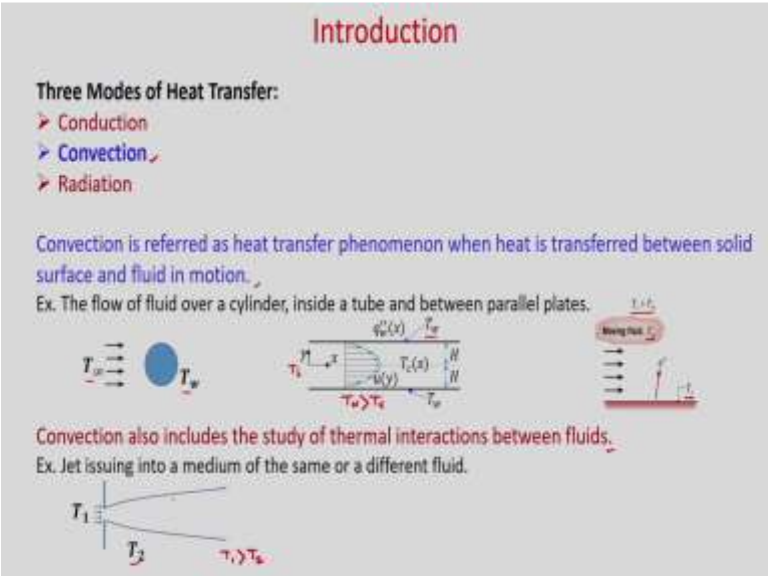
Fundamentals of Convective Heat Transfer
Prof. Amaresh Dalal
Department of Mechanical Engineering
Indian Institute of Technology, Guwahati

Module - 01
Introduction
Lecture - 02
Foundations of Heat Transfer

Hello everyone. So, in today's class we will first discuss about basic laws of 3 Modes of Heat Transfer. Then we will discuss about heat transfer coefficient which plays an important role in convective heat transfer, then we will discuss about some fluid dynamics equations which will be relevant in our course.

(Refer Slide Time: 00:57)



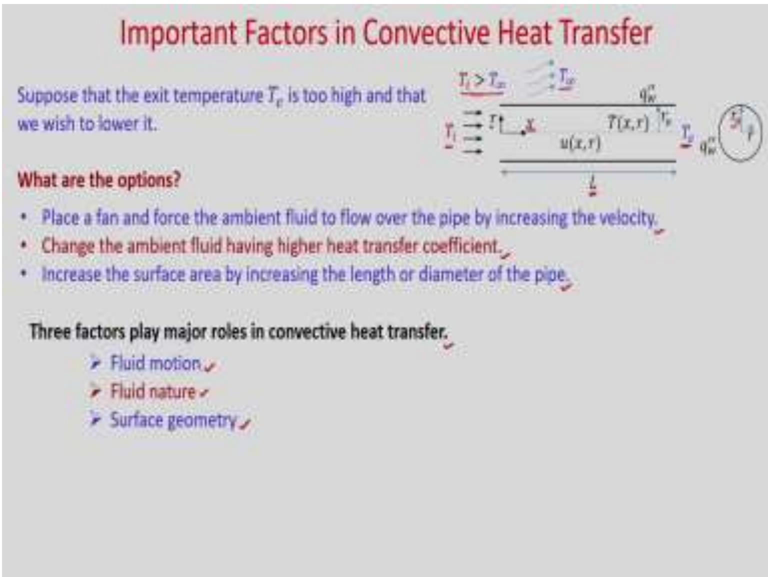
So, already we discussed that convection is one of the 3 modes of heat transfer. Convection is referred as heat transfer phenomena where heat is transferred between solid surface and fluid in motion. So, you can see there are some examples say the flow of fluid over a cylinder. Let us say cylinder is maintained at a higher temperature than the ambient temperature. So, $T_w > T_\infty$ and fluid flow is taking place. Obviously there will be heat transfer from the solid surface of the cylinder to the ambient fluid.

Similarly, if you see the flow inside a tube or parallel plates. So, in this case you can see fluid flow is happening inside 2 parallel plates. The wall is maintained at temperature T_w

and let us say at inlet you have temperature T_i and if $T_w > T_i$, then obviously when fluid flows inside these parallel plates there will be heat transfer from the solid surface to the fluid. Similarly, if you consider flow over a flat plate where your surface temperature is T_s and ambient temperature is T_∞ and $T_s > T_\infty$; obviously, there will be heat transfer from the solid surface to the fluid.

So, you can see in this phenomenon the heat is transferred between solid surface and moving fluid. Convection also includes the study of thermal interaction between fluids. So, you can see in this case jet issuing into a medium of the same or a different fluid. Let us say one fluid is entering through this jet whose temperature is T_1 and here another fluid is there which is having the temperature T_2 and let us say $T_1 > T_2$. Then when this fluid comes here, so in the jet you can see there will be a mixing and there will be heat transfer. So, you can see in this case the convection is taking place between the fluids.

(Refer Slide Time: 03:24)



Now, let us discuss some important factors in convective heat transfer. First let us consider flow inside a circular pipe. So, you can see this is a circular pipe of radius r_o. So, you can see that axial direction is x and radial direction is r and radial direction r is measured from the center of the cylinder. Here, let us say that steam is entering at high temperature T_i. The length of the pipe is L and the exit temperature is T_e that is to be determined. In the ambient temperature is T_∞ and obviously, as steam is entering here

$T_i > T_\infty$. Considering this situation let us say that when you measure the exit temperature T_e it is too high.

