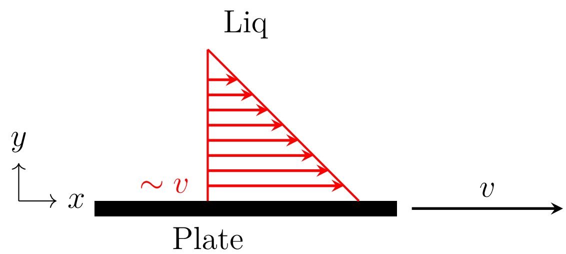
Transport phenomena

Angelantonio Squicciarini

0.1 Solution Models

If I put a drop of a heavy liquid on a surface, it does not spread. if you would like to stir a thick cream as compared to that of water in a glass, in the first case you are going to spin more energy (require more force) to turn the spoon inside the glass.
The concept of how these liquid molecules resist the relative motion in between them has given rise to the concept of viscosity.
I draw a simple system in which there is a solid plate moving with velocity $\bar{v}$ and on top of it. As the solid plate starts to move, the liquid layer will also start to move (natural tendency of the liquid in contact with the solid plate). Near the solid plate, the liquid will have the velocity which would be equal to the velocity of the solid plate but, as we move away from the plate, the effect of the plate will be felt lesser and lesser by the liquid layers on top of the plate. If I could draw the velocity profile, some sort of a very rough approximate velocity profile of the liquid initiated by the movement of the plate, it would probably look something like this



the velocity of fluid will progressively decrease, ultimately at a point far from the plate the velocity will be roughly zero.
There would be a liquid molecule which is associated with the first layer close to the wall, and due to its Brownian motion, there is a possibility that it would jump to the upper layer and similarly, a molecule from the upper layer, can come to the lower layer. The molecule when it goes from the lower layer to the upper layer, carries with it the momentum associated with the velocity of the bottom layer, so it carries with it more

momentum corresponding to that of any molecule existing on the top layer. This transport of momentum, with transport of molecule with velocity more than that of the upper region, will carry an additional momentum when it goes to the top plate, which would try to force the upper layer move with a velocity close to that of the bottom layer. In similar, on the other hand, when I have a molecule from the top layer coming to the bottom layer, the tendency of this molecule will be to slow the faster moving layer.
There will always interaction between the layers as a function of y (distance from the solid plate).
The origin of the viscosity is molecular in nature and the result of the viscosity is essentially transport of x momentum in the y direction.
The molecular transport of momentum is found to be proportional to the velocity gradient (variation of the x component of velocity with y). The shear stress experienced by the top layer, because of the difference in velocity between these 2 layers, is expressed in this form:

$$\tau \propto \frac{d\bar{v}_x}{dy} \tag{1}$$

some of the fundamental laws that we see come from our understanding of what is the cause and what is the effect.
I you think of heat transfer, for example conductive transport of heat, the amount of heat transported by conductive heat between 2 points, essentially depends not on the temperature but on the temperature gradient. If we consider 1 directional case, in which the temperature varies only with x, not with y or z, the cause is the temperature gradient. As a result of this temperature gradient, there would be some sort of heat transfer (heat flux) in between 2 points where there exists, and the law that relates the two is the Fourier's law where k is a constant called thermal conductivity:

$$Q = -k\frac{dT}{dx} \tag{2}$$

we also have a minus sign, denoting that heat always gets transported from higher temperature to lower temperature.

Similarly, when we talk about mass transfer, the diffusive mass flux of species A is the effect of the gradient in concentration of species A between 2 points. The gradient is important not just the difference in concentration of species. The law that relates the two is Fick's law:

$$N_A = -D_{AB}\frac{dC_A}{dx} \tag{3}$$

Where D_{AB} is the diffusion coefficient, diffusion of component A in B. Coming back to the momentum transfer we have the Newton's law of viscosity:

$$\tau_{yx} = -\mu \frac{d\bar{v}_x}{dy} \tag{4}$$

All fluids which obey this type of law for momentum transport because of difference in velocity, are known as the Newtonian fluid. There are some specific types of fluids which do not obey the Newton's law.

In which direction this momentum transport is taking place ? I understand that my velocity is in the x direction, so the momentum is also in the x direction, so the transfer of momentum that we talk about is essentially the transport of x momentum. The correct way to write is τ_{yx} where the first subscript denotes the direction in which the momentum gets transported and the second subscript denotes the directional momentum that we are talking about. τ_{yx} represents the x momentum getting transported in the y direction. Since there is a variation in velocity in the y direction, the x momentum gets transported in the y direction because of the existence of Thermo physical property which is known as viscosity. More the viscosity, there would be more transport of momentum, and these 2 layers are going to be more bonded, connected together such that they will oppose the relative motion between these 2 layers.

We should keep in mind that whatever we are discussing we are restricting ourselves for laminar flow where the principal reason of momentum transfer, heat transfer are molecular in nature. So due to the molecular motion which is Brownian motion, the momentum gets transported.

These 3 fundamental equations cannot be derived analytically, they are derived observing a large number of data points will you calculate the amount of heat transfer based on the temperature variant or amount of mass transfer based on the concentration gradient and so. These equations are phenomenological in nature.

These 3 questions are the same conceptually but there is one difference. Heat and mass transfer, they are identical in nature because mathematically T and C_A both are scalar quantities, therefore Q and N_A are going to be vectors. On the other hand, $\bar{v}_x$ is a vector quantity, therefore τ_{yx} is going to be a tensor.

The viscosity is a strong function of temperature and it is also, especially in the case of gases, a function of pressure as well. Ihere are various ways to measure the viscosity; for gases you can also predict what would be the value of viscosity using certain theories but, mostly we deal with variation of viscosity from a large number of experimental data. For liquids, viscosity is decrease with increase in temperature; hot water will slow faster for the same temperature gradient as compared to cold water.

0.2 VELOCITY DISTRIBUTION IN LAMINAR FLOW

In order to do that, I will introduce a concept which is known as the shell momentum balance. If you think of a control volume consisting of some fluid, we have that at steady state:

$$\text{RATE OF MOMENTUM IN} - \text{RATE OF MOMENTUM OUT} + \sum \text{FORCE ACTING ON C.V.} = 0 \tag{5}$$

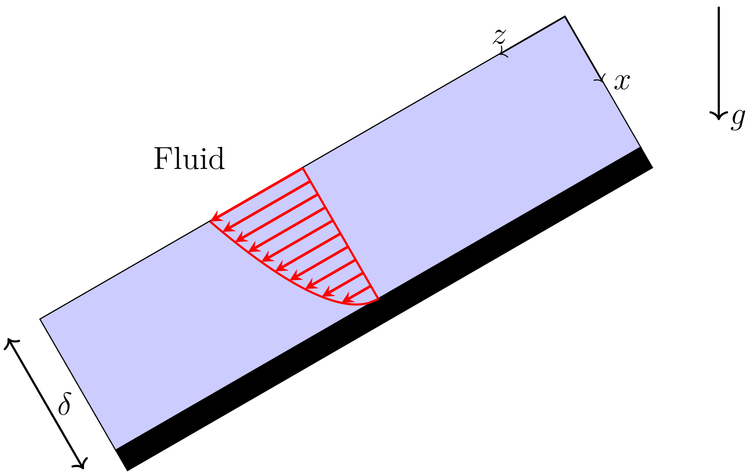
there is no unbalanced force acting on the control volume.
A control surface is like the paper, it has no mass of its own. Control surface is only used to define what is a control volume which has a fixed mass; anything that comes in the C.S. must go out and so the conservation equation for a control surface would be rate of something in must be equal to rate of something out, nothing gets stored in a control surface.
whereas, the control volume is something that has a finite volume of itself. Some amount of heat for example may come in, some amount of heat may leave this, some amount of heat may be generated in the C.V. it self, because non zero mass can absorb some amount of heat and all these will result in a change in internal energy of the control volume.
When we talk about a control volume consisting of a fluid, the algebraic sum of all forces acting on it at steady state must be equal to 0.
There could be a body force. A body force is something which depends on the mass (for example gravity). Gravity force acts on all points inside the control volume.
If you think of pressure, pressure acts only on surfaces, on the left side surface and on the right side surface and as a result of unbalanced pressure forces on two sides of the control volume, the control volume will either move in left direction or move in right direction depending on which side is at lower pressure. Pressure is a surface force on the other hand gravity is a body force.
Pressure and body forces are static in nature, they are always there. Apart from that, some amount of momentum may come in, which is like the shear stress exerted on the faces top, face bottom, two side faces of the control volume.
When I talk about rate, it is the time rate. Time rate of change of momentum which comes in, due to shear stress through any of these surfaces and the rate at which it leaves the surfaces, they also constitute some force which are acting on the control volume.
Forces can be exerted by a body force, by a surface force or by liquids which are coming in carrying some amount of momentum inside the control volume.
When you think of an object, you first have to identify which of these surfaces are taking part into this momentum transfer process. What is the body force which is acting on? What are the surface forces which are acting on relevant surfaces? If, for a control volume

I can identify all the component of forces, or time rate of change of momentum into the control volume, then if it is a steady state, the control volume has no acceleration, so at steady state the sum of all these must be equal to zero.
In a shell of a fluid, some fluid is coming in, some fluid is going out and there can be also variation in velocity which exists in our fluid. When something is flowing into a control volume, it comes in with a velocity, some mass of fluid coming in with a fixed velocity carries with it some amount of momentum. The time rate of this momentum which comes into the control volume and the time rate at which it leaves the control volume, there would be a net addition of momentum to the control volume per unit time. Also due to the variation in velocity, then can also be momentum fluxes as for the Newton's law of viscosity. There can be molecular transport of momentum which is given by the Newton's law and there can be the overall convective transport of momentum which is due to the flow.
if I can write in mathematical terms, it's going to give rise to an equation which is a difference equation, which is written in terms of Δx, Δy, Δz, where they are the components that specify the space and its also going to have a Δp (the pressure difference) and it will be the effect of g which is gravity.
if you can write this difference equation and then if you can convert the difference equation to a differential equation, what I would have is the governing equation for the flow over a flat plate for example, and if I can solve these fundamental equation with appropriate boundary conditions then I would get the velocity profile which is my ultimate aim.
In order to do that I first need to identify what are the relevant boundary conditions that you can think of when a solid and a liquid are in contact.
Lets assume that I have a solid plate, I have a film of liquid of thickness δ; the solid plate is slightly inclined so the liquid starts to flow along the solid plate.
The first BC is at $x = \delta$, since the solid plate is static, the velocity in the z direction $\bar{v}_z$ will also be equal to zero. No relative velocity between the solid and liquid. This boundary condition is commonly known as non-slip condition.
The second BC is at $x = 0$ which is the liquid-air interface. Here $\tau_{xz} = 0$, that is the non-shear condition.
There are some special cases where this no slip condition would not be valid. There is a special branch of fluid mechanics which deals which fluid mechanics and very small scale, where the flow through a nanotube or a micro tube is being considered, or where the flow of a gas is being considered at very low pressure. In some special cases like that, the no slip condition would not be valid and you are probably going to get a case in which there would be slip flow, there would be a non zero velocity component that may exist at the

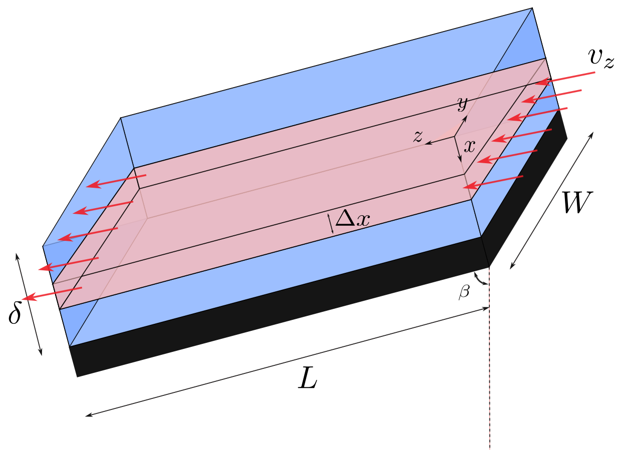
solid-liquid interface.
The other condition is normally. There exists a significant difference between the viscosities of the liquid and the gas. Because of this difference in viscosity, the momentum transfer τ_{xz} (z component being transported in the x direction) would be insignificant at the liquid-vapour interface. The moving liquid film will not be able to import momentum to the gas and vice versa, therefore even if the gas is flowing the liquid can still remain stationary and no momentum from the gas gets transported to the liquid and so on. The exceptions is when the relative velocity is high. When the relative velocity between the liquid and the vapor (the gas) is large, then this relation would not be valid. you would see the formation of the waves, the sharing of the entertainment of the liquid when there is a fast moving vapor on top of the static liquid film.

0.3 Flow of a falling fim

There is an inclined plate and the liquid is flowing because of gravity and there is no imposed pressure gradient at the left and right, as can seen in picture



We consirder a 1-D flow case, that means $\bar{v}_z \neq 0, \bar{v}_y = \bar{v}_x = 0$. We can see that $\bar{v}_z = f(x)$ but $\bar{v}_z \neq f(y) \neq f(z)$.
The smaller dimension of the imaginary control volume, will always be the dimension across which the variation in velocity is taking place. In this specific case, it does not matter what is the length of the control volume that you take or what is the width of the control volume that you take, but what you are going to take is the thickness of the film. The top layer is moving with the highest velocity and the bottom layer, which is in contact with the solid, is static, it does not move, so there is a variation in the x direction in velocity.
Let us take the following C.V. of thickness Δx, length L and width W



we are going to make the momentum balance of the C.V. We have that the velocity is different in $\bar{v}_z(x)$ and $\bar{v}_z(x + \Delta x)$.
The mass can cross just on the right and left face and not on bottom-up or side one (we are talking about convective flow).
The rate of momentum coming in through the right face is:

$$\{(W\ \Delta x\ v_z)\, \rho v_z\}_{z=0}$$

The rate of momentum coming out through the left face is:

$$\{(W\ \Delta x\ v_z)\, \rho v_z\}_{z=L}$$

If I assume that it is an incompressible fluid, then the density is constant along z and so the amount of mass that comes in must be equal to the amount of mass that goes out, otherwise if it is not so there will be accumulation of mass inside the control volume which should violate our steady state assumption.
Since that $\bar{v}_z\,(z = 0) = \bar{v}_z(z = L)$, the amount of convective mass in is equal to the amount of convective mass out. The only terms that remains are the conductive terms in and out because the velocity vary along x (that is the molecular transport of momentum or the shear stress).
The rate of momentum in by conduction is:

$$\text{Area}\ \tau_{xz} = L\ W \cdot \tau_{xz}$$

We have also a body force which is the gravity force:

$$F = (L\ W\ \Delta x)\, \rho g cos\,(\beta)$$

The momentum balance equation for the C.V. is:

$$(L\ W\ \tau_{xz})_x - (L\ W\ \tau_{xz})_{x+\Delta x} + \left(W\ \Delta x\ \rho\ v_z^2\right)_{z=0} - \left(W\ \Delta x\ \rho\ v_z^2\right)_{z=L} + (L\ W\ \Delta x)\,\rho g cos\left(\beta\right) = 0$$

If we divide all by $L\ W$ and take the limit when $\Delta \to 0$:

$$\lim_{\Delta x \to 0} \frac{(\tau_{xz})_{x+\Delta x} - (\tau_{xz})_x}{\Delta x} = \rho g cos\left(\beta\right)$$

We get the definition of the first derivative:

$$\frac{d}{dx}\left(\tau_{xz}\right) = \rho g cos\left(\beta\right)$$

If we integrate once:

$$\tau_{xz} = \rho g cos\left(\beta\right) x + c_1$$

The B.C. is that at the liquid-gas interface, at $x = 0, \tau_{xz} = 0$. In this way we obtain $c_1 = 0$:

$$\tau_{xz} = \rho g cos\left(\beta\right) x$$

For Newtonians liquid, we have that:

$$\tau_{xz} = -\mu \frac{dv_z}{dx}$$

To get:

$$\frac{dv_z}{dx} = -\left(\frac{\rho g cos\left(\beta\right)}{\mu}\right) x$$

Integrating once again:

$$v_z = -\frac{\rho g cos\left(\beta\right)}{\mu}\frac{x^2}{2} + c_2$$

Using the second boundary condition, no-slip, at $x = \delta,\ v_z = 0$, we get the expression of the velocity:

$$v_z = \frac{\rho g \delta^2 cos\left(\beta\right)}{2\mu}\left[1 - \left(\frac{x}{\delta}\right)^2\right]$$

The velocity is parabolic in nature. We are interested to know what is the average velocity. All the average velocity that we are going to refer from know on, will be area average velocities, whereas the area across which I am averaging is perpendicular to the principal direction of flow. Here the principal direction of flow is in the z direction so, it is an area which is perpendicular to the z direction so it is the z face across which I am doing the integration, with my understanding that the velocity varies with x, it does not vary with y. Even though I'm using a double integration in order to average it across the

z face, across y it does not vary, across x it does vary.
The expression for the average velocity mathematically would be:

$$< v_z >= \frac{(\int_0^W \int_0^\delta v_z \ dx \ dy)}{(\int_0^W \int_0^\delta dx \ dy)} = \frac{1}{\delta} \int_0^\delta v_z \ dx = \frac{\rho g \delta^2 cos(\beta)}{3\mu}$$

The volumetric flow rate would simply be:

$$Q = W \ \delta \ \ < v_z > = \frac{\rho g W \delta^3 cos \left(\beta\right)}{3\mu}$$

What is the force exerted by the moving fluid on the solid? The force exerted by the fluid on the plate in a direction perpendicular to the motion. Since the fluid is moving, it would try to take the plate along with it. In order to keep the plate stationary, you must apply a force in the reverse direction because the fluid flow would try to take the plate along with it.
The genesis of the force it must be due to the viscosity. The shear stress out of this viscosity is essentially shear stress exerted on the top area of the solid plate, is the one which is the force exerted by the liquid on the plate.
The total force exerted by the fluid on the solid plate is:

$$F_z = \int_0^L \int_0^W \left(\tau_{xz}\right)_{x=\delta} dydz = \int_0^L \int_0^W \left(-\mu \frac{dv_z}{dx}\right)_{z=\delta} dydz = \rho g \delta cos \left(\beta\right) L \ W$$

is interesting to see here, that this expression is nothing but the z component of the weight of the entire fluid for inter fluid present in the film on the plate.
Whenever we talk about, also be aware of the limits imposed by our simplified treatment. The first thing is that is not valid for very fast flow, its only valid for laminar flow. If you have turbulent flow, the liquid is moving at a very high velocity, then you are going to get waves at the top and which should probably not be able to use the concept that its zero share at the liquid vapour interface and so on. I am assumed that is one dimensional flow with straight stream lines. if it is 2 dimensional flow, then this simple analysis, you would not be able to use. Finally whether or not it's a laminar flow, turbulent flow, you would be able to get an idea by calculating what is the Reynolds number for the fluid and for certain range of Rainolds number, lower values of Reynolds number, this analysis is perfectly valid and this is an ideal example to show how it can be done. The other complexities that we did not considered is that the solid plate on which the inclined plate on which the flow takes place, is at elevated temperature. If it is at an elevated temperature, then the viscosity that we have used to express the shear stress using Newton's law, μ would also be a function of x. μ near the plates, since the temperature is more, would be lowered as

compared to the μ near the top, so this variation in the physical property, the transport properties of the system, can cause additional problems and you won't be able to obtain such a closed form simplified solution. If you have the plate at a higher temperature, then will set an heat transfer across the film as well. Not only the Thermo-physical properties would vary, the temperature would vary as a result of which the flow field would be different than what we have done here.

0.4 Example of Shell Momentum Balance

In Our introductory class, we have seen how important viscosity is in transfer of momentum. viscosity plays a critical role, whenever there is a difference in velocity between 2 adjacent layers. when 2 adjacent layers of fluids passed by one another, they would be some amount of momentum transfer due to molecular motion. Molecules with higher velocity would jump from one layer to the slower moving layer, thereby transferring the momentum from the faster moving layer to the slower moving layer and the same thing happens when molecules from a slower moving layer would come to the faster moving layer.

this has given rise to a transfer of momentum in a direction perpendicular to the flow, which we commonly call as the shared stress. we have seen how the shear stress is generally represented. its represented with a double subscript, the first subscript refers to the direction of motion, the principal direction of motion, and the second subscript refers to the direction perpendicular to the principal direction of motion in which the momentum gets transported. if you think of a layer which is moving in the x direction and another layer on top of it which is also being dragged the x direction but with a lower velocity, then the x momentum of flow gets transported because of viscosity in the y direction. layers above the faster moving layer, then the next layer, and the layer above that, all will start to move in the x direction as a result of the invisible strain, the viscosity which binds these 2 layers.

the momentum even though it is in the x direction, it gets transported in the y direction. this is also called the molecular transport of momentum and the double subscript is a very common way to represent the shared stress, that means the stress being filled, being exerted, by the moving fluid on the layer just above it. this area which is in contact with the layer below it, it gets the stress (some force per unit area) which is the direct result of viscosity. The defining equation of viscosity, as we have seen previously, is the Newton's law of viscosity, where the shear stress is directly proportional to the cause, which is the velocity gradient. the proportionality constant of this is known as the viscosity. the fluids which follow Newton's law, are commonly known as Newtonian fluids.

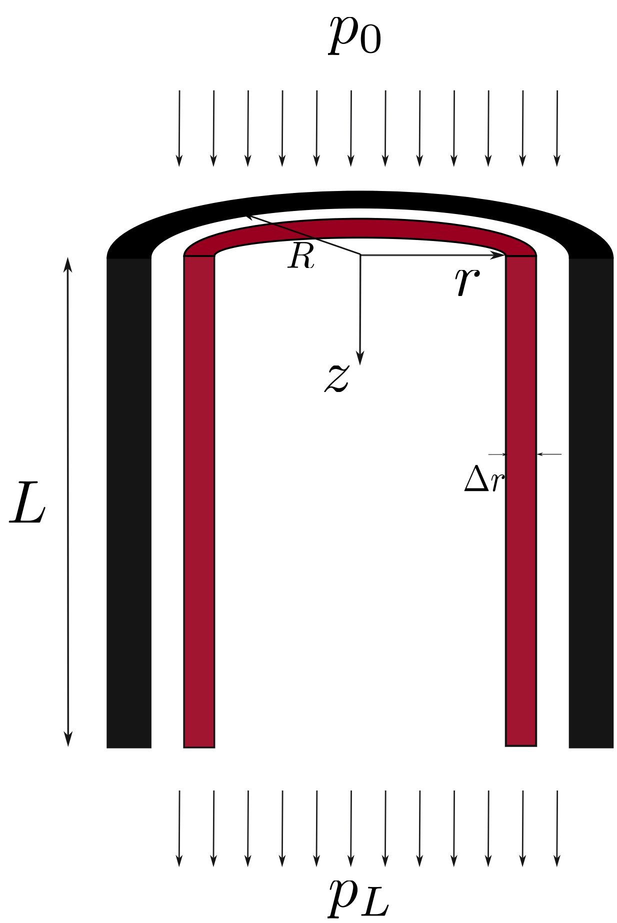
examples of different behaviours have quite common, there could be some fluids which would start without first resist motion but once a threshold stress is applied on it, it will automatically start to move and from that point outward, the stress is going to be proportional to the velocity gradient. so those kind of fluids which have a threshold stress, which must be applied for it to start its motion, are called Bingham plastics. in the common example of Bingham plastic is toothpaste.

in the subsequent classes what we have seen is that it is useful to define a shell in a moving fluid and find out what are the forces, what are the momentum, that are acting on that shell. these shells are generally defined in the smaller dimension of the cell and a shell is the is the direction in which the velocity is changing. in the previous class we solved the example of flow along a flat plate so they don't be a flow of a liquid along a flat plate and so obviously the velocity is going to change so at this point the boundary condition is going to be no slip condition which we know that at the liquid solid interface there cannot be any motion of the liquid molecule so the liquid molecules adjacent to the solid boundary will have zero relative velocity. at the other end where we have the liquid vapour interface, the shear stress across the interface would be equal to zero. This is true for most of the cases when the velocity differences are not too great or the air is not moving with a very high velocity, creating waves and so on. since the velocity varies in the x direction, the shell that we are going to define is going to be have a smaller dimension of Δx, it can be any length and any width and since we are assuming its one dimensional, laminar and incompressible flow (where the density remains constant), than the velocity is not going to be a function of the axial distance, it is not going to be a function of the width but it is only going to be a function of the depth of the film. for those 1D cases, its we should always define the shell as having any length L, any width W but the thickness dimension is going to be Δx since between x and Δx the velocity change substantial. we express the physics in the form of a difference equation. we understand that there is going to be some amount of fluid which comes into the shell because of its flow, but since I do not have any velocity component in other directions, nothing comes into flow through these 4 surfaces. Flow in, will carry some momentum along with it and flow out, will take some momentum out of the control volume. since the velocity is changing in the x direction, some shear stress will be felt by the bottom surface and some shear stress will be felt by the top surface. The forces due to the shear which would act on two sides of this shell would be τ_{xz} multiplied by the bottom area and τ_{xz} multiplied by the top area.

there may be other forces acting on, for example this is freely falling so there is no pressure difference between these two points, and so there is no surface force present on in it. However, since it is inclined, the component of gravity would act on the volume of the liquid contained in this control volume, so the only force which is acting on it would be

the gravity force (the body force).
at steady state the sum of all these would be zero. the fundamental equation that we are going to write for any initial is rate of momentum in minus rate of momentum out plus sum of all forces acting on it must be equal to zero at steady state. then we identify that the momentum can come in as a result of convection and as a result of conduction; convection is with the flow conduction is perpendicular to the flow due to the presence of the shear stress.
once we have the difference equation, we are successful in explaining or in expressing the physics of the problem in the form of a difference equation which contains the smaller dimension Δx.
the next step would be to divide both sides by Δx and taking the limit when Δx tends to zero, so that would essentially be using the definition of the first derivative and out of the difference equation you get the differential equation.
once we have the governing differential equation we need boundary conditions, since we are going to integrate that in order to obtain the expression for velocity for such a case. the two common boundary conditions then one would expect in momentum transport is no slip at the solid liquid interface and no share at the liquid vapour interface. so we have used those two conditions to obtain a compact expression for the velocity as a function of depth.
once we have the expression for velocity we would be able to obtain what is the maximum velocity and then we should also be able to express not the point value of velocity but we are more interested in to the average velocity. all such kind of averages are always done across the flow cross section, across the face which is perpendicular to the flow direction. so if I have flow in the z direction, then I need to find out what is the velocity at every point in an area that is perpendicular to the direction of flow. with that we have obtained what is the expression for average velocity and once we have the average velocity then we can calculate what is the mass flow of the following film.
Now we would like to see in this exercise, how we can use the shell momentum balance to obtain an expression for velocity in a tube through which the liquid is flowing. now we are going to assume in this case is that the pipe is vertical, so there would be effect of gravity that tries to pull the liquid downward direction and there is also a pressure gradient; there will be some pressure over and a slightly lower pressure at the bottom, so the pressure gradient is forcing the liquid to move downwards. the effect of pressure gradient and gravity is to create a flow in the downwards direction. As fluid starts to flow, it's going to interact with the walls of the tube, and the way it is going to interact is through viscous forces. when it reaches steady state, the sum of all forces acting on a control volume, suitably defined for flow in a pipe, must be equal to zero. so if I can

define a control volume for flow in a pipe, then we are going to find out what is the rate of momentum into the control volume by convection and by conduction, what is the surface force that is acting on the control volume namely pressure in this case, what is the force due to gravity that is acting on it. with that difference equation we should be able to obtain a differential equation and we should be able to use appropriate boundary condition for this case to obtain what is the velocity distribution of a flowing fluid in a tube subjected to pressure gradient and subjected to gravity.
For example if you are designing an experiment to measure the viscosity of a liquid, all of you probably have used capillary viscometer. what is the principle on which capillary viscometer works? so how can you relate viscosity with the flow rate?.



