Microscale Transport Processes

Prof. S. DasGupta
Department of Chemical Engineering
Indian Institute of Technology, Kharagpur

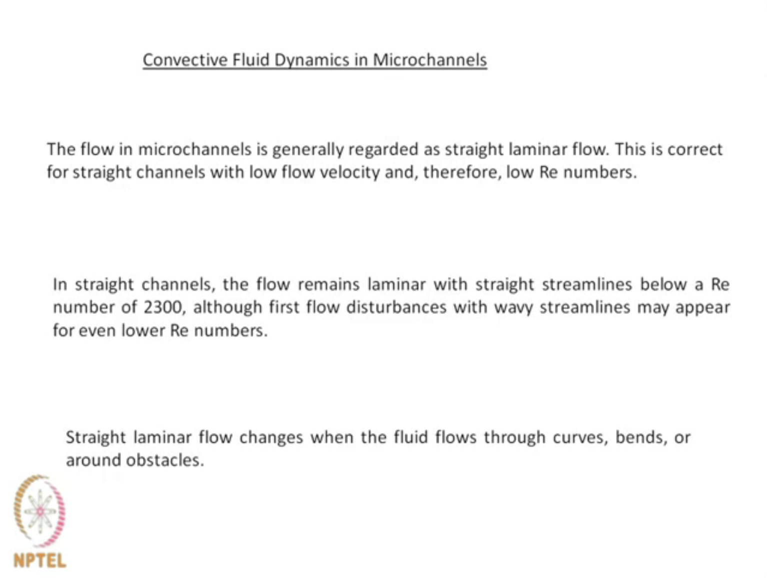
Lecture No. # 36
Convective Fluid Dynamics in Microchannels

In this class, we are going to talk about convective fluid dynamics in microchannels. Now, we all know that when we have flow through any normal size channels, the flow maybe laminar turbulent or in between. Now, in microchannels because of the predominance of viscous forces the flow can remain laminar for quite some time. And it is also not possible in most of the cases to have turbulent flow in such small devices. So, in other words, in order to obtain or in order to attain turbulent flow in microchannels one has to sacrifice a huge amount of pressure drop. So, most of the cases, we will encounter laminar flow in microchannels, but there will be situations in which the flow at times or at certain pockets may become turbulent or the mixing will become much more.

For example, if there are bends in the pipeline or if there are internals which are present in the path of the in microchannels or in the path of the fluid or there are entry and exit, then, it may lead to near turbulent situations specially near the corners. Now, this fact has been used to device mixers at microscale. Now, we all understand that if we have two fluids which are flowing side by side then mixing them in a microchannel where the flow is predominantly laminar could be a problem. So, one alternative could be to use these bends intelligently such that they themselves will act as mixers, but we also have to understand that once this mixing pattern sets in the flow will become stabilized, once we are away from that bend.

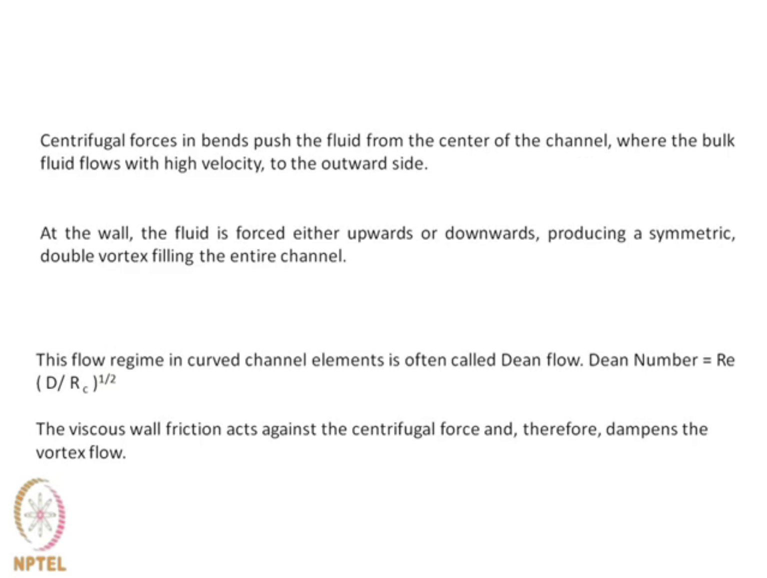
So, we need to have some specific studies in which we can quantify or we can study at depth, how the flow becomes turbulent or how the flow becomes irregular not straight streamline laminar flow near an near 90 degree bend and how it stabilizes at a certain length from the bend.

(Refer Slide Time: 02:52)



So, we are going to concentrate today, in on flow in microchannels which are generally regarded as straight laminar flow. But in straight channels, the flow remains laminar below a Reynolds number of 2300, although we start to see disturbances with various streamlines that may appear even lower Reynolds number and there is obviously, a range a Reynolds number beyond 2300 for which we may expect through the turbulent flow.

(Refer Slide Time: 03:35)

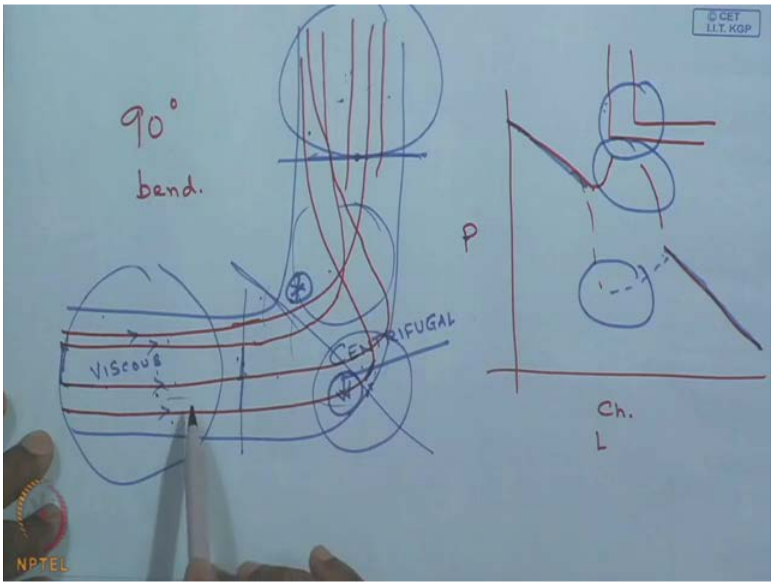


Now, as I was talking as I was telling you before this straight streamline or straight laminar flow changes when fluid flows through curves bends or around obstacles. Now, what happens at at the bends is that the centrifugal forces at bends pushes the fluid away from the bend. So, there will be alternatively positions of low pressure and high pressure at at different positions at the bend and this would give rise to mixing near the bends. So, at the wall the fluid is forced either upwards or downwards producing a symmetric, double vortex which fills the entire channel. Now, this vortex will soon dye down depending on the viscosity of the fluid and the dimension of the channel and may give rise to straight streamline flow once again.

So, in order to quantify the flow regime in curved channel elements, we often bring in a number, which is known as the dean number and the dean number is Reynolds number multiplied by D by R c to the power half. So, this D is the diameter or the diameter of this channel and R c is the radius of curvature for the bend. Now, this flow regime in curved elements are often called dean flow. And it is the viscous flow viscous wall friction which acts against the centrifugal forces present in those bends and they will bring order to the system. So, depending on the value of the dean number, we can classify the flow at curved bends under several different distinct regimes and the flow pattern in each of these systems has been probed.

And depending on the values of dean number, we could see rapid mixing between two liquids which are flowing side by side and suddenly come across R T junction or an 90 degree bend and I will give you examples of that in my subsequent slides.

(Refer Slide Time: 06:08)

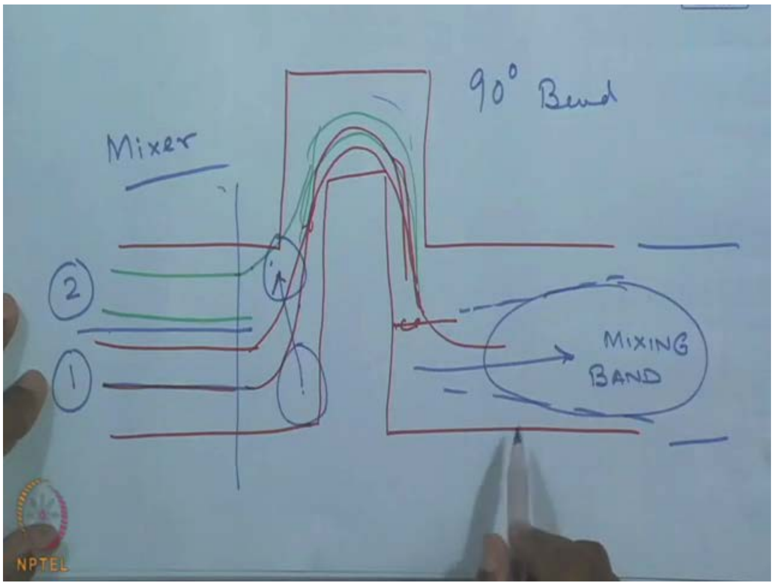


So, now we will let us say we moved to a situation in which there is a 90 degree bend and at this 90 degree bend, what we see is that the straight streamline of the flows before the bend suddenly get mixed in this region and then, afterwards once we go far from the bend then the flow becomes stabilized again. So, here is a the case where viscous forces are important and in this case the centrifugal forces would try to push the liquid towards this side and viscous forces will oppose the centrifugal force and will try to bring order over here. So, if we can see that, over here there would be if we could measure the pressure at every point, one can see that the pressure from here would be definitely have to be higher than the pressure at this point so that the flow can take place.