Now, you have to lower the temperature T_e. So, how can you do it? What are the possible ways you can decrease the exit temperature T_e? So, you can see here what the options are. Here, you have ambient temperature T_∞ and if you put a fan, then obviously its velocity will increase and more heat transfer is likely to take place. So, that is one possible way.

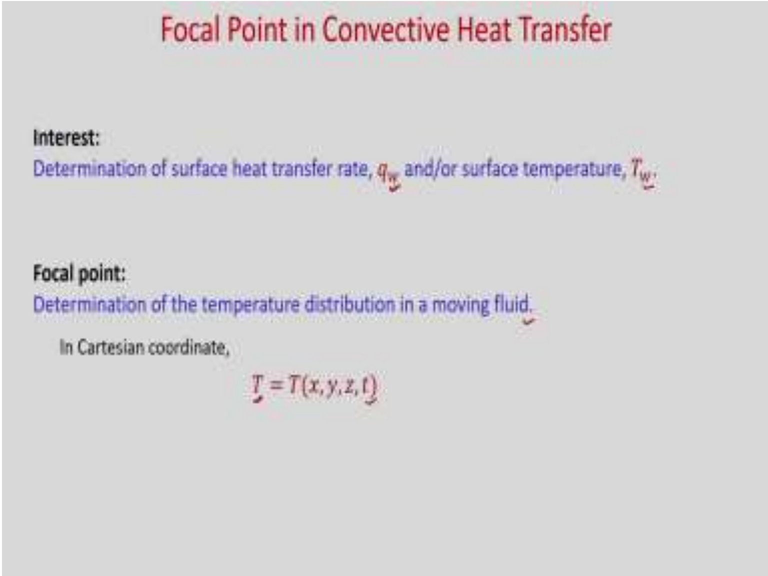
So, place a fan and force the ambient fluid to flow over the pipe by increasing the velocity. The other way is that whatever ambient fluid is there you just change that ambient fluid so that it can take away more heat from the pipe surface. So, obviously if you change the fluid you can get such a fluid whose heat transfer coefficient is high.

So, secondly you can change the ambient fluid having higher heat transfer coefficient. The other way to increase the heat transfer from the pipe surface, so that you can lower the temperature T_e is to increase the surface area.

So, which way you can increase the surface area? You can increase the diameter of the pipe or you can increase the length of the pipe. So, increase the surface area by increasing the length or diameter of the pipe. So, L you can increase or r_o you can increase so that your surface area increases and more heat transfer will take place, so that your exit temperature at T_e will decrease.

So, in this example you can see that 3 factors play major roles in convective heat transfer. One is fluid motion, then fluid nature or fluid properties and surface geometry. So, these are the 3 factors which play major roles in convective heat transfer.

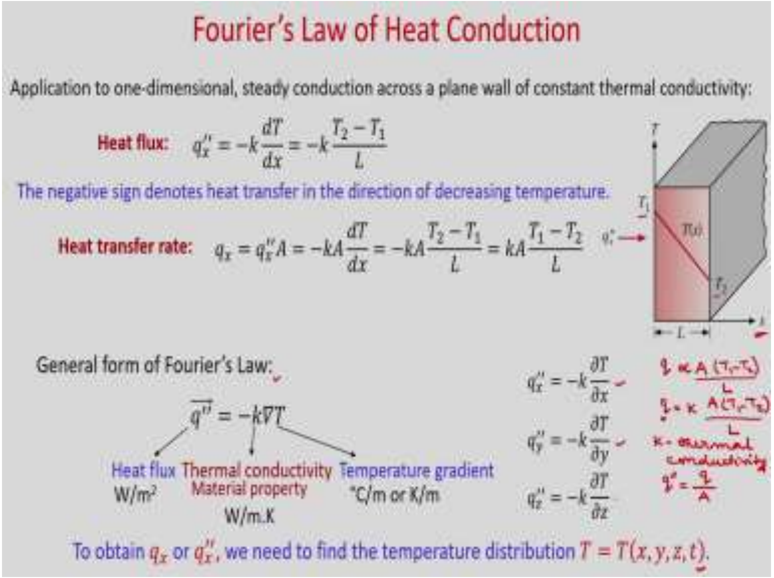
(Refer Slide Time: 06:36)



So, now next question is that why do we want to study this convective heat transfer? Why we are interested in studying this convective heat transfer and in which quantity we are interested? So, obviously you can see that we are more interested in finding the heat transfer rate while designing some industrial equipment or to know the surface temperature. And to determine the heat transfer rate or the surface temperature you need to know what the temperature distribution inside the domain is.

So, our interest is to determine the surface heat transfer rate q_w and or surface temperature T_w and focal point is the determination of the temperature distribution in a moving fluid. So, in Cartesian coordinate temperature will be function on the space x, y, z and may be with time. So, once you find the temperature distribution you will be able to calculate the heat transfer rate. Now, let us discuss the basic law in conduction. Already you have studied in the basic heat transfer course the Fourier’s law of heat conduction.

(Refer Slide Time: 07:59)



So, if you consider the solid of thickness L, the temperature in the left surface is maintained at temperature T_1 and right surface at T_2 and other walls are insulated so that there will be no heat transfer and heat transfer will take place only in one direction in x direction in this case. So, in experiment it is shown that the heat transfer rate is proportional to the temperature difference and directly proportional to the heat transfer area and inversely proportional to the thickness of the solid.

