 highlights the heat transfer distribution associated with the various aerodynamic flow regimes on the aerofoil. For the vane, the heat transfer coefficient increases from the leading edge to the suction side, reaches a peak value, and then decelerates towards the trailing edge. On the pressure side, it reduces from the leading edge and after transition, continuously accelerates up to the trailing edge. For the blade, the peak heat transfer coefficient value is at the leading edge, which then decreases gradually on the

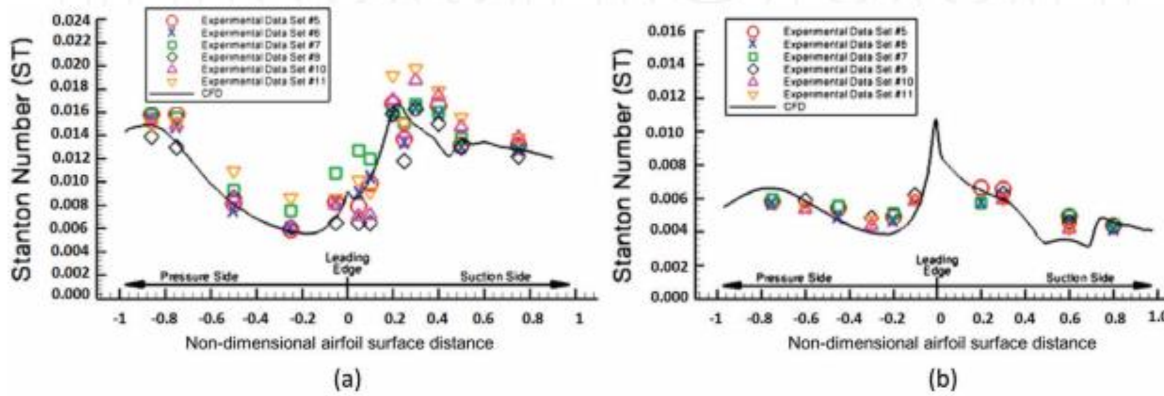


Fig. 5.11. External heat transfer measurements at 50% span, (a) 1st stage vane, and (b) 1st stage blade.

suction side until the trailing edge. However, on the pressure side, the heat transfer coefficient reduces rapidly from the leading edge, and then there is a transition to higher values until the trailing edge. These typical trends in the nonuniformity of the heat transfer coefficient are generally observed on most turbine vanes and blades. However, in addition to these generalized aerofoil heat transfer distributions, actual industrial gas turbines blades are also affected by several other parameters, such as; inlet pressure and temperature profiles, aerofoil shape and curvature, position of film cooling holes, thermal barrier coating roughness, transient wakes from upstream vanes, and blade passage turbulence intensity levels.

Endwall external heat loads

At the vane and blade endwall or platform, the aerodynamic flows are highly three-dimensional, transonic, and consist of areas where the hot gas flow strongly interacts with cooler rim purge and leakage flows. Fig 5.12(a) highlights the salient features of the hot gas path flow interactions on the platforms, which are largely pressure driven flows generated by the cross passage pressure differences on the pressure and suction side of neighbouring aerofoils. Over the last decade, there has been a significant experimental and numerical research effort to understand the behaviour and impact of these high speed endwall flows on the platform heat transfer. Some of the key factors which define the platform heat transfer include the inlet profile of hot gas temperature, pressure and turbulence intensities, film cooling, platform contouring, and impact of leakage and rim purge flows.

Fig. 5.12(b) shows the heat transfer and film cooling distributions on a first stage vane. The heat transfer coefficient distribution shows that the suction side shoulder and the pressure side trailing edge regions experience the highest heat transfer coefficients, which also correspond to the areas with the highest Mach numbers. For the vane platform film cooling effectiveness without the upstream purge flows, Fig 5.12(c) shows that the measured and predicted film cooling effectiveness compares quite well, and the films remain attached to the passage wall and are very effective in cooling the platform. Due to the three-dimensional nature of the endwall flows, Fig. 5.12 highlights that the magnitude and directions of the local velocity, temperature, and pressure distributions play a dominant role in the heat transfer distributions on aerofoil platforms

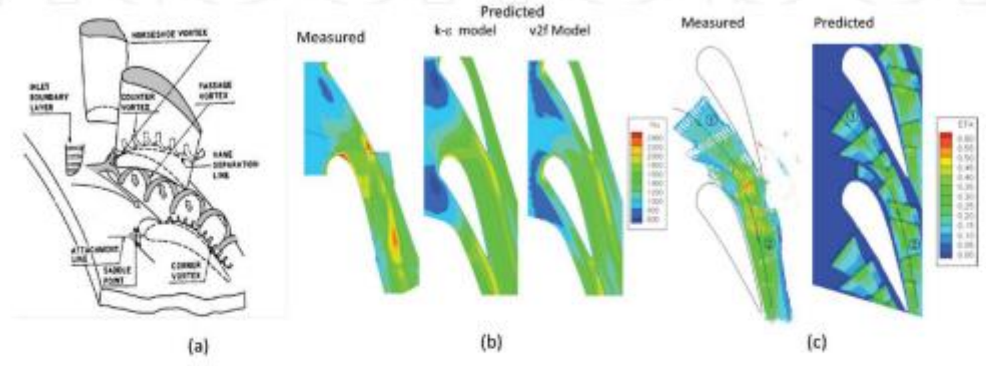


Fig. 5.12. Endwall flow and heat transfer, (a) flow structures, (b) heat transfer coefficients, and (c) film cooling effectiveness

Blade tip and endwall external heat loads

The blade tip and its neighbouring endwall regions are one of the most complex aerodynamic and heat transfer areas of the gas turbine. Figure 5.13 shows some typical blade tips designs ranging from flat tips, squealers and shrouded tips, and its impact on the turbine efficiency. For flat tip and squealer tip designs, this region is dominated by the pressure driven overtip leakage flow from the aerofoil pressure side to the suction side. This flow then travels through the narrow gap between the rotating blade and the static casing endwall, and subsequently interacts with the main cross passage flows to form a high speed vortex on the tip suction side. For the shrouded blade, the gas flow is from the leading to the trailing edge. The hot gas flows then interact within the rotating shroud fins with the shroud cooling air. Due to the complex flow structure and high heat loads at the blade tips, an accurate knowledge of the local aerodynamics and heat transfer is important for ensuring that the mechanical integrity of the blade tips are ensured for long operating periods, especially at higher gas turbine operating temperatures.

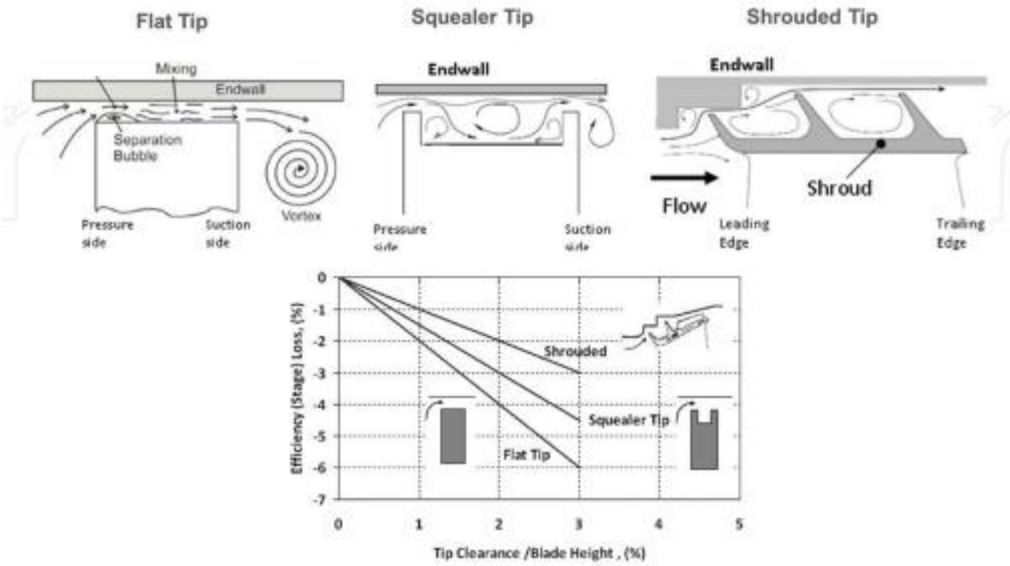


Figure 5.13. Typical blade tip designs and performance characteristics

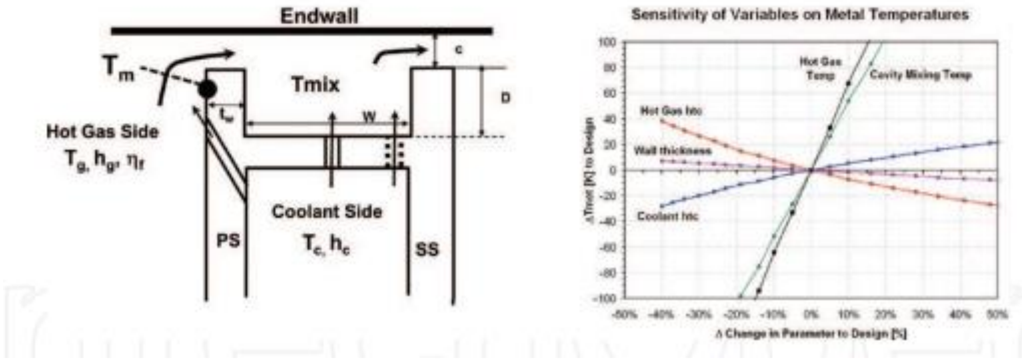


Figure 5.14. Sensitivity of operating conditions on blade tip heat transfer.