As can you see in the picture, the pipe has a radius equal to R, there is flow from the top, the pressure at the top is p_0 and pressure at the bottom is p_L, the length of the of the tube is L.
As the flow takes place in the tube, the velocity at steady state and for incompressible

cases, the velocity is a function of r only. The closer to the wall the liquid layer is, the lower is going to be its velocity and as it moves progressively towards the centre the velocity will increase.
the first job for this case where I have flow from the top as a result of pressure difference and as a result of gravity, is to define a shell and across the shell we are going to make the momentum balance. What would this shell look like? it could be of any length L, it does not matter since the velocity is not a function of L, the velocity $v_z \neq f(z)$, it is definitely a function of $f(r)$. since its a function of r, then my shell will have the smaller dimension as Δr.
Now you will try to see what would be momentum in term, momentum out term and all the forces due to the surface force and due to the body force acting on the liquid containing in this shell.
through the top surface of the imaginary shell, momentum gets in by convection, so the liquid is crossing the points of the top surface carrying with some momentum, but since the velocity is changing in the r direction, there would be a gradient in velocity between the inner and outer sides wall shell, and so there would be viscous transport of momentum.
if you think of the momentum which is in by convection, the area on which it is acting on must be equal to $2\pi r\Delta r$, whereas when we talk about viscous transport acting at the inner area, the corresponding area for that would simply be equal to $2\pi rL$ and the outer one it is going to be to $2\pi(r+\Delta r)L$. the volume of the fluid which is contained in this shall would simply be equal to $2\pi r\Delta rL$.
The convective momentum in to the system will be:

$$\text{Convective Momentum in} = (2\pi r\Delta r v_z\rho v_z)|_{z=0}$$

The convective momentum out of the system will be:

$$\text{Convective Momentum out} = (2\pi r\Delta r v_z\rho v_z)|_{z=L}$$

since that $v_z \neq f(z)$ then $v_z(z=0) = v_z(z=L)$ and so the two terms will cancel out each other when you do the balance.
The terms that remains are the following:

$$(2\pi rL\tau_{rz})|_r - (2\pi(r+\Delta r)L\tau_{rz})|_{r+\Delta r} + (2\pi r\Delta r p_0)|_{z=0} - (2\pi r\Delta r p_L)|_{z=L} + 2\pi r\Delta r L\rho g = 0$$

Now we divide both sides by the smaller dimension Δr and take the limit as $\Delta r \to 0$:

$$\lim_{\Delta r \to 0} \frac{(r\tau_{rz})_{r+\Delta r} - (r\tau_{rz})_r}{\Delta r} = \left(\frac{p_0 - p_L}{L} + \rho g\right) r$$

we can define $P = p - \rho g z$ where at $z = 0, P_0 = p_0$ and at $z = L, P_L = p_L - \rho g L$, and so becomes:

$$\frac{d}{dr}(r\tau_{rz}) = \left(\frac{P_0 - P_L}{L}\right) r$$

If we integrate once we obtain:

$$r\tau_{rz} = \left(\frac{P_0 - P_L}{2L}\right) r^2 + c_1$$

$$\tau_{rz} = \left(\frac{P_0 - P_L}{2L}\right) r + \frac{c_1}{r}$$

we understand that τ_{rz} must be finite at $r = 0$ and this can only happen if $c_1 = 0$. in certain cases, the physics of the problem is not just applying no shear and no slip at the two interfaces, but in some cases you can make a definitely statement about the nature of the velocity, nature of the flow, nature of shear stress, which give you an additional physical boundary condition which in this case we have used to obtain the first integration constants c_1.

Using the Newton's law we have:

$$\frac{dv_z}{dr} = -\left(\frac{P_0 - P_L}{2\mu L}\right) r$$

integrating again we have:

$$v_z = -\left(\frac{P_0 - P_L}{4\mu L}\right) r^2 + c_2$$

Applying the no-slip BC at $r = R, v_z = 0$, we get c_2 and the finally the expression of velocity is the following:

$$v_z = \left(\frac{P_0 - P_L}{4\mu L}\right) R^2 \left[1 - \left(\frac{r}{R}\right)^2\right]$$

where the point of maximum velocity is at $r = 0$ and the value of the velocity is:

$$v_{z_{max}} = \left(\frac{P_0 - P_L}{4\mu L}\right) R^2$$

In order to obtain an expression for the average velocity, I need to integrate the velocity across some area. the flow is in the z direction and the velocity varies with r, so the flow area which I need to incorporate in order to obtain the average velocity must be subarea which is perpendicular to this flow. If I take the cross section of the circle, then the circular area must be the flow area across which I need to integrate in order to obtain an average velocity. all velocities are area average velocities and all these areas are always perpendicular to the direction of flow

$$< v_z >= \frac{\int_0^{2\pi} \int_0^R v_z r dr d\theta}{\int_0^{2\pi} \int_0^R r dr d\theta} = \left(\frac{P_0 - P_L}{8\mu L} \right) R^2$$

The flow rate is:

$$Q = \pi R^2 < v_z >= \pi \left(\frac{P_0 - P_L}{8\mu L} \right) R^4$$

This is a famous equation that is called Hagen-Poiseuille equation.
This equation gives the volumetric flow rate for a fluid flowing through a vertical tube due to the presence of a pressure gradient and body force.
The first assumption that we have made is the laminar flow, you cannot have a very high pressure gradient because the flow inside becomes disturbed and so turbulent. The other assumption if that the fluid is incompressible nd so the density of the fluid is constant.
in order for the liquid column inside the pipe to move with a constant velocity, we have assumed that sum of all forces acting on it must be equal to zero. it is not the case, then the column of liquid which is flowing inside the tube will either accelerate or will slow down. The liquid column is going down because there is a difference in pressure between up and down, and there is also the gravity which is trying to move the liquid column in downwards. Both forces are acting in the same direction and so there must be an opposing force which is going to be equal to the combined effect of these 2 forces equal and opposite, only then it is going to move with a constant velocity. what is the opposite force? what is that force that opposes the motion of a liquid in such a way the velocity remain constat? the viscous force is the one which opposes any flow of the liquid.
if you would like to analytically find out what is the force that is acting on it, you need to find out what is the viscous force at $r = R$, that means at the solid liquid interface along the inner wall of the pipe. that force is τ_{rz} evaluated at $r = R$ multiplied with the area on which it is acting that is $2\pi RL$.

0.5 industrial problem: upward moving belt

let consider a belt which initially is stationary immersed into a liquid. At some point, the belt starts to move upwards with a velocity U_0. after you provide sufficient time, when it reaches steady state, the belt is going to carrying with it a thin film of liquid (stack to it) of constant value equal to h. We can see that $v_y \neq 0$ and $v_x = 0$. There is no pressure

gradient acting ,but the force which is acting is gravity.
the gravity would like to drain the liquid in the reverse direction; so viscous forces pull the liquid up, gravitational forces tend to drain the liquid.
we would like to find out what's going to be the velocity distribution v_y in the thin portion of the liquid.
in order to think of a shell, we need to first identify what is the direction in which the velocity is varying, because whatever be that direction that is going to be the smaller dimension of the control volume.
the velocity is varying with x and so the thickness of the shell will be Δx. It can be of any width W and any length L.
so some amount of convective momentum will come in and some of amount of the convective momentum will go out. these 2 must be equal to each other because of my assumption that $v_y \neq f(y)$.
At stady state we have:

$$(\tau_{xy})_x LW - (\tau_{xy})_{x+\Delta x} LW - LW\Delta x \rho g = 0$$

At the gravity force we have put a negative sign because is acting in the opposite direction of the y axis of th system.
We divide both sides by Δx and taking the limit of $\Delta x \to 0$, we get the governing equation:

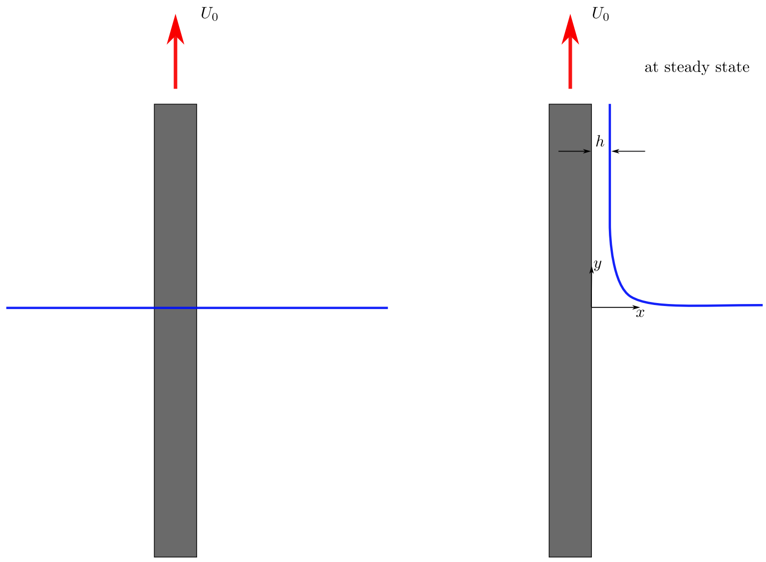
$$\frac{d}{dx}\tau_{xy} = -\rho g$$

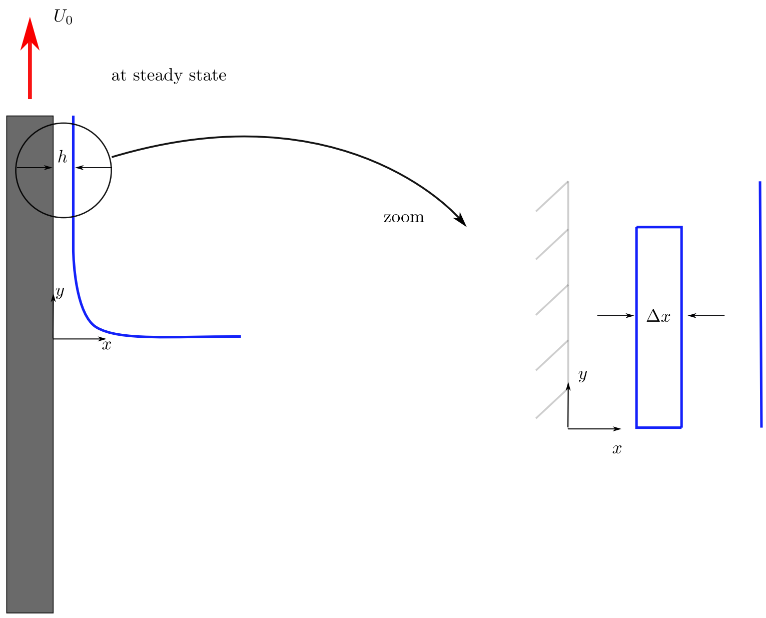
We can integrate it ones:

$$\tau_{xy} = -\rho g x + c_1$$

and the BC's are at $x = h, \tau_{xy} = 0$ where we obtain $c_1 = 0$. Using the Newton's law and plugging into it, integrating and inserting the other condition at $x = 0, v_y = U_0$ (no relative velocity), we have:

$$v_y = U_0 - \frac{\rho g h^2}{2\mu}\left[2\left(\frac{x}{h}\right) - \left(\frac{x}{h}\right)^2\right]$$





0.6 Example of momentum balance

Let consider 2 parallel plates, in which one of them is moving the other is stationary. the top plate starts to move and it would try to drag the liquid which is in the remaining space flow starts to establish and we will look at that flow when a steady state has reached. we are going to assume that there exists a pressure gradient as well, so the motion in the intervening space between the 2 plates one moving the other stationary is caused by 2 factors: one is the movement of the top plate and the second is the pressure gradient that exists in between 2 points. if the pressure decreases in the direction of the top plate motion, then the fluid is going to have an additional flow due to pressure gradient from left to right. so the top plate is moving from left to right, the pressure gradient is in such a way that could try to push the liquid from left to right. when that happens, we will

say that its a favourable pressure gradient. so in order for a favourable pressure gradient, the pressure has to be decreased as it moves in the axial direction. If instead the pressure increases, we will call it as an unfavourable pressure gradient.
the problem that we are going to deal with, is flow between 2 parallel plates, one plate is in motion with velocity U and there is a favorable pressure gradient. we would like to solve using shell momentum balance.
There is a pressure gradient that we call it as $\frac{dp}{dx} = -A$ and is constant. As we move in the x direction, the pressure will decrease and this also will force the fluid to moves to the right direction.
We need to find the shell and make the balances. The flow is in the x direction, but the velocity changes in the y direction, so the shell is going to have a thickness Δy, any area WL.
The convective momentum in and out are equal because $v_x = f(y) \neq f(x)$ and also is incompressible, and so the difference will be zero. The time rate of momentum in due to conduction minus the time rate of momentum out due to condition remains. Since the system is horizontal, the gravity do not acts in the direction of flow and so we do not have any body force acting on it. We have also a pressure gradient, where p_0 is at the left and p_L is at the right. At steady state we can write:

$$\tau_{yx_y} WL - \tau_{yx_{y+\Delta y}} WL + p_0(W\Delta y) - p_L(W\Delta y) = 0$$

We divide both sides by Δy and we take the limit as $\Delta y \to 0$:

$$-\frac{d}{dy}(\tau_{yx}) = \frac{\Delta p}{L}$$

and for Newtonian's fluid we obtain the following governing equation:

$$\frac{d^2 v_x}{dy^2} = \frac{1}{\mu}\frac{dp}{dx}$$

integrating two times we have:

$$v_x = \frac{1}{2\mu}\left(\frac{dp}{dx}\right) y^2 + c_1 y + c_2$$

at $y = 0, v_x = 0$ and we have that $c_2 = 0$; at $y = b, v_x = U$ and we have that $c_1 = \frac{1}{b}\left[U - \frac{1}{2\mu}\left(\frac{dp}{dx}\right) b^2\right]$. With this we obtain the velocity profile:

$$v_x = \frac{U}{b} y - \frac{1}{2\mu}\left(\frac{dp}{dx}\right) b^2 \left[\frac{y}{b} - \left(\frac{y}{b}\right)^2\right]$$

If there is no imposed pressure gradient, than $\frac{dp}{dx} = 0$ and so we get the Couette flow, in which we have a linear distribution of velocity:

$$v_x = \frac{U}{b} y$$

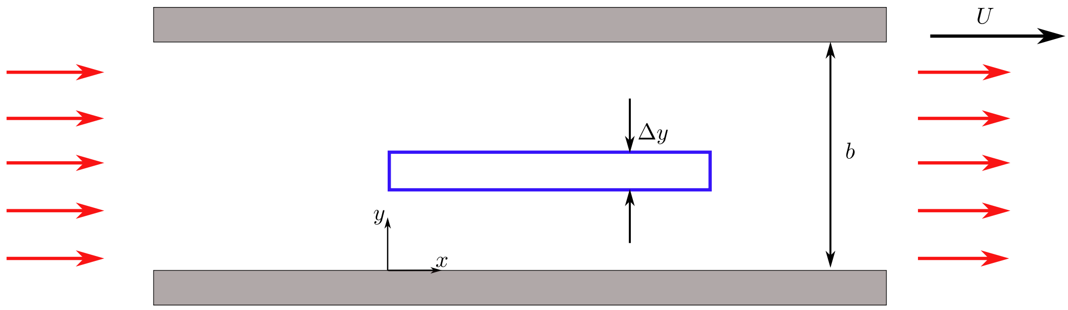
The first therm of the velocity is the Couette flow and the second term is pressure driven flow.
In the Couette flow the maximum of velovity is at $y = b$, but if you have an unfavorable pressure gradient or a favorable pressure gradient, the location of the maximum of velocity will be different.
In order to the get the maximum velocity we impose that:

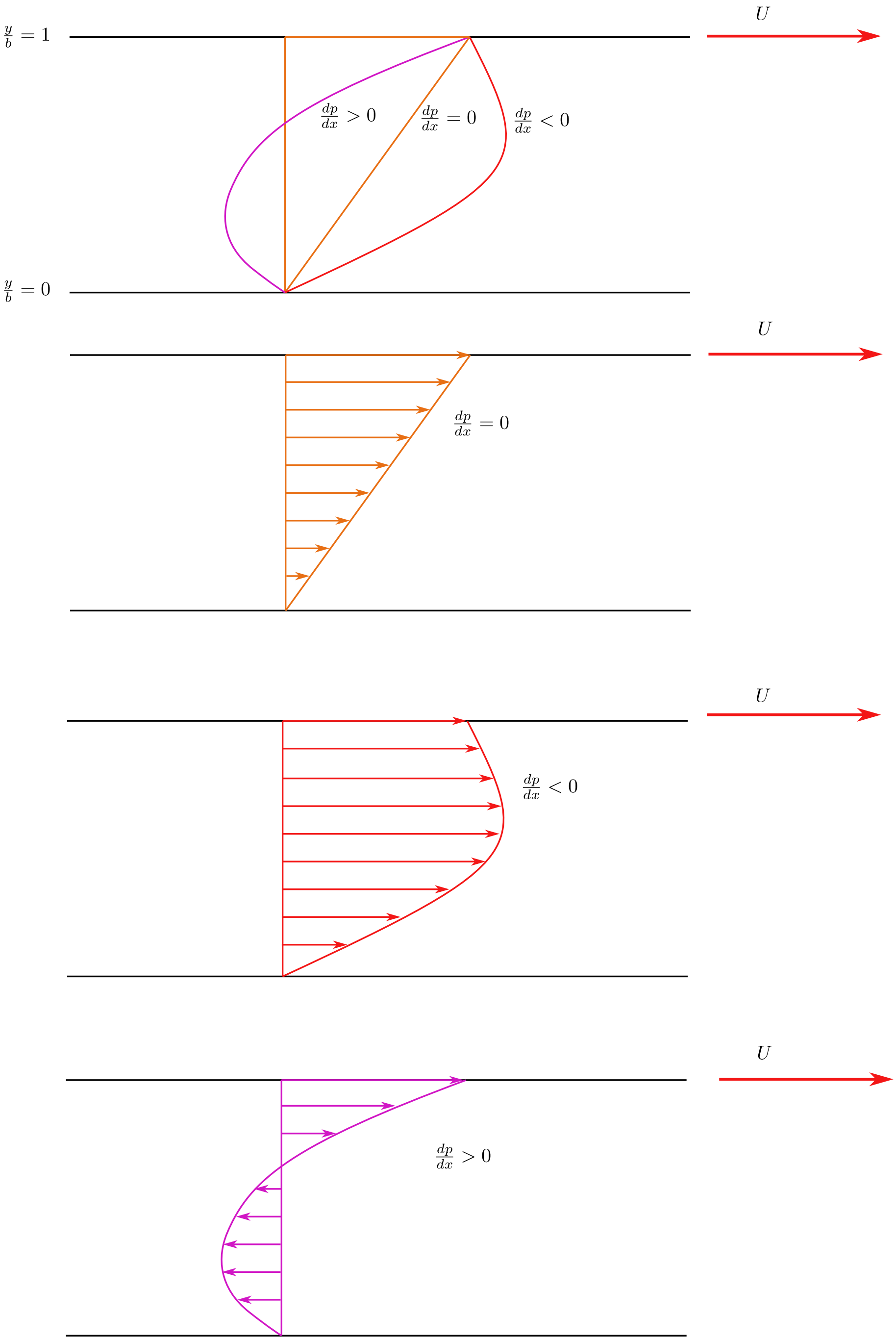
$$\frac{dv_x}{dy} = 0 \Rightarrow \frac{U}{b} = \frac{b^2}{2\mu}\left(\frac{dp}{dx}\right)\left[\frac{1}{b} - \frac{2y_{max}}{b^2}\right] \Rightarrow y_{max} = \frac{b}{2}\left[1 + \frac{2\mu U}{Ab^2}\right]$$

where $A = -\frac{dp}{dx}$.
What is pressure gradient that should be imposed in such a way there is n o net flow ($Q = 0$). We have to consider the unfavorable pressure gradient in which $\frac{dp}{dx} > 0$, where toward the bottom the liquid is flowing in the left and toward the top, due to couette flow, the fluid is moving to the right direction. We need to find the relation between the velocity of the top plate U and the imposed pressure gradient $\frac{dp}{dx}$ such that $Q = 0$. The algebraic some of th flow rate of the top part must be equal and opposite to the bottom part, so the area of the bottom and top curve must be equal. If $Q = 0 \Rightarrow < v_x >= 0$. The expression of the avarage velocity is:

$$< v_x >= \int_0^b v_x dx W = 0$$

If now the top plate is moving, the bottom plate is stationary and you have just a favorable pressure gradient, we can see notice that if we have a large $\frac{dp}{dx}$, than $v_{max} \neq U$. If $\frac{dp}{dx} = 0$ than $v_{max} = U$. If $\frac{dp}{dx}$ is small, than we could have that $v_{max} = U$ The velocity profile is not a straight line but the maximum value of velocity remains there. The question is to find the limiting value of $\frac{dp}{dx} = 0$ such that $v_{max} = U$. The condition to impose is that $\frac{dv_x}{dy}|_{y=b} = 0$. The other question could be, what is force needed to pull the plate in the case of $\frac{dp}{dx} = 0$ limiting?. You do not need any force to sustain the motion of the top plate; top plate will perpetually move if you can maintain the limit condition.


U
Δy
b
y
x


$\frac{y}{b} = 1$
U
$\frac{dp}{dx} > 0$
$\frac{dp}{dx} = 0$
$\frac{dp}{dx} < 0$
$\frac{y}{b} = 0$
U
$\frac{dp}{dx} = 0$
U
$\frac{dp}{dx} < 0$
U
$\frac{dp}{dx} > 0$

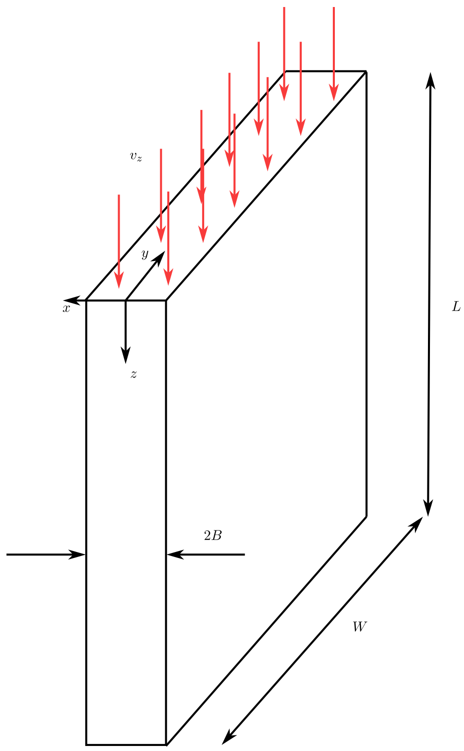
0.7 laminar flow in a narrow slit

a narrow slit is formed by two walls and the distance between them is quite small compared to their width or compared to that length. let's assume that the upper portion of the slit there is some applied pressure which is more than the pressure at the bottom and it's obvious that if I can keep it vertical, then there would be effect of gravity forces as well. as a result of which the fluid starts to flow in between these two slits. the downward velocity obviously is going to be a function of how far it is from either of these two plates. so if I assume one dimensional flow once again, if I assume that it is acted upon by gravity and pressure difference, however it's steady 1D incompressible flow, then the velocity is a function only on how far they are from the side walls of the slit, and it's not going to be a function of where it is in terms of the z location. we are going to start with our analysis of flow in a slit, pressure gradient, and the slit is very narrow such the variation in x direction is very important, variation in y direction can be neglected, and it is a 1-D steady flow so there is no variation in the direction of flow.

we can see that v_z is the only nonzero component of velocity and $v_z = f(x), v_z \neq f(z)$ and since it's a narrow slit, the dependence of v_z on y can be neglected; this means that W is too log compared with $2B$. the flow is principally one-dimensional and it's a steady state case. since your velocity varies with x, the shell is going to be of thickness Δx. there will be the conductive transport of momentum in and out. the amount of convective mass which comes in through the top and leaves at the bottom are equal so I do not write them because they are going to get canceled. there is an effect of pressure and there is an effect of gravity.

$$\tau_{xz_x} WL - \tau_{xz_{x+\Delta x}} WL + p_0(W\Delta x) - p_L(W\Delta x) + WL\Delta x\rho g = 0$$

The BC's are the following: at $x = B, v_z = 0$ that is the no-slip condition and at $x = 0$, the velocity should be maximum, and so $\tau_{xz} = 0$, because if the velocity is maximum we have that $\frac{dv_z}{dx} = 0$.