And from here to this point, the pressure may vary in an erratic fashion depending on whether we are measuring the pressure over here or we are measuring the pressure at this point. Beyond a certain length from the bend the pressure again becomes similar to what we have before the bend. So, these two pressure differences will behave in a similar fashion and the pressure gradient one can expect from this point to this point and beyond this region will be straight line will be linear in nature. So, what we have shown here is the plot of pressure against channel length and if we think of this as the 90 degree bend, what we have shown over here, then the pressure will linearly fall between the this point and the beginning of the bend and beyond this line the pressure will again fall linearly.

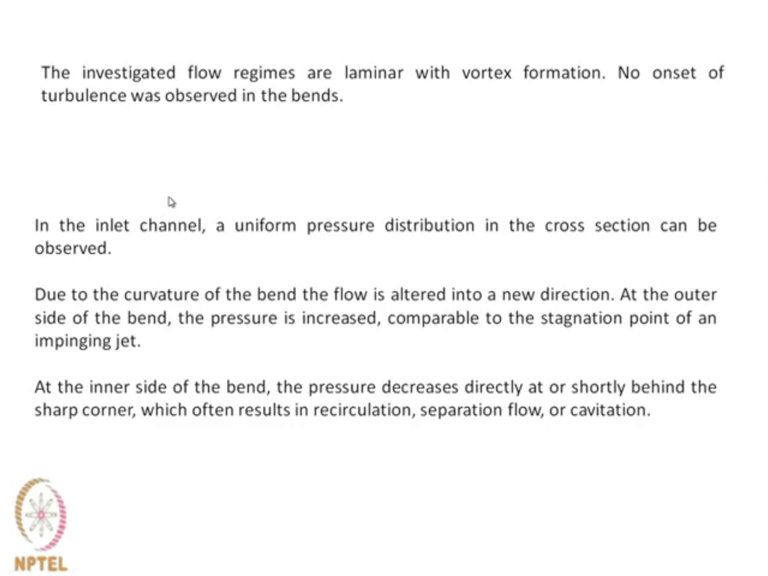
But if we think conceptually what is happening at the bend, at this point the pressure is going to be more and at this point the pressure is going to be less. So, across this, there is going to be a pressure jump near the wall where the liquid is; where the fluid is coming hitting the wall changing it is direction and moving in another direction and over here where a low pressure region is going to form.

(Refer Slide Time: 09:15)



So, what we see here is the pressure jump near the outer wall and a pressure a reduced pressure near the inner wall. So, this kind of pressure behavior is quite common even for laminar flow in a 90 degree bend. So, this concept that, change in flow direction is causing the straight streamlines of the flow to deviate from the straight path can be utilized in a number of devices and one such device is a mixer utilizing a 90 degree bend.

(Refer Slide Time: 09:23)


The investigated flow regimes are laminar with vortex formation. No onset of turbulence was observed in the bends.

In the inlet channel, a uniform pressure distribution in the cross section can be observed.

Due to the curvature of the bend the flow is altered into a new direction. At the outer side of the bend, the pressure is increased, comparable to the stagnation point of an impinging jet.

At the inner side of the bend, the pressure decreases directly at or shortly behind the sharp corner, which often results in recirculation, separation flow, or cavitation.

NPTEL

Now, what we can summarize? Our observation is that the investigated flow regimes are laminar with formation of vortex and we may not see any onset of turbulence in the bends. And at the inlet channel we have an uniform pressure distribution in the cross section and due to the curvature of the bend the flow is altered in a new direction and as I say that the outer side of the bend the pressure is increased compared to the inner side. And this difference in pressure at the curved bend can create mixing due to the presence of the 90 degree bend and which has further usage in micro flow in microchannels.

(Refer Slide Time: 10:11)

Approx. 100 μm behind the bend, a uniform pressure establishes in the cross section.

At this point, the vortices are already dampened and straight laminar flow is established again, as shown by the streamlines in left side

The pressure loss in the bend results in vortex formation and is the basis for further calculation in mixing theory.

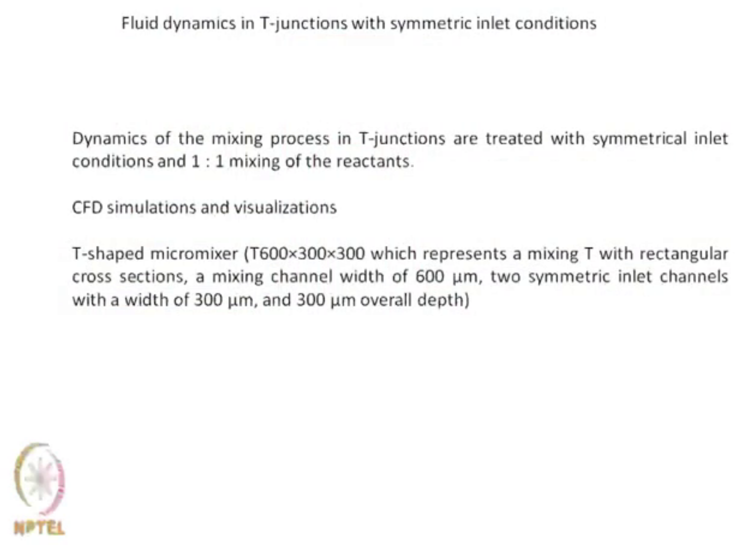
NPTEL

It has been shown that approximately 100 micron behind the bend, uniform pressure gradient pressure establishes in the cross section. So, in the at this point the vortices are already dampened by the presence of viscous forces and the straight laminar flow reappears and will continue till it encounters another bend in the path. So, the pressure loss in the bend results in vortex formation and in is a basis for further calculation in mixing theory. So, we may get more mixing, but definitely at the cost of increased pressure gradient in the path of the flow. Now, we can also; look at the fluid dynamics in T junctions by we will come to that later on.

Let us just think of how 90 degree bend works and this is some experiment which has been done with two different fluids. So, I have fluid 1 which is slowing and in and fluid 2 and the volumetric flow of these two are the same so, the fluids are clearly separated and then it has a 90 degree bend in it is path. So, what would happen is that the pressure gradient up to this point will be linear and over here, there is going to be a high pressure zone and a low pressure zone that would cause the fluid to move from one side to the other.

And therefore, the green fluid and the red fluid will mix and they will mix further and over here at the exit from the 90 degree bend, we are going to get a mixing bend which will propagate as we move along this. And very good mixing can be obtained utilizing a 90 degree bend in a microchannel and that is one of the principle ways to mix two fluids at the microscale. Because we know that in a very; how to mix two fluids in the bulk, but how do we mix two fluids which are in at microscale and when they are flowing side by side. So, when they are flowing side by side in a microchannel, the only way these two fluids can be mixed is by diffusion which we know is inherently a very slow process.

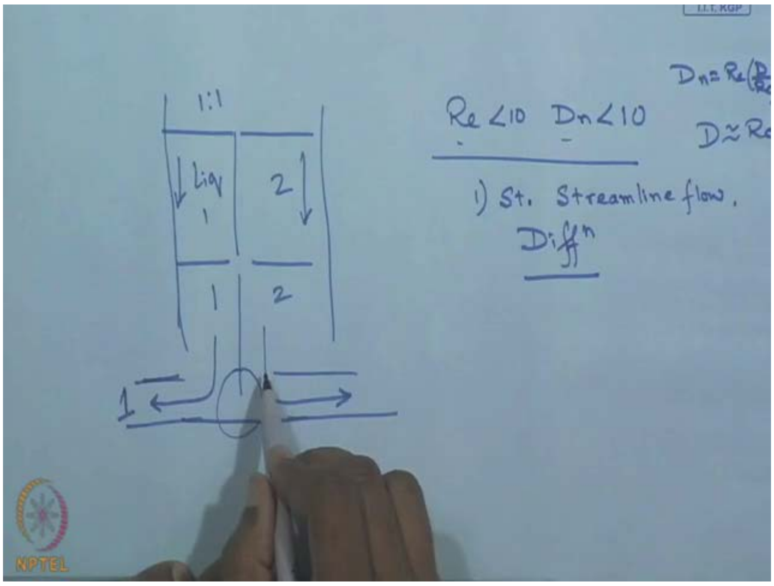
(Refer Slide Time: 14:30)



So, in order to speed up the diffusion process, in order to speed up the mixing in between the two purposefully a bend is placed in it is path so that we get very good mixing. Now, this has been used, this has been studied for the mixing of different protein solutions. And in T bends this mixing pattern, the study of this mixing pattern is very interesting results in term; and the results have been correlated in terms of mixing parameters and in terms of the dean number of the flow. So, I will show you a simple experimental setup in which two protein solutions are flowing side by side and suddenly, when it comes to a T bend the two protein solutions which were flowing side by side suddenly starts mixing and it may.