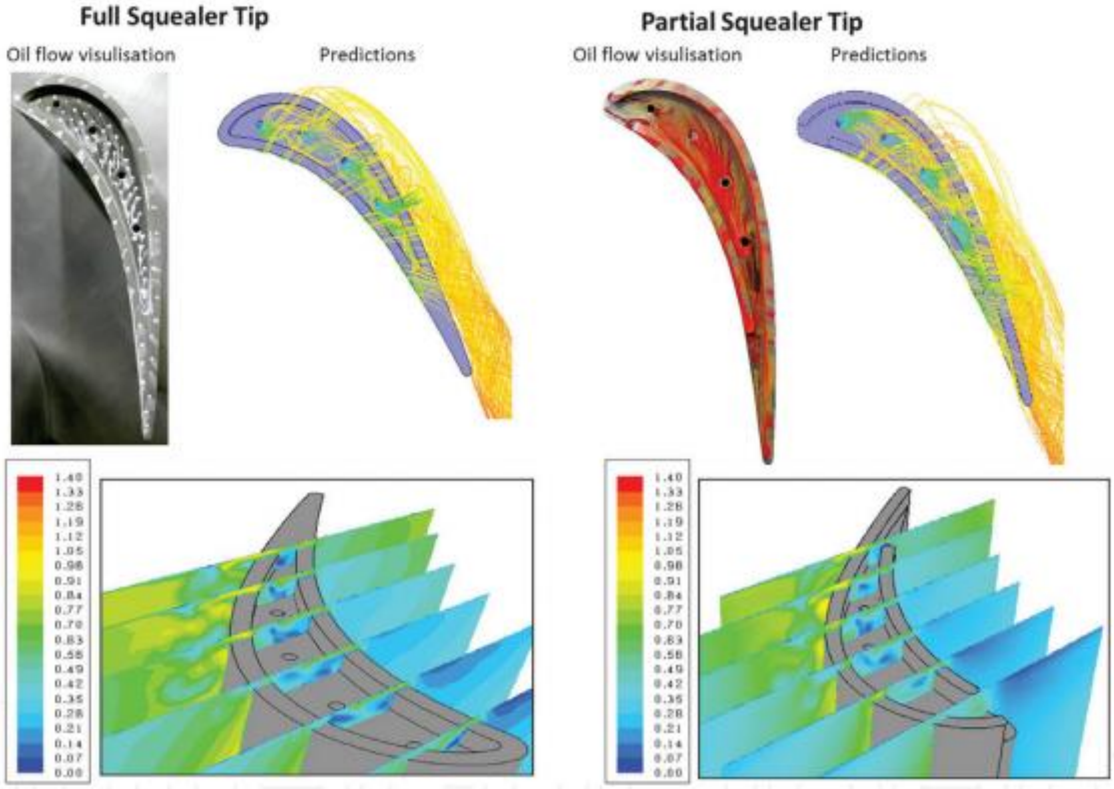


Figure 5.15. Flow structure and Mach number distributions for full and partial squealer blade tips

Figure 5.14 shows the sensitivity of the key parameters which influence the metal temperature of a typical squealer blade tip design. The main parameters influencing the tip metal temperatures are the hot gas temperatures and the cavity mixed temperatures. Other parameters such as the wall thickness and the heat transfer coefficients also play a major role in determining the tip metal temperatures. The heat transfer distribution on the blade tip and the endwall is highly nonuniform and driven largely by the local Mach number distributions and the tip geometry.

Figure 5.15 shows the flow distributions for two squealer tip designs and highlights the complex flow structure within the tip crown and the flow interactions between the tip leakage, main hot gas flows, and the coolant within the blade passage. For these two blade tip designs, Figure 5.16 also shows the measured and predicted heat transfer coefficients on the blade with film cooling. Both measurements and predictions show that on the blade tip, very high values exist in the leading edge regions and on the suction side rims. However, on the neighbouring endwall, the high heat transfer regions are largely location on the blade pressure side and towards the trailing edge.

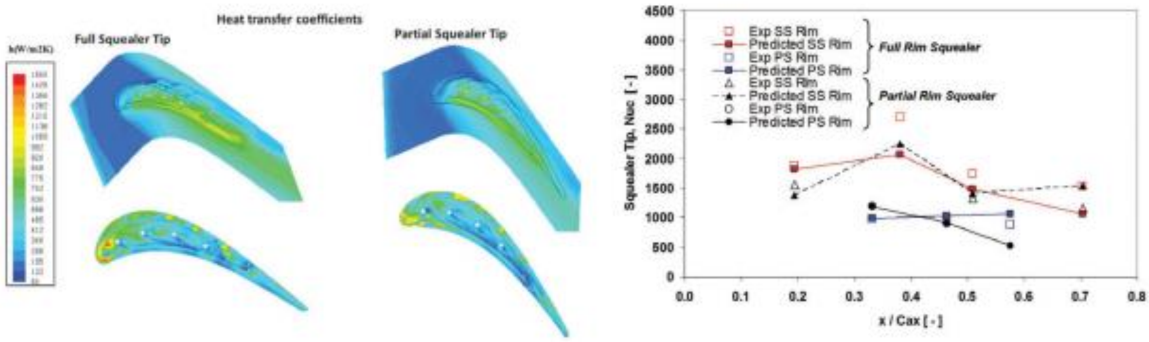


Figure 5.16. Heat transfer coefficient distributions on blade tip and endwall for full and partial squealer blade tips

<u>Thermal barrier coatings</u>

The use of high temperature thermal barrier coatings (TBC) for reducing the incident heat flux on both static and rotating gas turbines blades is extensive in gas turbines, particularly in the first and second turbine stages. There are essentially two main types of TBC, which are in widespread use in the gas turbine industry, namely air plasma sprayed (APS) and electron beam physical vapour deposition (EBPVD). For heavy duty gas turbines, the APS TBC is widely used with thickness which can range from 100 to up to 600 μm. The thermal impact of thermal barrier coatings on the turbine blade thermomechanical integrity is significant, and they therefore play an important role as a thermal protection system for gas turbine components. As highlighted previously, in the thermal analysis of turbine blades, the thermal barrier coating is generally represented as a thermal resistance to the incident heat flux, by modifying the hot gas transfer coefficient via the thermal barrier coating Biot number. Figure 5.17 shows that by increasing the thickness of the thermal barrier coating and reducing its thermal conductivity, the effective hot gas heat transfer coefficient can be significantly reduced. This results in a direct reduction of the incident heat flux on the turbine blade.

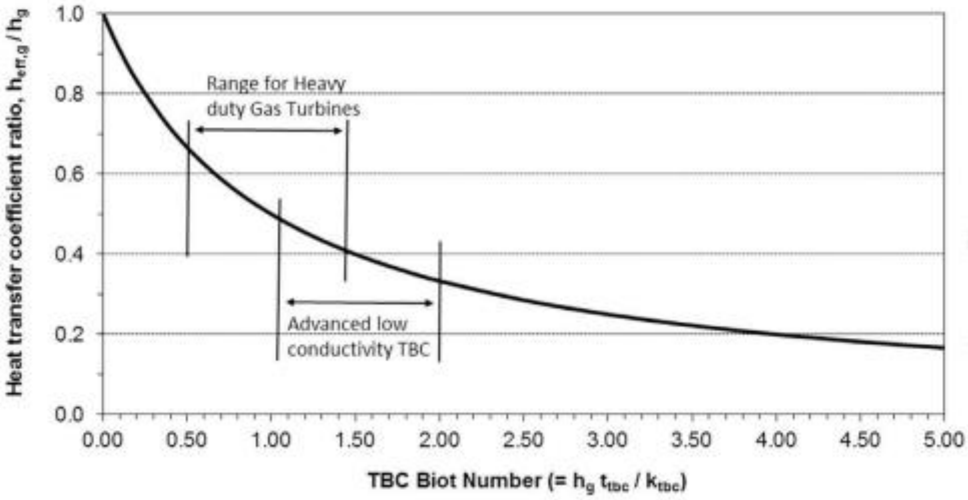


Figure 5.17. Effect of thermal barrier coatings (TBC) on heat transfer.

For typical heavy duty gas turbines, Figure 5.17 shows that the thermal barrier coating reduces the effective heat transfer coefficient by almost 50% compared to no application of the TBC. Additionally, Figure 5.17 shows that, for the new generation of advanced TBC's, with lower thermal conductivity, the effective heat transfer coefficient can be further reduced. It is clear from Figure 17, that thermal barrier coatings are an integral and significant part of the overall blade thermal design system.

Film Cooling

Film cooling is generally applied at different locations along the perimeter of an aerofoil by rows of discrete holes, through which coolant air is discharged into the aerofoil external boundary layer. The coolant, which is several hundred degrees colder than the hot gas, then creates a film of air on the aerofoil surface, whose temperature is significantly lower than the surrounding hot gas. Consequently, the incident hot gas temperature for heat transfer is reduced. Figure 5.18 shows an example of the application of a film row at the blade trailing edge and the key parameters which define the performances of film cooling. Figure 5.19 shows that as the average film cooling effectiveness on a turbine blade is increased, the film to hot gas temperature ratio reduces and the film temperature close to the wall can be reduced by several hundred degrees relative to the surrounding hot gas temperature.

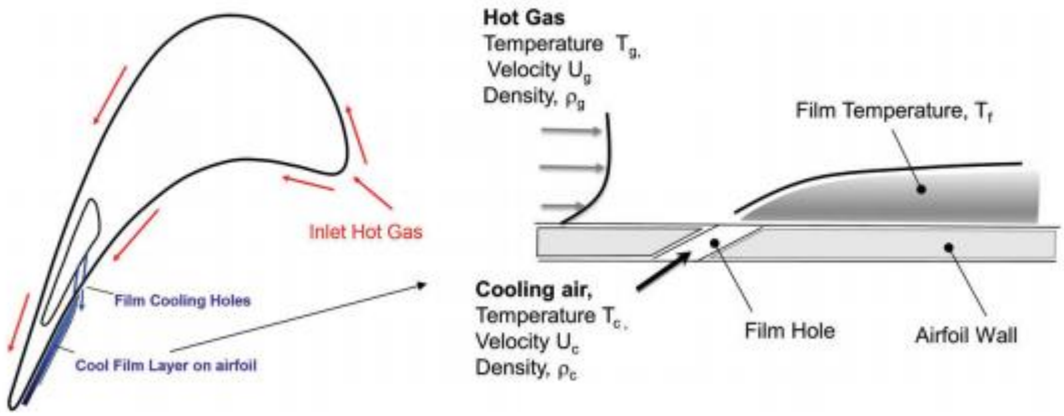


Figure 5.18. Film cooling of gas turbine blades.

Increasing the average film cooling effectiveness can be achieved by using many film rows, but this would be at the expense of high coolant consumption and reduced turbine efficiencies. Alternatively, increased film cooling effectiveness can also be achieved by using advanced film cooling hole designs without increasing the coolant consumption.

The film cooling effectiveness depends on the complex aerothermal interaction between the high speed hot gas flow and the ejected film cooling jets in the external gas boundary layer. It is also dependent on several geometrical and operational parameters such as film cooling hole shape, hole angle, velocity and temperature of the ejected coolant, temperature and velocity of the surrounding hot gas, blade curvature, and local turbulence levels. As highlighted in Figure 5.19, increasing the film cooling effectiveness results in significant reduction in the film to hot gas temperature ratio, and hence there continues to be a significant research effort on developing film cooling technology due to its significant benefits in reducing local near wall hot gas temperatures. Over the last decade, there has been a significant focus on airofoil, platform, and blade tip film cooling with more recent focus on advanced shapes of film cooling holes, such as three-dimensional shaped holes and trench holes. In a recent study, the multirow film cooling characteristic on a high lift vane and blade were demonstrated. Figure 5.20 shows that the use of three-dimensional advanced fan shaped holes can provide high airofoil average film cooling effectiveness and the use of only one or two row of shaped holes located upstream of the suction side shoulder can provide high film cooling effectiveness until the trailing edge.