The expression the velocity will be

$$v_z = \frac{B^2}{2\mu}\left(\frac{P_0 - P_L}{L}\right)\left[1 - \left(\frac{x}{B}\right)^2\right]$$

The expression for the avarage velocity will be:

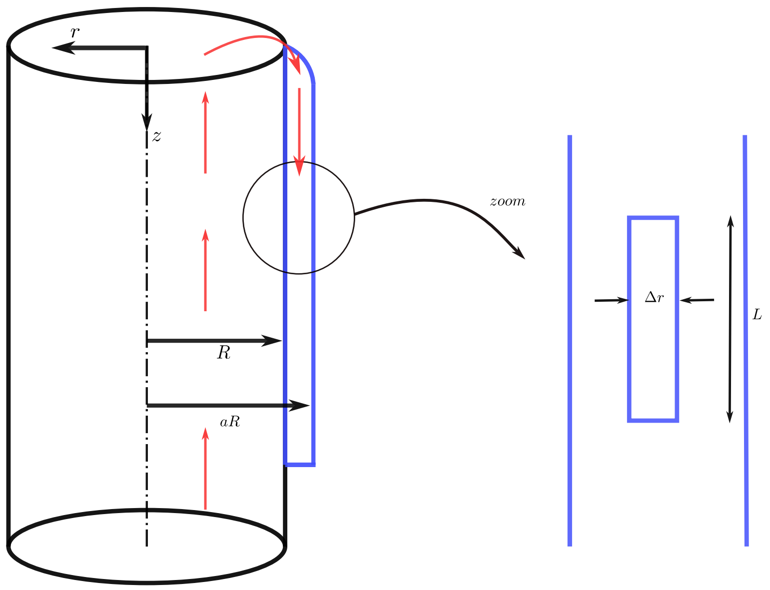
$$< v_z >= \frac{B^2}{3\mu}\left(\frac{P_0 - P_L}{L}\right)$$

The relation between average and maximum velocity will be:

$$< v_z >= \frac{2}{3}v_{max}$$

and the volumetric flow rate will be:

$$Q = \frac{2}{3}\left(\frac{P_0 - P_L}{\mu L}\right)B^3W$$



0.8 falling film of fluid pipe

there is a pipe and through this pipe, a liquid comes in from the bottom. there is a pressure gradient which forces the liquid to come to the top. as it comes to the top it spills over, it starts to fall along the outside of the pipe wall creating a film.
you are not dealing with what is happening inside the pipe, you are dealing with what is happening on the outside of the pipe. in the outside of the pipe the region is at the radius and beyond, not the point from 0 to R.
when you see the fall of the falling film outside of it, even though you have a pressure which is forcing the liquid to move up come to the top and then spill over, when it starts to fall it's a freely falling film, there is no imposed pressure gradient on the system, the liquid is falling on its own, only gravity is present.
the boundary conditions are: at $r = R$, there is no sleep, so the film is in contact with the outside of the wall. if it is falling as a film, then I must also have a liquid-vapor interface. the outside of the falling film and the air beyond that, is a liquid-vapor or liquid-air interface and the boundary condition to be used is that the shear stress is zero.

$$\tau_{rz_r} 2\pi r L - \tau_{rz_{r+\Delta r}} 2\pi (r + \Delta r) L + 2\pi r \Delta r L \rho g = 0$$

$$\frac{d}{dr}(r\tau_{rz}) = \rho g r = 0$$

$$\tau_{rz} = \frac{\rho g r}{2} + \frac{c_1}{r}$$

Applying the Newton's law we get:

$$v_z = -\frac{\rho g r^2}{4\mu} - c_1 ln(r) + c_2$$

The BC's are: at $r = R, v_z = 0$ and at $r = aR$, $\frac{dv_z}{dr} = 0$. Than we get the final expression of velocity:

$$v_z = \frac{\rho g R^2}{4\mu}\left[1 - \left(\frac{r}{R}\right)^2 + 2a^2 ln\left(\frac{r}{R}\right)\right]$$

the shell balance approach has some limitation for: unsteady State cases, for cases where there is change in the flow Direction resulting in situations where you can get multi-dimensional effects, and similarly we are assuming that it's laminar flow, that means all the transport of momentum is due to viscosity only, it's viscous transport of momentum; if it's a turbulent flow, then most of the momentum would be carried not by this viscosity (or the molecular momentum) but it would be due to the formation of eddies. eddies are packets of fluids which are generated in turbulent flow and which carry with them momentum from one point to the other. the transport of momentum by eddies, will supersede that by the simple molecular transport in laminar flow.
we can clearly see a need for a more generalized treatment for situations where we have multi-dimensional effects, the effect of unsteady behaviour and more, not laminar but beyond laminar, there may be turbulent flow as well,how do we take into account the additional momentum transport due to the formation of eddies.

0.9 The equation of change (isothermal system)

when we talk about equations of change, there are certain definitions which needs to be clarified.
the concepts of different derivatives. you are trying to measure something as a function of time and depending on where you are, what are you doing, the values of the quantity that you measure as a function of time, can greatly vary. so I'll try to give you an example so that you can have a clearer understanding of the different derivatives.
you pick the busiest intersection of someplace in your town, where roads have come from all sides at that crossing. now you are standing right at the center of this intersection and you have been told that please count the number of people who are wearing a blue

shirt. you are standing at the middle of a crossing and counting people who are wearing blue shirts. you are not moving, you are static at that point, and you're measuring the number of such people as a function of time. every second you try to see how many blue shirts you can see, while you are stuck at a position. so you are at the center of the reference frame which is static (which does not move) and any quantity that you measure as a function of time is known as the partial time derivative.
let's say c denotes the number of persons who are wearing a blue shirt. variation of that with respect to time when x, y, z are constant, is the partial time derivative $\frac{\partial c}{\partial t}$. This means that you are at the center of intersection of the 3 planes, and measuring what is the value of c.
let's say you while standing there for some time, I mean you are definitely bound to get bored, so you get bored and you have decided that enough of it I'm not going to be at that intersection for a long time, I need to walk around, a bit around in the area where I am trying to count the number of persons with a blue shirt. so you start to move around, you have a velocity of your own, so you can go in any direction that you want. you are still being a very conscious worker, you are still counting the number of persons with the blue shirt, but since you now you have a velocity of your own, the numbers that you are counting would definitely be different had you been stuck to the place where x, y, z are constant, so right at the center of the intersection. so if you measure the number of people while standing at the intersection and if you measure the number of people while you start to move around with a velocity of your own, these two numbers must be different. so when you measure the variation of the number of people wearing blue shirt as a function of time while you have a velocity of your own, by definition it's known as the total time derivative which is denoted by $\frac{dc}{dt}$ and is expressed in this way:

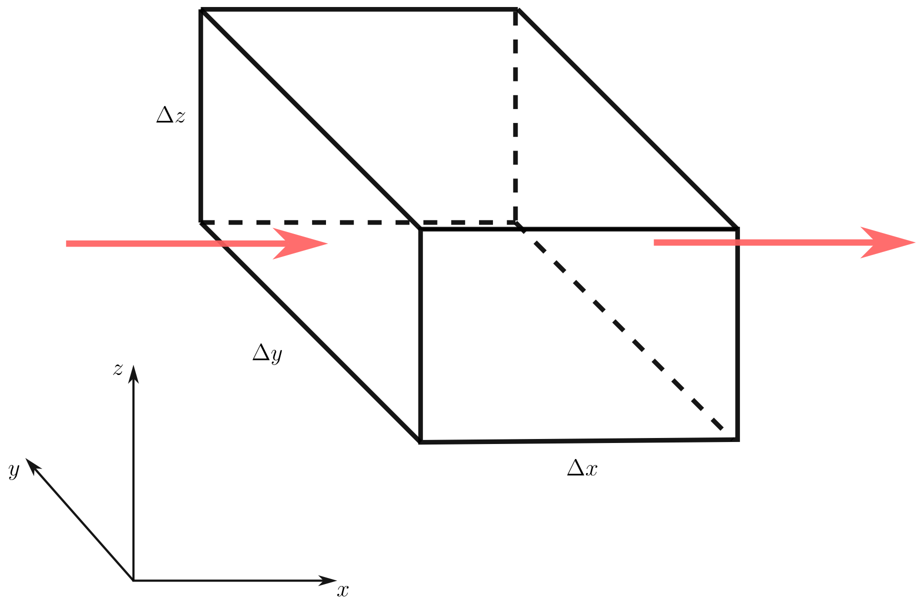
$$\frac{dc}{dt} = \frac{\partial c}{\partial t} + \frac{\partial c}{\partial x}\frac{dx}{dt} + \frac{\partial c}{\partial y}\frac{dy}{dt} + \frac{\partial c}{\partial z}\frac{dz}{dt}$$

where $\frac{dx}{dt}, \frac{dy}{dt}, \frac{dz}{dt}$ are the components of the your velocity.
How long you can stand so you decided to move with a velocity, how long you can move around. at some point of time you get tired and being a busy intersection there that's lot of crowd which are going in all possible directions. so at some point of time you decide that I had enough and I will simply float with the crowd so no matter which way the crowd is the maximum number of people are moving I'll move with them with their average velocity and at some point, at each point of time, I will always move with the velocity of the prevailing crowd at that point of time, but you are still counting the number of persons with the blue shirt. now you do not have a velocity of your own, whatever be the local fluid velocity, that is the velocity of the reference frame and the

numbers that you are counting is some sort of a derivative $\frac{dc}{dt}$ that is the time derivative of the number of persons, is a derivative where the reference frame moves with the fluid with its average velocity. that mean it can also be said as it's a derivative following the motion.
this is generally called the substantial time derivative:

$$\frac{Dc}{Dt} = \frac{\partial c}{\partial t} + v_x \frac{\partial c}{\partial x} + v_y \frac{\partial c}{\partial y} + v_z \frac{\partial c}{\partial z}$$

Where v_x, v_y, v_z are the velocities of the fluid at that point at that instant of time. The differences between this equation and the previous equation is that $\frac{dx}{dt}, \frac{dy}{dt}, \frac{dz}{dt}$ are velocities of you, that you have decided to move with this velocity; whereas v_x, v_y, v_z are that of the fluid surrounding you, so you let go yourself and Float with the fluid and the numbers that you count are now known as the substantial time derivatives or the derivative following the motion.
The partial time derivative the stationary frame is fixed, the total time derivative where the reference frame has a velocity of its own and the substantial time derivative where the reference frame has the velocity as that of the fluid.
with these concepts clearly understood, now we are going to derive what is going to be the equation of continuity. equation of continuity is nothing but a statement of conservation of mass; if I define a control volume fixed in space and allow fluid to come in and go out through all the possible faces, then the rate of mass of fluid coming in minus the rate of mass of fluid that's going out must be equal to the rate of accumulation of mass inside the control volume. in minus out must be equal to accumulation and this is nothing but this statement of conservation of mass and we are going to derive the equation of continuity for a system with a cartesian coordinate system and the dimensions of the volume is simply going to be $\Delta x, \Delta y, \Delta z$. so it's a box placed in a flow and the liquid is coming in through the x, y, z faces and through the face at $x + \Delta x, y + \Delta y, z + \Delta z$ and z+ Delta Z the fluid leaves the control volume and as a result of this there's going to be some amount of mass accumulation if possible within the system. we are going to write the balance equation for such a system and derive the equation of continuity.
Let define a coordinate system and there is a box like in te picture.



There should be some amount of mass coming in and some amount of mass coming out. Just for simplification we have consider the mass that enters inside the x face. The rate of mass in through the x face is:

$$\text{rate of mass in the x face} = \rho v_x|_x \Delta y \Delta z$$

instead the rate of mass out through the x face is:

$$\text{rate of mass out the x face} = \rho v_x|_{x+\Delta x} \Delta y \Delta z$$

We are going to have 3 in terms at x, y, z and 3 out terms at $x + \Delta x, y + \Delta y, z + \Delta z$. As result of these in and out, you are going to get some amount of accumulation inside the system, so the governing equation is:

$$\text{rate of mass accumulation} = \text{rate of mass in} - \text{rate of mass out}$$

In order to have mass accumulation inside the system, the density of the mass change. $\frac{\partial \rho}{\partial t}$ rappresent the change in density of the fluid contained withing the C.V. In order to have kg/s as unit of measure, we need to multiply it bu the volume:

$$\text{rate of mass accumulation} = \Delta x \Delta y \Delta z \frac{\partial \rho}{\partial t}$$

Finally we have the conservation of mass:

$$\Delta x\Delta y\Delta z\frac{\partial\rho}{\partial t} = \Delta y\Delta z\left[(\rho v_x)|_x - (\rho v_x)|_{x+\Delta x}\right] + \Delta x\Delta z\left[(\rho v_y)|_y - (\rho v_y)|_{y+\Delta y}\right] + \Delta x\Delta y\left[(\rho v_z)|_z - (\rho v_z)|_{z+\Delta z}\right]$$

We divide both sides by $\Delta x\Delta y\Delta z$ and we take the limit of it tending to 0, we get the definition of the first derivative:

$$\frac{\partial\rho}{\partial t} = -\left[\frac{\partial}{\partial x}(\rho v_x) + \frac{\partial}{\partial y}(\rho v_y) + \frac{\partial}{\partial z}(\rho v_z)\right]$$

I did not take ρ outside the brackets because we are not considered the condition in which the flow is incompressible. In a more compact form we can write the equation as:

$$\frac{\partial\rho}{\partial t} = -\left(\nabla\cdot(\rho\bar{v})\right)$$

The divergence of the mass flux vector, should be equal to the time rate of change density inside the C.V.
We can also expand the bracket of the previous equation to get:

$$\frac{\partial\rho}{\partial t} = -\left[v_x\frac{\partial\rho}{\partial x} + \rho\frac{\partial v_x}{\partial x} + v_y\frac{\partial\rho}{\partial y} + \rho\frac{\partial v_y}{\partial y} + v_z\frac{\partial\rho}{\partial z} + \rho\frac{\partial v_z}{\partial z}\right] \Rightarrow$$

$$\Rightarrow \frac{\partial\rho}{\partial t} + v_x\frac{\partial\rho}{\partial x} + v_y\frac{\partial\rho}{\partial y} + v_z\frac{\partial\rho}{\partial z} = -\rho\left[\frac{\partial v_x}{\partial x} + \frac{\partial v_y}{\partial y} + \frac{\partial v_z}{\partial z}\right]$$

The left hand side is nothing else the substantial time derivative simply replacing c by ρ

$$\frac{D\rho}{Dt} = -\rho\left(\nabla\cdot\bar{v}\right)$$

and this is the continuity equation both expressed in partial derivative form and substantial derivative form.
The equation of continuity for incompressible fluid is when ρ is not a function of x, y, z, t but is constant, and so we obtain:

$$\nabla\cdot\bar{v} = 0$$

0.10 Equations for isothermal system

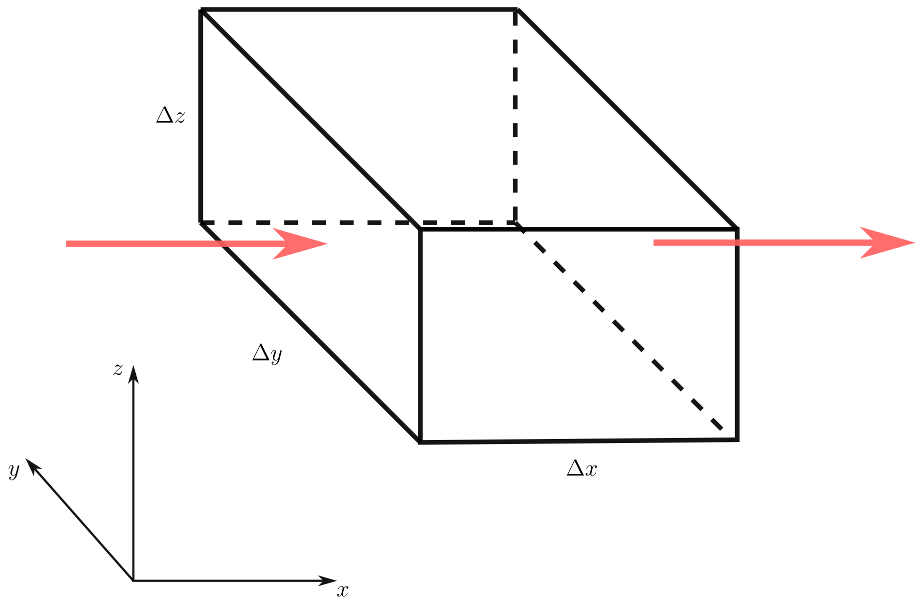
This class we are going to talk about equation of motion. So what is equation of motion and how it can be derived? The derivation itself is not that important because ultimately we are going to use the various forms of equation of motion, the one which is relevant to the system, to the geometry. We are trying to find out what's the velocity distribution. But for any system, if one wants to write the equation of motion essentially what we are doing for a fluidic system is we are writing the equation of Newton's 2nd law of motion for an open system in which the fluid is allowed to enter and leave the system. So there is some momentum which is being added to the system due to the motion of the fluid and there are two ways by which due to the motion of the fluid, momentum can come into the system which we have seen before. One is the convective momentum which is due to the flow of the fluid. So some amount of mass of fluid per unit time is coming, crossing the surface area. The amount of mass which crosses the surface area has a velocity at that point. So mass flow rate, mass $\times$ velocity would give you amount of momentum which comes in due to the actual motion, crossing the interface which is nothing but the convective flow of momentum into the control volume.

There are certain cases in which let's say the fluid is moving in upward direction (y direction) along a vertical plane which is the control surface and there is a gradient in velocity between two vertical planes. So y momentum gets transported in the x direction (horizontal direction) due to viscosity which is going to manifest itself as shear stress on the surface. So the y momentum getting transported in the x direction must also be taken into account as a source of momentum coming in to the control volume. Since its taking place in a direction perpendicular to that of the motion and since the principle reason by which this kind of momentum transfer takes place is molecular in nature, it's also known as the molecular transport of momentum or conductive transport of momentum. So we have two different momentum due to the flow of the fluid, one is the convective transport of momentum which is due to velocity and the second is conductive transport of momentum which is due to the velocity gradient. You remember Newton's law of viscosity in which the viscous transport is expressed by velocity gradient and not by velocity itself. So convective transport is due to velocity, conductive is due to velocity gradient. So these are different ways by which net rate of momentum coming into the control volume.

This control volume can also be acted upon by different forces. So, all the forces which are acting on the fluid inside the control volume should also be taken into account in the difference equation. As a result of all these, there may be an unbalanced force on the control volume and whenever there is an unbalanced force on the control volume, its momentum may change. So:

$$\begin{array}{c}\text{Rate of}\\ \text{momentum}\\ \text{accumulation}\end{array} = \begin{array}{c}\text{Rate of}\\ \text{momentum}\\ \text{coming in}\end{array} - \begin{array}{c}\text{Rate of}\\ \text{momentum}\\ \text{going out}\end{array} + \sum \begin{array}{c}\text{Forces acting}\\ \text{on the system}\end{array}$$

This is the most general form of equation of motion for a fluid which is nothing but again, Newton's 2nd law for an open system. That's what I have written over here. You can see that if this is defined as the control volume of side $\Delta x, \Delta y, \Delta z$, you understand that the equation that I have just described would be applicable on the system. So we would see how the convective momentum comes into the system.



If I only look at the face $\Delta y \Delta z$, and let's assume that the velocity here is v_x, so the amount of mass which comes in through the $\Delta y \Delta z$ face must be equal to $\Delta y \Delta z \rho v_x [kg/s]$. Now in order to obtain the $x-$momentum, multiply it with another v_x. So the x component of momentum must be equal to $\Delta y \Delta z \rho v_x v_x$. So, rate of momentum coming into the control volume is $\Delta y \Delta z \rho v_x v_{x_{|x}}$. And when we talk about the momentum that is going out, so this would be $\Delta y \Delta z \rho v_x v_{x_{|x+\Delta x}}$. Similarly when we talk about the $\Delta x \Delta z$ face, the amount of mass which is coming in through the face must be equal to $\Delta x \Delta z \rho v_y$. But this amount of mass has an x component of velocity. So in order to obtain the x component contribution of this much of mass, this must be multiplied with v_x. So mass coming in multiplied by the component of velocity in the x direction at that point would give me the amount of $x-$ momentum that comes in to the control volume through the face $\Delta x \Delta z$ at y. So, $\Delta y \Delta z \rho v_y v_{x_{|y}}$ is the total amount of $x-$ momentum coming in through the y face. And the x momentum that goes out of the control volume through the face at y would be $\Delta y \Delta z \rho v_y v_{x_{|y+\Delta y}}$.
Similarly I can write the z component, the amount of mass coming in through the z face is $\Delta x \Delta y \rho v_z$. Now this amount of mass has some x component of momentum associated

with it. In order to obtain that, I simply multiply it with v_x. So momentum in through the z face is $\Delta x \Delta y \rho v_z v_{x_{|z}}$ and out is $\Delta x \Delta y \rho v_z v_{x_{|z+\Delta z}}$. So all are x momentum. So these six terms tells us the x component contribution of momentum to the control volume:

$$\Delta y \Delta z \left(\rho v_x v_{x_{|x}} - \rho v_x v_{x_{|x+\Delta x}}\right) + \Delta x \Delta z \left(\rho v_y v_{x_{|y}} - \rho v_y v_{x_{|y+\Delta y}}\right) + \Delta x \Delta y \left(\rho v_z v_{x_{|z}} - \rho v_z v_{x_{|z+\Delta z}}\right)$$

Next we go into the contribution of molecular momentum (or conductive momentum or viscous momentum). Let's start with the y face. The area of the y face is $\Delta x \Delta z$. So the x component of momentum getting transported in the y direction (so τ_{yx}) must act on an area which is $\Delta x \Delta z$. So the x component contribution in the y direction must be equal to $\Delta x \Delta z \tau_{yx_{|y}}$. The one that goes out would also be same thing but evaluated at $y + \Delta y$.
Let's do the z face. The x component of momentum getting transported in the z direction (so τ_{zx}). In order to obtain the total amount of viscous momentum which is coming in through the z face, I must multiply τ_{zx} with the area of the z face which is $\Delta x \Delta y$. the rate of viscous momentum in through the z face is $\Delta x \Delta y \tau_{zx_{|z}}$. the rate of x momentum getting out of the z face is $\Delta x \Delta y \tau_{zx_{|z+\Delta z}}$.
τ_{xx} is the x component of momentum getting transported in the x direction. This is slightly unusual. Let us say three packets of fluids are coming towards the x face which is $\Delta y \Delta z$ and there is a variation on velocity between the 3 packets. So since there is a variation of v_x in the x direction, that means $\frac{\partial v_x}{\partial x} \neq 0$, then by Newton's law (our understanding of viscosity), there must be a stress between these 3 packets of fluids which will also be transmitted on this face. So this kind of stress where the principle direction of motion and the direction in which the momentum gets transported are identical, they are commonly called as the normal stress. So τ_{xx} is nothing but the normal stress exerted by the fluid on the x face due to a variation in velocity of the x component. There should be the in term evaluated at x and the out term evaluated at $x + \Delta x$

$$\Delta y \Delta z \left(\tau_{xx_{|x}} - \tau_{xx_{|x+\Delta x}}\right) + \Delta x \Delta z \left(\tau_{yx_{|y}} - \tau_{yx_{|y+\Delta y}}\right) + \Delta x \Delta y \left(\tau_{zx_{|z}} - \tau_{zx_{|z+\Delta z}}\right)$$

So these 6 terms in total would give you the amount of momentum which is coming in to the control volume as a result of viscous transport of momentum. So we have correctly identified in our equation of motion, the rate of momentum in and the rate of momentum out, both for convective motion and the conductive motion.
So what is left right now is to identify what is going to be the pressure forces and what is going to be the body forces.
So let us assume that the pressure at x is $p_{|x}$ and the pressure at $x + \Delta x$ is $p_{|x+\Delta x}$ So,

the difference in pressure force acting on the x face is

$$\Delta y \Delta z \left(p_{|x} - p_{|x+\Delta x}\right)$$

the other pressure differences do not contribute to the x component of forces. Only the pressure acting on the x face and on the $x + \Delta x$ face, they have contribution in the x direction

Please remember I am pointing out once again that we are writing this equation for the x component of the forces, be it pressure or be it other body forces. All the other components, y and z components, can be written in a similar fashion and you do not have to write each one of those components separately.
So we have identified the momentum, we have identified the pressure forces, the only thing that is remaining is the body force which is acting on it. Body force must be equal to the m (the total mass of the system), multiplied by the component of gravity in the x direction, if gravity is the only body force, since we are writing this equation for the x component of equation of motion. So the mass of the control volume would be $\Delta x \Delta y \Delta z$ (that is the volume), multiplied by the ρ which makes it $[kg]$, multiplied by g_x would be the x component of the body force

$$\rho g_x \Delta x \Delta y \Delta z$$

The rate of accumulation of $x-$ momentum inside the C.V. is:

$$\Delta x \Delta y \Delta z \frac{\partial}{\partial t} \left(\rho v_x\right)$$

Doing the balance we get the following compact form of the equation of motion in force per unit volume:

$$\rho \frac{D\bar{v}}{Dt} = -\nabla p - \left[\nabla \cdot \bar{\bar{\tau}}\right] + \rho \bar{g}$$

So how did we get to this equation from all the previous equations?. Remember what we have done for the case of x component of equation of motion. I have identified all the terms, those can be put into the equation of balance. So both sides can then be divided by $\Delta x \Delta y \Delta z$ and in the limit of $\Delta x \Delta y \Delta z \to 0$, then one can get a differential equation which is the x component of equation of motion. Hence in a similar fashion, one can write the y component of equation of motion and the z component of equation of motion. All these three equations can be added to obtain the compact equation of motion once you express them in a vector-tensor notation. So no new concepts are involved beyond what I have taught you in this part. So you can see the text and you can yourself see the simplifications that are made which are only algebraic in nature without the involvement of any additional concepts. So I did not derive the entire equation in this. I have given you enough pointers for the fundamental development of equation, and then I have told you how to combine these 3 equations in vector tensor notation and what you get is the equation of motion considering all the 3 directions. I would like to draw your attention to this equation once again, because each term of this equation is essentially force per unit volume and everything possible has been taken into account in here. So if in this equation you introduce the restriction of constant ρ and constant μ and if you add the equation of continuity:

$$\rho\frac{D\bar{v}}{Dt} = -\nabla p + \mu\nabla^2\bar{v} + \rho\bar{g}$$

which is the famous Navier-Stoke's equation.

There is one more simplification that can be thought of, is if viscous effects are absent, that means we are dealing with an inviscid fluid, in that case,

$$\rho\frac{D\bar{v}}{Dt} = -\nabla p + +\rho\bar{g}$$

0.11 Equation of change for Isothermal System

Navier–Stokes equation in Cartesian, cylindrical and spherical coordinates are given in any textbook. You can refer either to Bird, Stewart and Lightfoot or Fox McDonald, or any of the textbooks will contain the expression for Navier–Stokes equation in different coordinate systems.

So what you need to do in that is first see what kind of geometry you have in hand. Then accordingly choose whether it is Cartesian, spherical or cylindrical coordinate has to be chosen. Then find out what is the principal direction of motion. If it is one-dimensional flow, let's say in the z direction, then you choose the z component of equation of motion and simplify the terms which are not relevant in that context. If the flow is two dimensional, then you have to, analyze both, lets say, the x component of equation of motion as well as y component of equation of motion and then see what kind of simplifications you can suggest in order to make the set of equations solvable hopefully by analytic method, if not, we have to think of other methods including numerical techniques to solve such problems.