So, in this case you can see q if it is a heat transfer rate, then it is directly proportional to the area, it is directly proportional to the temperature difference and it is inversely proportional to the thickness. And you can see that from here if you equate this heat transfer rate, then a proportionality constant will come and that proportionality constant is known as thermal conductivity k.

So, $q = kA\frac{(T_1 - T_2)}{L}$

So, this k is your thermal conductivity of the material. So, you can see that if it is heat transfer rate, then heat flux you can write as q" is the heat transfer rate per unit area.

So, $\frac{q}{A}$ So, this is your heat flux.

So, you can see that heat flux in one dimensional steady conduction across a plane wall of constant thermal conductivity we can write $q''_z = -k\frac{dT}{dz}$

So, $\frac{dT}{dx}$ is the temperature gradient and you can see temperature gradient you can write $\frac{T_2 - T_1}{L}$

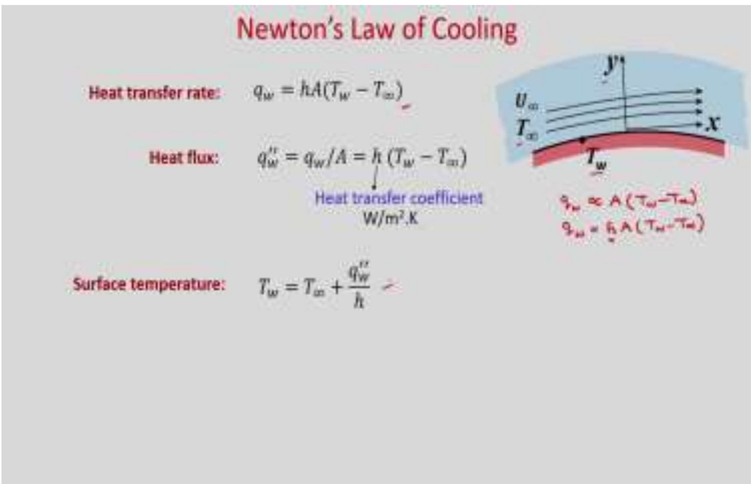
So, here you can see the negative sign these negative sign denotes heat transfer in the direction of decreasing temperature. So, heat transfer rate you can write as heat flux into the area.

So, it will be $-k\frac{dT}{dx}$ and hence you can write $kA\frac{T_1 - T_2}{L}$. In general form of Fourier's law you can write as a heat flux vector quantity.

So, ∇ $\vec{q''} = -k\nabla T$ where q'' is the heat flux in Watt per meter square, k is the thermal conductivity it is a material property. Its unit is W/mK and ∇T is the temperature gradient it is K/m or ^{o}C/m.

So, we can see to obtain q_x or q''_x; that means, heat transfer rate and the heat flux we need to find the temperature distribution T, then only you can calculate the temperature gradient. So, in this case you can see in x direction you can write $q''_x = -k\frac{dT}{dx}$, in y direction $q''_y = -k\frac{dT}{dy}$ and in z direction $q''_z = -k\frac{dT}{dz}$

(Refer Slide Time: 11:41)

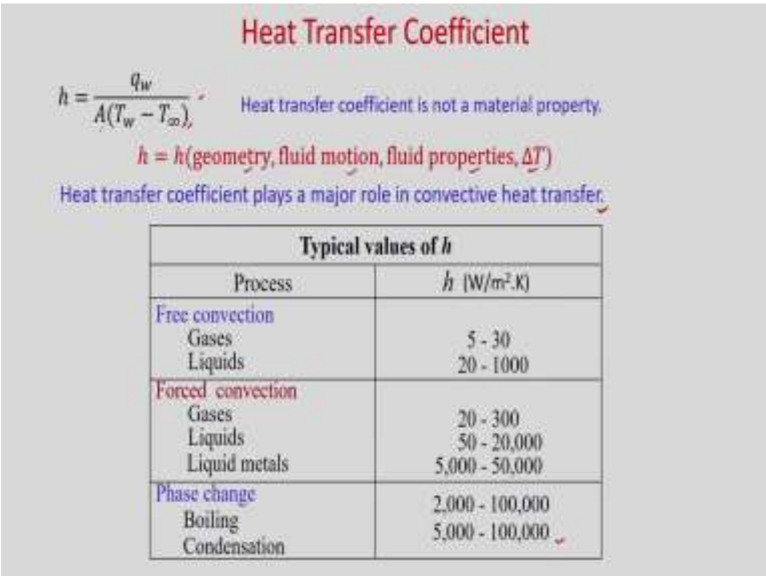


Now, in convection we have a basic law which is known as Newton's law of cooling. Consider this is a surface maintained at temperature T_w, x is along the surface and y is normal to the surface and you have fluid flow where free stream velocity is u_∞ and you have temperature T_∞. So, in experiments it is shown that your heat transfer rate at the wall is directly proportional to the area and the temperature difference and once you write equal to then you will get one proportionality constant into area into the temperature difference.

So, this proportionality constant is known as heat transfer coefficient. So, we can see heat transfer rate we can write $q_w = hA(T_w - T_\infty)$ and heat flux is the heat transfer rate per unit area. So, that you can write h (T_w -T_∞) where h is the proportionality constant and it is known as heat transfer coefficient its unit is W/m^{2}K. So, from here you can see that you can write the surface temperature $T_w = T_\infty + \frac{q''_w}{h}$.

So, from here you can define the heat transfer coefficient.

(Refer Slide Time: 13:17)



So, you can see heat transfer coefficient is the heat transfer rate per unit area per unit temperature difference. So, here you can see that heat transfer coefficient is not a material property. So, it is a transport property and it depends on many things. So, it depends on geometry, fluid motion, fluid properties and sometime on temperature difference. So, heat transfer coefficient plays a major role in convective heat transfer.