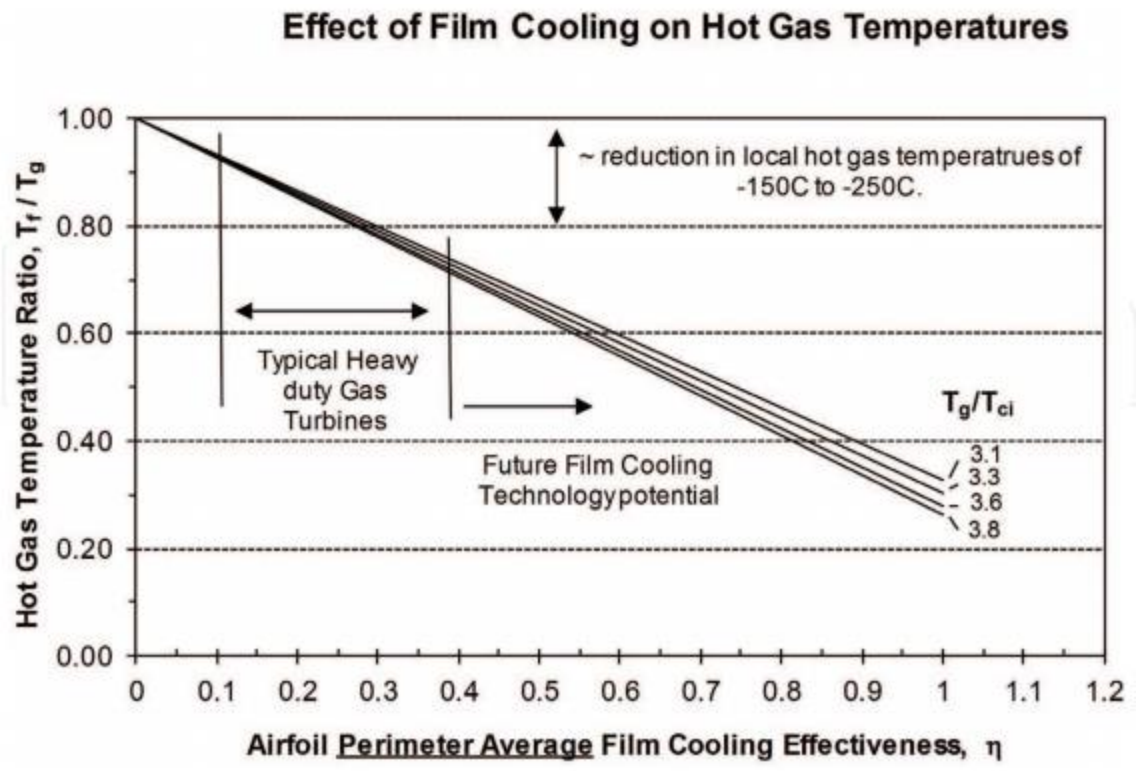


Figure 5.19. Effect of film cooling effectiveness on hot gas temperatures.

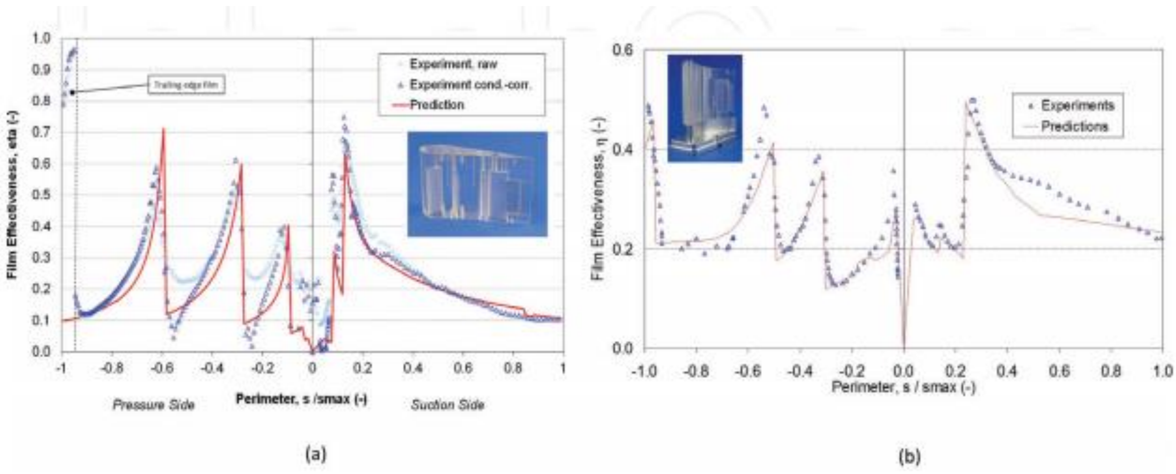


Figure 5.20. Multi row film cooling characteristics on a gas turbine (a) vane and (b) blade

Internal heat transfer of cooled turbine airofoils

The need for the internal cooling of gas turbine blades is primarily defined by the magnitude of the incident heat load on the aerofoils, which range from 0.5 to 5 MW/m^2 , and the requirements of the component durability for long operating hours against thermomechanical fatigue (TMF), low cyclic fatigue (LCF), creep, oxidation, and high cyclic fatigue (HCF).

While the external aerofoil profile defines the aerofoil aerodynamic performance, the internal cooling geometry is defined by the amount of coolant required to maintain the aerofoil at a certain material temperature and the temperature gradients across critical wall sections of the aerofoil. Figure 5.21 shows some typical examples of turbine vane and blade cooling designs. The internal heat transfer technologies used in these vanes and blades include impingement cooling, turbulators or ribs, pin or pedestal banks, dimples, shaped internal passages, and combinations of the above cooling features.

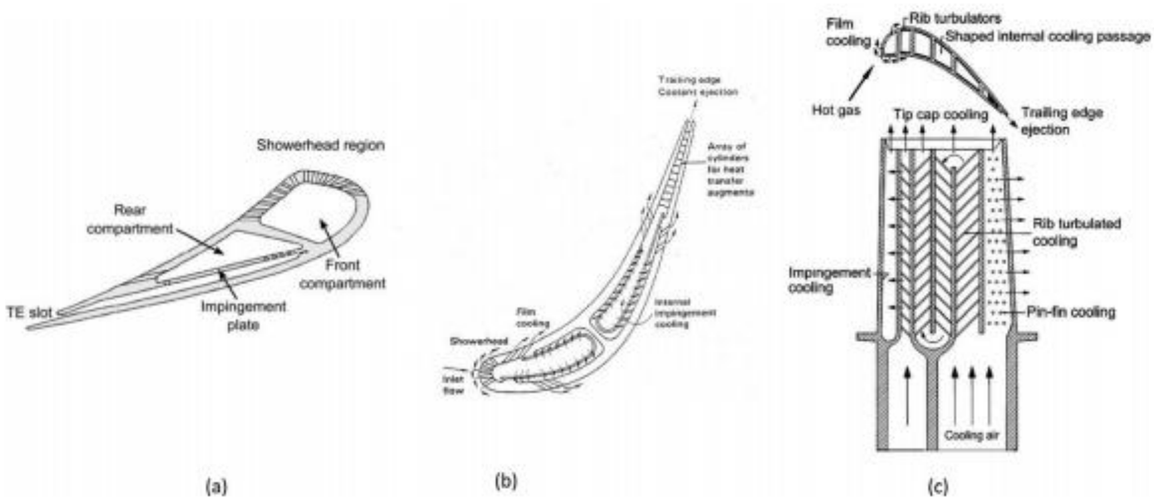


Figure 5.21. Typical blade cooling designs, (a) nozzle guide vane], (b) turbine vane, and (c) turbine blade

<u>Convective cooling with jet impingement</u>

Impingement cooling is widely used for the internal cooling of gas turbine components, particularly static aerofoils (vanes), heatshields (casing segments), combustor liners, and fuel nozzles. The impinging jets are generally formed through cylindrical holes in a thin wall insert, which is positioned adjacent to the aerofoil inner wall that is required to be cooled. They are normally directed as a single row of jets or as multiple rows of jets, and are generally injected normal to the target surface. For the midchord areas of aerofoils, impingement cooling is designed with multiple rows of jets and is directed on the pressure and suction sides of the aerofoil. The efficiency of the impingement cooling is defined by several parameters such as the standoff distance of the impingement jet relative to the target surface, the axial and radial pitch of the neighbouring impingement hole. For relatively flat surfaces, Figure 5.22 shows the impingement hat transfer for a flat surface with multiple impingement holes, which is based on the correlations. This figure highlights, that although jet impingement cooling is highly effective, the design of the impingement system requires careful consideration of several influencing parameters, such as the standoff distance from the target surface, the axial and lateral pitch of the impingement holes and the amount of crossflow from upstream jets.

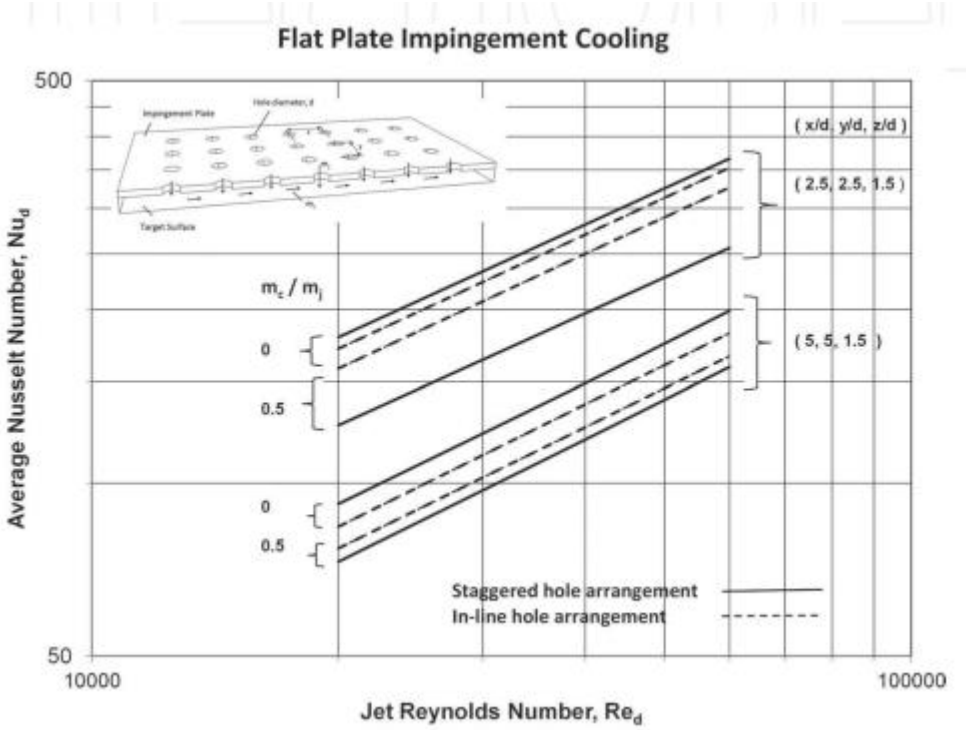


Figure 5.22. Impingement heat transfer with multiple rows on flat target surfaces.