The Navier-Stokes's equation in Cartesian coordinates is:

$$
\begin{aligned}
x: \rho\left(\frac{\partial v_x}{\partial t}+v_x\frac{\partial v_x}{\partial x}+v_y\frac{\partial v_x}{\partial y}+v_z\frac{\partial v_x}{\partial z}\right) &= -\frac{\partial p}{\partial x}+\mu\left(\frac{\partial^2 v_x}{\partial x^2}+\frac{\partial^2 v_x}{\partial y^2}+\frac{\partial^2 v_x}{\partial z^2}\right)+\rho g_x \\
y: \rho\left(\frac{\partial v_y}{\partial t}+v_x\frac{\partial v_y}{\partial x}+v_y\frac{\partial v_y}{\partial y}+v_z\frac{\partial v_y}{\partial z}\right) &= -\frac{\partial p}{\partial y}+\mu\left(\frac{\partial^2 v_y}{\partial x^2}+\frac{\partial^2 v_y}{\partial y^2}+\frac{\partial^2 v_y}{\partial z^2}\right)+\rho g_y \\
z: \rho\left(\frac{\partial v_z}{\partial t}+v_x\frac{\partial v_z}{\partial x}+v_y\frac{\partial v_z}{\partial y}+v_z\frac{\partial v_z}{\partial z}\right) &= -\frac{\partial p}{\partial z}+\mu\left(\frac{\partial^2 v_z}{\partial x^2}+\frac{\partial^2 v_z}{\partial y^2}+\frac{\partial^2 v_z}{\partial z^2}\right)+\rho g_z
\end{aligned}
$$

The Navier-Stokes's equation in cylindrical coordinates is:

$$
\begin{aligned}
r: \rho\left(\frac{\partial v_r}{\partial t}+v_r\frac{\partial v_r}{\partial r}+\frac{v_\theta}{r}\frac{\partial v_r}{\partial \theta}+v_z\frac{\partial v_r}{\partial z}-\frac{v_\theta^2}{r}\right) &= -\frac{\partial p}{\partial r}+\mu\left[\frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial}{\partial r}(rv_r)\right)+\frac{1}{r^2}\frac{\partial^2 v_r}{\partial \theta^2}+\frac{\partial^2 v_r}{\partial z^2}-\frac{2}{r^2}\frac{\partial v_\theta}{\partial \theta}\right]+\rho g_r \\
\theta: \rho\left(\frac{\partial v_\theta}{\partial t}+v_r\frac{\partial v_\theta}{\partial r}+\frac{v_\theta}{r}\frac{\partial v_\theta}{\partial \theta}+v_z\frac{\partial v_\theta}{\partial z}+\frac{v_r v_\theta}{r}\right) &= -\frac{1}{r}\frac{\partial p}{\partial \theta}+\mu\left[\frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial}{\partial r}(rv_\theta)\right)+\frac{1}{r^2}\frac{\partial^2 v_\theta}{\partial \theta^2}+\frac{\partial^2 v_\theta}{\partial z^2}+\frac{2}{r^2}\frac{\partial v_r}{\partial \theta}\right]+\rho g_\theta \\
z: \rho\left(\frac{\partial v_z}{\partial t}+v_r\frac{\partial v_z}{\partial r}+\frac{v_\theta}{r}\frac{\partial v_z}{\partial \theta}+v_z\frac{\partial v_z}{\partial z}\right) &= -\frac{\partial p}{\partial z}+\mu\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial v_z}{\partial r}\right)+\frac{1}{r^2}\frac{\partial^2 v_z}{\partial \theta^2}+\frac{\partial^2 v_z}{\partial z^2}\right]+\rho g_z
\end{aligned}
$$

The same equations expressed in term of shear stress are the following:

$$
\begin{aligned}
x: \rho\left(\frac{\partial v_x}{\partial t}+v_x\frac{\partial v_x}{\partial x}+v_y\frac{\partial v_x}{\partial y}+v_z\frac{\partial v_x}{\partial z}\right) &= -\frac{\partial p}{\partial x}-\left[\frac{\partial}{\partial x}\tau_{xx}+\frac{\partial}{\partial y}\tau_{yx}+\frac{\partial}{\partial z}\tau_{zx}\right]+\rho g_x \\
y: \rho\left(\frac{\partial v_y}{\partial t}+v_x\frac{\partial v_y}{\partial x}+v_y\frac{\partial v_y}{\partial y}+v_z\frac{\partial v_y}{\partial z}\right) &= -\frac{\partial p}{\partial y}-\left[\frac{\partial}{\partial x}\tau_{xy}+\frac{\partial}{\partial y}\tau_{yy}+\frac{\partial}{\partial z}\tau_{zy}\right]+\rho g_y \\
z: \rho\left(\frac{\partial v_z}{\partial t}+v_x\frac{\partial v_z}{\partial x}+v_y\frac{\partial v_z}{\partial y}+v_z\frac{\partial v_z}{\partial z}\right) &= -\frac{\partial p}{\partial z}-\left[\frac{\partial}{\partial x}\tau_{xz}+\frac{\partial}{\partial y}\tau_{yz}+\frac{\partial}{\partial z}\tau_{zz}\right]+\rho g_z
\end{aligned}
$$

The Navier-Stokes's equation in cylindrical coordinates is:

$$
\begin{aligned}
r: \rho\left(\frac{\partial v_r}{\partial t}+v_r\frac{\partial v_r}{\partial r}+\frac{v_\theta}{r}\frac{\partial v_r}{\partial \theta}+v_z\frac{\partial v_r}{\partial z}-\frac{v_\theta^2}{r}\right) &= -\frac{\partial p}{\partial r}-\left[\frac{1}{r}\frac{\partial}{\partial r}(r\tau_{rr})+\frac{1}{r}\frac{\partial}{\partial \theta}\tau_{\theta r}+\frac{\partial}{\partial z}\tau_{zr}-\frac{\tau_{\theta\theta}}{r}\right]+\rho g_r \\
\theta: \rho\left(\frac{\partial v_\theta}{\partial t}+v_r\frac{\partial v_\theta}{\partial r}+\frac{v_\theta}{r}\frac{\partial v_\theta}{\partial \theta}+v_z\frac{\partial v_\theta}{\partial z}+\frac{v_r v_\theta}{r}\right) &= -\frac{1}{r}\frac{\partial p}{\partial \theta}-\left[\frac{1}{r^2}\frac{\partial}{\partial r}\left(r^2\tau_{r\theta}\right)+\frac{1}{r}\frac{\partial}{\partial \theta}\tau_{\theta\theta}+\frac{\partial}{\partial z}\tau_{z\theta}+\frac{\tau_{\theta r}-\tau_{r\theta}}{r}\right]+\rho g_\theta \\
z: \rho\left(\frac{\partial v_z}{\partial t}+v_r\frac{\partial v_z}{\partial r}+\frac{v_\theta}{r}\frac{\partial v_z}{\partial \theta}+v_z\frac{\partial v_z}{\partial z}\right) &= -\frac{\partial p}{\partial z}-\left[\frac{1}{r}\frac{\partial}{\partial r}(r\tau_{rz})+\frac{1}{r}\frac{\partial}{\partial \theta}\tau_{\theta z}+\frac{\partial}{\partial z}\tau_{zz}\right]+\rho g_z
\end{aligned}
$$

Let's say for the case of flow through the tube that we have analyzed so far, the direction of the motion of the fluid was in the z direction. So first of all in a tube we must choose the Cartesian coordinate system. Once I choose the Cartesian coordinate system, I will also have to choose what is the principal direction of motion, that is the z component. So in that lists the different components of Navier–Stokes equation, I must choose the z component of the cylindrical version of Navier–Stokes equation because my principal direction is in the z direction and then cancel the terms which are not relevant. So in this tutorial part of the course, I will pick three problems that we have done using

shell momentum balance, one where a fluid flows along an inclined plane, it's a freely falling liquid film. That means there is no imposed pressure gradient in the direction of flow. The flow takes place only because of gravity. In the second problem, the one that we have done is where the flow is taking place in a tube and there is a pressure difference as well as the gravity is acting downwards. And the third problem that we will look at is where we have a tube, the flow is from below, the liquid reaches the top of the tube, spills over and starts falling along the sides of the tube.
So in these three problems we have put considerable effort in obtaining the difference equation and from the difference equation, the differential equation. We would see, and I am sure all of you would agree with me, towards with the end of this class is that the use of Navier–Stokes equation is the way to go for solving the problems of fluid mechanics, the differential fluid analysis of fluid motion.
So we start with the first case where there was flow along an inclined plane, no pressure gradient, only gravity. This is the plate and I have a flow of liquid. The x direction is perpendicular to the flow and the z direction is along the flow. In the y direction the plate is assumed to be really wide. The length of the plate is L and the angle is β. The thickness of the falling film is δ. So let's see if we can simultaneously find out which equation we need to choose from this table. First of all, one must see that I have to choose the coordinate system, so obviously it is going to be a Cartesian coordinate system. The principal motion is in the z direction. So I must choose the z component of equation of motion. So if I choose the z component of equation of motion, then I am going to write the z component, either the expression that you see here, everything is expressed in terms of velocity, or in the form of the equation where instead of everything is expressed in terms of shear stress.

$$z : \rho\left(\frac{\partial v_z}{\partial t} + v_x\frac{\partial v_z}{\partial x} + v_y\frac{\partial v_z}{\partial y} + v_z\frac{\partial v_z}{\partial z}\right) = -\frac{\partial p}{\partial z} + \mu\left(\frac{\partial^2 v_z}{\partial x^2} + \frac{\partial^2 v_z}{\partial y^2} + \frac{\partial^2 v_z}{\partial z^2}\right) + \rho g_z$$

The same set of equations in terms of shear stress can be written as:

$$z : \rho\left(\frac{\partial v_z}{\partial t} + v_x\frac{\partial v_z}{\partial x} + v_y\frac{\partial v_z}{\partial y} + v_z\frac{\partial v_z}{\partial z}\right) = -\frac{\partial p}{\partial z} - \left(\frac{\partial}{\partial x}\tau_{xz} + \frac{\partial}{\partial y}\tau_{yz} + \frac{\partial}{\partial z}\tau_{zz}\right) + \rho g_z$$

In some cases it would be beneficial to work with the shear stress form, in some cases it would be more convenient to use with the velocity gradient form. So you can choose which one you are comfortable with. But the conceptually they are the same. In order to bring parity to what we have done in our previous class while solving this problem, I am going to choose the z component of Navier–Stokes equation in Cartesian coordinate system and I will use the shear stress form of the equation. Because you would remember that the governing equation that we have obtained for that case was in terms of shear stress. The same problem can be done using the velocity form of the Navier–Stokes equation. There is conceptually no difference between the two.

What were the assumptions that we have made while solving these problems? It was a steady state, 1-D flow where you only have $v_z \neq 0$ but all other components are going to be equal to zero, no pressure acting on the system and you only have the component of gravity in downward direction.

So the first term would be 0 since the velocity in the z direction is not a function of time (steady state). Now if you look at the second and third terms, both v_x and v_y is zero, and so are 0. Since we are also going to assume this as an incompressible fluid, than $v_z = f(x)$ only and $v_z \neq f(z) \neq f(y)$ and so also the fourth term is 0.

So what I have, the entire left hand side is going to be equal to zero.

This is a case where no pressure is acting, no applied pressure gradient on the system, so the first therm on the right hand side could be equal to zero.

Now look at the shear stresses. For the z component to be transported in the y direction, there must exist a velocity gradient in the y direction. As there is no y variation of velocity, So $v_z \neq f(y)$, therefore $\tau_{yz} = 0$. Similarly, for z component of momentum to get transported in the z direction, v_z must be a function of z. But we know that $v_z \neq f(z)$, therefore $\tau_{zz} = 0$. Now, when we come about τ_{xz} then for z component of momentum to get transported by viscous means in the x direction, v_z must be a function of x. If we look at the picture over here, v_z is definitely a function of x, being zero at solid-liquid interface here and maximum at the distance δ. So there will be a transport of z momentum in the x direction since $v_z = f(x)$.

I have dropped the partial sign because it is only a function of x, not functions of y or z, and so we have remains:

$$0 = -\frac{d}{dx}(\tau_{xz}) + \rho g_z$$

So of the three terms in the viscous transport of momentum, only one will remain. Now g_z is the component of the body force in the z direction and from the figure you can clearly see that the component of gravity in the z direction is simply $gcos\beta$. So the final equation would be

$$0 = -\frac{d}{dx}(\tau_{xz}) + \rho gcos\beta$$

that we have obtained from shell balance in the last week, this is exactly the same governing equation. So there is no need at all to think about a shell, make balances along and across the surfaces, find out what are the pressure forces and so on. The only job that you need to do is simply you choose the right component for Navier–Stokes equation in the appropriate coordinate system. After that use your understanding, the description, the physics of the problem, cancel the terms which are not relevant. What you will be left with is the governing equation. So it's a very simple way to arrive at the governing equation and once you arrive at the governing equation the rest will be identical. That means we are going to integrate in the same way, we are going to use the same boundary conditions and you will end up with the same solution but in a much more structured and easy way.

So in the next problem what we are going to see is the same problem where we have a flow through a tube, in which there is going to be a pressure gradient and there's going to be the action of gravity which has led to the Hagen–Poiseuille equation in the problems that we have dealt with before. So our next problem is analysis of flow through a vertical tube when there is a pressure gradient active in the system. So we have a vertical tube where r is the radial direction and z is the axial direction from the central axis of the tube. You have some pressure p_0 at the top of the pipe and p_L at the bottom of the pipe and as a result of which you are going to have flow in and flow out of the system. If you remember previously we had to think of a shell and in that shell we have found out what is the amount of liquid coming through the annular top surface, what is the shear stress that is acting inside and so on. Here we need not do anything of that sort right now. What I need to do only is to find out the right component of the equation over here. This is a cylindrical coordinate problem, r, θ, z. So, I am going to choose the cylindrical coordinates. The principle direction of the motion is in the z direction

$$\rho\left(\frac{\partial v_z}{\partial t} + v_r\frac{\partial v_z}{\partial r} + \frac{v_\theta}{r}\frac{\partial v_z}{\partial \theta} + v_z\frac{\partial v_z}{\partial z}\right) = -\frac{\partial p}{\partial z} - \left[\frac{1}{r}\left(\frac{\partial}{\partial r}(r\tau_{rz}) + \frac{1}{r}\frac{\partial}{\partial \theta}\tau_{\theta z} + \frac{\partial}{\partial z}\tau_{zz}\right)\right] + \rho g_z$$

The assumptions that we used are:

- $1-D$ flow, flow only in $z$direction ;
- incompressible flow ;
- $v_z \neq 0, v_z = f(r), v_z \neq f(\theta, z), v_r = v_\theta = 0$;
- Steady state condition ;

The first left term should be zero since is a steady state problem. The second left term would also be zero since that $v_r = 0$. The third left term is zero because $v_\theta = 0$. The fourth term is zero because $v_z \neq f(z)$. The entire left hand side of the expression is zero (temporal and convective transport of momentum).We have a pressure varient that acts

on the sistem along z and so the first right term is not equal to zero. The z component get transported in the r direction is present because we have the that $v_z = f(r)$ and so the second right terms is not zero. The z component get transported in the θ direction can happen only if the velocity must change in the θ direction, but this is ot true because $v_z \neq f(\theta)$ and so the third right term is zero. The z component get transported in the z direction can happen only if the velocity must change in the z direction and this is not true so the fourth right term is zero. We have the z components of gravity acting and so is not equal to zero.
The governing equation that we have is:

$$0 = -\frac{\partial p}{\partial z} - \frac{1}{r}\frac{\partial}{\partial r}(r\tau_{rz}) + \rho g_z$$

and since $v_z = f(z)$ only, we can write the expression as:

$$0 = -\frac{\partial p}{\partial z} - \frac{1}{r}\frac{d}{dr}(r\tau_{rz}) + \rho g_z$$

The BC's are:

- $v_z = 0$ at $r = R$ (no-slip condition) ;
- τ_{rz} must be finite at $r = 0$;

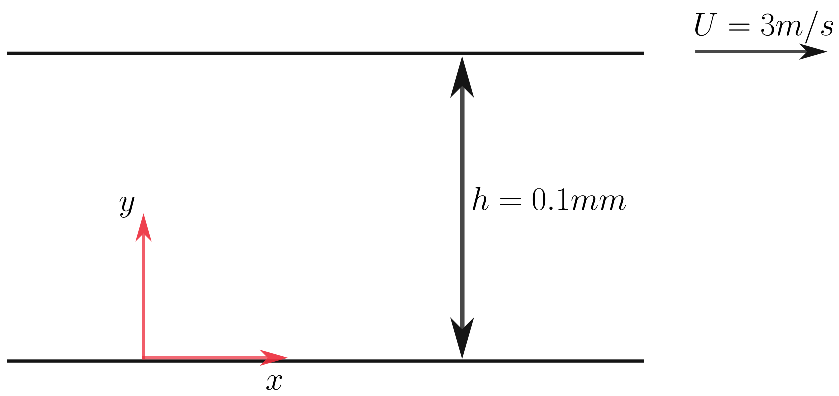
So once you reach this point your rest of the solution would be identical to the one you had done before. Now what you then see here is you do not need to think of a shell right now.

0.12 equation of change for isothermal systems

We have so far covered the Navier–Stokes equation, the equation of motion, the special form of which is Euler's equation and we have seen how in conjunction with equation of continuity, the Navier–Stokes equation can be simplified for different situations and to obtain the governing equation for flow. Once the governing equation is obtained, it's easy to solve with appropriate boundary conditions that defines the physics of the process. We, in the previous classes, have seen some of the simple problems which we have done using shell momentum balance. We have reworked those problems using Navier–Stokes equation.
In this and two more classes we would look at different problems which are slightly more complicated and therein we would be able to appreciate the utility of Navier–Stokes equation to obtain the governing equation for flow in such complicated systems. So I would draw your attention to this which you should be able to see more clearly in your

textbook where the equation of motion for a Newtonian fluid with constant density and viscosity are provided for Cartesian coordinate systems, cylindrical coordinate systems and for spherical coordinate systems. In the previous table the same equations are provided in terms of shear stress again in Cartesian, cylindrical and spherical coordinates. So depending on which kind of geometry you are handling, you would, you should be able to choose which equation out of these would be applicable for your case. In most of the cases what you do is you choose the equation which is in the direction of principal motion. There is flow in x direction, let's say on an inclined plane. Then you should choose the x component of the Navier–Stokes equation and cancel the terms which are not relevant to obtain the governing equation. What I would do in this class is I would give a problem for you to try on and I will also provide the answers.

So the first problem, It would be your job to arrive at the solution based on whatever we have discussed so far. So the problem that we have is a laminar flow between two infinite parallel plates. The upper plate moves to the right with a velocity $U = 3m/s$. There is no pressure variation in the x direction, i.e. in the direction of flow. y direction is perpendicular to the plate as shown in the figure below.



However there is an electric field which is given by $\rho B_x = 800N/m^3$. The body force provided by an electric field. The gap between these two plates of the liquid, is $h = 0.1mm$ and the viscosity of the liquid is $\mu_L = 0.02kg/ms$. You have to find out what is the velocity profile, $u(y)$, and the second part of the same problem is to compute the volumetric flow rate past a vertical section. And here you can assume the width of these plates to be $W = 1m$. Now you have to write the assumptions. It's one dimensional steady incompressible flow, so there is no variation of velocity, u, which is x component of the velocity, with x but you can clearly see that u is going to vary with y. It's going to be no-slip condition at the lower plate and at the top plate. So you should use the Navier–Stokes equation for Newtonian fluid and since the boundary conditions are in terms of velocities,

so probably it would be better if you use the velocity gradient form of the Navier–Stokes equation and not the shear stress. The shear stress form would probably have been useful or appropriate if you have, instead of a liquid solid interface at both ends, at one end you have a liquid vapor interface. Then the prevalent boundary condition at that point would be in terms of shear stress. So τ would be zero at the liquid vapor interface. So if that's the condition you have in your system then it is probably better to start with the the shear stress form of the equation of motion rather than the equation of motion expressed in terms of the velocity gradient. So look at the problem, see what are the boundary conditions you can use and then choose the relevant governing equation. Since, in our case, the motion is in the x direction, we must choose the x component of motion for Navier–Stokes equation for a fluid which is Newtonian and we understand the other parameters which are relevant here is the x component of velocity denoted by u is a function of y only. It does not vary with x, it does not vary with z. It's a horizontal system, so there is no gravity force present in the system. However there is an electric body force denoted by ρB_x, the value of which is provided. We also note that there is no pressure gradient present in the system. The first term on the right hand side which is $\frac{dp}{dx}$ part in the Navier–Stokes equation would be zero. And if you work out this problem then you would see as before the entire left hand side of the Navier Stokes equation which has one temporal term and the other three terms denoting the convective transport of momentum, they will be zero and you would be left with the right $\frac{dp}{dx}$, that would be zero as stated in the problem and then you have the viscosity terms, the terms which denote shear stress and a body force term. So you would clearly see that in this specific problem, the principle momentum is in the x direction. So, x momentum is getting transported because of a variation in the velocity in the y direction. So that's the only shear stress term that would be left in the governing equation, and the body force term. So these two terms would remain in your governing equation which you are going to solve and I will just simply give you the final expression for the velocity and the expression for the volumetric flow rate.
So the expression for velocity which you would find is

$$v_x = \frac{Uy}{h} + \frac{\rho B_x h^2}{2\mu}\left[\frac{y}{h} - \left(\frac{y}{h}\right)^2\right]$$

If you did not have the body force then the entire second term would be zero and what you end up with is the Couette flow expression which simply says that the velocity varies linearly with y and its maximum value of velocity would be at the top plate which is equal to U. So the first term is the Couette flow part and because of the body force you have these additional term present in the expression. So once you have the velocity you should also be able to obtain what is the area average velocity and you should get it to be in the form:

$$< v_x >= \frac{U}{2} - \frac{\rho B_x h^2}{12\mu}$$

This is the area average velocity that means if you put a plate which is perpendicular to the direction of flow and average the velocity out as whatever we have done before, this is what you are going to get as your average velocity and the volumetric flow rate would be the average velocity multiplied by area which in this case is:

$$Q =< v_x > hW$$

When you plug in the numbers the value of Q would be $1.5 \cdot 10^{-4} m^3/s$.
this is the quick problem which would give you some idea about how to use the Navier Stokes equation, how to get a form, the velocity, the average velocity and so on. So this problem is for you to work and to see if you are getting the right expression.

Next problem that we are going to deal with is slightly more complicated. Herein we have a piston and a cylinder. So it's a piston cylinder apparatus assembly where there is sufficient pressure which is generated or which is provided on the piston. As a result of which the piston slowly starts to come into the cylinder. The cylinder initially contains a liquid, and an oil, viscous oil which is used as a lubricant. So as this piston starts to come inside the cylinder the oil which is contained in the cylinder must come out in between the thin gap between the piston and the cylinder. So as the piston starts to come inside, since the liquid present inside is incompressible it must leak through the very small gap that exists between the the cylinder and the piston. So it's a piston cylinder assembly, very close fitting that means the outer diameter of the piston is slightly smaller than the inside diameter of the piston. Or in other words, the gap in between the piston and the cylinder is extremely small. We have to make an assumption in this case which is very common for systems in which the curvature is small compared to the radius of the system. Now if the piston and the cylinder are very close fitting and if the piston has a large radius, then what would happen is that for a very small section in the piston cylinder assembly the flow is going to be in between parallel plates. Or in other words, for cases where the gap between two surfaces are very small as compared to the curvature of the system, which is the case in the piston assembly, then a cylindrical coordinate problem can be transformed in a Cartesian coordinate problem. So the piston cylinder assembly or in any such situation where the gap is very small compared to the curvature of the system, you can assume those two surfaces which are essentially cylindrical in

nature but would behave as if they are flat plates. So a piston cylinder assembly like this, can now be opened so that they become plates. And once they become plates, then we will be able to use the Cartesian component of the Navier–Stokes equation. It is a very common practice in many cases to resolve the cylindrical problems into Cartesian coordinate system problems. It must be explicitly written or you must understand.
So what we have in this case is the piston going down into the cylinder with a very thin gap in between, so we can very safely assume that it is going to be the flow between two parallel plates where the plate which is representing the piston is going down and the plate which is representing the cylinder remains static. If you use the cylindrical system it is fine, you can still do that but the advantage of using the Cartesian coordinate system is the terms are simpler. Then it is fairly easy to handle a problem in Cartesian coordinate system as opposed to that in a cylindrical coordinate system. So we will always try to use Cartesian coordinate system as far as possible. So herein is a case which is ideal for transforming from cylindrical to Cartesian coordinate system. So I will draw the system and tell you the parameters and the problem.
The piston cylinder assembly is shown in the figure above. On top of this piston let's put a mass of M. We have the oil in between the gap. As the piston starts to come down, the oil has to leak in between the thin intervening space between the piston and the cylinder.

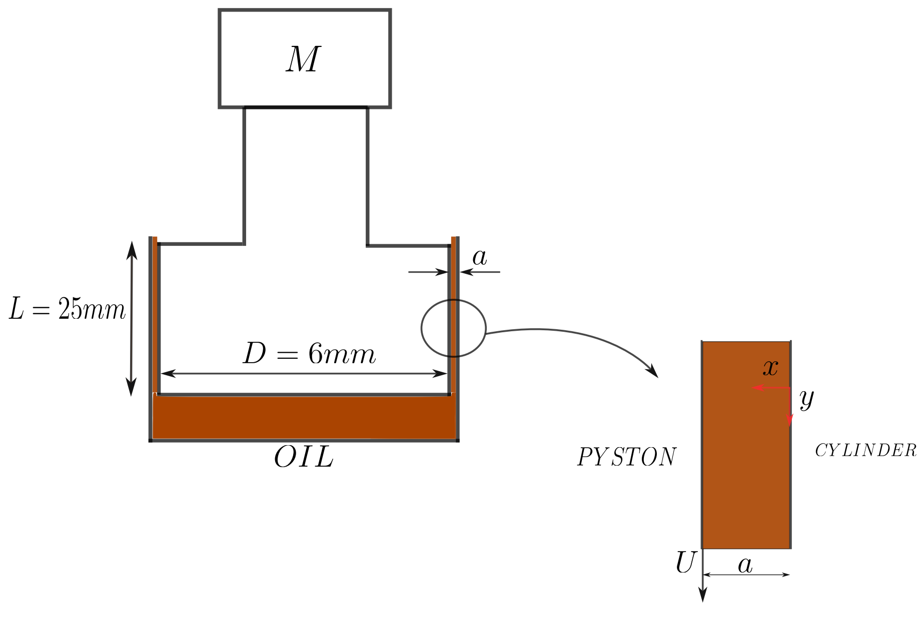
The diameter of the piston is $D = 6mm$ and the length of the piston is $L = 25mm$. The piston is coming down with a velocity of $1mm/min$. So you can see the piston is coming down at a very slow velocity and as a result of which the oil is leaking from the side walls. Now the first part of the problem is find the mass M that needs to be placed on the piston to generate a pressure equal to $\P_\gamma = 1.5MPa$ inside the cylinder.
The second part of the problem is to find the gap a between the cylinder and piston such that the downward motion of the piston is $1mm/s$.
So this is the entire problem. The piston is loaded with the mass that creates a pressure of $1.5MPa$ at the bottom and the piston is slowly coming down. There is a huge pressure difference between the point 1 (inside the oil bottom) and the point 2 (outside the exit of oil), which is open to atmosphere. At the same time you have the cylinder coming down but the cylinder velocity is fixed at $1mm/min$. So you must find out what is the space in between the cylinder and the piston that would allow the system to work in the specified form. So if your a is large then the oil will come out with a very high velocity and as a result of which the piston will fall with a higher velocity. On the other hand if a is very small then the leakage rate of oil would be such that it would not allow the piston to come down with a velocity of $1mm/s$. So there is only one value of a which would give you the correct oil leakage rate such that the piston is going to come down with the specified velocity while the pressure is maintained to be $1.5MPa$ inside the cylinder.

So let's first evaluate how do we get the value of M in this case. The value of M is going to provide a force equal to Mg on the platform on which it rests on the piston. So this force exerted on the piston by the weight must be supported by the pressure gradient which exists inside the cylinder and outside of the cylinder. So whatever be the area of the piston which has a diameter of $6mm$, that area of the piston multiplied by the pressure difference between the cylinder and the atmosphere must balance each other. In other words:

$$Mg = \frac{\pi D^2}{4}(p_\gamma - p_{atm})$$

So, so this formula should be used to obtain what is the value of the unknown mass that must be placed on the piston to generate the pressure of $1.5MPa$ inside the cylinder. So the only unknown is M and the value of M can be obtained as $4.32kg$. So the first part is done.

About the second part, as I said since the gap is very small in comparison with the curvature, then we are going to think of it as two parallel plates, the x and y directions are shown in the above figure.