So, happen that they will mix it may so happen that, they are going to overlap and it has also been shown that for very large values of dean number, the flow may go from one side and the other flow will go to the other side. So that the flow reversal that is the fluid which was flowing from the left it is going to move to the right and the other fluid is going to move to the left.

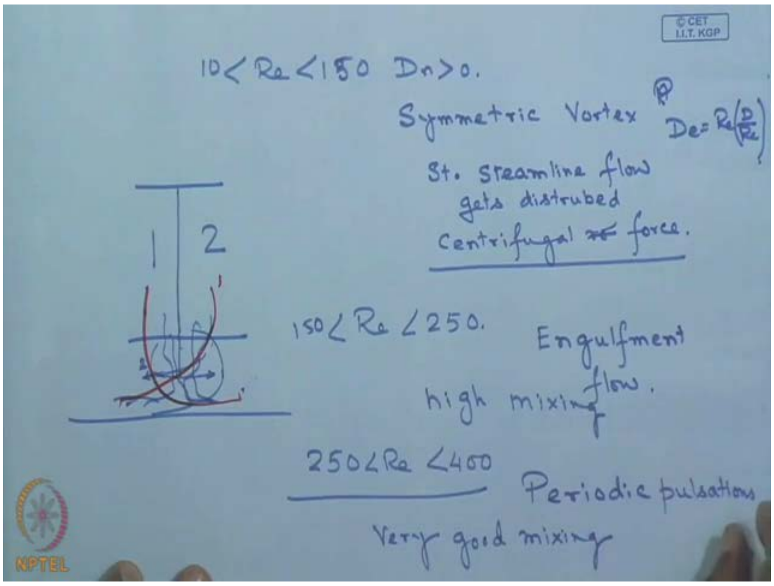
(Refer Slide Time: 14:32)



So, there are different kinds of mixing can which are possible, which can be seen in many such situations. So, the fluid dynamics in T junctions with symmetric inlet conditions they are like this. So, we have a T junction in which we have liquid 1 which is flowing in this direction and liquid 2 which is flowing in the other direction, if the flow is very small then, we will still have 1 that is moving from right to left and the 2 which will move in the other direction with very little mixing in between the two. And this happens, the classification of the of the flow patterns, the flow; these are 1 is to 1 mixture of two fluids. And the c f d simulations and visualizations simply tells us that for Reynolds number less than 10 and dean number less than 10 the where dean number is defined as Reynolds times D by R c to the power half.

So, this simply tells us that when both are of the same order then D is roughly equal to R c and if they this condition is satisfied then we have straight streamline flow even after even after the after the mixing. So, the diffusion dominates mass transfer in this case. So, this is a diffusion only process and you get very little physical mixing in between the two.

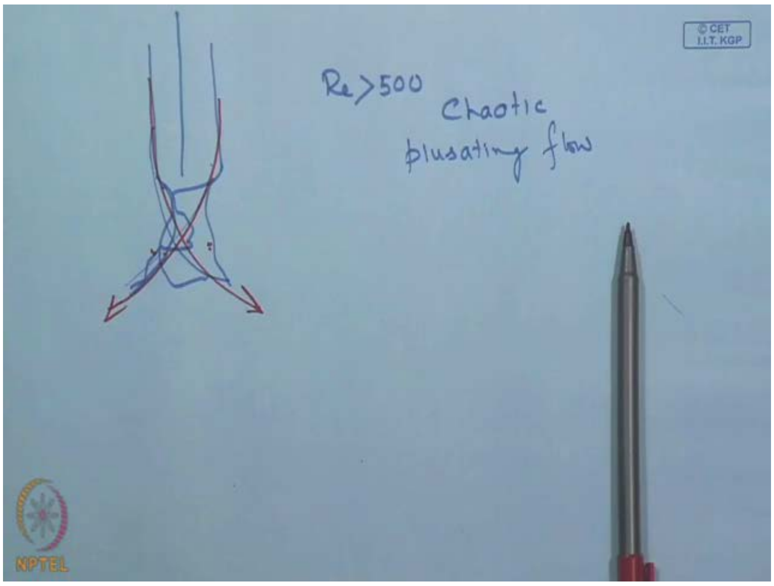
(Refer Slide Time: 16:12)



Now, when we move to a situation, where the dean number is; where the Reynolds number is greater than 10, but it is less than roughly about 150 or so, but the dean number is greater than 10 then we get Symmetric Vortex. Symmetric Vortex are formed in what you would see is that, if this is the boundary 1 and 2 and then I have the T junction then you are going to get some sort of mixing in between the two and part of 1 may go in this direction, a small amount of 2 may come in to the other direction. So, the straight streamline flow which was present before, straight streamline flow get us disturbed and this disturbance is due to the centrifugal force. As again as I said dean number is defined as Reynolds into D by R c. So, for sharp bends what you would get is the centrifugal force will start to cause mixing the this some sort of mixing at the mixing channel.

Now, the situation becomes even more interesting when the Reynolds number is greater than 150 and less than about 250 or so, then, this region is called the engulfment flow. And in the engulfment flow what you would see is that, the symmetry totally breaks up and the part of the fluid is going to move in the opposite direction and this 1 2 is going to come in the reverse direction. So, the fluid swaps to the opposite side and this leads to short diffusion length and very high mixing quality. So, the mixing increases in such systems for two; Reynolds number greater than 250, but less than 400 then the situation becomes even more chaotic and it the two will not come to this side on a continuous basis or the liquid one will not move into this direction. Rather we have periodic flows periodic pulsations of liquid column from one side to the other.

(Refer Slide Time: 19:39)

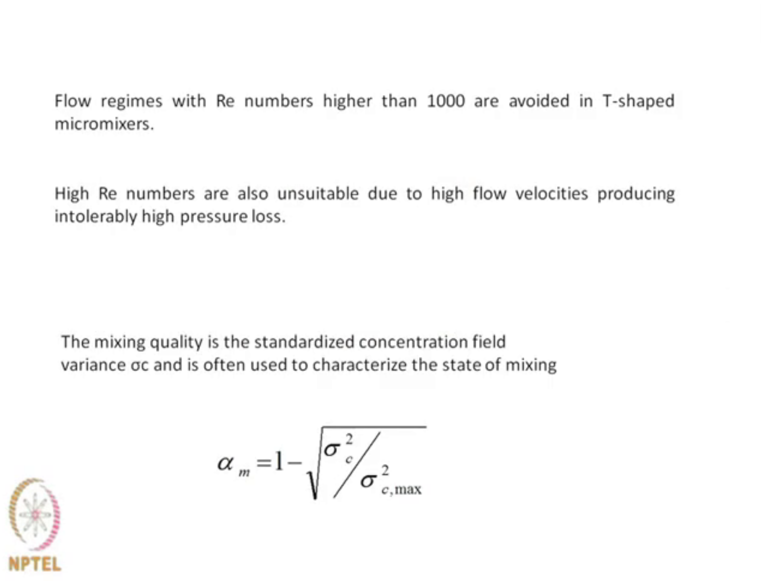


So, this pulsating flow has very high, very good quality mixing. And if the Reynolds number is greater than 500 then, what we have is a complete mixing of these two liquids. So, for Reynolds number greater than 500 it becomes chaotic pulsating flow, the vortex breaks down and you get the single flushes of fluid show ups to the other side. So, the liquid 2 will move to this side, liquid 1 will too move to the other side and this takes place at such a high pulsation that the mixing quality may actually decrease in mixing quality may actually decrease.

So, depending on the value of Reynolds number and depending on the value of the dean number one would be able to classify this type of flows into different specific domains. Generally with increase in Reynolds number the mixing quality increases, but that increase does not continue in definitely, at some point of time the engulfment of one flow to the other is has a periodic nature in it and as a result the mixing quality may decrease. But we have to keep in mind is that these are c f d simulations, these are highly specific to the geometry of the system, as well as nature of the fluid in question.