Aerofoil curved leading edges are normally subjected to very high heat loads, and at these locations, internal impingement cooling in combination with turbulators and film cooling is quite common. Figure 5.23 shows the dependency of the standoff distance and the leading edge curvature on the coolant Nusselt numbers at varying Reynolds numbers, based on the correlation. Highest stagnation heat transfer can be achieved if the jets are arranged very close to the target surface and the highest average heat transfer are achieved for aerofoils with small internal leading edge diameters. At the internal leading edges, there are also several other additional factors that influence the airofoil heat transfer, such as showerhead film cooling, turbulators, surface roughness, and the amount of impingement crossflows. More recently, there have been several studies on the use of narrow channel impingement passages and inclined impingement jets in variable shaped passages. Such design configurations can provide higher internal heat transfers and have been mainly driven by the introduction of near wall cooling features in gas turbine blades. Such configurations can be manufactured with 3D printing technologies such as selective laser melting (SLM) and direct laser melting (DLM), which allows greater manufacturing flexibility with geometrically complex cooling passages.

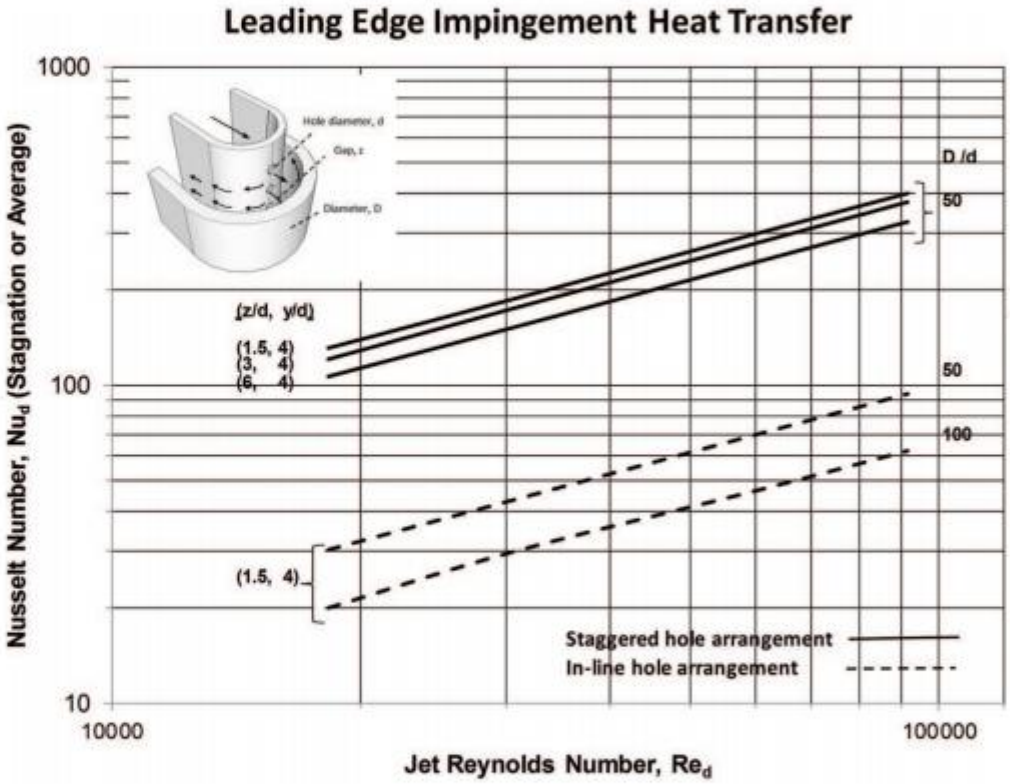


Figure 5.23. Impingement heat transfer on curved leading edge surface.

For narrow channel impingement, that in addition to the high heat transfer from the target surface, the heat transfer from the impingement cavity side walls can also be significant. Figure 5.24 shows that for a narrow channel with in-line impingement holes, the heat transfer from the side walls can be up to 50% of that from the target plate. The use of inclined impingement jets on shaped turbulators in irregular shaped passages can also result in very high heat transfer. In a recent study, and as shown in Figure 5.25, it was highlighted that directed inclined impingement can result in relatively high heat transfers from the target walls and additionally produces intense convective fluid mixing within the passage. In a turbine blade, such a combination can result in greater total heat removal by the coolant from the hot airofoil walls. Figure 5.26 shows that directing a double impingement jets on a curve leading edge with showerhead cooling results in high heat transfer coefficients at the pressure and suction surfaces, and additionally generates significant turbulent mixing within the leading edge passage. Based on the many different design variations of impingement cooling, it is expected that impingement cooling systems will continue to play a significant role in gas turbine heat transfer technology.