One wall represents the piston and other wall is the cylinder. So in between I have oil present inside. The piston is slowly coming down with the velocity $1mm/s$, the gap in in between the cylinder and the piston is a. We have to write the balance equation to obtain what is the expression for a. Now you can clearly see that as the motion is in the y direction, you have to choose the y component of Navier–Stokes equation and as before

it's a steady state, 1D flow problem and therefore the entire left hand of Navier–Stokes equation would be equal to zero. So what you have on the right hand side is $\frac{dp}{dx}$. This must be taken into account because we have a huge pressure gradient in between the two ends of the piston. At one end, its $1.5 \cdot 10^6 Pa$ at the other end you have the atmospheric pressure, and dx is the length of the piston which has also been provided. Then you have the viscous term in which you have y directional velocity which is varying in the x direction. So the term $\mu\frac{d^2v_y}{dx^2}$ will be present in the Navier–Stokes equation and since it's vertical, you are going to have the effect of gravity as well.
So the governing equation for such system can be written as:

$$\mu\frac{d^2v_y}{dx^2} - \frac{dp}{dx} + \rho g = 0$$

There is one more thing which you have to keep in mind is that at times the problem becomes simpler if you would be able to cancel some terms based on their magnitude. So this order of magnitude analysis tells you about which of the terms in the Navier–Stokes equation, eventhough its present, even though its non-zero but it's so small in comparison to the other terms that it can be neglected. There are certain cases which would be apparent specially if you look into this problem. So I have two terms, one is the pressure gradient term, other is the ρg term. So what's the rough approximation value of ρg? So the term ρg is going to be of the order of 10^4. The pressure difference is of the order of 10^6 Pascal and dx, which is the length of the piston is $25mm.$ so roughly this $\frac{dp}{dx}$ term is going to be of the order of 10^8. So you could see the difference in order between these two terms. It is safe to drop this ρg term from your governing equation as well.
So sometimes order of magnitude analysis of the different terms present in the governing equation lets you further simplify the Navier–Stokes equation which you should always look for. So your governing equation now becomes

$$\mu\frac{d^2v_y}{dx^2} = \frac{\Delta p}{L}$$

and therefore

$$v_y = \frac{1}{2\mu}\frac{\Delta p}{L}x^2 + c_1x + c_2$$

and the two boundary conditions are at $x = 0, v_y = 0$ that is the no slip condition, which will give me $c_2 = 0$ and the other BC is at $x = a, v_y = U$ ans so i will get c_1.

$$v_y = \frac{1}{2\mu}\frac{\Delta p}{L}\left[x^2 - xa\right] + U$$

and the average velocity is:

$$< v_y >= \frac{1}{a}\int_0^a v_y dx = -\frac{1}{12\mu}\frac{\Delta p}{L}a^2 + \frac{Ua}{2}$$

So the governing equation with the relevant boundary condition should give you the local velocity and the average velocity. Once you have the average velocity, then you should be able to obtain what is the flow rate:

$$Q =< v_y > a\pi D = \frac{\pi D^2}{4}U$$

1 Heat Transfer

So what is heat transfer? Whenever we talk about heat transfer, the other thing comes to our mind is thermodynamics. So if you have a system at a state $'A'$ and a system at a state $'B'$ and if you know the conditions of the state, the thermodynamics would tell you what has changed between $'A'$ and $'B'$. So thermodynamics essentially deals with the end states of the process. But how you are going to get heat transfer from one point to the other, through the transport of heat from system $'A'$ to system $'B'$ is defined as heat transfer. So the transport of energy from a system to another system is described by the heat transfer process. Thermodynamics deals with the end states but heat transfer deals with how and when the energy can get transported from system 1 to system 2. So that is why we call heat transfer as its energy in transit.

Now when we talk about heat transfer, we also know there are three types of heat transfer which are possible.

One is the conductive heat transfer in which you require a medium, but there is no net movement of the medium. So the heat conduction, like most of the other conduction processes in other fields, there is no net motion of the molecules and heat gets transported from one point to the other point through a medium, it could be solid, liquid or even gas and heat always travels from high temperature to low temperature. And whether or not heat can pass easily through a system is denoted by a property of the system which is the thermal conductivity of the system. The fundamental law which dictates the amount of conductive heat transfer from point 1 to point 2 is Fourier's law of conduction. The Fourier's law of conduction is a phenomenological equation. You cannot derive Fourier's Law from first principles. It's a result of seeing and analyzing a large amount of experimental data and to establish a relation between the heat flux (which is the amount of heat flowing per unit area per unit time). If you have heat transport only in the x direction, so the $q_x^{''}$, the amount of heat that gets transported in the x direction, is proportional to $\frac{dT}{dx}$. So it's not proportional to the temperature difference but the temperature gradient. Then you plug in the proportionality constant which is going to be the property of the medium that would dictate the ease or difficulty with which heat gets transported is known as thermal conductivity and the complete form of equation in 1-D is:

$$q_x^{''} = -k\frac{dT}{dx}$$

The double prime denotes the flux and not total quantity. The minus sign denotes the physical observation that heat always flows from high temperature to low temperature.

In difference form we can express is as:

$$q_x^{''} = -k\frac{T_2 - T_1}{L}$$

Next we come to convection in which we still require a medium but the medium is moving. Therefore, many of the common examples of what we see around us is a combination of conduction and convection process. In some cases convection is more common than conduction. So if you are sitting in a room listening to my lecture and if you have a fan or an air conditioner, the flow of air above you which helps to cool or reduce your body temperature, is an ideal example of convection. So you have movement of the medium past you and that is the basic requirement of convective heat transfer process. But you still require a medium. One more thing I would like to point out here is that you can never have convection without conduction. You can have a system in which you have only conduction. For example, if you have a solid object and if you maintain one side of it at a higher temperature as compared to other side, there would be transport of heat even when there is no convection. But let's say the same solid object is at 100°C and you keep it in a room, in which a fan, a blower is making the air moving over the hot object with a certain velocity, then you are going to have convection. But even at that point, the air molecules which are very close to the solid surface, they will cling to it due to no-slip condition and they are going to gain, energy from the hot object by means of conduction and then it will transfer that energy to the mobile molecules just above it by means of convection. So in the convection process between the solid and the convective flow of air above it, you have a layer of molecules which due to no-slip condition is not moving. So through that layer you have conduction. So conduction is there in convection process but you can have a purely conductive heat transfer. But you do not have something called purely convective heat transfer. There would be one stagnant layer which is going to get energy or lose energy through the adjoining surface by means of conduction. The law which describes the convective heat transfer process is known as the Newton's law of cooling, which simply tells us the amount of heat flux lost from the surface per unit time is:

$$q^{''} = h(T_s - T_\infty)$$

where h is the convective heat transfer coefficient, T_s is the temperature of the solid substrate and T_∞ is the temperature of the fluid at a point far from that of the solid. The amount of heat transfer by convection from a surface would be different based on whether you have laminar flow or you have turbulent flow around the solid object. And of course if your velocity is more, if your flow is in the turbulent region, you will lose or

gain more energy by convection. So a natural convection or a free convection is going to dissipate lesser amount of heat by convection as compared to the forced convection method in which you are forcing the fluid by an external agency to move over the solid at a higher velocity and thereby creating the right conditions for additional heat transfer. Whereas in free or natural convection you are not forcing, there is no external agency which forces the fluid to move. The fluid adjoining to a hot plate simply gets heated and it will rise to be replaced by cooler air from the surrounding. So a hot object placed in a room full of static air will create a current in that static room due to the change in buoyancy of the gas or the air, let's say of the air, which is caused by its interaction with another solid object of higher temperature. So this convective process will start in without the aid of an external agency and its known as the natural or free convection.
The third one is radiation which does not require the presence of a medium and therefore the common law of radiation which expresses the amount of heat flux, which gets transported as a result of the temperature of the substrate is given as:

$$q^{''} = \epsilon \sigma T_s^4$$

where σ is Stefan Boltzmann's constant and T_s is the temperature of the solid substrate in Kelvin. However for real surface, there is a factor emissivity ϵ which is brought into this formula to emphasize that real surfaces do not emit heat radiation as efficiently as that of an ideal substrate where the value of emissivity is 1. So you have different values of emissivity for different surfaces. The radiative heat transfer is in itself a separate subject of heat transfer in which you would have the concept of the transmittivity, the reflectivity and so on. There is a concept of black body, gray body and you have probably done the network method of radiation exchanges between surfaces which are forming, let's say an enclosure, the concept of view factor and so on.
So I will not discuss about them in this transport phenomena course. In transport phenomena I will restrict myself to conduction, and convection and try to write generalized equations and develop models which would describe the convection process and so on. Same as in the case of momentum transfer, we would see that defining or assuming a shell and making a balance of all the heat inflow and outflow terms and the amount of heat generation in the system etc., after a while it becomes very difficult to visualize and solve a system assuming a shell only. So the same way, a generalized method in the form of Navier-Stokes equation was used. Similarly for heat transfer also, we will develop an equation which not only takes care of all the heat flow in and out and heat generation, but it would also take into account the work done by the system or work done on the system because that would also affect the total energy content of the control volume.

So generalized equation which would take the heat as well as the work form of energy into considerations, that can be used for any system undergoing conduction or convection, free or forced, in presence or absence of body forces and so on, that equation we are going to derive in this course and we would see then, as in the case of Navier–Stokes equation, a simplification of the energy equation for the problem at hand would make our life a lot simpler. So you would simply write the energy equation in the correct coordinate system, cancel the terms which are not relevant for the problem that we are discussing and then what we will have in the end is the governing equation. Once we have the governing equation, we will also try to see what would be the pertinent boundary conditions for that specific problem. Then it is a question of simply solving it to obtain the temperature profile. Once you have the temperature profile, you can find out the gradient of temperature at a specific location to obtain how much heat that surface is receiving or losing and how do we connect the amount of heat loss to the heat transfer coefficient and thereby obtain a relation that contains h, the convective heat transfer coefficient and the length scale and property of the system so as to combine the h and other properties including the geometry of the system. We will bring the concept of Nusselt number. So the expression for Nusselt number is the most sought after while describing convective heat transfer process. So our whole emphasis will be to start the energy equation and obtain if possible an expression for Nusselt number for the heat transfer taking place in that specific geometry under that specific conditions. So that is what we are going to do in our treatment of heat transfer from now on.

From point of view of notation we have that $q^{''}$ is the heat flux measured in $\frac{W}{m^2}$, then we have the heat flow q measured in W which simply is:

$$q = q^{''}A$$

than we have the heat generated per unit volume that is $\dot{q}$ measured in $\frac{W}{m^3}$ and finally k measured in $\frac{W}{mK}$.

1.1 Heat Transfer

So the fundamental study of heat transfer starts with our identification of the conservation of energy principle. So, for any object we first have to appreciate the conservation of energy principle and that conservation of energy principle when expressed in terms of mathematical relations could give rise to the complete equation describing the energy transport from or to the object. And energy conversion inside the object if that is relevant as a result of which, the total energy content of this object will change.

Energy can also be changed if you include the work done to or by the system. So work done on the system will enhance its energy and work done by the system will reduce the total energy content of the object. So, all these considerations must be taken into account while writing the conservation equation for any system.
So, we are going to start with the simplest conservation equation to begin with where we will at the moment not consider any work effects. So all our consideration is the tangible form of energy which comes in or out, to or off the control volume as a result of conduction or convection. So, we start with the conservation of energy relation which simply says that:

$$\dot{E}_{in} + \dot{E}_g - \dot{E}_{out} = \dot{E}_{st}$$

the rate (the dot represents essentially the time rate) of energy coming into the system by either convection, conduction or a combination of both ($\dot{E}_{in}$), plus the rate generation of energy inside the system ($\dot{E}_g$), that could be due to the ohmic heating or due to the presence of a nuclear source in the control volume, minus the rate of energy which is going out of the control volume, as a result of conduction and/or convection, and the sum total of all these three is the rate of change of stored energy in the system.
So when we talk about steady-state, the rate of energy stored is going to be equal to 0 ($E_{st} = 0$) that means there would not be any net storage of energy as a result of these few processes, the temperatures could be a function of space coordinates, that means the temperature could be different at different points in the control volume but temperature will not be a function of time and so $\frac{\partial T}{\partial t} = 0$. we understand that T could be a function of the three space coordinates $T(x, y, z)$.
If you think of a solid surface and air in contact with it, and let us assume that the air temperature is T_∞ at a point far from the solid edge. The inside temperature is T_1 at $x = 0$ and T_2 at $x = L$. So, you would see later on, that in absence of any heat generation in the solid, the temperature profile will be linear, but then it would sharply reduce and asymptotically merge with the temperature T_∞, that is the temperature at a point far from it.

So, if you consider the region which is very close to the solid surface in the fluid where the temperature sharply changes from T_2 to that of T_∞ far from the plate. And here you see that the temperature asymptotically approaches the free stream temperature, so this by analogy with our previous discussion, you can clearly see that this essentially establishes the concept of a thermal boundary layer.
So same way as in velocity boundary layer, where the velocity increases as we move away from the flat plate and reaches the value equal to the free stream velocity, the same way the temperature changes from that of the solid and it gradually approaches to the bulk temperature or T_∞, that is the temperature at a point far outside of the effect of the solid

present in contact with the fluid. So, the distance over which this temperature variation takes place, will in a proper way be expressed as the extent in which the heat transfer is taking place. Because outside of that the temperature does not vary with distance anymore. So, no heat transfer, convective or conductive is taking place in the region where the temperature reaches T_∞. So ultimately we will try to bring in the same concept as that of velocity or hydrodynamic boundary layer in the case of heat transfer as well in the form of thermal boundary layer.

But coming back to the problem that we were discussing about, so if this is my control volume, this expression is applied to the control volume, $\dot{E}_{in}$ is the rate of energy coming from the left of the figure, plus $\dot{E}_{gen}$ if this is a nuclear fuel element, then some amount of heat would be generated, if it is not, then $\dot{E}_{gen} = 0$, $\dot{E}_{out}$ is the energy which goes out to the air surrounding it as a result of which the net energy content of the control volume may change and if it does not change, then what we have is a steady-state situation. So, in absence of energy generation and if it is a steady-state system, we have that $\dot{E}_{in} = \dot{E}_{out}$.

What is a control surface? The dotted line which I have drawn can be thought of a control surface which by the definition does not have any mass of its own. So if it does not have any mass of its own, then it cannot store any energy and no energy is generated in it. So, at the control surface, we have always $\dot{E}_{in} = \dot{E}_{out}$.

1.2 Heat diffusion equation

If I have a control volume of size $dxdydz$ and if we consider only conduction, let us say $q_x^{''}$ is the amount of flux of heat which is coming through x face and $q_{x+dx}^{''}$ is going out and also for the other directions.

$q_{x+dx}^{''}$ can simply be expressed in the Taylor's series expansion:

$$q_{x+dx}^{''} = q_x^{''} + \frac{\partial q_x^{''}}{\partial x} dx$$

same way can be done for other directions. So, you can simply write it and if we assume that $\dot{q}$ is the energy generated per unit volume inside this by some means, could be nuclear, then the total amount of heat generation in this control volume would simply be:

$$\dot{E}_g = \dot{q} dxdydz$$

and the energy stored would be:

$$\dot{E}_{st} = \rho c_p \frac{\partial T}{\partial t} dxdydz$$

where $\rho dxdydz$ is simply equal to the mass m of the control volume. So, $mc_p \frac{\partial T}{\partial t}$ is the time rate of change of energy stored in the system. When you write the balance equation:

$$\dot{E}_{in} + \dot{E}_g - \dot{E}_{out} = \dot{E}_{st}$$

$$\frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right) + \frac{\partial}{\partial z}\left(k\frac{\partial T}{\partial z}\right) + \dot{q} = \rho c_p \frac{\partial T}{\partial t}$$

The first three terms are the amount of energy, net conductive heat flux into the control volume from the x direction, from the y direction and from the z direction. So if we assume that it is a constant k case, that is the thermal conductivity of the system is constant, then k can simply be taken out of it and what you get is:

$$\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} + \frac{\dot{q}}{k} = \frac{1}{\alpha}\frac{\partial T}{\partial t}$$

So, there is a reason why $\alpha = \frac{k}{\rho c_p}$ is expressed in here. I simply write $\frac{1}{\alpha}$ because α has units of m^2/s and is called the thermal diffusivity of the system. When we spoke about the momentum diffusivity, which was denoted by $\nu = \frac{\mu}{\rho}$, (which was the kinematic viscosity), its unit was also m^2/s.
So these two, the momentum diffusivity or the thermal diffusivity having the same units as m^2/s, it came from the concept of diffusion coefficient which we would discuss when we will talk about the Newton's law, when we will talk about the mass transfer process, the conductive mass transfer process which is Fick's 1st law for diffusion which connects the amount of diffusive transport of mass is equal to $-D_{AB}\frac{dC_A}{dx}$, where D_{AB} is the diffusion coefficient of A in B, if it is a one-dimensional conduction, one-dimensional mass transfer process. And this D_{AB} is the mass diffusivity and has units of m^2/s. So simply by rearranging the terms in the form of $\frac{\mu}{\rho}$ in momentum transfer and $\frac{k}{\rho c_p}$ in heat transfer and D_{AB}, since all of them have the same units as m^2/s, so borrowing the term from mass transfer, $\frac{\mu}{\rho}$ is called the momentum diffusivity, $\frac{k}{\rho c_p}$ is called the thermal diffusivity and D_{AB} is simply called the diffusivity or mass diffusivity.
So that is why that the equation of energy is generally expressed not in terms of k, ρ or c_p but the combination variable which is called the thermal diffusivity.
So, this is the equation what you would get for only for heat conduction, where the temperature can vary in x, y and z direction, the fourth term is the heat generated per

unit volume and the right-hand side is the total amount of energy stored or lost in the control volume per unit volume.
If it is at steady-state, then, $\frac{\partial T}{\partial t} = 0$ and what you will get is:

$$\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} + \frac{\dot{q}}{k}$$

If it is 1-D conduction in steady-state and let's say the heat is only getting transported in the x direction, and without heat generation, then what you would get is:

$$k\frac{d^2T}{dx^2} = 0$$

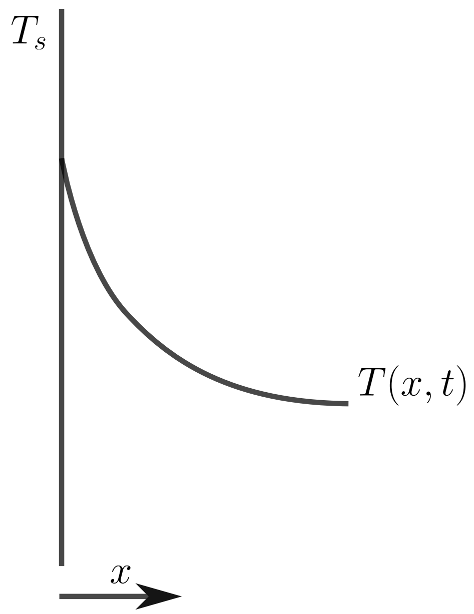
$$\frac{d}{dx}\left(k\frac{dT}{dx}\right) = 0 \quad \Rightarrow \quad \frac{dq_x^{''}}{dx} = 0$$

The double prime denotes the flux.
So, these expressions are very common and would give rise to several other simplifying cases. So this is in Cartesian coordinate system, so if you write it in cylindrical coordinate systems, then you should be able to see it from the text:

$$\frac{1}{r}\frac{\partial}{\partial r}\left(kr\frac{\partial T}{\partial r}\right) + \frac{1}{r^2}\frac{\partial}{\partial \phi}\left(k\frac{\partial T}{\partial \phi}\right) + \frac{1}{z}\frac{\partial}{\partial z}\left(k\frac{\partial T}{\partial z}\right) + \dot{q} = \rho c_p \frac{\partial T}{\partial t}$$

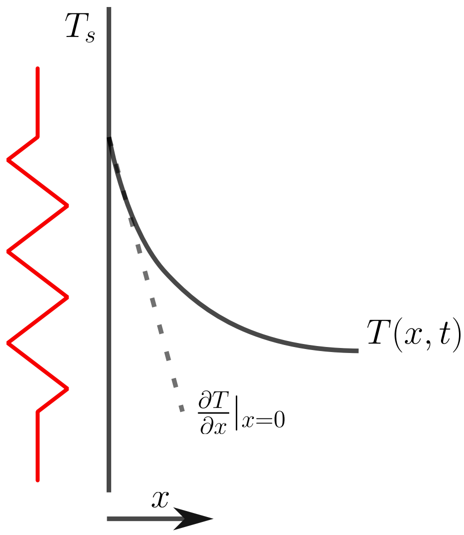
there was no new concept involved in the cylindrical coordinate expression, it is only that when you transform from Cartesian coordinate system to cylindrical coordinate system, you express everything in terms of r, ϕ and z. Where r is the radial distance, ϕ is rotational direction and z is the axial direction. So, when you transform the coordinate, the form of the equation gets slightly more complicated but fundamentally it still remains the same.
So one should start with the right equation, cancel the term which are not relevant and then apply the relevant boundary conditions to obtain what would be the temperature distribution in a solid, be it a rectangular, a cylindrical or a spherical solid and to get a complete picture of the temperature profile, present in such a case.
So, when we talk about boundary conditions which are present in the system experiencing heat transfer, let us see what are the possible boundary condition that can be there.
The first boundary condition is the temperature at a point can be specified, so you precisely know what is the temperature at a specific point. So, if a solid let us say is in contact with a fluid and you know what is the temperature of the solid at the edge, at $x = 0$, when it is in contact with the liquid, then the temperature at this point is specified. So, the first condition that you should look for while solving the equation is, if the temperature at any point in the control volume is provided to us. So, the first

boundary condition which is relevant in heat transfer is surface temperature known.

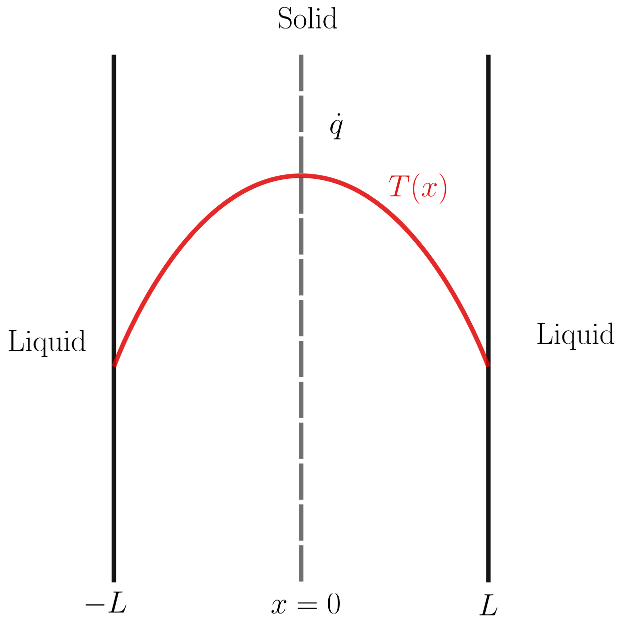


So, let us say this is the x direction and this T_s is known and over here the temperature may vary as a function of x and t but we know that at $x = 0, T(x,t) = T_{0_s}$ that is a constant.

The second type of condition can be a constant surface heat flux. That means you have a way, by creating a condition, let us say this is the temperature profile where $T(x,t)$ is known and if I draw a slope to this line, so this is essentially $\frac{\partial T}{\partial x}|_{x=0}$, and so $q_s^{''} = -k\frac{\partial T}{\partial x}|_{x=0}$ which is nothing but the heat flux at the surface. So it could be such that this value of $q_s^{''}$ is provided to you. The meaning it could be that you have a resistance heater in here. The resistance heater is providing a certain amount of heat into the solid which is in contact with the liquid (the solid is on the left and the liquid is on the right). So at steady-state, the amount of heat which is supplied by the heater has to come and has to go out of the surface. So at this point which is the junction of liquid-solid interface, the heat flux is a constant. And if this is the profile of temperature, then as we understand that you can never have convection without conduction, so conduction is the means by which heat gets transferred to the 1st layer of liquid molecules. So the heat flux lost by the solid to the liquid, this must be equal to the amount of heat which is supplied to the solid by the resistance.



There is the third type of boundary condition is where by an artificial means you do not allow any heat to cross a specific interface. So, you have this solid and you are going to place a perfect insulation at the right side, so if you place a perfect insulation, no heat can cross through this insulation and come out on the other right side. So, if you can apply a perfect insulation on one side of it, then going back to what we have said, so if you have an insulated surface, then $q_s^{''} = 0$, and this would give rise to $\frac{\partial T}{\partial x}|_{x=0} = 0$. If by some means you could insulate a surface perfectly which is an idealized condition, then no heat can cross this. So, this is known as insulated surface or an adiabatic surface.
In some cases, I am describing this adiabatic surface in slightly more detail, let us say you have a solid, on two sides of it you have liquids and some amount of heat is being generated in the solid uniformly. So everywhere there is some amount of heat which is generated. I would show you later on but the profile would look something like this



So, when you consider the plane which is located at $x = 0$, if you see this region, T at $x = 0$, what you would see is $\frac{dT}{dx} = 0$. The nature of the curve tells you that for the special case when you have a solid and some amount of heat is generated uniformly and you would see later on that the profile would be like an inverted parabola, then the apex of the parabola which is located on $x = 0$, will have a slope 0 at $x = 0$. So the $x = 0$ plane, no heat crosses from the left to the right or from the right to the left, so the $x = 0$ plane is known as the adiabatic plane. So, for an adiabatic plane or for an insulated surface as we have seen before would be the same which is in the case of one-dimensional conduction case, $\frac{dT}{dx} = 0$.

The third condition could be that I have a convection surface condition where you have surface which is solid, on this side you have a liquid and the profile of temperature is this, where you have T_∞, and the heat transfer coefficient involved is h. So heat will come from left to the boundary by means of conduction and from the boundary to right by means of convection. So what happens at steady-state is then:

$$-k\frac{\partial T}{\partial x}|_{x=0} = h\left[T(0,t) - T_\infty\right]$$

where the left term essentially denotes the amount of heat which comes in by conduction from the interior up to this point and has to be equal to the heat which goes out of the convection, which is the right side, and if I use Newton's law we have that equation. So, the convective heat loss must be equal to the conductive heat which is coming to the interface. So by conduction you have some amount of heat coming in and by convection the same amount of heat is going out.

1.3 Heat Transfer

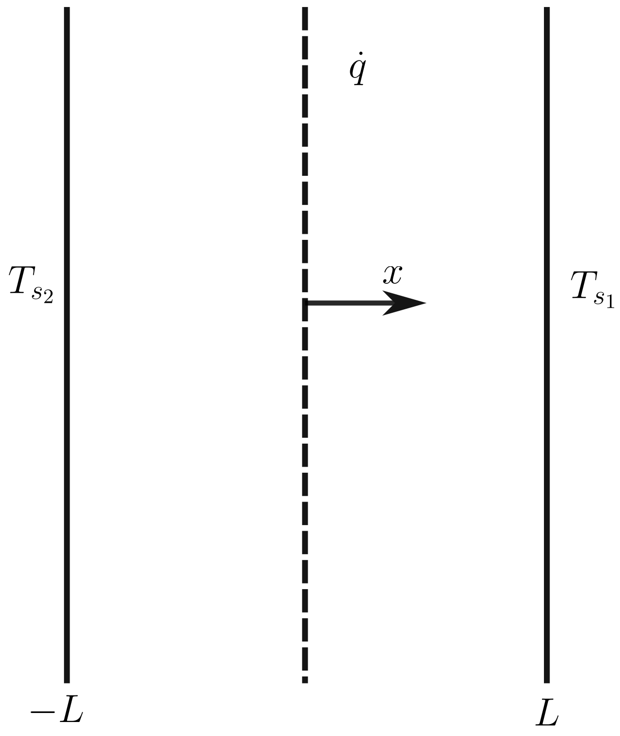
we have a plane wall with a heat source that is distributed uniformly inside the control volume, so some heat is going to be generated in this volume and on these two sides ($x = L$ and $x = -L$) it is open to, let us say atmosphere and we would like to solve the heat diffusion equation at steady-state for this system in which you have some amount of heat which is generated inside the control volume $\dot{q}$ for whatever reasons, it could be ohmic heating or it could be a heat source system which is distributed inside the control volume. If we assume it is a one-dimensional steady-state condition and this is a solid and if this is my x direction as in the picture, then $T \neq f(y, z, t)$. So the governing equation for conductive heat transfer in a plane wall system where the heat conduction is one-dimensional, it is only in the x direction and it is at steady-state is:

$$\frac{d^2T}{dx^2} + \frac{\dot{q}}{k} = 0$$

And let us assume that the boundary conditions which are available to us are $T(x = L) = T_{s_1}$ and $T(x = -L) = T_{s_2}$.