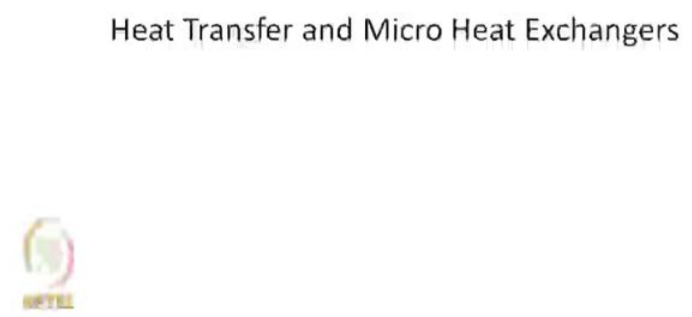
So, the limits of Reynolds number and dean number that I have proposed so far is system specific. And one has to be careful in conducting these numerical studies before one can design an actual mixer based on these phenomena, where centrifugal forces help in the mixing and the viscous forces will try to reduce any or to try to any suppress in a vortex formation. So, these are interesting areas of research, where one can think of use this simple concepts to the design of micromixers.

(Refer Slide Time: 22:07)



Next, we are going to as I was mentioning about the different ranges of Reynolds number. So, for example, Reynolds number higher than 1000 are usually avoided in T shaped micromixers as it leads to very high pressure losses and that kind of pressure loss cannot be sustained in most of the microfluidic devices. And mixing quality as I was referring to is the is the standardized concentration field and can often be used to characterize the state of mixing.

(Refer Slide Time: 22:50)

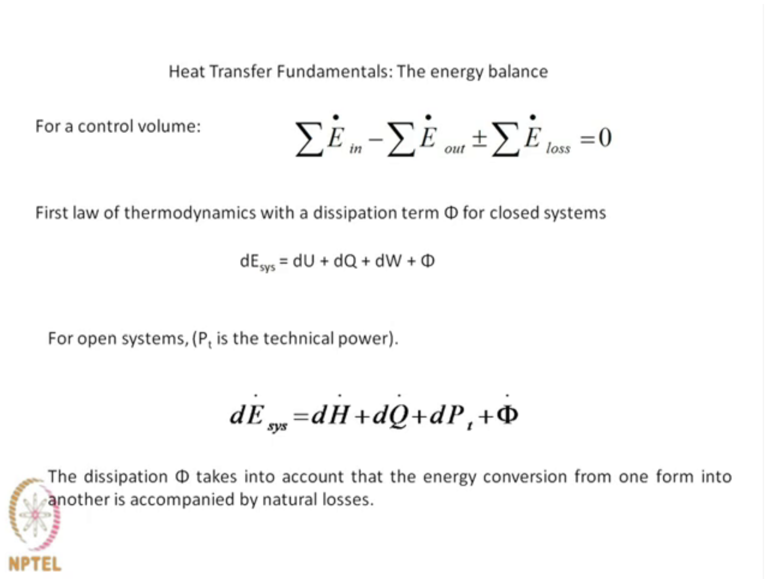


Now, we are going to come into another subtopic, which is my heat transfer and micro heat exchangers so this is a very interesting application of microscale transport processes. Now, we all know that the Nusselt number let us say, for flow through a tube there are two distinct conditions, one is a constant heat flux and the other is a constant wall temperature.

Now, in all cases after the flow becomes fully developed, thermally fully developed the Nusselt number defined as H T by K, where d is the length scale, by K is a constant. The value of which is a 3.66 and 4 point something and depending on whether you have a uniform heat flux condition or uniform wall temperature condition. Now, as we know that in microchannels, the value of d is quite small. So, if the value of d is small then all and if Nusselt number is constant then the value of H has to increase. So, this is the reason, that in microchannels the value of heat transfer coefficient convective heat transfer coefficient can be very high. This fact or this phenomena of high convective heat transfer coefficient has have been utilized in devising a number of equipments devices. So, to extract or to dissipate large quantities of it for very small area,

The examples that we can think of are for the cooling of microchips where a small amount of heat maybe generated, but the area where it is generated is very small. So, the heat flux that needs to be removed can be very large. So, a small amount of heat generation can lead to a situation where a large heat flux has to be removed in order to maintain the integrity of the device, integrity of the I c chip. And for some such situations microscale heat exchangers based on the or developed on the principle of high heat transfer coefficient can be a very effective tool to dissipate the large heat flux produced in such systems. So, in this part of in this subtopic we are going to look into the heat transfer in and micro heat exchangers, how do they perform? What are the governing equations for such cases and so on.

(Refer Slide Time: 25:39)



So, we look into, the first the fundamental heat transfer fundamentals the energy balances which we are very much familiar with. So, for any control volume the rate of energy in minus rate of energy out and plus minus any kind of loss generation etcetera, all are rates it is going to be equal to zero. So, if we put into the first law of thermodynamics, then we can add we need to add a dissipation function phi for close systems and for open systems this the; these are the two equations for close systems and for open systems with phi denoting the dissipation term. So, the dissipation phi takes into account the energy conservation from one form to another form accompanied by natural losses. For example, if we have flow in a microchannel with a bend, then the dissipation function over there essentially tells us it is related to the pressure losses in such a system. And the frictional losses, which is going to change the internal energy content of the fluid. So, these are dissipation functions which could also be viscous losses need to be added or need to be included in the energy equation so as to describe the energy content of the inflow; of the inlet fluid and the outlet fluid and which are essentially all the fundamental relations, thermodynamic relations for energy balance.

(Refer Slide Time: 27:17)

For a process device with mass flow rate *m*, heat flux *q* over the boundary, technical work W_t, or mechanical power *P*, and chemical reaction, the energy equation is written as

$$\dot{Q} = \left(\dot{m}(u_2 - u_1)\right) + \left(\dot{m}\left(\frac{p_2}{\rho} - \frac{p_1}{\rho}\right)\right) + \left(\dot{m}g(z_2 - z_1)\right) + m\left(\frac{\alpha_2 \overline{V_2^2}}{2} - \frac{\alpha_1 \overline{V_1^2}}{2}\right) + \dot{E}_q + P$$

The temporal energy change in a system consists of

- net energy flowing
- pressure,
- gravity,
- KE,
- energy produced
- technical work

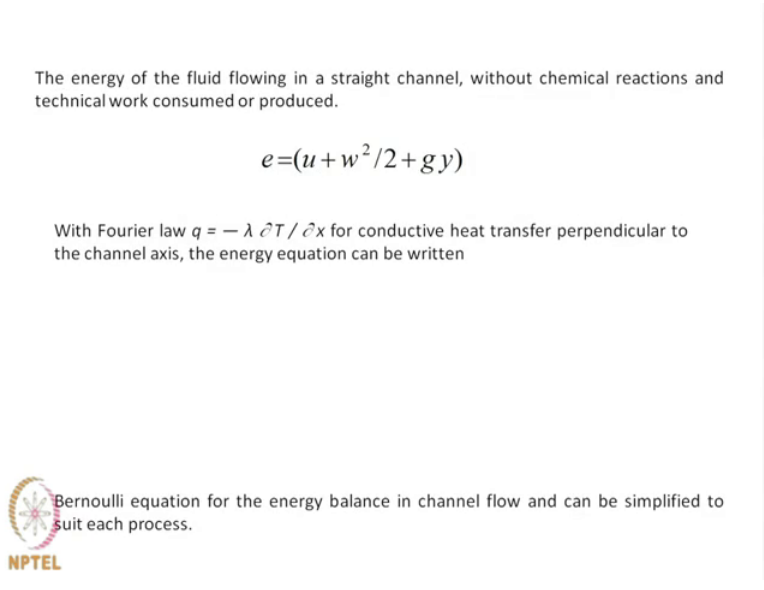
NPTEL

So, let us see, we have a process device, where the mass flow rate is m, the heat flux q is over the boundary. So, if it is a tube then, we can think of this q as the heat flux that is being added through the walls of the tube and the technical work which is W t or the mechanical power which is denoted as the p, and there may be a chemical reaction and therefore, we can write the energy equation as given. So, if you see here the Q dot is the amount of heat, the amount of heat that is being the change in that m; change in the energy and this change in the energy has several components. So, if you see the first term on the right hand side. So, it is the net energy flowing to the control volume as a result of flow, as a result of convective flow and could be also as a result of conduction across the system boundaries.