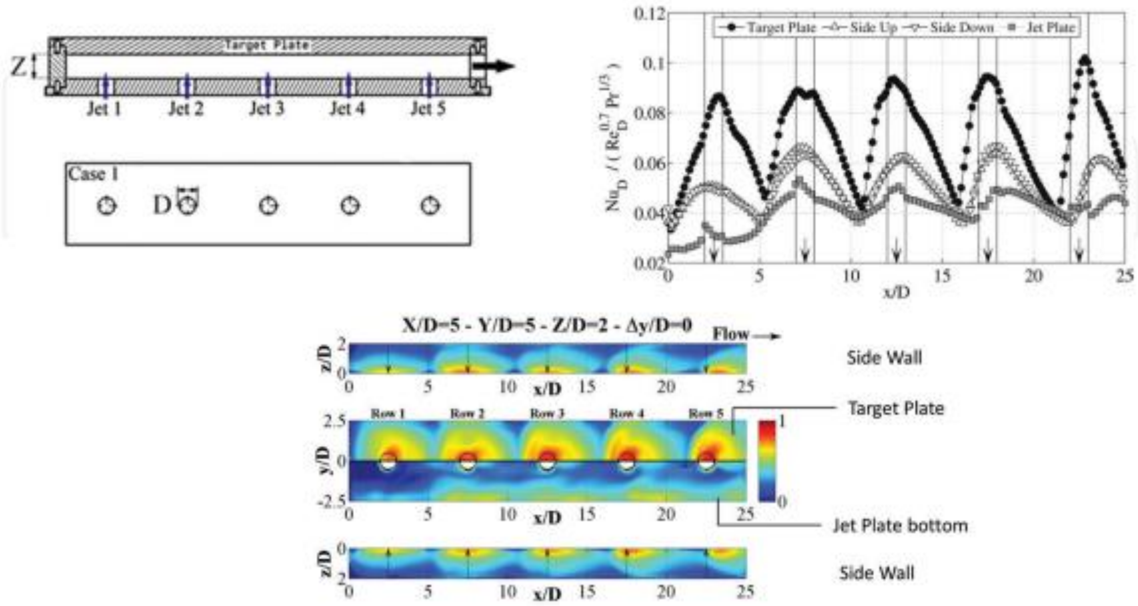


Figure 5.24. Impingement heat transfer in narrow channel passages

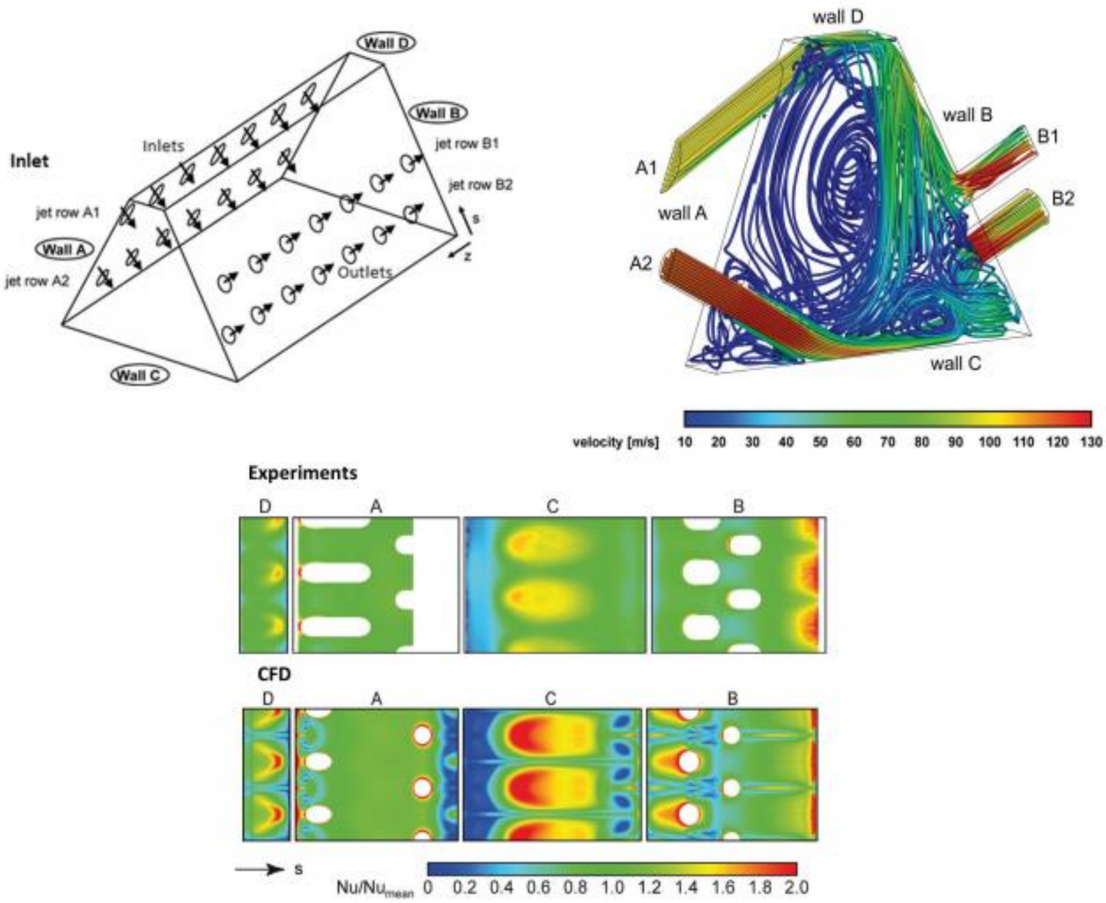


Figure 5.25. Impingement heat transfer in irregular passages

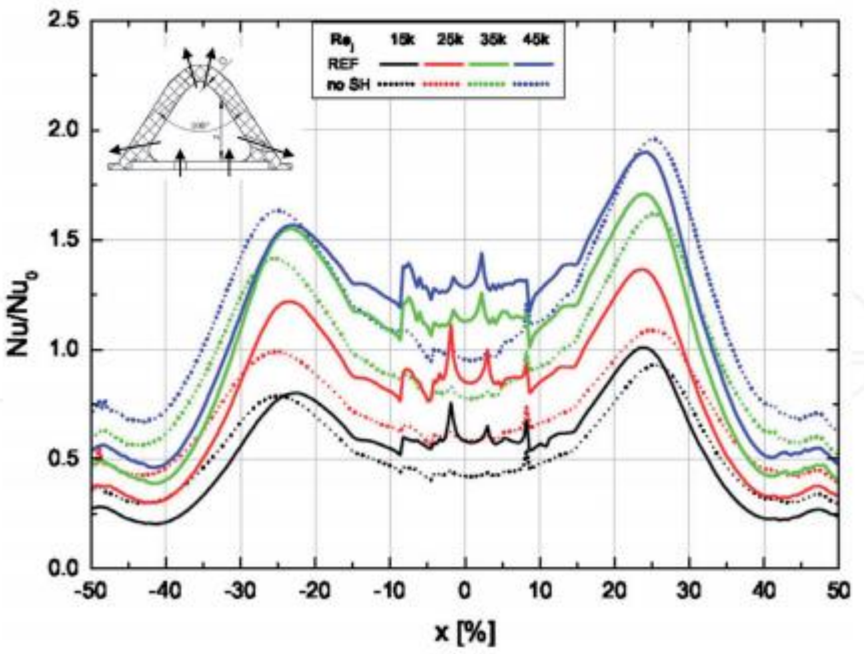


Figure 5.26. Impingement heat transfer in leading edge channels

<u>**Convective cooling with turbulators**</u>

The use of ribs or turbulators for cooling gas turbine blades is a major heat transfer technology and has been employed largely in rotating blades with radial passages as shown in Figure 5.27. The passages are typically arranged as multiple radial passages and commonly referred to as serpentine passages or multipass systems, and the turbulators are generally designed on the pressure and suction surfaces of the passages. The key function of the turbulators is to create regions of flow separation downstream of the turbulators, which promotes intense regions of turbulence, secondary flows, and rapid mixing between the air warmed by the heated walls and the core coolant flow. The nature of the flow structure and the amount of heat transferred in the passages with turbulators is dependent significantly on the turbulator shape, configuration pitch, height, angle of orientation, flow Reynolds number, rotation, passage shape, and its aspect ratio. There has been a significant amount of research conducted on the application of turbulators in gas turbine blades, including the effects of rotation, shapes, sizes, orientation, entrance length effects, position of film cooling holes, presence of bends and other enhancement devices, and operating parameters.

To assess the relative impact of different turbulators on the heat transfer and frictional characteristics, Figure 5.27 shows the ribbed wall heat transfer and passage frictional enhancement of turbulators in various aspect ratio passages. Figure 5.27 highlights that the turbulator angle and the shape of the passage have a major effect on both the passage heat transfer and pressure loss. Although smaller aspect ratio ducts (W/H = 0.25) generally give higher heat transfer enhancement on the ribbed walls and lower pressure losses, the passage average heat transfer values can be lower due to the larger perimeter of the nonribbed walls, which have much lower heat transfer enhancement. Although the results shown in Figure 5.27 are for low Reynolds number, and with idealized geometry, care needs to be taken when implementing such results in real turbine blades with cast geometries, where the dimensions, shape, and position of the turbulators can be different from the predicted idealized geometries.

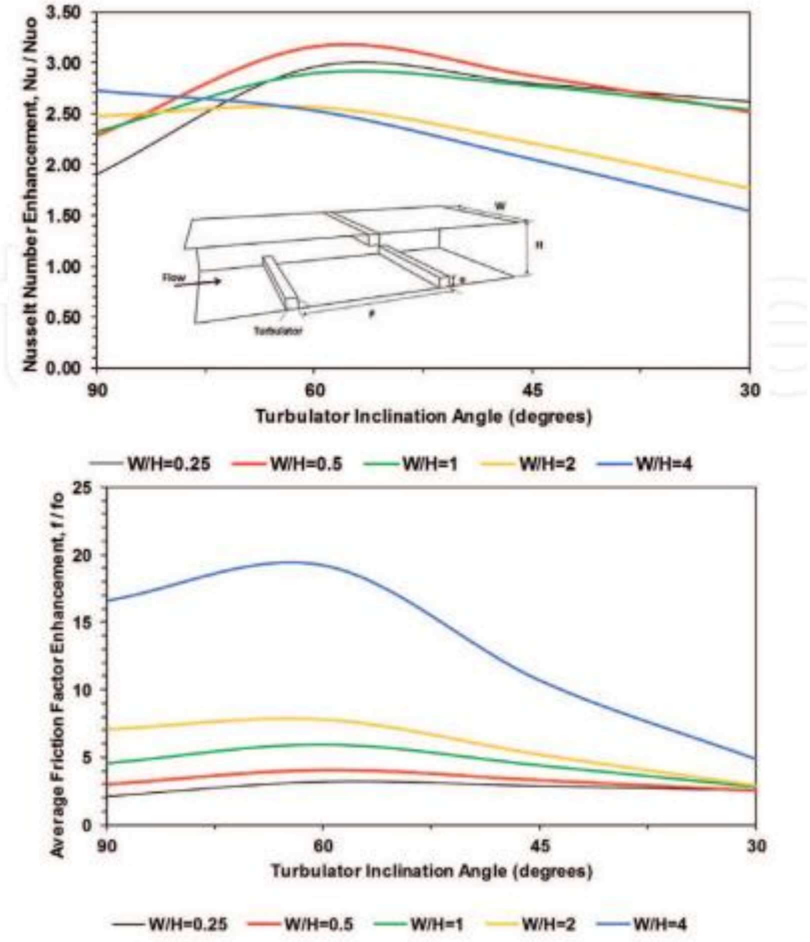


Figure 5.27. Passage heat transfer and frictional losses due to turbulator design and passage shape.

The impact of rotation on the heat transfer from gas turbine blades can be significant and is dependent on several additional parameters such as the rotational and buoyancy numbers.

Figure 5.28 shows a schematic overview of the impact of rotation on the flow field in a twopass rotating passage of a gas turbine blade. Under rotating conditions, Coriolis and buoyancy effects can significantly alter the temperature and velocity profiles within the passages. Figure 5.28 shows that with the coolant flowing radially upwards, the heat transfer with increasing rotation numbers, increases on the pressure side and reduces on the suction surfaces. Similarly, when the coolant flows radially inwards, the heat transfer increases on the suction side and reduces on the pressure side, especially for smooth passages. However, for passages with turbulators, the turbulators tend to dampen the effect of rotation and the heat transfer enhancement on the pressure and suction sides. This overall trend of rotation with different type of passages and turbulators has been observed by several studies, and these effects play an important role in the design of gas turbine blades.

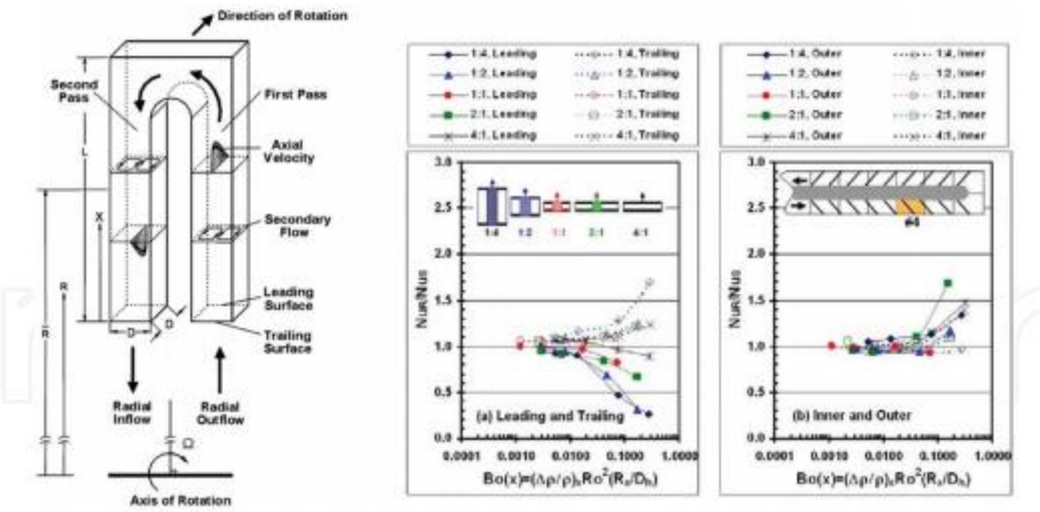


Figure 28. Impact of blade rotation on passage heat transfer

The trailing edge regions of rotating blades are in general the most difficult to thermally design, largely due to the thin aerofoil geometry, complex internal flow geometry, and the coolant flow conditions. The trailing edge region generally consists of coolant passages with very high aspect ratios, typically between 4 and 7, and which have cooling features such as turbulator and pedestals. Previous studies in such triangular and wedge-shaped passages with various turbulator shapes reported significant variation in the heat transfer distribution in both stationary and rotating cases. In a recent study, it has been shown that the impact of high Reynolds number typically found in heavy duty gas turbines can have a significant effect on the overall thermal performances of angled, broken, and chevron turbulators in a very large aspect ratio passage. Figure 5.29 shows the complex flow structures and the high three-dimensional heat transfer distribution which exist within the high aspect ratio triangular passages with different turbulator shapes. Figure 5.30 shows the comparison of the average thermal performances and shows that at high Reynolds numbers, the differences between the various designs are very similar. When considering the investigated turbulator design for gas turbine cooling applications, all three configurations show comparable levels of heat transfer performances.