So when you solve these equations and these boundary conditions, the temperature profile that you are going to get is:

$$T(x) = \frac{\dot{q}L^2}{2k}\left(1 - \frac{x^2}{L^2}\right) + \frac{T_{s_2} - T_{s_1}}{2}\frac{x}{L} + \frac{T_{s_2} + T_{s_1}}{2}$$



So this is the complete profile of the temperature inside this solid. So if you see that for the case where the temperature at the boundaries are different. So if you have a symmetric situation in which the two end temperatures are identical or in other words, $T_{s_2} = T_{s_1} = T_s$, so we have:

$$T(x) = T_s + \frac{\dot{q}L^2}{2k}\left(1 - \frac{x^2}{L^2}\right)$$

The temperature distribution is symmetric around $x = 0$. So the maximum temperature in this case that I call as T_0, it is clear from the equation that the maximum temperature exists where $x = 0$, that means at the mid-plane and it would simply be:

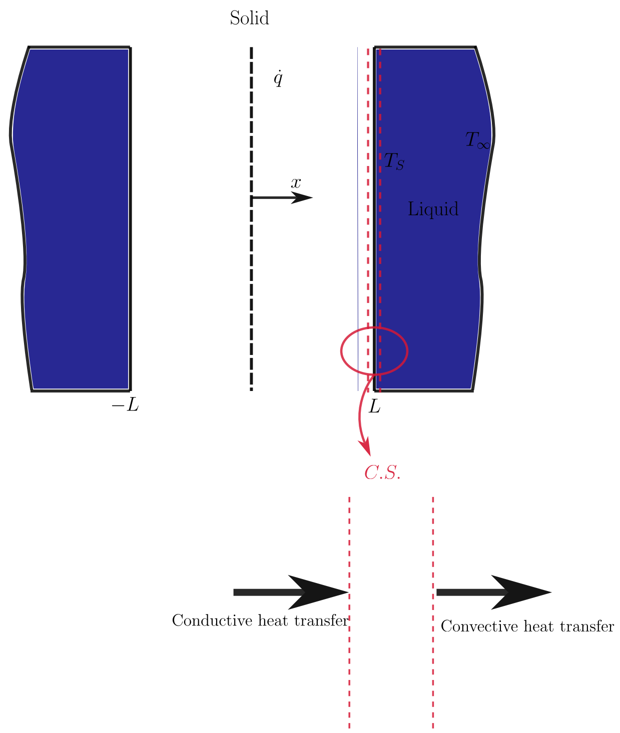
$$T_0 = T_s + \frac{\dot{q}L^2}{2k}$$

And if you just rearrange the two equations, what you get is:

$$\frac{T(x) - T_0}{T_s - T_0} = \left(\frac{x}{L}\right)^2$$

And at $x = 0$ plane, what you see here is that, $\frac{dT}{dx}|_{x=0} = 0$ can be called as an adiabatic surface. No heat travels or no heat losses, no heat flow process this plane in either direction. So this is truly an adiabatic surface.

In some cases this plane wall which is generating heat, the two end temperatures are difficult to evaluate. So if we have a situation in which the end temperatures of this wall are not known, however they are placed in a liquid, whose temperature, the surrounding temperature T_∞, this is known. So we need to express our new boundary condition that we are going to use must concentrate on the surface of the object. So we are going to take this surface as our control surface, and what we have done is whatever heat comes to the control surface must be equal to the heat that gets convected out of the solid into the liquid.



And at steady-state, the conductive and convective heat transfer must be equal. So if these two must be equal, what I can write for the conduction of heat towards the control surface would simply be:

$$-k\frac{dT}{dx}|_{x=L} = h\left[T_S - T_\infty\right]$$

where T_∞ is the temperature of the fluid at a distance far from the solid and T_S is the temperature of the wall that we do not know. So I am expressing T_S in terms of T_∞ by invoking the equality of conduction and convection at the control surface located at the junction of the solid and the liquid. So we already know the temperature profile:

$$T(x) = T_s + \frac{\dot{q}L^2}{2k}\left(1 - \frac{x^2}{L^2}\right)$$

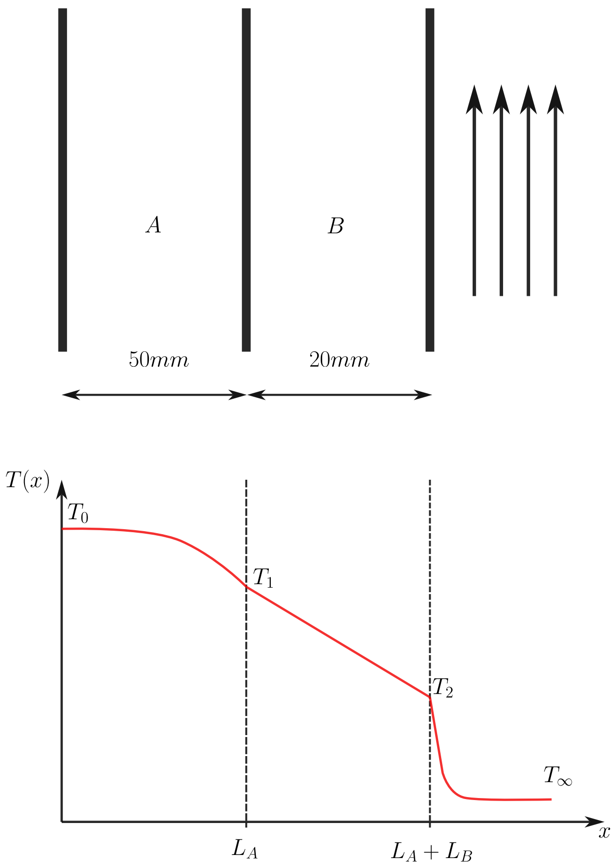
so when we do do the derivative of the temperature profile and evaluate it at $x = L$ and I put that in the previous relation I get:

$$T_S = T_\infty + \frac{\dot{q}L}{h}$$

1.4 Problem

I have two surfaces A and B and left side of A is insulated, which simply means no heat crosses from A to the outside and let us assume that the temperature at the left wall of A is T_0, the temperature at the right wall of A is T_1 and the temperature at the right wall of B is T_2. So we have to use your simple logic to find out whether or not T_1 is going to be more or less than T_2. The heat always travels from high temperature to low temperature and that would give you some indication.
I have some amount of heat which is generated in A that I denote as $\dot{q}_A = 1.5 \times 10^6 W/m^3$ with the thermal conductivity $k_A = 75\frac{W}{mK}$ with the thickness of the plane wall A equal to $50mm$. We have no heat genereted inside B, which means $\dot{q}_B = 0$ with the thermal conductivity $k_B = 150\frac{W}{mK}$ with the thickness of the plane wall B equal to $20mm$.
Outside of surface B, I have flow of air or flow of any liquid which is moving past this outer surface, and $T_\infty = 30řC$ and convection condition outside of B maintains a convective heat transfer coefficient of $h = 1000\frac{W}{m^2K}$.
What you have to do is to sketch the temperature distribution that exists at steady state condition in the system and find out T_0 and T_2.
If you look the solid A, you have a uniform generation of heat and no heat crosses the left side. At that wall, the convective heat must be 0 $-k\frac{dT}{dx}|_{x=0} = 0$.
Whatever be the profile of temperature, the temperature function must approach the left wall of A with a zero slope.
Since inside B there is no heat generation, than we have that $\frac{dT^2}{dx^2} = 0$ which would give rise to a linear temperature profile inside B. The slope of this linear profile would depend on the value of k_B. More the value of thermal conductivity, lesser is going to be the slope of the straight-line connecting T_1 and T_2. And if you have a very low conductive wall then this slope will be even more.
Beyond B, it is going to be only convection. And we know from our discussion in previous classes, that there would be a thin layer of liquid very close to the hot wall in which most of the transport processes are going to take place. Beyond that thin layer, nothing will happen, there would be no transport, no effective transport of energy from that point to the bulk. So the temperature T_2 will asymptotically reach T_∞ over a very short distance near the wall which is essentially the concept of thermal boundary layer. So the temperature of the solid wall in contact with the liquid, will approach the liquid

temperature over a very thin layer and over a very small distance from the solid wall itself. So there is going to be sharper drop of temperature in a region close to the outside of the wall and then the temperature will asymptotically reach the value of free stream/fluid which is moving at some velocity.



After calculating you get: $T_0 = 140$ř$C, T_1 = 105$řC.

1.5 Problem

heat conduction with an electrical heat source. This is a very common situation in which let us say I have an electrical wire through which a current is being passed, it is cylindrical in nature, the radius of this wire is equal to R. So this is an electrical wire, which obviously has a resistance and when current passes through it, there is going to be some amount of heat generation, and we will call this heat generation per unit volume through the electric wire with the symbol S_e.
We would like to find out from first principles what would be the solution of temperature,

the form of the temperature distribution inside such a wire.
What we have seen for the case of plane wall, now we are going to do it for a cylindrical system in which we consider the radial distribution of temperature. So, what we assume is that the temperature varies only with r, it does not vary with z or with θ and that we are dealing with a steady-state system.
So, the same way we have done for the case of shell momentum balance, here also I will have to assume a shell of some thickness, of some small dimension and write the corresponding terms of in and out and generation if any and then equate them to at steady-state to 0. And the same way for the case that we have done for the case of momentum transfer, the trick is to choose the smaller dimension of the control volume is to realise in which direction the temperature is changing.
Since the temperature is changing with r, my choice of the control volume should have a thickness which we can call it as Δr. So, this is a cylindrical shell which is coaxial with the axis of the electric wire and we are talking about this shell.
So, if you think of this cylindrical shell, then I am going to have some amount of heat which is coming in by conduction, the heat which is going out of the shell by conduction and some amount of heat will be generated in the control volume because of the heat generation term which is heated generation per unit volume. So, if we think $q_r^{''}$ which is the heat flux at r which is entering and $q_{r+\Delta r}^{''}$ is the heat flux at $r+\Delta r$ that is leaving this interface. The total amount of rate heat in would simply be:

$$\text{Total amount of rate heat IN} = (2\pi r L)q_r^{''}|_r$$

The total amount of rate heat out would simply be:

$$\text{Total amount of rate heat OUT} = 2\pi(r+\Delta r)Lq_r^{''}|_{r+\Delta r}$$

The rate heat generated per unit volume would simply be:

$$\text{Rate Heat Generated per unit volume} = (2\pi r\Delta r L)S_e$$

we know that:

$$\text{IN} - \text{OUT} + \text{GEN} = 0$$

The difference in question would definitely be converted into the differential equation:

$$\frac{d}{dr}\left(rq_r^{''}\right) = S_e r$$

So, this becomes the governing equation for heat transfer when you have generation of heat inside a cylindrical system. This can be integrated to obtain:

$$q_r^{''} = \frac{S_e r}{2} + \frac{c_1}{r}$$

where c_1 is the constant of integration.
At $r = 0$, $q_r^{''}$ has to be finite which implies $c_1 = 0$. So you get:

$$q_r^{''} = \frac{S_e r}{2}$$

and substitute Fourier's law which is:

$$q_r^{''} = -k\frac{dT}{dr}$$

you get:

$$-k\frac{dT}{dr} = \frac{S_e r}{2}$$

and you integrate it once to get:

$$T(r) = -\frac{S_e r^2}{4k} + c_2$$

And the second boundary condition could be that at $r = R$ that is the at the outer edge of the electrical wire, the temperature is some known temperatures T_0 and if you use this boundary condition, then the temperature distribution would simply be,

$$T(r) - T_0 = -\frac{S_e R^2}{4k}\left[1 - \left(\frac{r}{R}\right)^2\right]$$

The temperature distribution which is a function of r, T_0 is the temperature of the outside of the wire is a parabolic function of r and terms such as the thermal conductivity of the solid wire and the heat generation per unit volume due to the passage of current through it. So, it is a parabolic distribution and with the maximum temperature, T_{max} would obviously be at a point where $r = 0$, with value:

$$T_{max} = T_0 + \frac{S_e R^2}{4k}$$

If you are interested not only in the temperature at every point but some sort of an average temperature rise and all these averages are mostly area averages

$$< T > -T_0 = \frac{\int_0^{2\pi}\int_0^R [T(r) - T_0]\, r dr d\theta}{\int_0^{2\pi}\int_0^R r dr d\theta} = \frac{S_e R^2}{8k}$$

And the last part that remains is what is the heat flow at the outer surface? The heat flow at the outer surface is:

$$Q|_{r=R} = 2\pi r L q_r^{''}|_{r=R} = \pi R^2 L S_e$$

If you look carefully, is simply the volume of the control volume $\pi R^2 L$ and S_e is the heat generated per unit volume. So, the heat flow at the outer surface, must be equal to the heat generated. That is a condition which must be maintained in order to reach the steady-state. So, at steady state, all the heat which is generated inside the control volume must be conducted out of the control volume.

However the moment we have this is true for the case of conductive heat transfer, the moment we have convective heat transfer the system starts to get complicated and a simple shell balance will probably turn out to be inadequate in addressing slightly more complicated problems.

1.6 Problem

Let us assume that you have again a plane wall where one surface is maintained at T_0 located at $x = 0$ and the other surface is insulated located at $x = L$. And it says that at exposed surface (surface at $x = 0$) is subjected to microwave radiation which causes heat generation inside the wall (absorbed) according to the function of the heat generation:

$$\dot{q}(x) = \dot{q}_0(1 - \frac{x}{L})$$

The things that you have to find is derive equation for the temperature profile and second is obtain the temperature profile with appropriate boundary conditions.
So as before you can either think of a thin shell of thickness Δx and do the balance or you can directly write, since it is a conduction only case, you can directly write the governing equation:

$$\frac{d^2T}{dx^2} + \frac{\dot{q}}{k} = 0$$

Put in the expressions of heat generation, integrate, use the appropriate boundary conditions which I would not discuss and what you would get is, the answer you should get is:

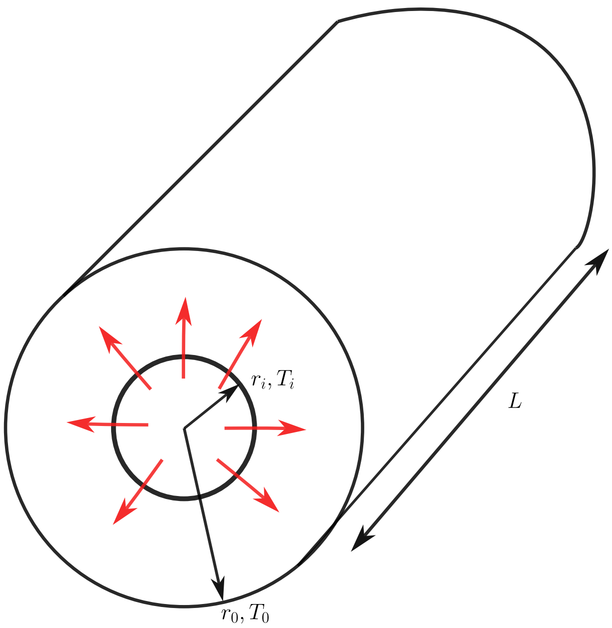
$$T(x) - T_0 = \frac{\dot{q}_0 L}{2k}\left[\frac{x}{L} - \frac{x^2}{L^2} + \frac{1}{3}\frac{x^3}{L^3}\right]$$

1.7 Temperature Distribution in Radial Systems

Temperature distribution at Steady-State in radial flow of heat. The mode of heat transfer is conduction but it cal always have convection at the outer or the inner edges of a radial of a wall. And therefore, it in that case it is going to become a convection-conduction problem, but for the time being let us start first with conduction only case in a radial system.

So, the figure that you see over here is that of a that of a hollow cylinder whose inner radius is equal to r_i and the outer radius is r_0 and the temperature inside is maintained at T_i whereas the temperature at the outside is maintained at T_0.

The length of the hollow cylinder is L and we are trying to find out what is the flow of heat through this area to outside. So, if we assume that T_i is greater than T_0, than the radial direction in which heat will flow through the solid material of the cylinder is shown if the figure.



For steady state condition, no heat generation and the case where $T = T(r)$, the heat diffusion equation for radial systems reduces to only this (governing equation):

$$\frac{1}{r}\frac{d}{dr}\left(kr\frac{dT}{dr}\right) = 0$$

one can integrate this equation and what you get is:

$$T(r) = c_1 ln(r) + c_2$$

where the two BC's are that $T(r_i) = T_i$ and $T(r_0) = T_0$. The final expression that we get is:

$$T(r) = \frac{T_i - T_0}{ln\left(\frac{r_i}{r_0}\right)} ln\left(\frac{r}{r_2}\right) + T_0$$

the temperature distribution unlike the case of planar system is not a linear function of position, but it is going to be a logarithmic function of the radial location of the plate. as you move in the direction towards the outer radius, the area available for heat flow continuously increases. So, since it increases the temperature is not going to be linear and temperature here you can see it is going to be linear function of position.

since we are trying to correlate cause and effect, the cause being the temperature difference and the effect being the heat flow and we are trying to find a relation or express our results in the form of something similar to ohms law between cause and effect. So, if you do that to this equation in the automatically the resistance to conductive heat transfer in radial systems will come out.

From the previous equation I can write:

$$\frac{dT(r)}{dr} = \frac{T_i - T_0}{ln\left(\frac{r_i}{r_0}\right)} \frac{1}{r}$$

using Fourier's law we have that the heat in the radial direction is:

$$q_r = -kA\frac{dT(r)}{dr} = \frac{-k2\pi rL(T_i - T_0)}{ln\left(\frac{r_i}{r_0}\right)} \frac{1}{r}$$

or in the same way:

$$q_r = \frac{k2\pi L(T_i - T_0)}{ln\left(\frac{r_0}{r_i}\right)}$$

So, if I bring $k2\pi L$ in the denominator what you have is something similar to ohms law, where the cause is the thermal potential difference which is expressed as temperature difference, and the effect is the radial flow of heat. the denominator simply is the resistance to heat transfer for radial systems

$$q_r = \frac{(T_i - T_0)}{\frac{ln\left(\frac{r_0}{r_i}\right)}{k2\pi L}} = \frac{(T_i - T_0)}{R_{th}}$$

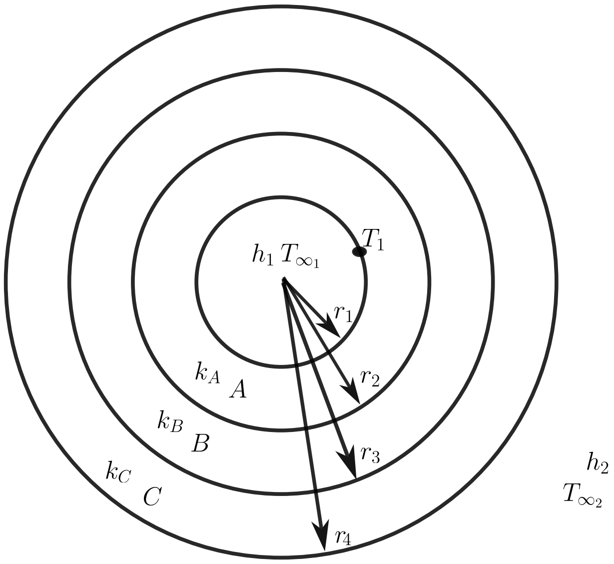
Let us assume that you have a composite situation in which you have a number of walls like this with the proper radius and thermal conductivity. You have convection inside as well as convection outside, with the inside condition given as the temperature of the of the fluid inside is T_{∞_1} which is creating a convection coefficient of h_1 at the inner surface

of the composite wall. On the other side, you have T_{∞_2} as the temperature of the fluid which is made to flow on the outside which is also producing convection coefficient of h_2. This is quite common in many of the practical situations for example, in heat exchangers which we would see where the hot fluid let us assume that it is flowing through a tube and somehow we it is going to come in contact in thermal contact with a cold fluid with which it will exchange heat, and therefore, the cold heat is going to be heated going to absorb going to gain energy out of this hot fluid. And let us see hot fluid, fluid is flowing from point a to point b from where it is produced to where let us say it is going to a reactor. So, in the transit you want to maintain the heat loss to a minimum you want to keep the heat loss to a minimum. So, how do you do that? you put insulation on top of that tube. So, this insulation the pipe and the insulation they are may be 2 or 3 different types of insulation one is an insulation which is going to protect heat loss and in order to protect the insulation you may have another outer cover.
As the heat travels from the inside to the outside it is going to experience different materials as it travels, and it is going to also see materials of different thermal conductivity. And as it travels it will see that the heat transfer area will keep on changing.
So, the radial flow of heat is going to be due to the temperature difference inside minus temperature difference outside, divided by the sum of all resistances (conduction and convection).
In the case of convective the ohm law can be derived in this way:

$$q_{r_1} = h_1 A \Delta T = h_1 2\pi r_1 L (T_{\infty_1} - T_1) = \frac{T_{\infty_1} - T_1}{\frac{1}{h_1 2\pi r_1 L}} = \frac{T_{\infty_1} - T_1}{R_{th}}$$

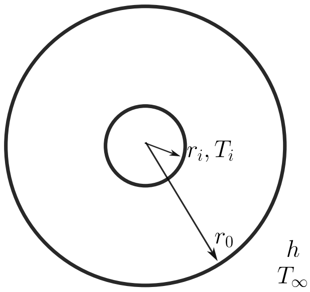
and so the total will be:

$$q_r = \frac{T_{\infty_1} - T_{\infty_2}}{\frac{1}{h_1 2\pi r_1 L} + \frac{ln\left(\frac{r_2}{r_1}\right)}{k_A 2\pi L} + \frac{ln\left(\frac{r_3}{r_2}\right)}{k_B 2\pi L} + \frac{ln\left(\frac{r_4}{r_3}\right)}{k_C 2\pi L} + \frac{1}{h_2 2\pi r_4 L}}$$



Now what I would do next is something very interesting is normally what we feel is that, if I am feeling cold I just wear a sweater which is nothing but an insulation and this reduces the loss of heat from my body to the ambient and I do not feel cold anymore. So, in the way the purpose of the sweater or the jacket is to ensure that the heat flow from my body to the ambient get slowered that is what is, that is what insulations do, they will use the flow of heat from the hot object to the cold object, but is it ever possible that by adding an insulation you are making more or higher flow of heat from the hot object to the cold object?. So, this is counter intuitive, but it may happen and I am going to show how under what conditions by adding insulation you simply increase the rate of flow of heat through that insulation. So, that is a very interesting concept it is known as the critical thickness of insulation. So, if your thickness of insulation is below a certain limit then by adding insulation you increase heat flow, if it is above the critical insulation thickness by adding insulation what you get is what we commonly expect that the heat flow rate reduces, but the interesting part is that region in which your size is below that of critical thickness of insulation so, by adding insulation you are increasing the loss of heat.

Let us say I have a solid cylinder and an insulation. The solid cylinder has radious r_i and let us assume that the temperature at the junction between the solid rod and the insulation is maintained at T_i. the insulation radius is r_0 and on the outside it is exposed to a convection environment with temperature T_∞ and heat transfer coefficient as h.



So, as before, I can write that the heat flow is simply going to be:

$$q_r = \frac{T_i - T_\infty}{\frac{ln\left(\frac{r_0}{r_i}\right)}{2\pi kL} + \frac{1}{h2\pi r_0 L}}$$

then I am going to see is it possible to mathematically get at which point my heat flow is going to be maximum?. So, if I can do that then I will probably get an idea of what is the concept of critical thickness of insulation.

I am going to differentiate this and set this to be equal to 0. So, I am trying to find out is there a radius r_0 where r_0 is the insulation thickness which would maximize the flow of heat which is denoted by q_r:

$$\frac{dq_r}{dr_0} = \frac{-2\pi L(T_i - T_\infty)\left[\frac{1}{kr_0} - \frac{1}{hr_0^2}\right]}{\left[\frac{ln\left(\frac{r_0}{r_i}\right)}{k} + \frac{1}{r_0 h}\right]^2} = 0$$

This means that the numerator should be equal to 0:

$$\frac{1}{kr_0} = \frac{1}{hr_0^2}$$

where we get:

$$r_0 = \frac{k}{h}$$

This r_0 is the critical thickness of the insulation which would maximize heat flow is known as the critical insulation thickness. there exists thickness when if you add insulation the heat flow will increase, go beyond that and you add insulation and the expected things will take place, that is the heat flow will decrease then why it should happen? this can only happen when the thickness is very small. So, when the thickness is very small by adding another layer of insulation you are increasing the resistance for flow of heat through that added layer of insulation; that means, you are increasing the conduction resistance by making the layer thicker; however, with putting the layer of insulation on

the outside you are making more area available for conduction because the area available for conduction is simply $2\pi rL$, where length is a constant. So, as r increases your area available for convection increases. So, with increase in r the conduction resistance will increase. So, the convective heat transfer is helped by adding insulation because of the additional area conduction heat transfer is going to be reduced by adding the insulation. So, these are 2 parallel mechanisms which compete with each other and for certain value of r as we have been previously the result is you get the maximum heat transfer. So, k for insulation material is roughly about $0.03\frac{w}{mK}$ and the convection h close to the wall outside, is mostly going to be the convection in air which is the order of $10\frac{W}{m^2K}$. So, the critical thickness of insulation is going to be about $3mm$.
when you think of very thin wires which conduct electricity, the diameter can be less than 3 millimeters. So, when current passes the heat is generated the ohmic heat is the joule, joule heat due to joule heating some amount of heat is generated and you want to dissipate that ok, but at the same time we do not want to live wire without any electrical insulation put on it. So, what you do is on the thin wire you put a layer of insulation, but what you get is something very interesting you not only make it safe for the wire to be safe since you have put an electrical insulation this electrical insulation is going to act as a thermal insulation as well, because the heat that is generated is going to dissipate to the atmosphere by means of conduction through the electrical insulation and convection to the outside. Now, if your radius of the electricity carrying wire is less than 3 millimeter, then you are enhancing the heat loss from the system by putting an electrical insulation on top of it. So, you serve 2 purposes, you cover the electrical wire with an insulation and by doing so, you are increasing the heat transfer and therefore, the wire itself now can be at a lower temperature which will probably prolong its life and will be less hazardous. So, critical thickness of insulation are mostly relevant in very thin electrical wires where you put an insulation on top of it, but for most of the practical systems since the dimension that we deal with are and the dimensions are more than $3mm$ you do not get that in most of the applications.