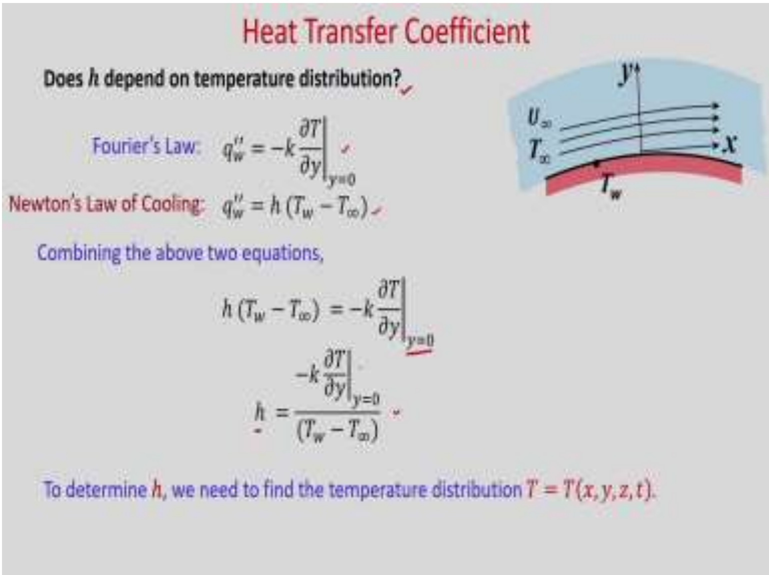
So, you can see that h is function of geometry, fluid motion, fluid properties and temperature difference. So, you will not get any particular value for any situation. So, some rough idea about the value of this heat transfer coefficient in different situation is tabulated here.

You can see if you have a free convection; that means, natural convections then for gases h is varies 5 to 30 W/m^2K. For liquids, obviously it is more it varies 20 to 1,000 W/m^2K. If it is a forced convection, then for gases it varies between 20 and 300 W/m^2K.

For liquids, in the range of 50 to 20,000 and in liquid metals 5,000 to 50,000 and if phase change takes place like boiling and condensation then you will get a very high heat transfer coefficient. You can see for boiling you can achieve the heat transfer coefficient in the range of 2,000 to 1 lakh and in condensation also 5,000 to 1 lakh you can achieve the heat transfer coefficient.

So, from here you can see that if you need to remove very high heat flux then you need to use phase change boiling or condensation.

(Refer Slide Time: 15:22)

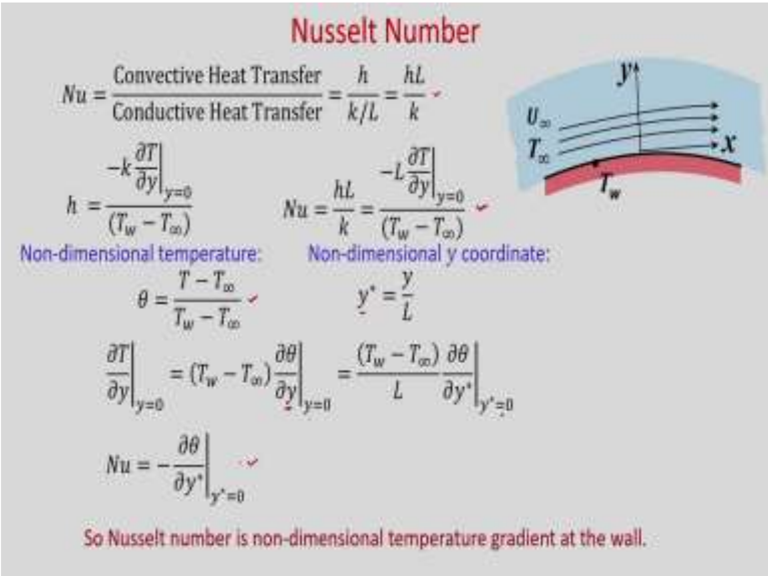


Now, the question is does h depend on temperature distribution. So, let us see in Fourier's law the heat flux at the wall we can write $q''_w = -k \frac{\partial T}{\partial y}|_{y=0}$. So, at the wall. From Newton's law of cooling also you can write the heat flux at the wall $q''_w = h(T_w -$

T_∞). So, if you equate these 2 you can write $h\left(T_w - T_\infty\right) = -k\frac{\partial T}{\partial y}|_{y=0}$; that means, at the wall.

From here you can see h you can determine from this relation $h = \frac{-k\frac{\partial T}{\partial y}|_{y=0}}{\left(T_w - T_\infty\right)}$. So, here you can see that you need to calculate the temperature gradient at the wall to find the heat transfer coefficient. So, obviously to determine the heat transfer coefficient we need to know the temperature distribution inside the fluid domain.

(Refer Slide Time: 16:24)



We have discussed about the heat transfer coefficient and very often we write this heat transfer in non-dimensional form and this non-dimensional number is known as Nusselt number. Nusselt number is the ratio of conductive to convective heat transfer in a fluid. So, you can see

Nusselt number, $N_u = \frac{ConvectiveHT}{ConductiveHT} = \frac{h}{k/L} = \frac{hL}{k}$

h is the heat transfer coefficient, k is the thermal conductivity and L; L is the characteristic length. So, this characteristic length varies depending on the different geometry.