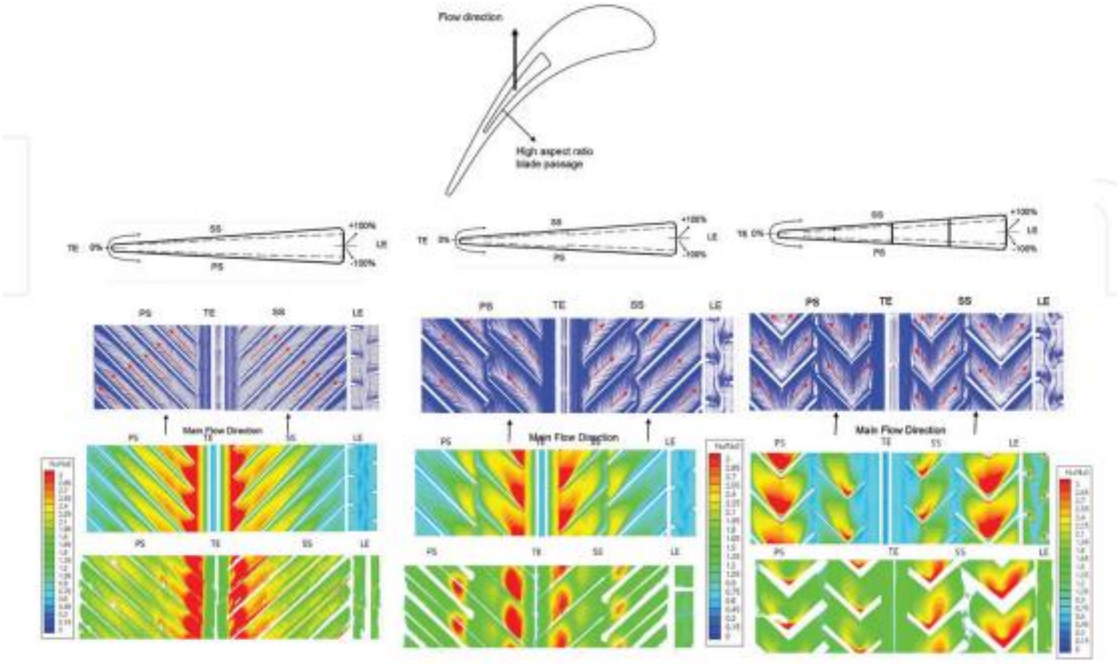


Figure 5.29. Impact of turbulator design in trailing edge passage

For leading edge passages of gas turbine blades, the application of turbulators is also widespread. However, due to the leading edge geometry, the heat transfer is significantly different to that in midchord or trailing edge passages. In a recent study several turbulator geometries were tested at engine representative Reynolds numbers. Figure 5.31 shows that the flow structure is significantly modified due to the presence of the turbulators, and this dominates the strength and distribution of the local and average heat transfer coefficients.

The final selection and implementation of turbulator designs in a turbine blade are dependent on several additional complex requirements. These include blade metal temperature, metal temperature gradients, cooling flow pressure margins, and the amount of required cooling flow. A further key criterion is for the fulfilment of the blade mechanical integrity, which is determined by the blade low cycle fatigue and creep behaviour, both of which are driven by the local metal temperature gradients and the absolute metal temperatures. An optimal balance of these factors is therefore needed to select the best turbulator concept for a blade design system.

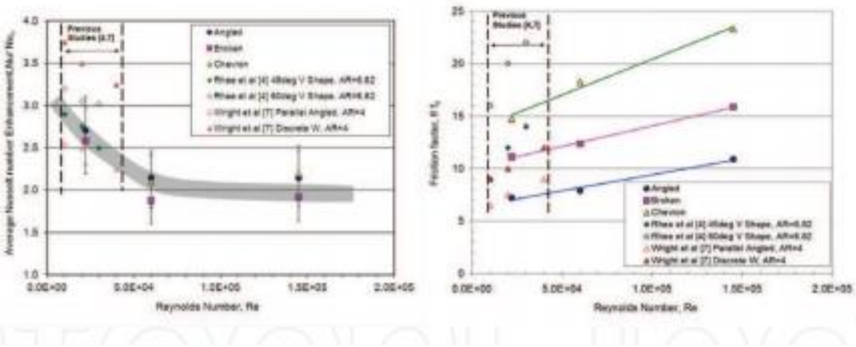


Figure 5.30. Average heat transfer and frictional loss in trailing edge passages

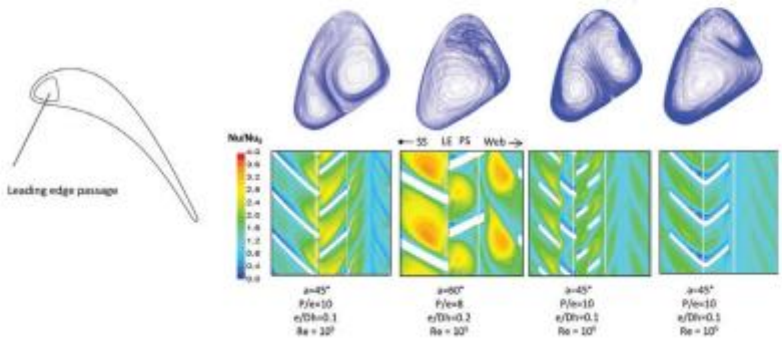


Figure 5.31. Heat transfer in leading edge passages

__Convective cooling with pins and pedestals__

The use of pins and pedestals for enhancing the internal heat transfer in gas turbine blades and vanes is quite common particularly at the aerofoil trailing edges, which generally demands aerodynamically small wedge angles and thin trailing edge diameters. Pin banks and pedestals are sometimes the only method of cooling in the space constrained narrow, converging trailing edges. They are also preferred from a manufacturing point as they tend to offer structural stability for casting. An additional advantage is that the pin banks also provide good mechanical integrity of the blades due to the robust structural support between the pressure and suction surfaces of the aerofoil. In general, the pins are cylindrical in shape, tend to be thick relative to their height, has fillets imposed at the interface with blade walls, and are typically arranged in a staggered pattern. Although they provide high heat transfer due to a combination of high heat transfer coefficients from the base wall and the pins, they also entail high pressure losses. The latter disadvantage is generally not a major issue for convectively cooled blades, where there is availability of higher coolant pressure ratios relative to the surrounding hot gas pressure at the blade trailing edge.

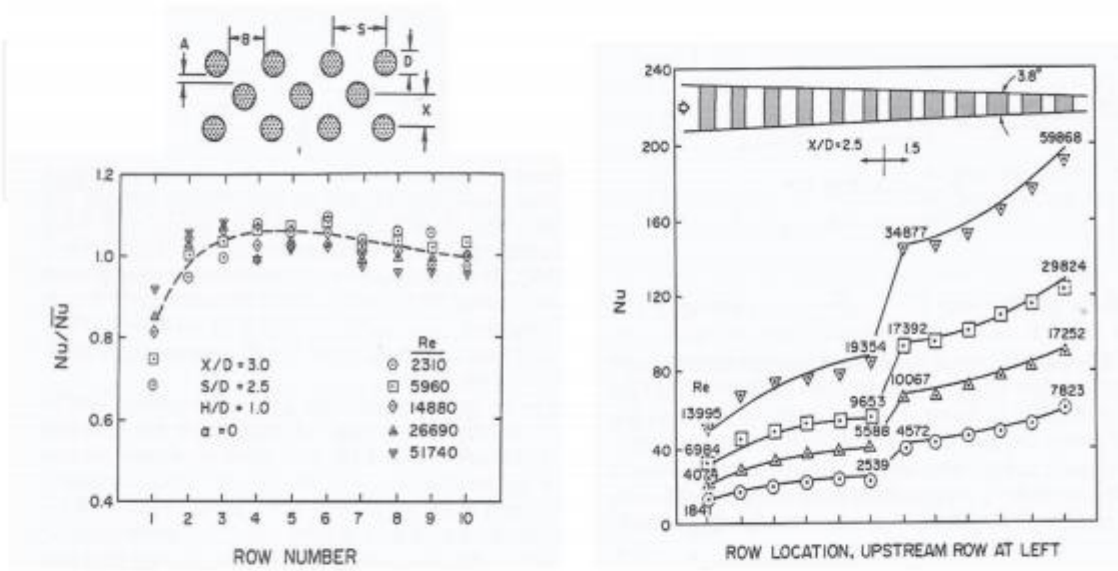


Figure 5.32. Heat transfer in pin banks and pedestals

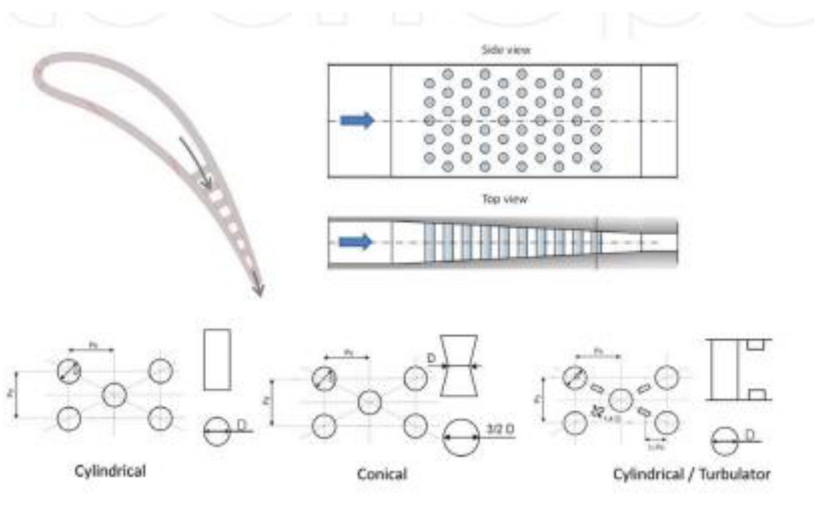


Figure 33. Trailing edge passages with different pin bank configurations

There have been a large number of heat transfer studies on pin banks which have addressed the influences of pin geometry, channel shape, arrangements, pin shapes and combination of pins with dimples and turbulators. For straight passages with pin banks, the local heat transfer generally increases from the first row of pins until the second to third row and then starts to decrease. For converging ducts, Figure 5.32 shows that with a converging duct, the heat transfer increases in the downstream section. Additionally, by using a thicker pin in the rear portion of the passage, further increase in heat transfer can be achieved, which is driven by the increased Reynolds number.