If you think of the second term, second bracketed term on the right hand side it refers to pressure. So, m dot times delta p that tells you the contribution of pressure to the overall measure of energy of a system, the third term on the right hand side tells us about the hydrodynamics, tells us about how what is the head? The hydrodynamic head which is being suppose the fluid is moving from low to high then what is it is going to be, it is potential energy content. The fourth term on the right hand side refers to the kinetic energy, the change in kinetic energy of the flow in alpha 1 and alpha 2 are the correction factors. So, these are essentially the kinetic energy terms multiplied by the mass flow rate and it tells us the change in kinetic energy between location one and location two.

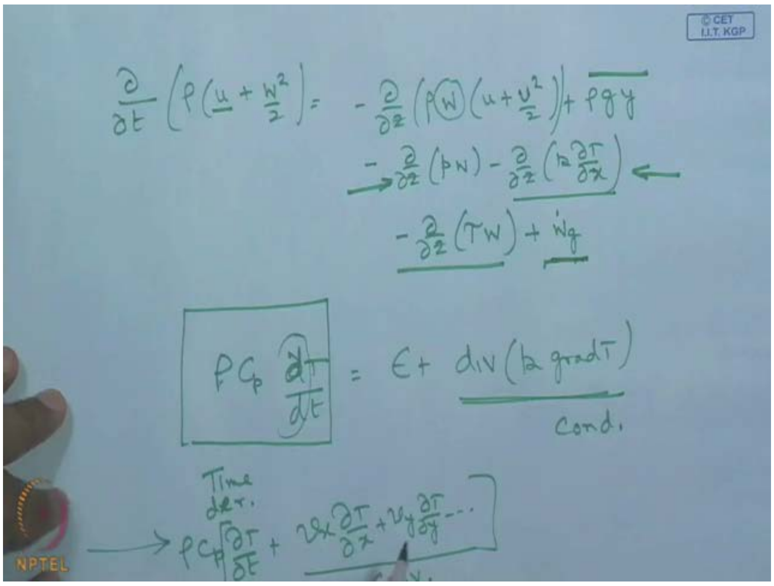
The fourth term e dot q is the energy produced or consumed, the energy produced can be a result of a reaction well so a chemical reaction so it could be an exothermic or an endothermic reaction and e dot q simply takes into account the amount of energy produced or absorbed in such a system. And if you look at the last term it is a technical work, it is the work that is being added to the system by a pump or extracted from the system by a turbine. So, this equation then gives us the complete description of the net energy change, the special energy change for a flowing fluid, considering the internal energy, the kinetic energy, the potential energy, the work done against surface forces. Such as pressure the heat generated due to let us say reactions and the any amount of work that is being done on the system, for example a pump or by the system, for example, by a turbine.

(Refer Slide Time: 30:52)



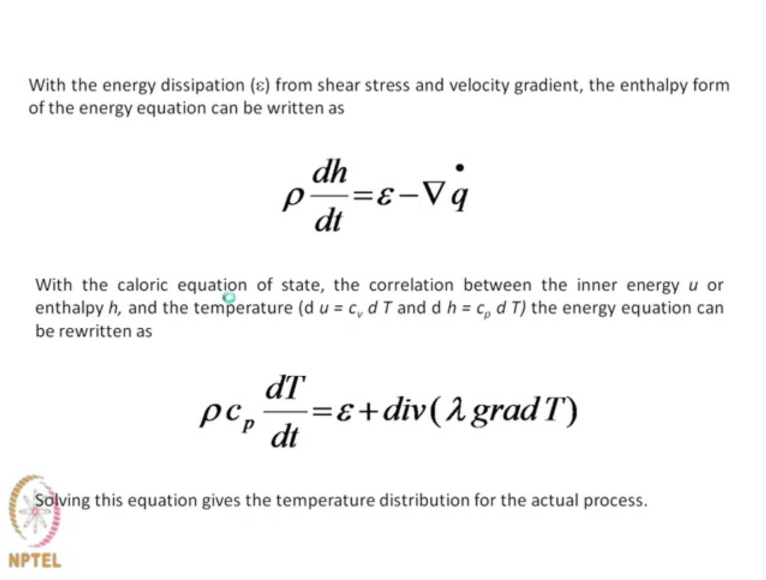
So, such energy equation would be equally valid for microscale processes as well with certain additional terms which we will see later on. So, the energy of the fluid which flows in a straight line, without chemical reactions and technical work is simply the sum of the internal energy, the kinetic energy and the gravitational energy.

(Refer Slide Time: 31:18)



So, we can add Fourier's law to it, for conductive heat transfer perpendicular to the channel axis and one can write the energy equation in the following form. Which is simply going to be the time rate of change of internal and kinetic energy would be equal to the spatial change of, this W is the mass flow rate internal plus kinetic energy then there would be a gravitational force component minus del del z of pressure times W minus del del z of k times del t del x. So, this refers to the conductive heat which is being added to the system minus del del z of tau times w. So, tau times W it gives us the dissipation plus W dot g so any amount of work done. So, this is the gravitational term, this is due to pressure, this is due to conduction, this is due to dissipation and this is the amount of work that is being done by the system or on the system.

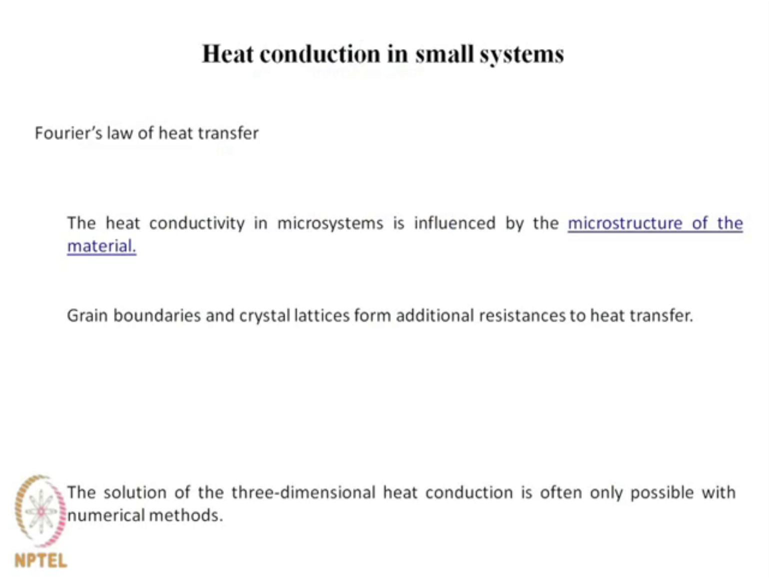
(Refer Slide Time: 32:34)



So, this can also be thought of it is, as if it is the Bernoulli equation for the energy balance in channel flow and can be simplified to suit each process. Now, if we try to get a more meaningful equation to this, which is more common to us, we can write it as rho C p d T by. So, these are substantial derivatives should be equal to epsilon plus divergence of k times gradient of T. So, this is the conduction and in here since, this is a substantial derivative, we have the time derivative since, this is the substantial derivative we have the time derivative as rho C p del temperature over del time plus we have v x times del temperature over del x plus v y times del temperature over del y and so on. So, this is the time dependent term and this is the convection so, this refers to the convection and this refers to conduction in a microchannel.

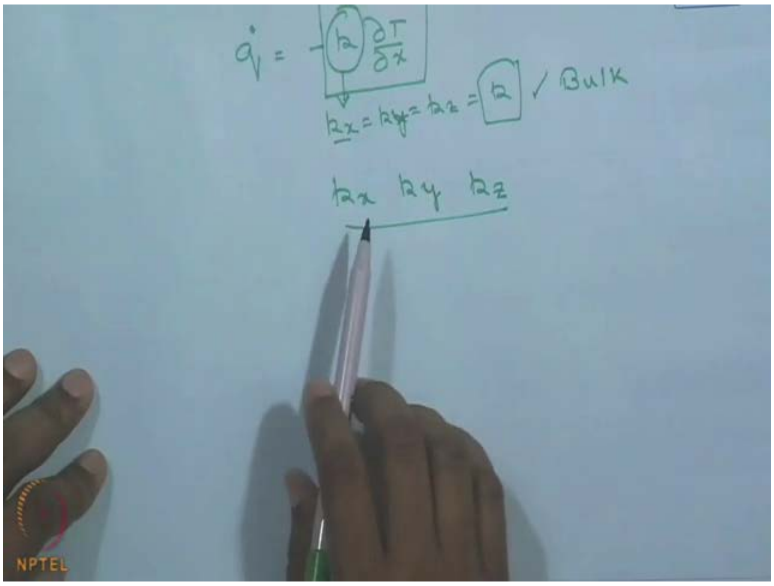
So, solving the; this equation essentially gives us the temperature distribution, if you could solve this problem then it gives us the temperature distribution for any real process.

(Refer Slide Time: 33:56)



Now, heat conduction in small system so, these are common for large systems as well as for small systems, but in small systems there would be a complicating factor and the complicating factor is something that we have needs, we need to be very careful about.