So, if you consider flow over a flat plate, then your characteristic length will be the length of the plate, but if you consider flow inside a circular pipe then your characteristic length may be the diameter of the pipe. So, we have already shown that $h = \frac{-k \frac{\partial T}{\partial y}\big|_{y=0}}{(T_w - T_\infty)}$.

So, if you put this h in this expression, then you can write $N_u = \frac{hL}{k}$

So, $N_u = \frac{hL}{k} = \frac{-L \frac{\partial T}{\partial y}\big|_{y=0}}{(T_w - T_\infty)}$

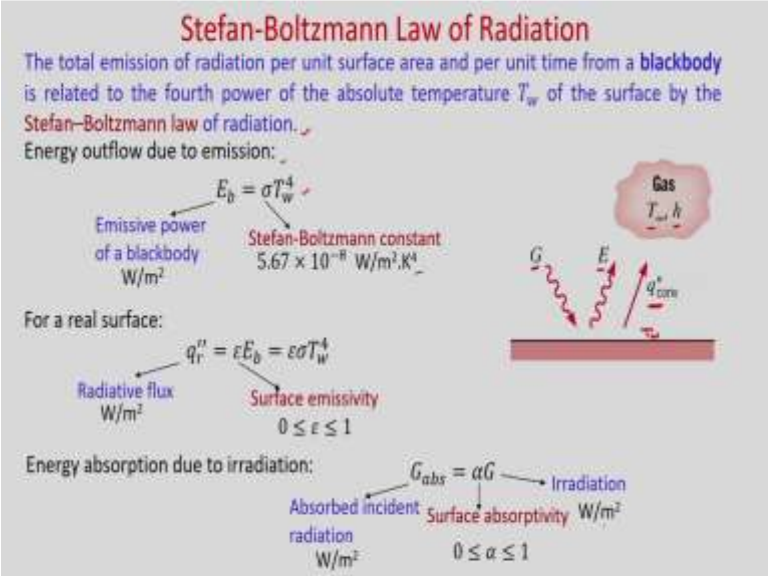
Now, let us write Nusselt number in terms of some non-dimensional quantities. So, now, let us define the non-dimensional temperature $\theta = \frac{(T - T_\infty)}{(T_w - T_\infty)}$ and non-dimensional y coordinate is $y^* = \frac{y}{L}$ where L is the characteristic length.

So, from this expression you can calculate the temperature gradient $\frac{\partial T}{\partial y}\big|_{y=0} = (T_w - T_\infty)\frac{\partial \theta}{\partial y}\big|_{y=0} = \frac{(T_w - T_\infty)}{L}\frac{\partial \theta}{\partial y^*}\big|_{y=0}$

So, this $\frac{\partial T}{\partial y}$ now if you put in this expression then you will get $N_u = -\frac{\partial \theta}{\partial y^*}\big|_{y^*=0}$

So, you can see that Nusselt number is non-dimensional temperature gradient at the wall. So, if you are solving non-dimensional equations and non-dimensional energy equations then you will get the Nusselt number directly as the temperature gradient because non-dimensional temperature we have considered. So, non-dimensional temperature gradient that you will give you the value of Nusselt number.

(Refer Slide Time: 19:31)



Now, let us discuss about the Stefan Boltzmann law of radiation. Here you can see this is the surface let us say the surface temperature is T_w. So, obviously at any temperature this plate will emit radiation and that is your E and some radiation will come to this surface and whatever is coming that is your irradiation G and ambient temperature is T_∞ and obviously your convective heat transfer coefficient is h.

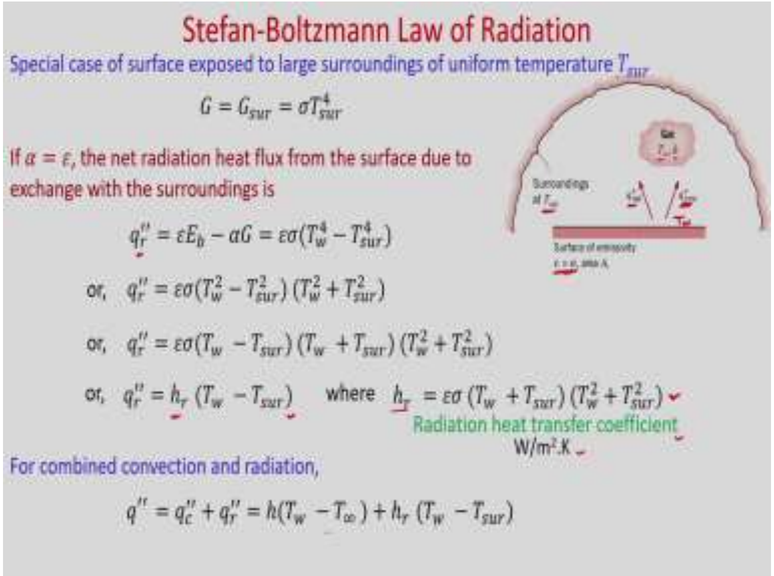
So, you can see there will be heat flux due to the radiation as well as you have heat flux due to convection. So, what is Stefan Boltzmann law? The total emission of radiation per unit surface area and per unit time from a black body is related to the fourth power of the absolute temperature T_w of the surface by the Stefan Boltzmann law of radiation.

So, we can see energy outflow due to emission due to Stefan Boltzmann law you can write $E_b= \sigma T^4_w$ where E_b is the emissive power of a black body and its unit is W/m^2 and T_w is the surface temperature and σ is the Stefan Boltzmann constant and its value is $5.67x\ 10^{-8}\ W/m^2.K^4$.