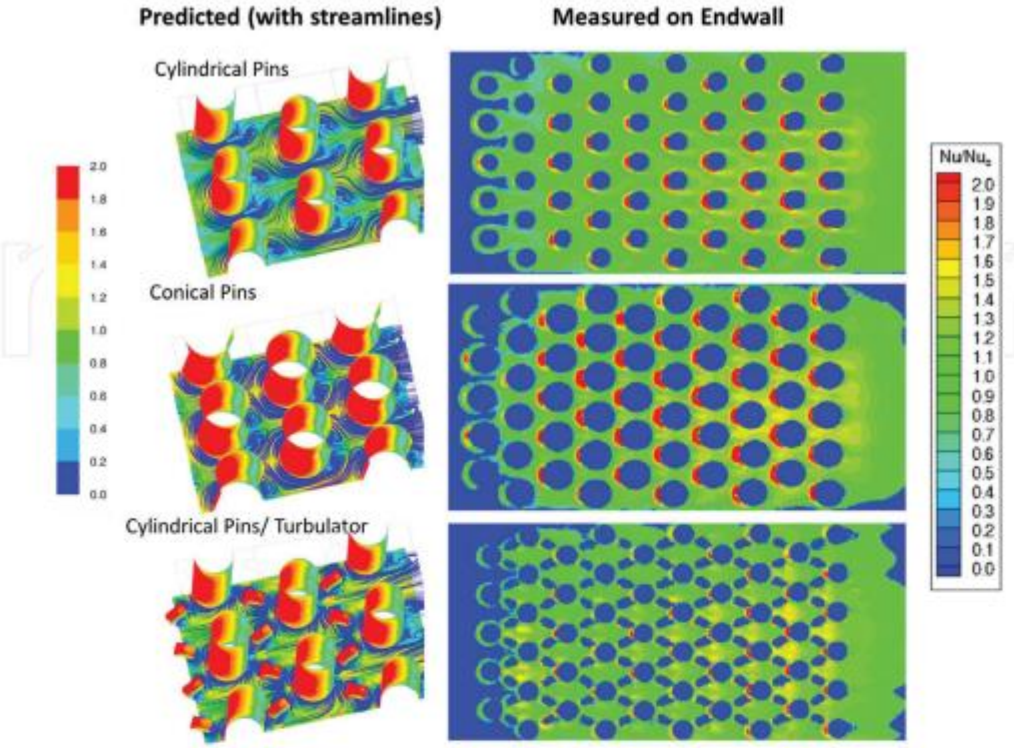


Figure 5.34. Heat transfer in trailing edge passages with different pin bank configurations

In a recent study, several pin fin configurations were investigated in a trailing edge converging channel which consisted of cylindrical pins, conical pins, and a hybrid cylinder pin/turbulator configuration. Figure 5.33 highlights the investigated geometries. Figure 5.34 shows from both predictions and measurements that the flow is highly turbulent downstream of the pins and that the complex heat transfer distribution exists on both the endwall and the pins. High levels of local heat transfer occur at the leading edge stagnation point of the pins and at the leading edge endwall. Lower heat transfer coefficients were predicted and measured in the wake region of the pin trailing edge. Laterally, averaged local distributions of the heat transfer enhancement are shown in Figure 5.34 for the tested geometries, and the nonuniform nature of the heat transfer in the pin banks are further highlighted.

5.3 Ablation

In aerospace design, ablation is used to both cool and protect mechanical parts that would otherwise be damaged by extremely high temperatures. Examples are heat shields for spacecraft, satellites, and missiles entering a planetary atmosphere from space and cooling of rocket engine nozzles. Basically, ablative material is designed to slowly burn away in a controlled manner, so that heat can be removed from the spacecraft by the gases generated by ablation, whereas the remaining solid material insulates the spacecraft from superheated gases. There has been an entire branch of spaceflight research involving the search for new fireproof materials to achieve the best ablative performance. Such a function is critical to protect the spacecraft occupants from otherwise excessive heat loading. The physical phenomena associated with ablation heat transfer depend on the application, but most involve in-depth material pyrolysis (charring) and thermochemical surface ablation. In numerical modeling, it is required to solve an energy equation, including effects of pyrolysis on a domain that changes as the surface ablates. Fig. 5.35 gives a general view of the ablation process and the involved mechanisms. As the material is heated, one or more components of the original composite material pyrolyze and yield a pyrolysis gas and a porous residue. The pyrolysis gas percolates away from the pyrolysis zone. The residue is often a carbonaceous char. The solution procedure is in principle a transient heat conduction calculation coupled to a pyrolysis rate calculation and subject to boundary conditions from the flow field.

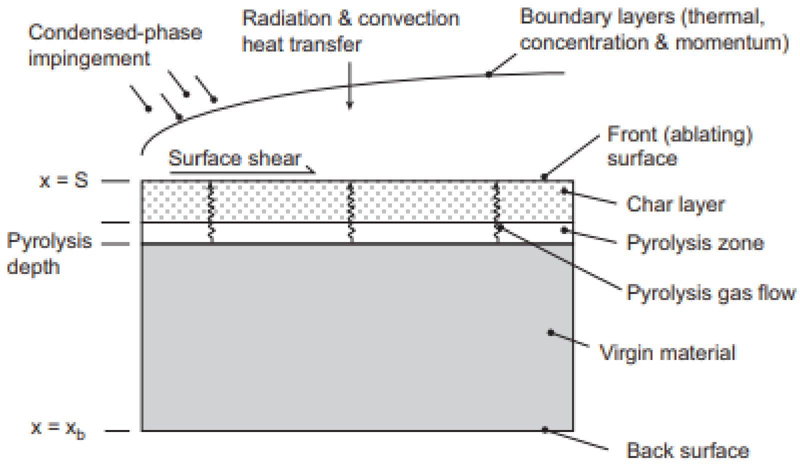


Figure 5.35 Ablation process.

In terms of heat transfer, the following mechanisms are involved: (a) convection in the boundary layer, which gives rise to the main thermal load; (b) radiation; and (c) conduction in the virgin material. In addition, resin decomposition and fiber decomposition may occur. In the boundary layer, shocks may appear, and in some cases, there may be combustion.

Ablation is affected by the freestream conditions, the geometry of the reentry body, and the surface material. The vehicles range from blunt configurations, such as spacecraft, to slender sphere-cone projectiles. At low heating values, ablators of Teflon are used, whereas at high heat loads, graphites and carbon-based materials are used. The most common ablative materials are composites, i.e., materials consisting of a high-melting-point matrix and an organic binder. The matrix might be glass, asbestos, carbon, or polymer fibers braided in various ways. A honeycomb structure filled with a mixture of organic and inorganic substances and with high insulating characteristics can be used. Advances in chemistry and materials technology extend the possibilities of selecting improved ablative materials.

Also surface movement occurs by, e.g., material spallation. Various methods exist to solve the moving boundary problems, commonly referred to as the Stefan problems. Basically two methods are considered: the fronttracking methods and the front-fixing methods. With the front-tracking methods, the ablating surface (the front) is tracked as it moves into the material.

<u>An Illustrative Example of Ablation</u>

Basically a transient heat conduction analysis is presented. The transient thermal response of the material exposed to a high-energy environment is important knowledge in the design of heat shields for reentry vehicles. The surface of a semi-infinite solid is heated by applying a constant heat flux q0 (caused by frictional or aerodynamic heating) as shown in Fig. 5.36

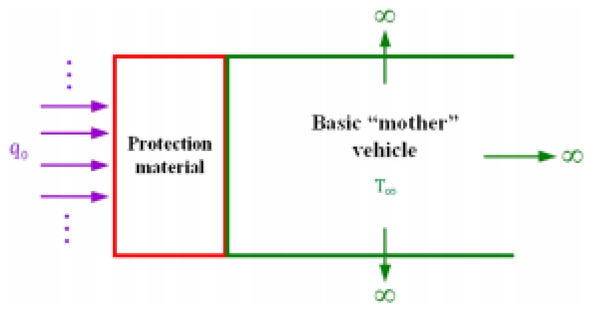


Figure 5.36 Simple illustration of an ablation process.

At time $\tau = 0$ the surface temperature has risen to the melting temperature Tp and phase change is initiated. The melted material (liquid) formed is completely removed by the aerodynamic forces. In this case the surface recedes with time but the surface temperature remains constant at the phase change temperature. A temperature distribution exists only in the remaining solid as conjectured in Fig. 5.37.

At a certain time the solid surface is located at x = X(τ). The temperature variation in the solid material penetrates to a depth x = δ(τ). The temperature of the solid at far distances, x > δ(τ), from the surface is kept at the constant initial temperature T_∞.

Introduce θ(x, τ) = T- T_∞, which implies that θ(x, τ) = T_p - T_∞ = $\theta_{p.}$ TN ¼. The one-dimensional unsteady heat conduction is then governed by

$$\frac{\partial\theta}{\partial\tau} = \alpha\frac{\partial^2\theta}{\partial X^2} \qquad \text{.....................5.26}$$

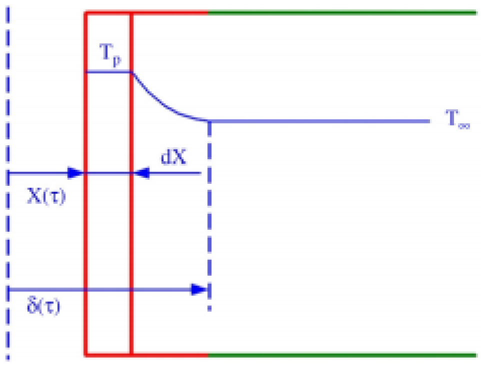


Figure 5.37 Temperature distribution in the simplified ablation process.

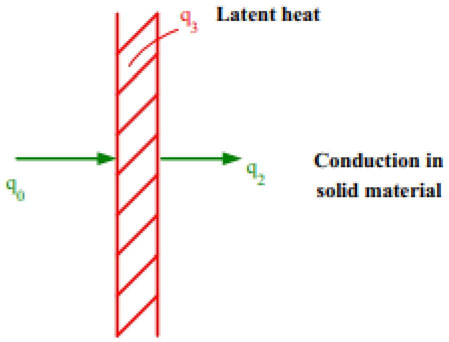


Figure 5.38 Heat balance at the liquid-solid interface.

In Fig. 5.38 the heat balance at the interface is presented. The thermal balance at the interface can be expressed as below:

$$q_0 - q_2 = q_3$$

$$q_0 + k\frac{\partial\theta}{\partial X} = \rho Q_L\frac{dX}{d\tau} \qquad \text{...............5.27}$$

The solution of Eq. (5.26) can be found by some standard procedures but here the integrated form is used. Details are given below.