(Refer Slide Time: 34:17)



For example, in Fourier's law we all know about Fourier's law we simply says that the heat flux is equal to minus k times delta T by delta x. So, this Fourier’s law assumes, that the k in this case it assumes, that the k the thermal conductivity is independent of position it is same as in all directions. So, there is no question of writing k x k y k z, because all of them are equal and equal to one value of thermal conductivity, but heat. So, this is true for a bulk system, but when we think of a microsystem, the microsystem is the thermal conductivity like many other physical property is influenced by the microstructure of the material.

So, you have grain boundaries and crystal lattices which form additional resistances to heat transfer. And the solution of the three-dimensional heat conduction is often possible with numerical method. So, what you need to do then is, the right form of Fourier's law would be the tensor nature, where the k x k y k z etcetera are will be allowed to vary and they are going to be a functions of position. And the solution of this three-dimensional heat conduction equation is often possible only with numerical methods.

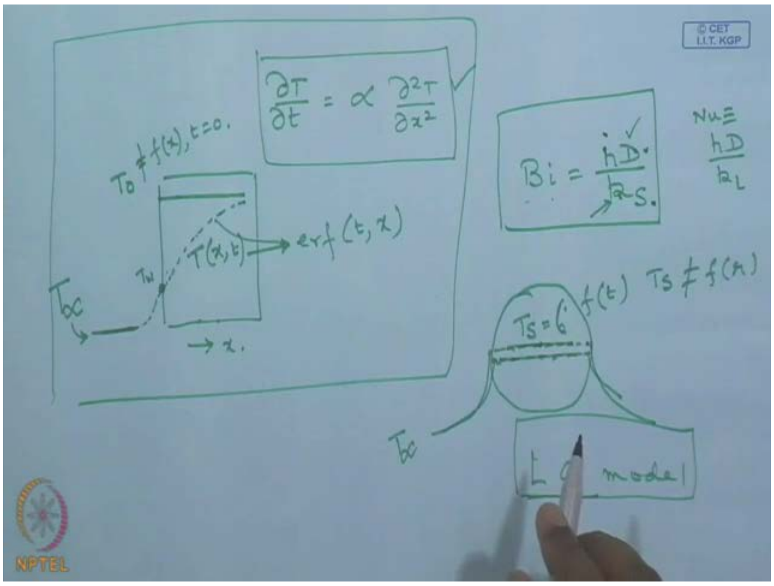
(Refer Slide Time: 36:06)

Miniaturization will not influence the temperature development and the heat flux for a semi-infinite body.

For small bodies (high Fo and low Bi number), the temperature distribution can be approximated by asymptotic solutions.

So, Fourier's equation of heat transfer has to be expanded to the tensor notation with the heat conductivity tensor. But, there is one more thing that one would; I would like to mention is that the miniaturization will not influence temperature development and the heat flux of a semi infinite body. Now, what is a common form of semi infinite body? That we know of like.

(Refer Slide Time: 36:19)



The transient temperature development in a semi infinite body is in one dimensional form is delta temperature by delta time is the alpha times del 2 T by del x square. So, this equation needs to be solved, where the temperature is a function of x as well as a function of time. So, let us say we have a solid object where this is the x direction and then what we need to have is then; this is the in constant initial temperature T 0 at time t equal to 0 for all values of x. So, t 0 is not a function of x and it is at time t equal to 0.

Now, when we allow this solid block of; solid block to come in contact with another fluid whose temperature is lower than T 0 then with time what you would see is that, the temperature will fall, this is the temperature of the wall and there would be a thermal boundary layer, a thin thermal boundary layer and then the temperature will be equal to T W and this growth of this thermal growth of this equation the temperature variation can be expressed by this equation. And there are solutions for semi infinite semi infinite of body and which describes the temperature development in a solid body. So, with constant wall temperature T W becomes constant and the dimensionless temperature distribution is govern by an error function, which is an error function in t and x. So, this T which is a function of x and time can be denote can be expressed by an error function.

But, what is important here to know is that the same equation can be used for a semi infinite body and for small bodies. There are certain numbers which we need to; for small bodies which are small values of Biot number and Biot number is defined as h D by k, where this k is the k of this solid, unlike Nusselt number which is also h D by k, but this k is for the liquid. So, for a small value of Biot number let us say, we are talking about a sphere now in a sphere the diameter is small. Now, if the diameter of the sphere is small or if the value of k is large then, what one can think of is that the temperature inside the sphere will remain constant, will remain invariant with time and this is the temperature T infinity, where as this is the temperature of the solid.

Now, one can see that for small value of Biot number that is small d or large case or small value of h, this T s is going to be a function of time, but it T s is not going to be a function of let us say r in this case. So, at one instant of time this is going to be the temperature profile, at the next instance the temperature will reduce to another value, but still inside the solid the temperature will not vary with r and that is only possible if your Biot number value is small. And physically it tells us that if the thermal conductivity of the solid is large if the size of the system is small or if you are dealing with a small value of heat transfer coefficient then the small value of Biot number indicate that I can assume space wise Isothermality in any solid.

And once I assume space wise Isothermality then it this modeling that system becomes quite easy and that is known as the lumped capacitance model. So, the lumped capacitance model or L C model in short allows us to get analytical solution of the temperature profile inside a solid object. And this is extremely useful, this is very useful to obtain the behavior of a fluid, behavior of a solid when it is quenched. So, let us say, we have a very small steel bowl that hot steel bowl which is put into a liquid and how would the temperature of the steel bowl change with time that is extremely important from an engineering perspective.

Similarly, for micro system how does the temperature inside the micro system vary with temperature? If you could assume that inside the small micro systems since the length scale is small there is no variation of temperature then, we would be probably, we would be able to obtain an analytic solution of temperature variation with time inside such a system.

(Refer Slide Time: 41:58)

The time dependent situation - second Fourier law

Fourier and Biot number

$$\mathrm{Fo} = \frac{a\,t}{x^2} \qquad \mathrm{Bi} = \frac{\alpha\,x}{\lambda_s}.$$

Miniaturization will not influence the temperature development and the heat flux for a semi-infinite body.

For small bodies (high Fo and low Bi number), the temperature distribution can be approximated by asymptotic solutions.

NPTEL

So, the two numbers of relevance here are Fourier number, which is alpha t by x square where alpha is a thermal diffusivity t is the time, x is the length scale and Biot number as I have already explained. So, Fourier number tells us how fast a temperature found will penetrate into a solid, where as in Biot number tell us how the temperature of a solid is changing. So, these two are important parameters, the time dependent important parameters in the second Fourier law.

(Refer Slide Time: 42:32)

Convective heat transfer in microchannels

The total heat transfer in microstructured devices consists of heat conduction through the walls and convective heat transfer from the wall into the fluid in microchannels.

For straight laminar flow, the dimensionless heat transfer coefficient, the Nu number, is constant.

For constant wall heat flux, Nu_q = 4.3, for constant wall temperature Nu_T = 3.66.

In a wide gap or narrow slit, the Nu number is 7.54 *(q* = const.) and 8.24 *(T* = const.) for double-sided heat transfer,

and 4.86 *(q* = const.) and 5.39 *(T* = const.) for single-sided heat transfer.

NPTEL

Now, this these are something, sorry these are, these I have already described partially. Is that the total heat transfer in micro devices can consist of heat conduction through the walls and convective heat transfer from the wall into the fluid in the microchannel. So, for example, if you are making a micro device and while you are making a micro device if you consider only the convective heat losses from let us say it is a tube and it is micro scale in size. So, the diameter of the tube is 100 micron and it is length is 2 centimeters.

So, now, when you are; it is made of silicon. Now, when you are; when it is let us say v grouped micro heat pipe. Now, there are situations one has to consider not only convective heat losses from the outer wall of the microchannel, but the heat which is which get us transported through the wall. So, in micro devices, unlike in macro devices the conduction heat transfer through the body of the device itself needs to be taken into account to calculate the total amount of heat loss or total amount of heat transfer from such a system.

(Refer Slide Time: 44:16)

For smaller channels, the heat transfer coefficient increases due to the constant Nu number.

At the entrance of a channel, at channel junctions, expansions etc. the disturbed flow enhances the radial transport

- increased pressure loss as well as heat or mass transfer.

Secondly, this we as I said for straight laminar flow, the dimensionless heat transfer coefficient is a constant and it is 4.3 or 3.66 for a wide gap or a narrow slit the Nusselt number can vary for double sided heat transfer. And for smaller channels the heat transfer coefficient increases due to the constant Nusselt number as I have already described. And at the entrance of a channel at channel junctions expansions etcetera the disturbed flow enhances the radial transport.