Now, for a real surface the radiative flux $q''_r = \varepsilon E_b$ where E_b is the emissive power of a black body and ε is the surface emissivity and it varies between 0 to 1 and for a black surface obviously, your emissivity is 1 and E_b if you put this then you will get $q''_r= \varepsilon\sigma T^4_w$

And if you see that whatever irradiation is coming G, so it will be absorbed some part of it by this surface and if you consider the surface absorptivity as α, then you can write energy absorption due to irradiation $G_{abs}= \alpha G$ where G_{abs} is the absorbed incident radiation W/m^2 and α is the surface absorptivity. Obviously, it varies between 0 and 1 and G is the irradiation in W/m^2.

(Refer Slide Time: 22:13)



Now, let us consider a special case. Surface exposed to large surrounding to uniform temperature T_{sur}. So, you have the surface whose temperature is T_w. So, due to convection there will be heat flux q''_{conv}. Due to radiation there will be heat flux q''_{rad}. Your ambient temperature is T_∞ heat transfer coefficient is h; surrounding temperature is maintained at T_{sur}.

Now, surface emissivity is ε and from Kirchhoff's law you can see that ε will be equal to the α and the area is A. So, obviously, you can see that whatever from irradiation is coming from the surrounding so that you can write $G= \sigma T^4_{sur}$ because it is coming from the surrounding and its fraction αG is absorbed by this surface.

So, now if you assume that $\alpha=\varepsilon$, then the net radiation heat flux from the surface due to exchange with the surrounding is, so this is your radiative heat flux. So, whatever emission is happening that is εE_b minus whatever radiation is absorbed that is αG. So, $q''_r= \varepsilon E_b - \alpha G$.

So, $\alpha = \varepsilon$ and G = sigma T^4_{sur} and $E_b = \sigma T^4_w$.

So, you can write $q''_r = \varepsilon\sigma(T^4_w – T^4_{sur})$

So, this you can write in this way $q''_r = \varepsilon\sigma(T^2_w - T^2_{sur}) (T^2_w + T^2_{sur})$

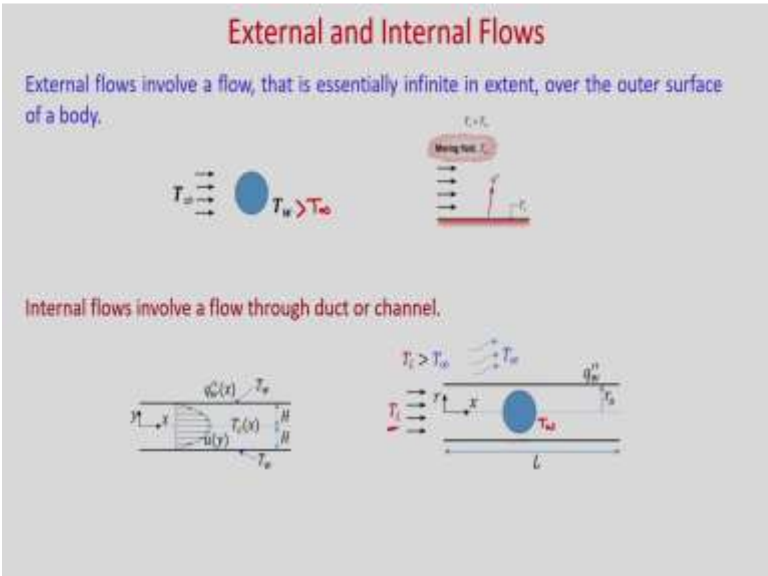
So, these quantity again you can write $(T_w - T_{sur}) (T_w + T_{sur})$. So, now we can write equivalent to Newton's law of cooling you can write the radiation heat flux as,

$q''_r = h_r (T_w - T_{sur})$

So, this $h_r = \varepsilon\sigma(T_w + T_{sur}) (T^2_w + T^2_{sur})$. So, where h_r is known as radiative heat transfer coefficient and its unit is also W/m^2K. So, you can see from the radiative heat flux we have written this expression similar to Newton's law of cooling, so that we can define radiative heat transfer coefficient and this is expression is this one.

So, now, for combined convection and radiation where it is taking place heat flux due to convection, heat flux due to radiation. So, $q'' = q''_c + q''_r = h(T_w - T_\infty) + (T_w - T_{sur})$

(Refer Slide Time: 25:22)

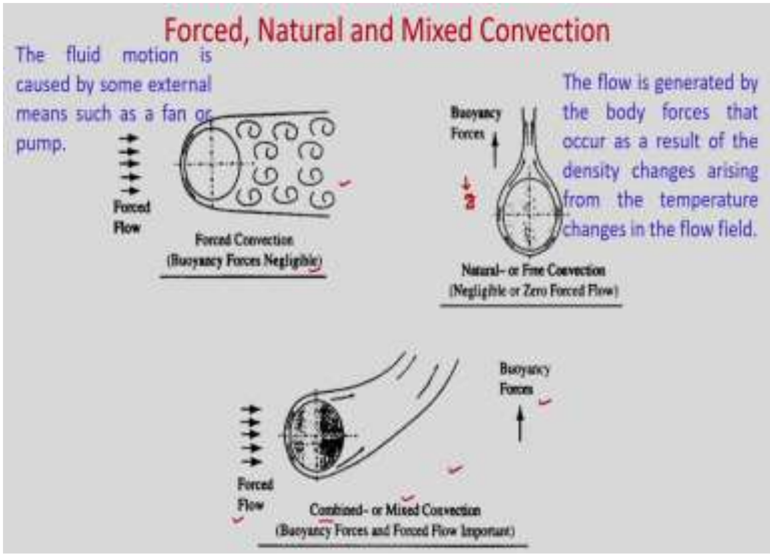


So, there are 2 types of flows external and internal. External flows involve a flow that is essentially infinite in extent over the outer surface of a body. So, you can see this example flow over a circular cylinder. Circular cylinder temperature is T_w which is greater than the ambient temperature T_∞. So, heat transfer will take place from the cylinder surface to the ambient and flow is taking place over this body. So, this is one example of external flows. You can see here that flow over a flat plate and heat transfer is taking place from the hot plate to the ambient fluid and in other direction it is infinite. So, this is external flow. Internal flows involve a flow through duct or channel.