2 Shell Heat Balance

So, the analysis that we are going to see in this class is about a reaction, it could be an exothermic reaction or an endothermic reaction which is taking place in a fixed bed reactor.
it is a catalytic reaction, so the reactants would enter the fixed bed reactor and then the reaction would take place in presence of the catalysts and the products will leave. And if it is an exothermic or endothermic reaction, then the temperature is going to be

a function of the axial position. So for example if it is an exothermic reaction, then as we move into the fixed bed reactor, the temperature of the reactant product mixture will progressively increase. And in order to give it a general flavour, it has been assumed that at the beginning and at the end of the reactor we also have an identical reactor attached to it. But these two reactors will not contain any catalyst particle. However, these reactors are filled up with inert particles of the same size, same porosity, same arrangements as those of the catalyst particles in the actual fixed bed reactor.

So the figure that we would use to solve the problem is that we have this catalyst bed reactors in which the blue ones are the catalyst particles, so the reactor starts at $z = 0$ and up to a length of $z = L$, the reactor is filled with catalyst particles and then from L onwards, we have certain length, a considerable length of the reactor in which inert particles of the same size, type, porosity, etc. of the catalyst particles are packed in there. Similarly, prior to $z = 0$, all the way up to $Z = -\infty$, mathematically speaking, that means whatever the large length, there would be another portion of the reactor in which all these particles, same size as those of the catalyst particles, same arrangement and so on, these are all inert particles. So, we have a reaction that is taking place from $z = 0$ to $z = L$ and the reactants come through a section of the reactor in which no reaction takes place and then once the reaction zone is over, the products would travel through similar reactor but without any reaction and go out of the reacting system.

So, we will assume that the temperature of the reactants which enter, they are entering at T_1 and it has been mentioned that the side walls of the reactor are perfectly insulated, so we are not going to have any heat loss or gain from the surrounding. So therefore the reactants, no reaction in zone 1, reaction endo or exothermic in zone 2 and again no reaction in zone 3.

So, we would assume that there would be some amount of generation or consumption of heat which has been denoted by S_c. So, S_c is the volumetric heat generation if it is an exothermic reaction that is taking place inside the reactor containing the catalyst particles. And our job is to find out $T(z, \text{other parameters})$ and of course the other parameters would consist of the temperature at which the reactants are coming in, the amount of heat generation if it is an exothermic reaction (due to reaction) and few other properties/other conditions, such as what is a flow rate of the reactants that is coming in and other factors. But principally we would like to find out how does temperature varies axially in the reactor and whether having an exothermic reaction in the zone 2 would affect the temperature profile of the reactants entering the reaction zone or the temperature profile of the products leaving the reaction zone.

Now if you look into this problem, you would realise that we are going to use, if we have to use a shell momentum balance, in the shell momentum balance, we are not only going to have conduction, we also are going to have convective heat transfer, because there is a flow from left to right and this flow carries with it some amount of energy, so if I assume the control volume, somewhere in between, somewhere in the reaction zone, then some amount of energy will enter the control volume because of flow, some amount of energy will enter because of conduction, there would be a generation of heat inside the control volume due to the reaction and there would be two out terms, the convection out and the conduction out. if we consider steady-state conditions, then the algebraic sum of all these terms must be equal to 0.

So the first thing, certain basic assumptions which one has to make in order to simplify the problem. We realise that this is a heterogeneous system, so any property that we refer to, is going to be the property of the catalyst or the property of a gas mixture which flows past this catalyst particles. For example let us take thermal conductivity. So when we say thermal conductivity, which thermal conductivity are we talking about? Is it that of the gas, that of the catalyst particles or some sort of combination of the catalyst particles and the gases which are flowing past the catalyst. So whenever we refer to any physical property in this, we are referring to some sort of an average property, for example the average value of thermal conductivity and so on. We also realise that this is a heterogeneous system, so somewhere you are going to have a solid particle, somewhere you are going to have a gas and so on. If you reduce the system size to a differential element, then whatever we refer to in this specific problem would refer to some sort of a radially average property.

So, the other assumption, that we are going to refer in making the balance equation, all the properties are radially averaged properties.

Secondly, we will assume the temperature is not a function of r, temperature is a function only of z, that is the axial position. And having the walls of the reactor perfectly insulated, essentially helps us in arriving or in using this condition.

So if we are going to use a shell, then of course we have to use a shell whose smaller dimension would be the direction in which the temperature is changing. So, the shell across which we are going to make our heat balance, across which we are going to write the conservation equation will have a thickness equal to Δz, however, it extends all the way up to R.

So, if we take this and draw a cylindrical shell of size Δz in thickness, you have flow of reactants from left and products will go out from right and the reaction is taking place inside this shell. So if you consider this, then we have to first identify what are the mechanisms by which the heat can enter or leave.

So, when you think of a circular disk like shell which we have assumed through left side, I am going to have some amount of conduction, which, let us say we represent as q_z. So that is a conductive heat flux which is entering the disk shaped control volume which we have assumed. So, the amount of heat which comes in by conduction through the face of the disk, the left-hand side of the disk would simply be area times conductive heat flux. So, the amount of heat or thermal energy which comes in by conduction would simply be:

$$\text{Thermal energy in by conduction} = \pi R^2 q_z^{''}|_z$$

And the thermal energy out by conduction, would simply be:

$$\text{Thermal energy out by conduction} = \pi R^2 q_z^{''}|_{z+\Delta z}$$

And next we would see what is the thermal energy in by convection. And when we mention convection, what I refer to is essentially due to the flow of the stream, the reactants which are coming in to this shell. So the amount of material which comes into the shell must be equal to in volumetric terms the area multiplied by the local velocity. the total volume of reactant mass which is entering the disk-shaped control volume due to flow is $\pi R^2 v_z|_z$. I can convert it in terms of mass, I simply multiply it with density. So, the mass flow rate of reactants entering the control volume would simply be:

$$\text{mass flow rate in by convection due to flow} = \dot{m} = \rho \pi R^2 v_z|_z$$

That is the mass flow rate which enters due to convection or due to flow. Now what is the thermal energy which enters because of the entry of this amount of mass into the control volume?. The simple expression is $\dot{m} c_p \Delta T$, where c_p is the heat capacity and $\Delta T = T(z) - T_0$ is the temperature of the stream that enters the control volume minus a reference temperature, it could be any reference temperature but we have to be consistent in its use throughout our analysis. So, energy is always expressed with respect to datum, so therefore the amount of convective heat that enters due to convection would simply

$$\text{Thermal energy in by convection due to flow} = \rho_1 \pi R^2 v_1 c_p (T(z) - T_0)|_z$$

where T0 is some datum temperature which we are going to use for calculation of the energy content of the stream.

Now I could have used velocity at z but what I have done is I have expressed ρ_1 at the entry conditions, velocity, v_1 at the entry condition. And since continuity has to hold at every section of this reactor system, so whatever be the mass flow rate that is entering, it should remain constant so the mass conservation will always be there as you move in this direction. So even if I reaction takes place, mass is always going to be conserved, moles may not be conserved because that would depend on the stoichiometry of the equation but mass will always be conserved at every section of the reactor that I have drawn. So, if I express my mass flow rate in terms of the entry conditions, the same mass flow rate will flow through every section of the reaction system.

$$\text{Thermal energy out by convection due to flow} = \rho_1 \pi R^2 v_1 c_p (T(z) - T_0)|_{z+\Delta z}$$

The thermal energy produced due to reaction would simply be, S_c where it is the volumetric generation of heat so I multiply it with the volume of the of the control volume that we have chosen, volume of the disk which. So, this is rate of heat generation per unit volume, so when you multiply rate of heat generation per unit volume with the volume of the system, what you would get is simply the rate of heat generation:

$$\text{Thermal energy produced} = \pi R^2 \Delta z S_c$$

And therefore if I take the algebraic sum of these, the balance equation at steady-state is:

$$\text{heat in} - \text{heat out} + \text{heat generated} = 0$$

And then as you can see, πR^2 will cancel from all the terms and you are going to divide both sides by Δz, so what you have then is:

$$\frac{q_z^{''}|_{z+\Delta z} - q_z^{''}|_z}{\Delta z} + \rho_1 c_p v_1 \frac{T|_{z+\Delta z} - T|_z}{\Delta z} = S_c$$

So this is the difference equation that describes the net flow of heat by conduction and the net flow of heat by convection and the heat generation term. So you take in the limit as $\Delta z \to 0$, the governing equation is

$$\frac{dq_z^{''}}{dz} + \rho_1 c_p v_1 \frac{dT}{dz} = S_c$$

we are assuming that it is a continuum model, but we understand that the resulting equation describes average values of q_z, T, v_1, etc. So, in order to use $\Delta z \to 0$, the system has to be be uniform. But we realise the system is not uniform, it has catalyst particles, then it has a void space and so on, so we are imposing a continuum condition as $\Delta z \to 0$, that means all the values that we refer to it this equation are essentially average values. This is something which we have to keep in mind.
So, we have this equation and then when you substitute $q_z^{''}$ by Fourier's law, what you get is:

$$-k_{z_{\text{eff.}}}\frac{d^2T}{dz^2}+\rho_1 c_p v_1\frac{dT}{dz}=S_c$$

I would I would say that k is the effective because it considers both the solid particles as well as the void space. So the equation will have different forming in zone 1, zone 2 and zone 3.

In zone 1 and in zone 3, there would not be any heat generation term, so it would simply be:

$$-k_{z_{\text{eff.}}}\frac{d^2T'}{dz^2}+\rho_1 c_p v_1\frac{dT'}{dz}=0$$

$$-k_{z_{\text{eff.}}}\frac{d^2T'''}{dz^2}+\rho_1 c_p v_1\frac{dT'''}{dz}=0$$

In zone 2:

$$-k_{z_{\text{eff.}}}\frac{d^2T''}{dz^2}+\rho_1 c_p v_1\frac{dT''}{dz}=S_c$$

And the boundary conditions that we can use are:

$$\begin{aligned}
T^{'} &= T_1 && \text{at } z=-\infty\\
T^{'} &= T^{''} && \text{at } z=0\\
k_{z_{\text{eff.}}}\frac{dT^{'}}{dz} &= k_{z_{\text{eff.}}}\frac{dT^{''}}{dz} && \text{at } z=0\\
T^{''} &= T^{'''} && \text{at } z=L\\
k_{z_{\text{eff.}}}\frac{dT^{''}}{dz} &= k_{z_{\text{eff.}}}\frac{dT^{'''}}{dz} && \text{at } z=L\\
T^{'''} &= \text{finite} && \text{at } z=\infty
\end{aligned}$$

where the apex means the zone.

So these are essentially continuity of temperature and heat flux.

To solve this problm we have to non-dimensionalized the equation. So what is done is the new dimensionless numbers are:

$$Z=\frac{z}{L}\ \ ,\ \ \theta=\frac{T-T_0}{T_1-T_0}\ \ ,\ \ B_0=\frac{\rho v_1 c_p L}{k_{z_{\text{eff.}}}}\ \ ,\ \ N=\frac{S_c L}{\rho v_1 c_p (T_1-T_0)}$$

Let's apply these to the governing equation in zone 2. dividing by $k_{z_{\text{eff.}}}$ and substituting the dimensionless variables, we get:

$$-\frac{1}{L^2}\frac{d^2T''}{dZ^2}+\frac{\rho_1 c_p v_1}{k_{z_{\text{eff.}}}L}\frac{dT''}{dZ}=\frac{S_c}{k_{z_{\text{eff.}}}}$$

Now, let's express the derivatives of T'' in terms of θ:

$$\frac{dT''}{dZ} = (T_1 - T_0)\,\frac{d\theta''}{dZ}$$

$$\frac{d^2T''}{dZ^2} = (T_1 - T_0)\,\frac{d^2\theta''}{dZ^2}$$

Substituting these into the equation and also using the definitions of B_0 and N, we obtain the dimensionless governing equation for zone 2:

$$-\frac{1}{L^2}(T_1 - T_0)\frac{d^2\theta''}{dZ^2} + \frac{B_0}{L^2}(T_1 - T_0)\frac{d\theta''}{dZ} = \frac{\rho_1 v_1 c_p (T_1 - T_0) N}{L k_{z_{\text{eff.}}}}$$

Simplifying by multiplying by $L^2/(T_1 - T_0)$, we get:

$$-\frac{d^2\theta''}{dZ^2} + B_0\frac{d\theta''}{dZ} = B_0 N$$

The characteristic equation for the homogeneous part of this second-order linear non-homogeneous differential equation is:

$$-m^2 + B_0 m = 0$$

which gives the roots:

$$m(B_0 - m) = 0 \implies m_1 = 0, \quad m_2 = B_0$$

Therefore, the homogeneous solution is of the form:

$$\theta_h^{(2)} = Ae^{0\cdot Z} + Ce^{B_0 Z} = A + Ce^{B_0 Z}$$

Now we need to find a particular solution $\theta_p^{(2)}$ for the non-homogeneous equation:

$$-\frac{d^2\theta''}{dZ^2} + B_0\frac{d\theta''}{dZ} = B_0 N$$

Since the right-hand side is a constant, let's assume a particular solution of the form:

$$\theta_p^{(2)} = DZ$$

Then, $\frac{d\theta_p^{(2)}}{dZ} = D$ and $\frac{d^2\theta_p^{(2)}}{dZ^2} = 0$. Substituting this into the non-homogeneous equation:

$$-0 + B_0 D = B_0 N$$

If $B_0 \neq 0$, then $D = N$. So the particular solution is $\theta_p^{(2)} = NZ$. Therefore, the general solution for $\theta^{(2)}$ in zone 2 is:

$$\theta^{(2)}(Z) = A + Ce^{B_0 Z} + NZ$$

The same procedure can be done to solve the other two equations, and after applying the boundary conditions we get:

$$\begin{aligned}
\theta^{(1)}(Z) &= 1 + \frac{N}{B_0}\left(\frac{e^{B_0} - 1}{e^{B_0}}\right) e^{B_0 Z} \\
\theta^{(2)}(Z) &= \left(1 + \frac{N}{B_0}\right) - \frac{N}{B_0 e^{B_0}} e^{B_0 Z} + NZ \\
\theta^{(3)}(Z) &= 1 + N
\end{aligned}$$

so the temperature in the zone 3 remains constant.

2.1 Viscous dissipation

In this class we are going to solve another problem which is quite common in many of the bearing systems. So, let us say we have two cylindrical elements, coaxial, one inside the other and there is a very thin gap in between. One of them, let us say the outer one is rotating at a higher angular velocity while the inner one is stationary, so a stationary inner cylinder and a rotating outer cylinder.

It is a very common occurrence in many applications, in order to reduce the friction in between the two cylinders, one of the most common ways to reduce the friction is to fill the gap in between the two cylinders by a properly chosen lubricant. So, the lubricant essentially reduces the friction between the two cylinders. The choice of the lubricant in this is very important. And let us also assume that the outer and the inner cylinders are maintained at two different temperatures.

So, suppose the two cylinders, none of the cylinders is moving, then simply I am going to have a temperature of the outer cylinder and a temperature of the inner cylinder. The gap between the two cylinders is too small and we have seen before while working with the problems of momentum transfer that if the separation between the two surfaces is very small in comparison to the radius or in other words, if the curvature of the system is not too small, then you can convert a radial system into a planar system.

So, the cylindrical system where we talk about two cylinders and the gap between them is extremely small, we can simplify the system by simply opening up the cylinder and making it as if it is a system of two parallel plates separated by the small distance.

So, this is how we have done some of the problems of the cylindrical bearings and if one of the cylinders is moving, then a Couette type flow will be established in the intervening space between the two cylinders.

So first of all, if the radius of the cylindrical system is large in comparison to the gap in between the two cylinders, then I can simply cut open the cylinders and make them as if they are two parallel plates with one plate moving with some velocity, the other is stationary. And any liquid in between, in between in the intervening space is simply going to going to have a Couette type flow because of the motion of the top plate.

So, in this specific case we can also treat the system as if it is a case of flow between two parallel plates, a lubricant which is placed between two parallel plates, one plate is moving with some velocity and the temperatures of the two plates are different.

So, the difference in this problem as compared to the problem that we have done in fluid mechanics is that here is that here the temperatures of the two plates are different.

If the temperatures of the two plates are different and if the gap in between them is very small, then viscous forces will ensure that there is going to be a negligible effect of convective heat transfer and most of the heat transfer between the two plates due to the difference in temperature will be due to conduction.

So, this is a situation in which conduction will prevail and the entire problem can be thought of as if it is a flow between two parallel, which are maintained at two different temperatures with a liquid in between where there is no convection. So, if it is a conduction problem, then we know that in absence of any heat generation in the liquid in between, it is simply going to be linear distribution of temperature, because in this case there is no variation of temperature with z, no variation with y, only with respect to x the temperature will change. So, if I think of the equation that describes conductive heat transport at steady-state in absence of any heat generation, we simply have:

$$k\frac{d^2T}{dx^2} = 0$$

and this would give rise to a linear temperature profile and the two constants of the profile can simply be evaluated by through the use of boundary conditions that at one plate the temperature is T_0 and on the other plate the temperature is T_1.

Now whenever we have a fluid, lubricant which is placed in between two rotating surfaces and the gap is small, what you see is that there would be a very strong velocity gradient present in the system. the velocity is zero at the bottom plate and the top plate moves at a very high velocity and the gap in between them is very small. So, the velocity gradient, which is in this case:

$$\frac{V-0}{x-0} = \frac{V}{x}$$

V is large and x is small, so the value of the velocity gradient would be very large. If the value of the velocity gradient is large, then the adjacent layers would start to slip past one another, move past one another with a very high relative velocity difference. Now whenever if you think of a solid object which is being pulled over another solid surface with some velocity, you are going to raise the temperature of the solid block due to friction. So, friction will ensure that the frictional losses will manifest itself into a temperature rise of both these solids. So, these two surfaces are going to have an increase in temperature due to solid friction. The same thing we can also or may also take place for the case of liquids when the layers in laminar flow slip past one another at a very high velocity. So, this is something which can loosely be called as liquid friction. And this type of frictional heat generation is quite common. It is a volumetric heat generation, due to friction. So, this kind of volumetric heat generation will have to be taken into account whenever you have the velocity gradient present in the system is very high.

So, we do not see the heat generation effect during the flow of the liquid quite often. But only in very special cases, for example in the flow of the lubricant or when a spacecraft re-enters the Earth's atmosphere, the velocity gradient is so high that you get substantially high generation of heat and the entire spacecraft will glow red due to the temperature increase. So, in this specific case we understand that the temperature profile is going to be linear but due to the volumetric heat generation due to viscosity its linear nature of the distribution will no longer remain linear.

So, we have to recall, in the governing equation itself we cannot now neglect $\dot{q}$ which is the heat generation per unit volume. The heat generation which is due to viscosity, the heat generation due to the property of the fluid which resists the motion of adjacent layers. So that is why it is called the viscous heat generation.

$$k\frac{d^2T}{dx^2} + \dot{q} = 0$$

And we know that since the top plate is moving with a constant velocity, the velocity profile would also be linear:

$$v_z = \frac{x}{b}V$$

This is the axial velocity profile imposed by the motion of the top plate.

Now, next is you have to think of shell, since the velocity is varying in the x direction, my shell is going to be of size Δx, it could be any area A, heat is going to come in, it is a conduction only process, no convection is to be taken into account, And this is $q_x^{''}$, and some amount is going out $q_{x+\Delta x}^{''}$ and due to friction, let us say some amount of volumetric heat generation is present which is denoted by S_v. So, at steady-state:

$$q_x^{''}A|_x - q_{x+\Delta x}^{''}A|_{x+\Delta x} + S_v A\Delta x = 0$$

So, this equation can now be expressed as in terms of the differential equation:

$$\frac{dq_x^{''}}{dx} = S_v$$

Now this, the expression for S_v, the volumetric generation of heat, can be expressed as the velocity gradient square:

$$S_v = \mu\left(\frac{dv_z}{dx}\right)^2$$

I will not be able to explain why this is so unless and until we derive equation of energy. Until and unless energy equation is introduced, the form of the viscous heat generation due to the presence of a velocity gradient would not be clear to you.
So we can write the eqaution as:

$$\frac{dq_x^{''}}{dx} = \mu\left(\frac{dv_z}{dx}\right)^2$$

but since:

$$v_z = \frac{x}{b}V \Rightarrow \frac{dv_z}{dx} = \frac{V}{b}$$

And so the equation becomes:

$$\frac{dq_x^{''}}{dx} = \mu\left(\frac{V}{b}\right)^2$$

The presence of the viscous heat generation essentially couples two equations, the momentum transfer equation and the heat transfer equation get coupled because of the presence of velocity or velocity gradient in the energy equation. So, which also means that you have to solve for the velocity profile, you have to solve the momentum equation first before you can attempt to solve the energy equation.
So, the coupling that we see here is one-way coupling, that is energy equation is coupled to the momentum equation but if you look at the momentum equation, the momentum equation is not coupled with the energy equation.
So, in most of the cases, you would see the presence of one-way coupling. You have to solve for the momentum equation first, get the velocity profile and then derive the energy equation where a velocity expression would arise either because of the presence of convection which we are not considering at this moment or the coupling due to the presence of the viscous dissipation term.
So viscous dissipation term appears only in specialised cases where the velocity gradient is very large, in most of the normal ordinary energy equations we do not need to include

that term. But if we do include, then the velocity expression must be obtained up a-priori before we attempt to solve the energy equation. So as long as the thermo physical properties of the system remain constant, there will always be a one-way coupling between momentum transfer and heat transfer. But if the properties start to change, then those equations, the momentum equation and the energy equation will have to be solved simultaneously. But as long as the equations are coupled in only one direction, you need to solve independently the expression for velocity and then plug that in to the energy equation that we have just derived which is the case here in.

So, with this I would be able to express this in terms of Fourier's law and with the Fourier's Law, the temperature expression can be obtained:

$$T = -\frac{\mu}{k}\left(\frac{V}{b}\right)^2 \frac{x^2}{2} - \frac{c_1}{k}x + c_2$$

And the two boundary conditions that we have are

$$\begin{aligned} T &= T_0 && \text{at } x = 0 \\ T &= T_b && \text{at } x = b \end{aligned}$$

With these boundary conditions, this temperature profile be written in dimensionless form as:

$$\frac{T - T_0}{T_b - T_0} = \frac{x}{b} + \frac{1}{2}Br\left(\frac{x}{b}\right)\left[1 - \frac{x}{b}\right]$$

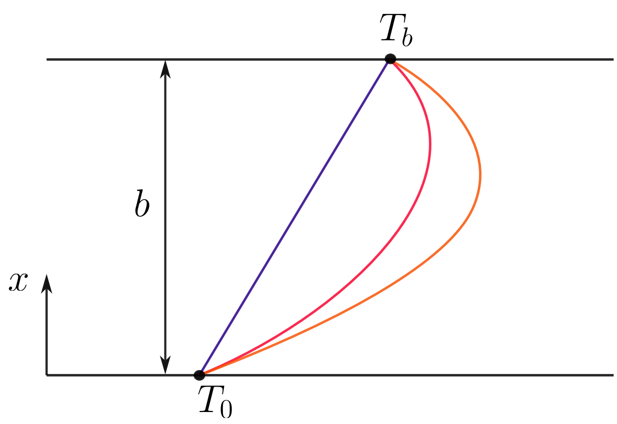
This B_r, is known as the Brinkman number, which is defined as:

$$Br = \frac{\mu V^2}{k\left(T_b - T_0\right)}$$

So, the Brinkman number essentially tells you how far viscous heating is important relative to the heat flow from the imposed temperature difference.

If you do not have any viscous heat generation in the system, what is going to happen to the expression that we have just derived? If you do not have any heat generation present in the system, the entire second term on the right-hand side would be zero, and what we would get is a linear distribution of temperature. Since you have viscous heat generation where the Brinkman number essentially tells you the importance of the generation of viscous heat with respect to the heat that would flow because of an imposed temperature difference, I am going to have a non-linear term present in the expression for temperature. So, Brinkman number tells me how important viscous heating is. And if viscous heating is important, the value of Brinkman number would increase and an interesting thing can be seen for a value of $B_r > 2$. If value of $B_r > 2$ then you would see the existence of a maximum temperature between the top and the bottom plate.

So here I have two plates, the top one is at T_b and the bottom is at T_0. Normally I would get a temperature profile in blue. So, the maximum temperature would be at the temperature of the top plate. But as the effect of viscous heating starts to become important, that means the value of Brinkman number starts to increase, there would be a value of $Br > 2$, and for all values of Brinkman number greater than 2, the profile would probably look something like in red or orange (if Br increase again). That means the maximum temperature which was at the top plate when Brinkman number is 0 or < 2, and for these cases in red, the $Br > 2$, as it progressively becomes more and more, the maximum is going to be somewhere in between the top plate and the bottom plate.



So now comes the question of the selection of the lubricant. Each lubricant has a specific value of temperature up to which it will retain its lubrication properties. So, if it is within the lubrication zone, if it is within that temperature, the lubricant will work perfectly. But if for some reason the temperature of the lubricant exceeds that of the that of the higher temperature surface and it will keep on increasing as the relative velocity between the two plates, then you may get a temperature which is more than the safe operating temperature of the lubricant. before you choose the lubricant, you first find out what is the temperature at which the lubricant can work safely. And then try to solve the problem with the known velocity differences between the top and the bottom plates and see what is the maximum temperature that can be attained by the lubricant due to the motion of one of the plates. So, the frictional heat generation plays a critical role in the choice of the lubricant. And for the performance of the lubricant, what kind of a velocity difference in a lubricant can sustain that is something which one has to consider before choosing the lubricant And Brinkman number would tell you, whether or not you are going to get a higher temperature in the lubricant as compared to any of the two temperatures of the two solid plates which are in motion. So viscous heat dissipation is can be important in some applications.

2.2 Transient conduction

let's say this is a coolant liquid whose temperature is T_∞, it is a very large tank in which the temperature does not change. And I have an object, it is a spherical ball whose temperature is T_i. At $t < 0$ the temperature of the solid object is equal to T_i. Then you drop the ball into the coolant liquid, then what you see is that the temperature of the solid object is now a function of time. So, any $t > 0$, the temperature would simply be a function of time. So as long as the temperature is going to be a function of time, this is a transient conduction problem, where the temperature is a function of time as well as the temperature could be a function of x, y, z.
Any problem that deals with the temperature, where the temperature can vary with respect to time is commonly termed as the transient conduction. Spatial as well as time-dependent, it is called the transient conduction.
Now, if we think of a spherical ball which is dropped and the liquid coolant at a temperature is lower than the temperature of the hot spherical ball, then if I write the conservation equation as:

$$\dot{E}_{in} + \dot{E}_g - \dot{E}_{out} = \dot{E}_{st}$$

there is no $\dot{E}_{in}$, that means no rate of heat that come into the spherical ball because the nergy of the ball is higher than the surrounding fluid, there is $\dot{E}_{out}$, there is no $\dot{E}_g$, there is $\dot{E}_{st}$, there is a change in the energy stored in the spherical object because it is now in contact with the cooler liquid. So, the governing equation, which would describe transient conduction is:

$$-\dot{E}_{out} = \dot{E}_{st}$$

The $\dot{E}_{out}$ is the rate of energy that goes out of the spherical balls is mostly by convection:

$$-hA(T - T_\infty) = \rho V c \frac{dT}{dt}$$

where ρ is the density of the solid, V is the volume of the solid, c is the thermal capacity of the solid. If we define $\theta = T - T_\infty$ then this equation takes the form as:

$$\frac{\rho V c}{hA_s} \frac{d\theta}{dt} = -\theta$$

Now one thing has to be mentioned here is that the assumptions we can make is that $T = f(t), T \neq f(x, y, z)$. If this assumption is valid, then the governing equation can simply be transformed to this and can be integrated. But the assumption that the temperature of the solid object is a function of time but it is not a function of positions,

this is called the Lumped capacitance model.
This allows me to simply integrate the equation with respect to time while assuming that at any point of the solid object is space wise isothermal. That is the temperature of the solid ball is going to vary depending on time but I take a specific time, there is no variation of temperature inside the solid ball. Its centre at a point halfway between the centre and the periphery, all these temperatures are the same. If that assumption is valid, then it would be easy to solve the problem of transient conduction and the assumption that the temperature of the solid object is space wise isothermal, it depends only on time, this assumption is known as the lumped capacitance model.