So, so far we have seen that the at the entrance of the channel or at a 90 degree bend how the fluid dynamics get us altered resulting in larger pressure drop. Here, what we are saying is that, not only the fluid dynamics get us altered, the altered fluid dynamics helps in transporting more heat due to the presence of the 90 degree bend. So, an obstruction in the path or alteration of the fluid path not only increases pressure drop it also increases heat transfer in microchannels. So, you we get more pressure loss, but at the same time we have increased heat or mass transfer and it is just a balance of which one we would want for our system.

Now, one more thing is that, this is fine. But, when we talk about Nusselt number, the constant value of Nusselt number and the smaller length scale giving rise to high value of heat transfer coefficient that is fine, but is it really beneficial is the high heat transfer rate helping us in obtaining large heat removal the answer is yes and no. The high heat transfer coefficient definitely helps us to extract more heat, but the area in consideration is quite small since, we are talking about micro devices. So, we need to somehow increase the area of contact to utilize the full potential of a high heat transfer coefficient now, how that can be done. So, there has to be numerous let us a numerous tubes, micro tubes which needs to be employed simultaneously to extract more heat.

Now, whenever that happens it gives it. So, it is a fabrication problem, how do we fabricate a large number of such channels which will utilize the high heat transfer coefficient, high convective heat transfer coefficient to extract more heat. And it is not only a fabrication issue it is also an issue of how do we feed the coolant liquid equally into all such channels or tubes. So, proper feeding, uniform feeding of liquid through all the fabricated channels, at the same time with minimum pressure drop that is the major challenge being faced by engineers who are trying to build smaller and smaller devices to take the help of the higher convective heat transfer coefficient. We also know that when the flow starts taking place it is not fully developed, it is not hydro dynamically fully developed or thermally fully developed.

All the relations or correlations that we have so far after fully developed flow and for a micro device it may so happen that before the flow gets fully developed we reach end of the channel. Now, if that happens then what is the equation, what is the relation to be used for calculating the Nusselt number which is essentially a design parameter. So, the normal convention is to use the expression for Nusselt number for fully developed flow and multiply it with some factors in order to obtain the Nusselt number for the zone in which the flow is developing more thermally and Hydrodynamically.

So, for a large tube in a micro system it is not a problem, because the entrance length part the portion where the flow becomes fully developed is quite small compare to the overall length of the channel. But in the microchannel when a micro system the length is so short that we are not, we may not provide enough length for the flow to become fully developed.

(Refer Slide Time: 49:03)

The mean Nu number in the entrance flow Nu_{me} is calculated with the mean Nu number in straight channel flow Nu_m according to the following correlation

$$Nu_{me} = \frac{Nu_m}{\tanh\left(2.432\, Pr^{1/6} X^{*\,1/6}\right)}.$$

This equation is valid for the entire channel length X^* and $Pr > 0.1$

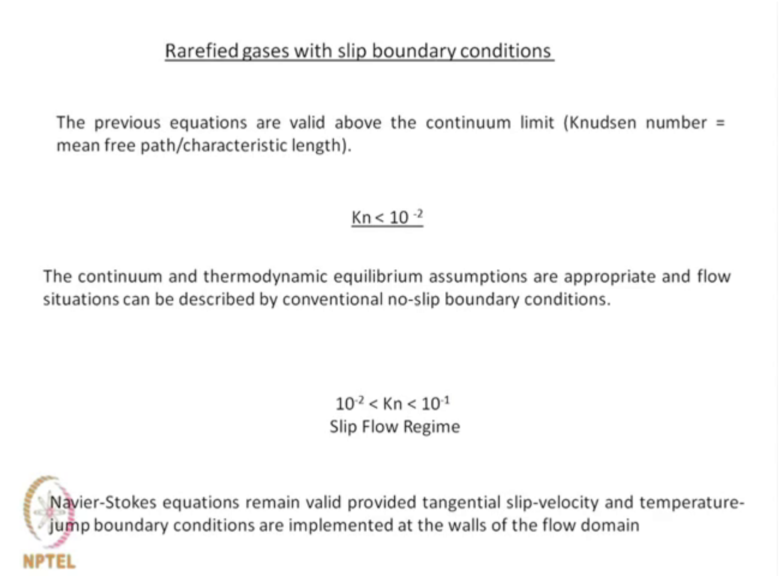
Fully-developed flow

Turbulent flow does not often occur in microchannels, near turbulent conditions at some locations.

NPTEL

So, we have these equations which tells us the Nusselt number, the Nusselt number which is developing is the Nusselt number which is straight channel flow and there are certain correlations which are available. So, this is the Prandtl number and this is the dimensionless length and the equation is valid for the entire channel length X star and for Prandtl number greater than 0.1.

(Refer Slide Time: 49:46)

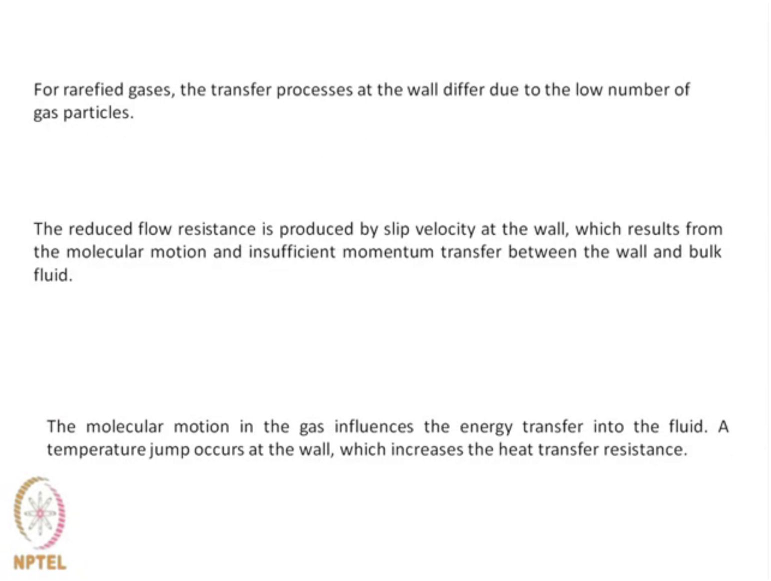


So, since turbulent flow is rarely encountered in micro channels. So, this kind of a situation may not arise, but there are some turbulent conditions at locations special near the bends and so on. Now, comes the question of when we start reducing the size of the devices smaller and smaller and letting it operate at very low pressures, what happens is that we will probably reach a situation where we have struck the continuum limit.

Now, when we have continuum limit and as we know that the continuum limit is generally expressed in terms of Knudsen number which is the ratio of the mean free path and the characteristic length. So, if the value of Knudsen number is less than 10 to the power minus 2; that means, the mean free path is small compared to the characteristic length and for such cases the continuum and the thermodynamic equilibrium assumptions are appropriate and the flow situations can be described by conventional no-slip boundary condition. If the Knudsen number is in between the two that is 10 to the power minus 2 to 10 to the power minus 1, then we come to the regime which is known as the slip flow regime.

So, in the slip flow regime the Navier-Stokes equation will still remain valid provided we alter we modify our no slip condition with a tangential slip-velocity and a temperature jump incorporated into our boundary condition. So, you may not be able to use that u and v the velocity in the x and the y component are both 0 at y equal to 0, where y equal to 0 denotes the wall. But we will have to provide a slip-velocity and there are several models which take into account the possibility of slip. Similar, to slip-velocity we would also have to incorporate a temperature jump across the interface.

(Refer Slide Time: 51:47)



So, previously for total, for continuum cases, the temperature of the solid and the temperature of the liquid or the fluid adjacent to it at the same at the interface, but if the flow is verified then we have a temperature jump across the interface. So, the reduced flow there since we have a slip velocity. So, therefore, the flow resistance will be rather less and the molecular motion etcetera, there is going to be insufficient momentum transfer between the wall and the bulk fluid.

(Refer Slide Time: 32:34)

(Refer Slide Time: 52:05)

Accommodation Coefficient, β

When a particle impacts with a surface energy is transferred in the form of heat and stress, which leads to 2 main types of accommodation coefficients thermal and transverse momentum.

The thermal accommodation coefficient is the fraction of heat transferred between the surface and the molecule. "The ratio of the average energy actually transferred between a surface and impinging gas molecules scattered by the surface, to the average energy which would theoretically be transferred if the impinging molecules reached complete thermal equilibrium with the surface.

Transverse Momentum Accommodation Coefficients (TMAC)

The TMAC is the fraction of the momentum normal to the wall that is transferred to the wall in terms of stress. This stress is more commonly known as pressure. By creating a pressure on the wall some of the vertical momentum is lost.