So, if it is confined by wall, then it is internal flows you can see flow inside 2 parallel plates ok. So, here this is confined by 2 parallel plates and flow is taking place inside this domain. So, this is one example of internal flows. Here, this example you can see one cylinder is placed inside this pipe. This is a circular pipe r is the radius radial direction and r_o is the radius of the circular pipe. So, this is one example of sphere kept inside the circular pipe and this sphere you can see that it is maintained at some temperature T_w. Obviously, heat transfer will takes place from sphere to this fluid which is entering at temperature T_i.

Now, you can see this is also example of internal flows for the circular pipe, but at the same time the flow is taking place over this sphere. So, it is kind of external flows. So, in this case you cannot separately tell whether it is internal or external flows, but; obviously, it is confined flow. So, it is kind of internal flows.

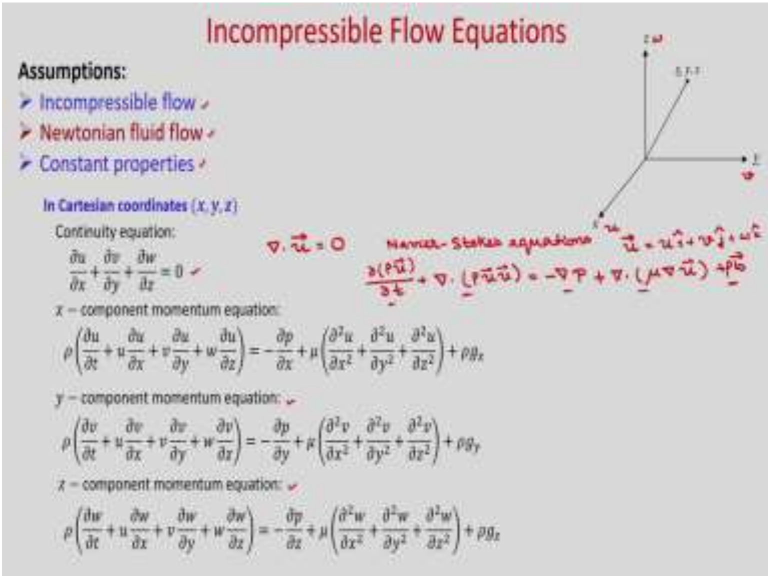
(Refer Slide Time: 27:31)



Convective heat transfer rate depends on the type of flow. So, you can see depending on the type of flow you can get forced, natural and mixed convection. In forced convection, externally there will be pump or fan from where this fluid flow will take place and buoyancy force is negligible. So, the fluid motion is caused by some external means such as a fan or pump. So, this is purely forced convection is taking place here, but if buoyancy force is present let us say you have a gravity is acting in negative y direction.

So, buoyancy force will act and this is known as natural or free convection, so and there is absence of externally induced flow. The flow is generated by the body forces that occur as a result of the density changes arising from the temperature changes in the flow field. Now, if you have both forced and natural; that means, buoyancy force is present as well as forced flow is present, then that is known as mixed convection or combined convection. So, buoyancy force and forced flow both are important in this case.

(Refer Slide Time: 28:51)



So, now, let us discuss some important governing equations in fluid mechanics. So, the main assumptions we are taking incompressible flow, Newtonian fluid flow and constant properties. So, in this case we are considering Cartesian coordinate. So, in x direction you have velocity u in y direction you have velocity v and in z direction you have velocity w.

So, if you define a vector velocity $\vec{u}=u\hat{i}+v\hat{j}+w\hat{k}$. So, in general you can write the continuity equation for incompressible flow as $\nabla\vec{u}=0$. So, u is a vector quantity and divergence of u you can be written in differential form as $\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0$. So, this is the continuity equation.

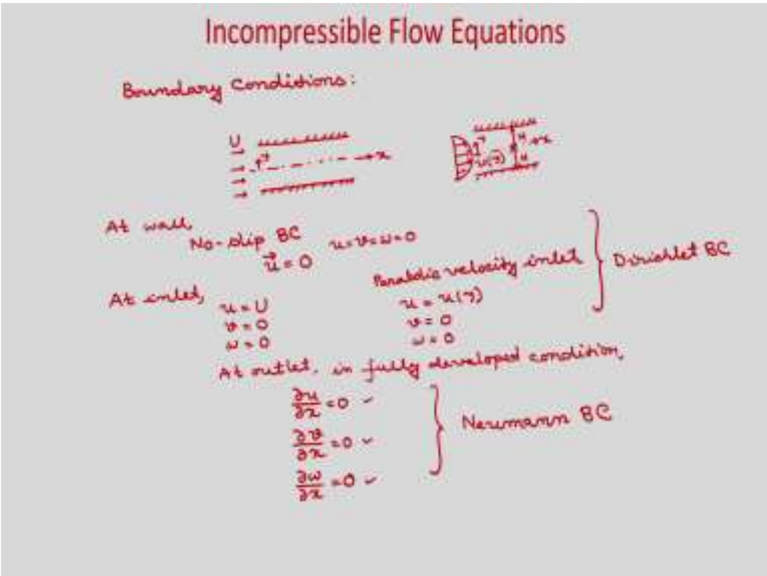
Now, you can write the momentum equation or Navier Stoke's equations in vector form as $\frac{\partial(\rho\vec{u})}{\partial t}+\nabla(\rho\vec{u}\vec{u})=-\nabla p+\nabla(\mu\nabla\vec{u})+\rho b$