The integrated form of Eq. (5.26) can be written as,

$$\frac{d}{d\tau}\left[\int_{X}^{\delta}\theta(x,\tau)dx - \theta(\delta,\tau)\cdot X\delta(\tau) + \theta(X,\tau)X(\tau)\right]$$
$$= \alpha\left[\frac{\partial\theta}{\partial x}(\delta,\tau) - \frac{\partial\theta}{\partial x}(X,\tau)\right]$$

……;.. 5.28

A second-order polynomial temperature profile is assumed as

$$\frac{\theta(x,\tau)}{\theta_P} = \left[1 - 2\frac{x-X}{\delta-X} + \left(\frac{x-X}{(\delta-X)}\right)^2\right]$$ **……….5.29**

With the conditions $\theta(\delta, \tau) = 0$ and $\theta(x, \tau) = \theta_P$. By combining the interface heat balance Eq. (5.27) with Eq. (5.28) and the assumed temperature profile Eq. (5.29), one obtains

$$\frac{d}{d\tau}\left[\frac{1}{3}(\delta - X)\right] + \left(1 + \frac{Q_L}{c\theta_P}\right)\frac{dX}{d\tau} = \frac{q_0}{\rho c\theta_P}$$

………………….5.30

Further combination of the equations gives,

$$\rho QL\frac{dX}{d\tau} = q_0\frac{2k\theta_P}{-\delta - X}$$ **………………………5.31**

Eqs. (5.30) and (5.31) are simultaneous equations, making the solutions of $X(\tau)$ and δ-X possible. The initial condition for X is X(0)=0. The initial condition for $\delta(\tau)$, the specification for δ when the surface reaches the melting temperature Tp, can be obtained from solutions of a semi-infinite solid exposed to a time varying surface heat flux by the equations

$$\delta(\tau) = \sqrt{6\alpha\tau}$$

which gives the value of $\delta(\tau)$ if a parabolic temperature profile is assumed, and

$$\delta(\tau) = \sqrt{1.5\alpha}$$

which gives the specification for the surface temperature as a result of a constant heat flux input qo. Using the above equations the value of $\delta(\tau p)$ when the surface temperature reaches θp may be calculated as

$$\delta(\tau_P) = 2\frac{k\theta_P}{q_0}$$ **………………5.32**

which is the remaining required initial condition.

Eqs. (5.30) and (5.31) have a steady-state solution if $dX/d\tau = A$ is a constant. Thus from Eq. (5.31), the value of δ -X must be constant, and then Eq. (5.30) gives

$$\frac{dX}{d\tau} = A = \frac{q_0}{\rho(Q_L + c\theta_P)} \qquad \text{..................5.33}$$

Goodman has solved Eqs. (5.30) and (5.31) by eliminating $dX/d\tau$ between them to obtain finally

$$\Omega = -\frac{1}{3}\left[\zeta - 2 + 2(1+\nu)\ln\frac{2(1+\nu)-\zeta}{2\nu}\right] \qquad \text{..................5.34}$$

where

$$\Omega = \frac{\tau q_0^2}{\rho k \theta_P Q_L}, \quad \nu = \frac{Q_L}{c\theta_P}, \quad \zeta = \frac{(\delta - X)q_0}{k\theta_P}$$

Substituting Eq. (2.9) into Eq. (2.6) yields

$$\lambda = -\frac{1}{3}\left[\zeta - 2 + 2\nu\ln\frac{2(1+\ \nu)-\zeta}{2\nu}\right] \qquad \text{..................5.35}$$

Where

$$\lambda = \frac{Xq_0}{k\theta_P}.$$

The results above are depicted in Fig. 5.39. It presents, in dimensionless form, the melt-line location versus time, and the parametric influence of the variable v introduced as $v = Q_L / (c\,\theta p)$ is also clearly depicted.

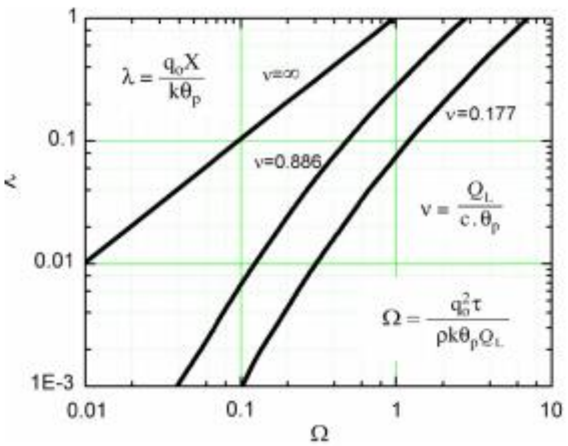


Figure 5.39. The melt-line location versus time in dimensionless form.

5.4 Aerodynamic Heating

The aerodynamic heating is heating of a body which moves through air or another gas at a high velocity. Aerodynamic heating results from the fact that air molecules flying against the body cause localized braking of the body. If a flight proceeds at a supersonic velocity, the effect of braking is primarily that of a shock wave, which is produced in front of the body. Further deceleration of air molecules occurs at the very surface of the body, in the so-called boundary layer. As a result of deceleration, the thermal energy of air molecules increases; that is, the temperature of the gas near the surface of the moving body increases. The maximum temperature to which a gas in the vicinity of the moving body can be heated is close to the so-called stagnation temperature:

$$T_o = Tn + v^2/2Cp$$

Where Tn is the temperature of the impacting air, v is the velocity of the body in flight, and Cp is the specific heat capacity of the gas at constant pressure. Thus, for example, in the flight of a supersonic aircraft at three times the speed of sound (about 1 km/sec), the stagnation temperature amounts to about 400ºC, whereas in the re-entry of a spacecraft into the earth’s atmosphere at the first cosmic velocity (8.1 km/sec), the stagnation temperature reaches 8000ºC. If in the first case for a sufficiently extended flight the temperature of the shell of the aircraft reaches such a temperature, then in the second case the surface of the spacecraft will surely disintegrate as a result of the inability of the material to withstand such high temperatures.

Heat is transferred to the moving aircraft from the region of superheated gas, resulting in aerodynamic heating. Two forms of aerodynamic heating exist—convective and radiative. Convective heating is a consequence of heat transfer from the outer “hot” part of the border layer to the surface of the body. Quantitatively, convective heat flow is defined by the equation

$$Q_k = \alpha(Te - Tw)$$

Where Te is the equilibrium temperature (the limiting temperature to which the surface of the body would be heated if there were no energy outflow), Tw is the actual temperature of the surface, and a is the coefficient of convective heat exchange, which depends on the velocity and altitude of the aircraft, on the shape and dimensions of the body, and on other factors. The equilibrium temperature is close to the stagnation temperature. The type of dependence of the coefficient a on the enumerated parameters is determined by the conditions of flow in the boundary layer (laminar or turbulent). In the case of turbulent flow, convective heating becomes more intensive. This is bound up with the fact that besides molecular heat conduction, an essential role in the transfer of energy is played by turbulent velocity pulsations in the boundarylayer.

As the aircraft velocity increases, thetemperature behind the shock wave and in the boundary layer grows, as a result of which dissociation and ionization of molecules occur. This produces atoms, ions, and electrons which are diffused into a colder region—against the surface of the aircraft. At that point, the reverse reaction occurs (recombination), proceeding with the liberation of heat. This also contributes to the process of convective aerodynamic heating.

At aircraft velocities on the order of 5,000 m/sec, the temperature behind the shock wave becomes significant, and the gas begins to radiate. As a consequence of the transfer of radiant energy from the area of superheated temperatures to the surface of the aircraft, radiative heating occurs. Radiation in the ultraviolet regions of the spectrum plays an important role in radiative heating. For an aircraft in the earth's atmosphere at a velocity below the first cosmic velocity (8.1 km/sec), radiative heating is small as compared with convective. At the second cosmic velocity (11.2 km/sec), their values become close, and at aircraft velocities of 13–15 km/sec and higher, corresponding to the speed of return to earth after flights to other planets, radiative heating makes the major contribution.

A special case of aerodynamic heating pertains to the heating of a body moving in the upper layers of the atmosphere, where the streamline condition involves free molecules; that is, the length of the free path of air molecules is commensurate with and even exceeds the dimensions of the body.

A particularly important role is played by aerodynamic heating in the entry of spaceships into the earth's atmosphere (for example, Vostok, Voskhod, Soiuz). Spaceships neutralize aerodynamic heating through the installation of special heat-shielding systems.

5.5 Convective and Radiative Wall Heat Transfer in Liquid Rocket Thrust Chambers

Wall heat transfer is one of the crucial items in liquid rocket engine (LRE) design because the typical temperature of the reaction products may exceed 3500 K, whereas the wall material can bear at best 850 K. This implies adopting an active cooling system, which must be sized on the basis of the anticipated distribution of heat flux all over the chamber walls. In this context, reliable numerical prediction tools can help alleviate the number, and cost, of experimental tests required for a safe design. To ensure reliability, numerical prediction tools must be validated against available experimental data. Wall heat transfer in LRE is mainly due to convection, and its contribution has been investigated by the authors in previous works, yet thermal radiation has been found to give a minor but nonnegligible contribution, estimated between 11 and 33% of the total heat flux in oxygen/RP1 engines and between 5 and 12% in oxygen/hydrogen ones. This underlines, if need be, that the weight of the radiative contribution depends on the particular propellant combination under consideration. Although hydrogen-fed combustion emits radiation in gas bands only, hydrocarbon combustion also features an overwhelming contribution from soot radiation, which can enhance radiation by a factor in between 2 and 10. Methane combustion is an exception, due to its mildly sooting nature, because it typically results in soot yields at least one order of magnitude lower than other hydrocarbons, owing to its exceptionally high H/C ratio. Accordingly, in open air/methane flames, gas radiation is still dominant with respect to the soot one. Anyway, the balance might be different in rocket chamber-like conditions, which deserves further investigation. It is worth noting in this regard that firing tests show a relatively nonluminous plume; although the plume radiates mainly in the infrared due to its lower temperature, one would expect a significant tail in the visible range in the presence of strong soot radiaton, which is not the case. The interest for methane stems from the fact that it is currently being considered as a possible cheaper and denser replacement for hydrogen as a liquid fuel .

Radiative heat transfer modeling involves several complex features, including directivity, spectral dependence, coupling to gas phase, sooting, possibly nonequilibrium, etc. In an engineering approach, attention is focused on phenomena most relevant to the determination of heat transfer, aiming at a solution approach both sufficiently accurate and unduly expensive in terms of computer time. Moving on these grounds, different computational models have been developed. The constraint on computer time almost invariably restricts the choice to a “gray gas” approach, which assumes that the optical properties of the medium are independent of radiation frequency (or wavelength), thereby enabling to describe the phenomenon in terms of a single radiative intensity, rather than a number of spectral intensities (typically, from a few tens to several thousands). As far as the solution of the radiative transfer equation (RTE) is concerned, several methods are available, including zonal method, flux method, discrete ordinate method, spherical harmonic method, Monte Carlo method, and discrete transfer method (DTM). Notice that, with increasing computing power, the first four methods converge to the solution of the method (vitiated by the specific assumptions introduced), whereas the last two converge to the actual solution of the problem. However, the Monte Carlo method seeks for the solution by injecting packets of photons into the flowfield and tracking their path under some stochastic assumptions, which makes coupling to a computational fluid dynamics (CFD) solver difficult.