NPTEL

So, in order to characterize the amount of transfer, amount of momentum transfer and heat transfer there is a concept called accommodation coefficient beta. Which is defined as the ratio of the average energy which actually get us transferred between a surface and the gas molecule, to the average energy which would theoretically be transferred if the impinging molecules reached complete equilibrium with the surface. Now, what happens here is that the molecule comes and hits the surface and reflects back into the bulk. Now, if this is a continuum case then this would come stay here for some time or stay here for sufficient amount of time to reach complete thermal equilibrium and then get will get reflected back, but the due to the rarefied condition it comes and then immediately it goes back. So, it cannot take enough the total amount of momentum, total quantity of momentum or total quantity of energy.

(Refer Slide Time: 53:11)

For rarefied gas flow (0.01 < Kn < 0.1), the boundary condition of the gas velocity at the wall is described by, (slip length ζ)

$$w(x=0)=\zeta\left(\frac{\partial w}{\partial x}\right)_{y=0}$$

The slip length ζ can be calculated with the accommodation coefficient β and the mean free path Λ of the molecules. The accommodation coefficient β describes the efficiency of the momentum and energy transfer from molecules to the wall and vice versa.

$$\zeta \approx \frac{2-\beta}{\beta}\Lambda$$

$\beta = 2$, from Kinetic theory for continuum regime

Experimental data for β can be found in literature.

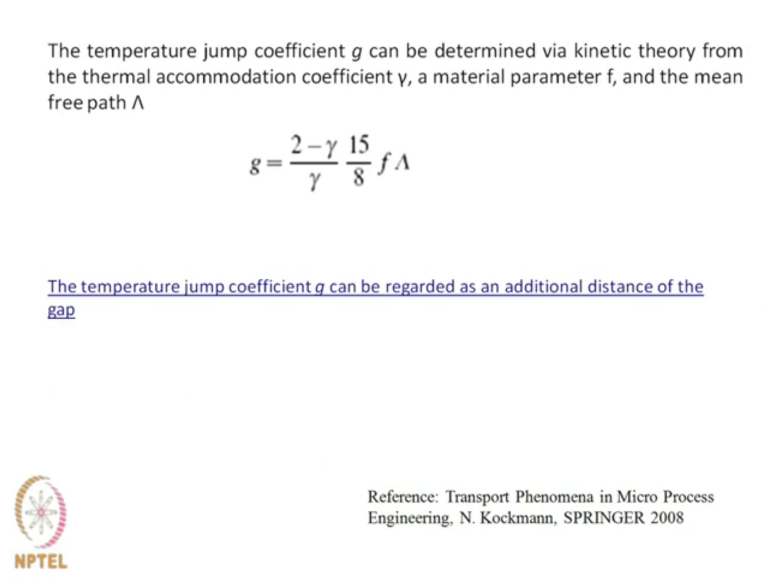
The temperature jump at the wall is described in a similar way with the temperature jump coefficient g

$$T(x=0)=T_r=T_W+g\left(\frac{\partial T}{\partial x}\right)_{x=0}$$

So, this beta essentially tells us about how much, what fraction of energy or what fraction of it can take due to the rarefied condition. So, for rarefied flow the boundary condition, the velocity is going to be a function of zeta this which is called the slip length and the slip length can be calculated from the accommodation coefficient and the mean free path of the molecules.

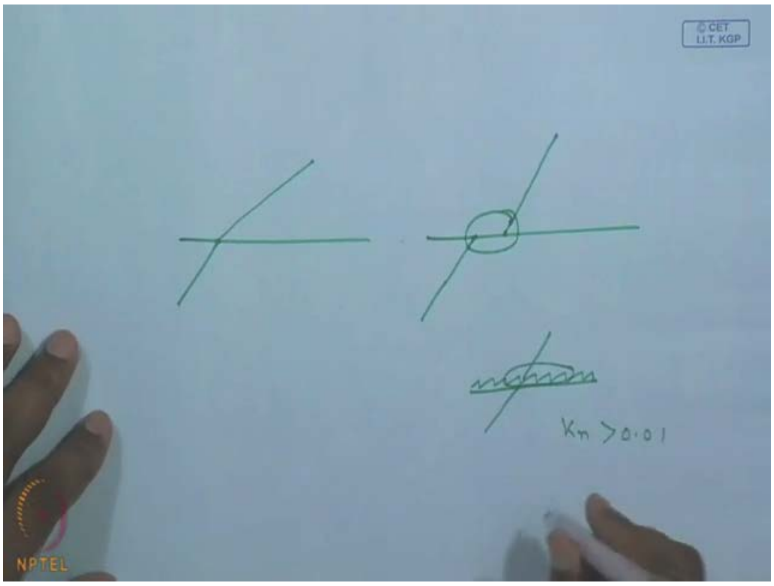
So, the experimental data on the accommodation coefficient beta it is defines the efficiency of the momentum and energy transfer from molecules to the wall and vice a versa. So, beta can be calculated from kinetic theory and there are enough large quantity of data on beta which can be found in the literature and similar to the jump the slip there is also temperature jump across the wall.

(Refer Slide Time: 54:23)



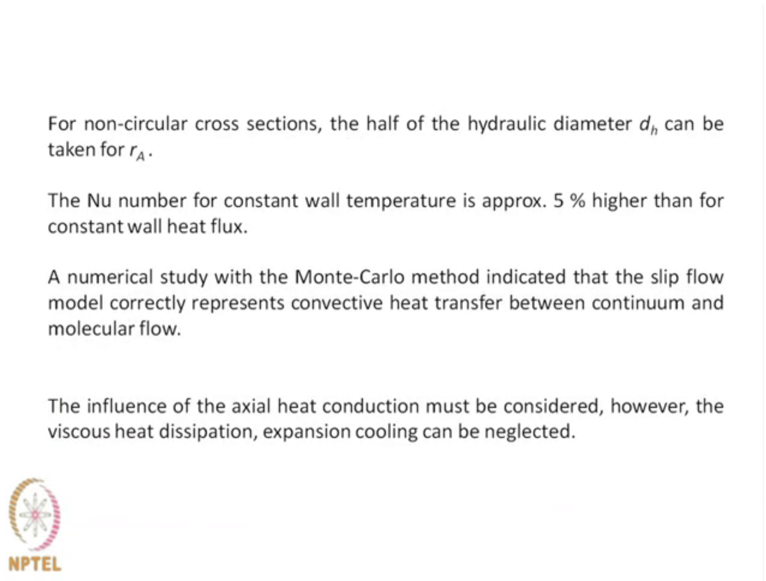
So, that the temperature at x equal to 0 is not t w, but g which is a temperature jump coefficient like the coefficient zeta in the hydrodynamic case times the gradient at the wall. So, the presence the temperature jump coefficient can be regarded as an additional distance to the gap.

(Refer Slide Time: 54:32)



So, it may look something like this that at the interface, we have the temperature profile in the solid and the temperature profile in the fluid. But, in the case of rarefied condition, we have the temperature profile in the solid and a temperature profile in the liquid with a change in temperature on the solid side of the interface and on the liquid side of the interface. So, this is the temperature jump that we are talking about. So, we could think of a situation in which I am adding an additional resistance over here and this additional resistance is the temperature jump in rarefied gases, for cases where Knudsen number is greater than 0.01. So, this temperature jumps coefficient g it can be regarded as an additional distance to the curve.

(Refer Slide Time: 55:28)



So, for non circular sections, in order to calculate these half of the hydraulic diameter can be taken for r A. And since, we have more mobility of the molecules in rarefied condition the Nusselt number for constant wall temperature is 5 percent higher than for the constant heat flux. And numerical study with Monte-Carlo method indicated that slip flow model correctly represents convective heat transfer between the continuum and the molecular flow. And the influence of actual heat conduction must be considered, but the viscous heat dissipation and evaporative cooling and expansion cooling can be neglected for flows in microchannel.

So, in summary what we can say is that, heat transfer in microchannel is essentially similar with most of the equations of macroscale that can be used except for certain cases. Where for Fourier's law the thermal conductivity k needs to be the three dimensional variation of thermal conductivity has to be incorporated and the tensor form of Fourier's law needs to be used with numerical solution only possibility is numerical solution. And Secondly, during a rarefied condition, the accommodation concept of accommodation coefficient tells us the efficiency of momentum transfer when a molecule hits a surface. And the accommodation coefficient thermal accommodation coefficient tells us that the impinging molecule and the wall has not reached complete thermal equilibrium and this can be modeled by having the presence of a velocity at the wall where it is a slip which is the function of the slip length and the gradient in velocity.

And similarly, the temperature can be expressed as a function of a temperature jump at the interface which is as if we are adding a thin layer at the interface. So, these are some of the factors that need to be considered for modeling or for using convective flow both the hydrodynamics and heat transfer in microchannels. Thank you.