So, you can see this is the Navier Stoke's equations. So, in Navier Stoke equation this first term is your temporal term because it is a time bearing term. This is the convective term ok, this is your pressure term because there will be a pressure gradient which is the driving force for the fluid flow and this is the viscous term and if some body force is present that you can incorporate in this term.

So, this Navier Stoke equations if you write for the constant properties, then rho you can take it outside and this mu which is your fluid viscosity dynamic viscosity you can take it outside and rho is the fluid density and P is the pressure. And in non-conservative form and in differential form you can write x component of momentum equation as $\rho(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z})=-\frac{\partial p}{\partial x}+\mu(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2})+\rho g_x$.

So, here you can see that this is your temporal term this is your convective term, this is your pressure gradient term, this is your viscous term and this is your body force term. Similarly, you can write y component of momentum equation and z component of momentum equation. So, this we have written for incompressible Newtonian fluid flow.

(Refer Slide Time: 32:30)



So, when you solve these equations you need to have the boundary conditions. So, if we discuss the boundary conditions for fluid flow. So, if you consider flow between parallel plates. So, these are the 2 parallel plates in third direction let us say it is infinite. So, you have fluid flow in axial direction x, this is your y and so obviously if these are walls then at the wall what will be the boundary condition; at wall what will be the boundary condition? So, you can see we are writing the boundary condition for the fluid flow equations. So, when the fluid particle resides on the wall, then it will have the same velocity as the wall.

So, if wall is stationary you will have velocity as 0 and if wall is moving with some particular velocity then that fluid particle will have the same velocity. So, this is known as no slip boundary condition. So, at the wall you have no slip boundary condition ok. So, at the wall the fluid particle which are residing on the wall, so immediate fluid particles will have the same velocity at the wall. So, if wall is stationary then you will have the velocities as 0. So, that means, your x direction velocity u, y direction velocity v and z direction velocity w all will be 0 at the wall and it is known as no slip boundary condition.

At the inlet, generally we define the velocity boundary condition. So, this is kind of Dirichlet boundary condition you have a constant value. So, you can have either uniform velocity inlet like constant velocity inlet. So, at inlet, so you have you can have in this particular case you can write u is equal to let us say U. So, at inlet you can have U where U is constant and v will be 0 and w will be 0. Also you can have some if you have a fully developed boundary condition. So, if you have a fully developed boundary condition, then you can have a parabolic profile. So, at the inlet you can have some parabolic profile.

So, this is your y, this is your x, x is the axial direction and let us say you have a parabolic velocity inlet which is function of y. So, for parabolic velocity inlet you can have u as u function of y ok. So, that you can determine depending on the condition here. So, if it is h and this is your h, then you can write the parabolic velocity boundary condition v will be 0 and w will be 0. At the outlet, most of the time we specify the pressure ok. So, generally p = 0 you put and if you want to write the velocity boundary condition.

So, generally we say that it has reached fully developed condition and at the outlet if it is a fully developed condition, then we can write

$$\frac{\partial u}{\partial x} = 0, \ \frac{\partial v}{\partial x} = 0, \ \frac{\partial w}{\partial x} = 0$$

because x is the axial direction and the velocity gradient in the axial directions are 0 if it is at outlet in fully developed condition.

So, obviously, if it is a channel flow ok, so at the outlet you will get the fully developed condition and you can use these boundary conditions for velocity. So, you can see that at wall we have no slip boundary condition where we are specifying the velocities 0; $u = v = w = 0$ and this is known as Dirichlet type boundary condition and at outlet you can see that we are specifying the velocity gradient.

So, this is kind of Neumann boundary condition and at inlet we are specifying the velocity either it is a uniform velocity inlet u or a parabolic velocity inlet which u is function of y. So, these 2 also we are specifying the velocity. So, this is also known as Dirichlet type boundary conditions. So, these 2 are Dirichlet boundary conditions. So, that means, the value of the velocity is specified at the boundary and this is your gradient specified. So, it is known as Neumann type boundary condition.

So, today we started with the definition of convection, then we have discussed about why we are interested to study the convective heat transfer, then we discuss the basic laws of 3 modes of heat transfer.

At first, we discussed about Fourier's laws of heat conduction and we defined heat transfer rate and heat flux, then we discussed about the Newton's laws of cooling and from there we defined the heat transfer coefficient and we have shown some typical values of heat transfer coefficient in different flow situations. Then we discussed about non-dimensional number, Nusselt number and finally, we have shown that Nusselt number is the non-dimensional temperature gradient at the wall.

Then we discussed about the Stefan Boltzmann law and we have shown the net radiation exchange between a surface and the surrounding and from there we defined the radiation heat transfer coefficient and for combined convection and irradiation we have defined the heat flux. Finally, we discussed about the fluid flow equations, continuity equation and Navier Stokes equations in vector form as well as in differential form.

Thank you.