So when I use this lumped capacitance model, the question would obviously come is, when we can say that the lumped capacitance model is valid? What exactly happens in the case of lumped capacitance or when it would be prudent to use the lumped capacitance model? let us consider a solid object which is in contact with a liquid, the temperature of the solid at $x=0$ is T_{s_1} and the temperature at the other end of the solid which is in contact with the liquid at $x=L$ is T_{s_2} and the temperature of the liquid far from the solid wall is T_∞. There is a flow of a fluid past the solid object which would carry heat away from the solid because of convection. So, the boundary $x=L$ of the solid is experiencing a convective heat transfer process due to its interaction with the fluid and the flow of the liquid maintains a convective heat transfer coefficient denoted by h. it is further assumed that $T_{s_1} > T_{s_2} > T_\infty$.
So, if I do this surface energy balance at $x=L$, what I can say is that at steady-state, the amount of heat which comes from the location $x=0$ at a temperature T_{s_1} to a location $x=L$ which could be a temperature of T_{s_2}, and the conductive flow of heat would simply be:

$$q_{\text{ cond.}} = \frac{kA}{L}\left(T_{s_1} - T_{s_2}\right)$$

must be equal to the heat convected out:

$$q_{\text{ conv.}} = hA\left(T_{s_2} - T_\infty\right)$$

and so we have:

$$\frac{kA}{L}\left(T_{s_1} - T_{s_2}\right) = hA\left(T_{s_2} - T_\infty\right)$$

I can also write it in this way:

$$\frac{\left(T_{s_1} - T_{s_2}\right)}{\left(T_{s_2} - T_\infty\right)} = \frac{\frac{L}{kA}}{\frac{1}{hA}} = \frac{R_{\text{cond.}}}{R_{\text{conv.}}} = \frac{hL}{k} = \frac{hL_c}{k}\text{Bi}$$

where Bi is the Biot number and is a dimensionless number and L_c is the characteristic length.
in order to have less temperature gradient between in the solid, the value of the Biot number must be small. So, if your conduction resistance is quite small in comparison with the convective resistance, then the temperature drop in the solid is going to be small. In order for LC model to be valid, you would like to have small $T_{s_1} - T_{s_2}$, the temperature drop in the solid. Therefore, in order for LC to be valid, Bi must be small. So, a small value of Biot number essentially tells us that we can simplify the system using the lumped capacitance model. So Biot number has to be quite small in comparison to 1 ($Bi << 1$) and the small value of your number is an indication of whether or not we can used lumped capacitance model. Normally $Bi < 0.1$. experimental values suggest that one can safely use lumped capacitance model if the Biot number is less than 0.1.
So, while trying to solve any transient conduction problem, the first thing one should do is to check what is the value of the Biot number. If it is less than 0.1, then the radiation of temperature inside the solid object or the spatial distribution of temperature inside the solid can be neglected and therefore at any instant of time the solid can be treated as space wise isothermal. At every point the temperature would be the same if I fix the time. But obviously with time, the temperature of the solid will change and depending on whether or not it is exposed to a hot environment or a cooler environment, the temperature may reduce or may increase, however it will remain spacewise isothermal as long as $Bi < 0.1$.
Normally $\frac{hL}{k}$, we term it, the more common one is the Nusselt number. So, what is the difference between Nusselt number which is also explained by Biot number. The difference between the two is that for Nusselt number, the k that you have in the denominator of the expression refers to that of the fluid, whereas in the Biot number the k that you have is the thermal conductivity of the solid. So depending on what you use, whether it is the thermal conductivity of the solid or the thermal conductivity of the surrounding liquid, you either have Biot number or you have Nusselt number. Biot number therefore plays a very important role in transient conduction and the value of which would let you know whether or not lumped capacitance model is valid and can be used.
the characteristic length is defined as:

$$L_c = \frac{V}{A_s}$$

where V is the volume of the solid and A_s is the surface area.
Therefore, a simple geometry would tell you that for a plane wall of thickness $2L$ with convection from both sides, $L_c = L$ where L is simply the half of thickness.
If you take a long cylinder, $L_c = \frac{r_0}{2}$ where r_0 is the radius of the cylinder.
For the case of a sphere, $L_c = \frac{r_0}{3}$.

But the common practice is to make your assumption more conservative, the length scale is chosen across which the maximum temperature difference takes place. So this is a conservative estimate of what would be the characteristic length.
So therefore you can see the maximum temperature difference for plane walls takes place over L and therefore this is correct, but if we think of a long cylinder, the maximum temperature difference would take place between the centreline and the outside, therefore, for this case, the conservative estimate would take the $L_c = r_0$, and the same for the sphere.
if you integrate the expression of the problem:

$$\frac{\rho V c}{h A_s} \int_{\theta_i}^{\theta} \frac{d\theta}{\theta} = -\int_0^t dt$$

$$\frac{\rho V c}{h A_s} \ln\left(\frac{\theta_i}{\theta}\right) = t$$

where:

$$\frac{\theta}{\theta_i} = \frac{T - T_\infty}{T_i - T_\infty} = \exp\left[-\left(\frac{h A_s}{\rho V c}\right) t\right]$$

where T_i is the initial temperature of the solid. therefore the temperature of the object would change in exponential fashion with time and the time constant of the process is simply the inverse of what I have written inside the bracket:

$$\tau_t = \frac{1}{h A_s} \rho V c t = R_{\text{cond.}} C_t$$

where C_t is some sort of thermal capacitance of the system. So, the behaviour of the variation of temperature with time is analogous to the voltage decay that occurs when a capacitor is discharged through a resistor in an electrical circuit. So of course, the higher the value of τ_t, the system will respond slowly in terms of the change in temperature with time. But this gives you a very simple way to treat the transient conduction in solid, provided you can use the lumped capacitance model. We can see the brackets of the exponential in another way:

$$\frac{h A_s}{\rho V c} t = \frac{h L_c}{k} \frac{k}{\rho c} \frac{t}{L_c^2} = Bi \frac{\alpha t}{L_c^2}$$

where α is the thermal diffusivity of the system. since on the left hand side we have a dimensionless number and on the right hand side Bi is dimensionless, of course also what multiply Bi is dimensionless and is called Fourier number expressed in this way:

$$Fo = \frac{\alpha t}{L_c^2}$$

So, the change in temperature of the solid object as a result in convection outside and conduction inside and if lumped capacitance model is valid, is expressed in terms of two dimensionless quantities:

$$\frac{\theta}{\theta_i} = \frac{T - T_\infty}{T_i - T_\infty} = \exp\left(-BiFo\right)$$

So, the Fourier number compares a characteristic body dimension with an appropriate temperature wave that penetrates, that gives you how fast the temperature wave will penetrate into the depths of the solid.

So, if $Bi > 0.1$ then there is no straightforward way to calculate what is the temperature. So, you have to write the equation, it will be partial differential equation and you have to solve it numerically. Fortunately for practising engineers, the solutions of transient cases when the $Bi > 0.1$ is present is available in all textbooks of heat transfer in the form of charts which are called Heisler charts.

3 Probel

We consider a thin cylindrical wire that has a specified electrical resistance. The wire is submerged in an oil bath whose temperature is lower than that of the wire, and an electric current flows through it. For this case, the convective heat transfer coefficient is provided. The problem is divided into two parts:

1. Determine the steady-state temperature of the wire.
2. Determine the time required for the wire to reach a temperature within 1 °C of its steady-state value.

The given data are:

- **Wire diameter:** $D = 1$ mm $= 0.001$ m.
- **Electrical resistance per unit length:** $R'_e = 0.01\ \Omega/\text{m}$.
- **Oil bath temperature:** $T_\infty = 25$ °C.
- **Convective heat transfer coefficient:** $h = 500\ \text{W/m}^2 \cdot \text{K}$.

- **Current through the wire:** $I = 100$ A.
- **Material properties of the wire:** density $\rho = 8000$ kg/m^3, specific heat $C = 500$ J/(kg $\cdot$ K), and thermal conductivity $k = 20$ W/(m $\cdot$ K).

4 Verification of the Lumped Capacitance Model

Before proceeding, we must verify that the lumped capacitance model is applicable by calculating the Biot number. For a cylinder, the characteristic length is taken as the distance over which the maximum temperature change occurs. When a thin wire is immersed in a cooler oil bath, the maximum temperature is at the center of the wire, and the minimum is at its surface. Consequently, the characteristic length is the radius:

$$r = \frac{D}{2} = \frac{0.001}{2} = 0.0005 \text{ m}.$$

The Biot number is defined as

$$\mathrm{Bi} = \frac{hL_c}{k} = \frac{h\,r}{k}.$$

Substituting the values:

$$\mathrm{Bi} = \frac{500 \times 0.0005}{20} = 0.0125.$$

Since $\mathrm{Bi} < 0.1$, the lumped capacitance model is valid and the temperature inside the wire can be assumed to be uniform (i.e., only a function of time).

5 Steady-State Temperature Analysis

At steady state, the heat generated in the wire due to electrical resistance is balanced by the heat lost by convection.

Heat Balance

The heat generated per unit length is:

$$q'_{\text{gen}} = I^2 R'_e.$$

The convective heat loss per unit length is:

$$q'_{\text{loss}} = \pi D\, h\,(T_s - T_\infty).$$

Balancing generation and loss gives:

$$I^2 R'_e = \pi D\, h\, (T_s - T_\infty).$$

Solving for the steady-state temperature T_s:

$$T_s = T_\infty + \frac{I^2 R'_e}{\pi D\, h}.$$

Substitute the numerical values:

$$T_s = 25\ ^\circ\mathrm{C} + \frac{(100)^2 \times 0.01}{\pi \times 0.001 \times 500}.$$

Simplify the fraction:

$$\frac{100^2 \times 0.01}{\pi \times 0.001 \times 500} = \frac{10000 \times 0.01}{1.5708} \approx \frac{100}{1.5708} \approx 63.7\ ^\circ\mathrm{C}.$$

Thus,

$$T_s \approx 25\ ^\circ\mathrm{C} + 63.7\ ^\circ\mathrm{C} \approx 88.7\ ^\circ\mathrm{C}.$$

6 Transient Analysis

For the transient behavior, we develop an energy balance on the wire (using the lumped capacitance assumption). In the transient case, the rate of energy storage is equal to the net energy in minus rate of energy out (due to convection) plus rate of heat generated. Nothing comes in to the solid.

Energy Balance Equation

The energy stored per unit length is:

$$E = m\, C\, T,$$

where the mass per unit length is

$$m = \frac{\pi D^2}{4} \rho.$$

Thus, the rate of change of stored energy is:

$$\frac{dE}{dt} = \frac{\pi D^2}{4} \rho C\, \frac{dT}{dt}.$$

The balance per unit length is then:

$$\frac{\pi D^2}{4}\rho C\,\frac{dT}{dt} = I^2 R_e' - \pi D\,h\,(T - T_\infty).$$

Defining the excess temperature $\theta = T - T_\infty$, the equation becomes

$$\frac{\pi D^2}{4}\rho C\,\frac{d\theta}{dt} + \pi D\,h\,\theta = I^2 R_e'.$$

Dividing both sides by $\frac{\pi D^2}{4}\rho C$ gives:

$$\frac{d\theta}{dt} + \underbrace{\frac{4h}{D\rho C}}_{A}\,\theta = \underbrace{\frac{4I^2 R_e'}{\pi D^2\rho C}}_{B}.$$

and than the expression of the governing equation would simply be:

$$\frac{d\theta}{dt} + A\theta = B$$

This ordinary differential equation can be solved using an integrating factor:

$$\frac{d}{dt}\left(\theta e^{At}\right) = Be^{At}$$

and integrating ones fro 0 to generic t we obtain:

$$\theta(t)e^{A\cdot t} - \theta(0)e^{A\cdot 0} = \frac{B}{A}\left(e^{At} - 1\right)$$

and divinding both terms by $e^{A\cdot t}$ we get:

$$\theta(t) = \theta(0)e^{-A\cdot t} + \frac{B}{A}\left(1 - e^{-At}\right)$$

Step 2: Application of the Initial Condition

Recall that we defined the temperature excess as

$$\theta(t) = T(t) - T_\infty.$$

The general solution obtained for $\theta(t)$ is

$$\theta(t) = \frac{B}{A}\left(1 - e^{-At}\right) + \theta(0)e^{-At},$$

where the initial condition is given by

$$\theta(0) = T(0) - T_\infty = T_i - T_\infty.$$

In this problem, it is physically assumed that the wire is initially in thermal equilibrium with the oil bath, i.e.,

$$T_i = T_\infty.$$

Therefore, the initial condition becomes

$$\theta(0) = T_i - T_\infty = 0.$$

Substituting this result into the general solution, we obtain

$$\theta(t) = \frac{B}{A}\Big(1 - e^{-At}\Big) + 0 \cdot e^{-At} = \frac{B}{A}\Big(1 - e^{-At}\Big).$$

Finally, recalling that

$$T(t) = T_\infty + \theta(t)$$

and that at steady state (when $t \to \infty$) we have

$$T_s - T_\infty = \frac{B}{A} = \exp\left(\frac{-4h}{\rho CD}t\right)$$

we can write the particular solution for the temperature profile of the wire as

$$T(t) = T_\infty + \Big(T_s - T_\infty\Big)\Big(1 - e^{-At}\Big).$$

This expression completely describes the evolution of the wire temperature starting from $T_i = T_\infty$.

5. Time to Reach Within 1°C of the Steady-State Value

We now determine the time t_r required for the wire to reach a temperature that is within 1°C of its steady-state value. Starting from the transient solution for the temperature, we have

$$T(t) = T_\infty + \theta_{ss}\Big(1 - e^{-At}\Big),$$

where the steady-state temperature excess is defined as

$$\theta_{ss} = T_s - T_\infty = \frac{B}{A},$$

and

$$A = \frac{4h}{\rho C D}.$$

To find the time t_r when

$$T(t_r) = T_s - 1,$$

we write

$$T_\infty + \theta_{ss}\left(1 - e^{-At_r}\right) = T_s - 1.$$

Subtracting T_∞ from both sides yields

$$\theta_{ss}\left(1 - e^{-At_r}\right) = \theta_{ss} - 1.$$

Dividing both sides by θ_{ss} (assuming $\theta_{ss} \neq 0$) gives

$$1 - e^{-At_r} = 1 - \frac{1}{\theta_{ss}},$$

which rearranges to

$$e^{-At_r} = \frac{1}{\theta_{ss}}.$$

Taking the natural logarithm on both sides, we obtain

$$t_r = \frac{1}{A}\ln\left(\theta_{ss}\right).$$

Substituting back the expression for A, we finally have

$$t_r = \frac{\rho C D}{4h}\ln\left(\theta_{ss}\right).$$

7 Conclusions

- The steady-state temperature of the wire is approximately $T_s \approx 88.7$ °C.
- The time required for the wire to reach within 1 °C of the steady-state temperature is approximately $t_r \approx 8.3$ s.

8 General Remarks on Non-Standard Cases

Always be on the lookout for cases that differ from the standard ones. If you truly understand the physics of the process, you will be able to incorporate any needed changes in your derivation so as to obtain an expression that is valid for the specific case at hand. This concludes our treatment of the transient conduction process. While there are many aspects of transient conduction that were not covered in this course, you can find a comprehensive discussion in any heat transfer textbook in the chapter on transient conduction. The treatment presented here is taken from Incropera and DeWitt.

9 Introduction to the Forced Convection Problem

Next, we move on to a slightly more involved problem, which will be continued in the next class as there is likely not enough time today. This is the case of forced convection.

Consider a cylindrical pipe through which a fluid at a given temperature T_0 enters. A constant heat flux $q_1^{''}$ is supplied through the side walls of the pipe. As the fluid moves, it receives energy by conduction in the radial direction, and its temperature increases also because of convection.

In this system both conduction and convection are present. The constant heat flux added through the side walls causes the fluid to gain energy continuously, manifesting as an increase in temperature. However, unlike the cases we have dealt with before, the fluid temperature profile is not solely a function of the axial position (denoted by z); it also varies in the radial direction (r). Thus, the temperature distribution depends on both z and r.

10 Velocity Distribution and Its Role in Convection

Whenever convection occurs, the fluid's velocity plays a critical role. In a pipe, fluid flow—induced by gravity and an imposed pressure gradient—yields a laminar, parabolic velocity profile. From fluid mechanics, the velocity in the z-direction is expressed as:

$$v_z(r) = v_{z,\max}\left[1 - \left(\frac{r}{R}\right)^2\right]$$

where $v_{z,\max}$ is the maximum velocity (occurring at the centerline) and R is the radius of the pipe. Note that the maximum velocity is related to the pressure drop; in fact, the difference $P_0 - P_L$ represents the combined effects of the imposed pressure, gravity, and other forces.

$$v_{z,\max} = \frac{(P_0 - P_L)R^2}{4\mu L}$$

It is advisable to review the derivation of the Hagen–Poiseuille equation, which describes flow through a pipe in the presence of both a pressure gradient and body forces.

Understanding the velocity distribution is a prerequisite for solving the heat transfer problem because convection depends directly on the fluid velocity. Therefore, one must first solve the fluid mechanics problem to obtain the velocity profile, which is then used to analyze convective heat transfer.

Since the temperature in the forced convection problem is a function of both the axial coordinate z and the radial coordinate r, we must consider a control volume (or shell) that reflects this two-dimensionality.

Assume a thin shell with the following dimensions:

- Axial thickness: Δz (to capture the variation in the z-direction).
- Radial thickness: Δr (to capture the radial variation).

The top surface of the shell, which is annular with an area given by

$$A_{\text{top}} = 2\pi r\,\Delta z,$$

experiences both convective heat transfer (due to the moving fluid) and conductive heat transfer (due to the axial temperature gradient). On the side wall of the shell (of thickness Δr), there is only conductive heat transfer in the r-direction, because the flow is predominantly in the z-direction and no convection occurs across this boundary.

Thus, for one boundary of the control volume, both conduction and convection must be considered, while the opposite boundary involves only conduction. In writing the overall energy balance, contributions from:

- Conduction in the z-direction,
- Conduction in the r-direction, and
- Convection in the z-direction,

must all be taken into account to derive the governing energy equation.

This approach underlines the benefit of developing a generalized energy equation—much like the Navier–Stokes equations in momentum transfer—which allows us to eliminate irrelevant terms to arrive at the final governing equation for complex geometries without resorting to a detailed shell balance for every case.

Energy Balance for Forced Convection in a Heated Tube

We begin our analysis of forced convection by considering a tube in which a constant heat flux is supplied through the walls. Two types of boundary conditions can be imposed:

1. **Constant temperature condition:** The tube walls are maintained at a fixed temperature.
2. **Constant heat flux condition:** An external agency provides a constant heat flux to the tube walls.

In our analysis, we assume the latter.

As the liquid flows through the tube, its temperature continuously increases. We assume that the flow is laminar, driven by both an imposed pressure gradient and gravity. For example, in a vertical tube the liquid flows downward because the pressure at the top is higher than at the bottom, with gravity further aiding the motion. Thus, even though the flow is laminar, it is influenced by both surface forces (pressure) and body forces (gravity).

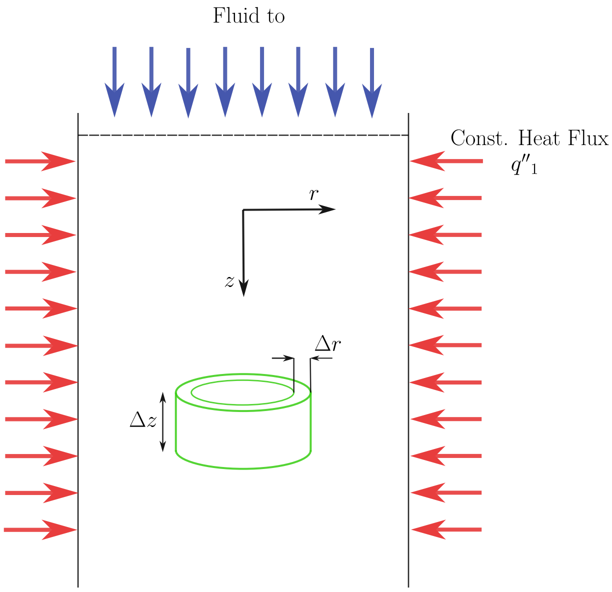
From fluid mechanics it is known that such conditions yield a parabolic velocity profile, where the axial velocity varies with the radial distance from the centerline. This velocity profile must be determined *a priori* so that it can be used in the subsequent energy balance analysis.

The first step is to define a control volume (or "shell") over which the energy balance will be performed. Since there is no internal heat generation in this problem, the energy balance reads:

$$\text{Heat in} \; - \; \text{Heat out} \; = \; \text{Rate of energy storage}\,.$$

Because the tube walls are heated, a temperature gradient exists in the axial (z) direction. This gradient gives rise to a conductive heat flow along the z-axis. In addition, since the temperature is highest at the tube wall and decreases toward the center, a radial (r) temperature gradient exists that causes conductive heat flow from the wall toward the center. (If instead a hot fluid were being cooled by a cold tube wall, the direction of the heat flow would be reversed.) In the present case, temperature varies with both z and r, so the control volume naturally has dimensions Δz and Δr.

Since the fluid is moving, convective transport is also present along the z direction. However, as we assume that the flow is one-dimensional, the radial velocity component is zero ($v_r = 0$) and no convection occurs in the r direction—only conduction does.



Conduction in the Radial Direction

For conduction in the radial direction, we adopt the convention that energy flows from r to $r + \Delta r$. The conductive heat flux at radius r is denoted as $q_r''(r)$, and it enters the control volume through the lateral surface with area

$$A_{\text{in}} = 2\pi r\,\Delta z\,.$$

Thus, the rate of heat entering by conduction at r is

$$Q_{r,\text{in}} = q_r''(r) \cdot 2\pi r\,\Delta z\,.$$

Similarly, the conductive heat flux at $r + \Delta r$, namely $q_r''(r + \Delta r)$, leaves the control volume through the area

$$A_{\text{out}} = 2\pi (r + \Delta r)\,\Delta z\,,$$

so that

$$Q_{r,\text{out}} = q_r''(r + \Delta r) \cdot 2\pi (r + \Delta r)\,\Delta z\,.$$

Conduction in the Axial Direction

Similarly, conduction in the axial direction follows:

$$Q_{z,\text{in}} = q_z''(z) \cdot 2\pi r \Delta r \tag{6}$$

with heat leaving at $z + \Delta z$ given by:

$$Q_{z,\text{out}} = q_z''(z + \Delta z) \cdot 2\pi r \Delta r \tag{7}$$

10.1 Convective Heat Transfer

The volumetric flow rate is given by:

$$\dot{V} = 2\pi r(\Delta r)v_z \tag{8}$$

Multiplying by density ρ, we obtain the mass flow rate:

$$\dot{m} = \rho\dot{V} = 2\pi r(\Delta r)v_z\rho \tag{9}$$

Energy transfer via convection is described by:

$$Q_{\text{conv}} = \dot{m}C_p(T - T_0) \tag{10}$$

where:

- C_p is the specific heat capacity,
- T is the fluid temperature,
- T_0 is the reference temperature.

At position z, the convective heat transfer is:

$$Q_{\text{conv,in}} = 2\pi r(\Delta r)v_z\rho C_p(T - T_0)\Big|_z \tag{11}$$

While at position $z + \Delta z$:

$$Q_{\text{conv,out}} = 2\pi r(\Delta r)v_z\rho C_p(T - T_0)\Big|_{z+\Delta z} \tag{12}$$

This v_z is also evaluated at the point $z + \Delta z$, but from our study of fluid mechanics we understand that v_z is a function of r only and is not a function of z. Therefore, for fully developed flow—which we have assumed to obtain our expression for the velocity profile—this v_z can be taken out of the expression, and consequently the term $(T - T_0)$, where T changes with z, represents the temperature difference in the z direction. So, T is a function of z but v_z is not a function of z, so we can take v_z out of the expression. And T_0 is just the datum temperature. Therefore, we add all the four terms that we have – that is, two terms for conduction (in and out) and two terms for convection (in and out). When we do this and express, then divide both sides by

$$2\pi\, r\, \Delta z,$$

what we get is

$$\frac{r\, q_r''\Big|_{r+\Delta r} - r\, q_r''\Big|_{r}}{\Delta r} + \frac{r\, q_z''\Big|_{z+\Delta z} - r\, q_z''\Big|_{z}}{\Delta z} + r\,\rho\, C_p\, v_z\, \frac{T\Big|_{z+\Delta z} - T\Big|_{z}}{\Delta z} = 0.$$

So, when you take the limit as $\Delta r \to 0$, you convert the finite-difference equation into a differential equation. In this limit, the equation becomes

$$-\rho\, C_p\, v_z\, \frac{\partial T}{\partial z} = -\frac{1}{r}\frac{\partial}{\partial r}\left(r\, q_r''\right) - \frac{\partial q_z''}{\partial z}$$

In this expression, the right-hand side terms (involving q_r and q_z) represent the conductive heat flow, while the left hand side represents the convective heat flow. Next, by applying Fourier's law

$$q_r'' = -k\,\frac{\partial T}{\partial r} \quad \text{and} \quad q_z'' = -k\,\frac{\partial T}{\partial z},$$

the heat fluxes become expressed entirely in terms of temperature gradients. We note that in this system the temperature T is a function of z (as the temperature increases along the flow direction) and also a function of r (since the temperature becomes higher near the heated wall). In other words, $T = T(r, z)$.

Therefore, we substitute these expressions for q_r and q_z into our governing equation.

Furthermore, from our study of fluid mechanics we know that, for a fully developed flow, the axial velocity v_z is a function of r only and does not vary with z. In fact, the velocity profile is given by

$$v_z(r) = v_{\max}\left[1 - \left(\frac{r}{R}\right)^2\right],$$

where R is the radius of the tube and $v_{\max}$ is the maximum velocity at the centreline.

Substituting the velocity profile into the expression for the convective term, the final form of the governing differential equation is obtained as

$$k\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\,\frac{\partial T}{\partial r}\right)+\frac{\partial^2 T}{\partial z^2}\right]=\rho\,C_v\,v_{\max}\left[1-\left(\frac{r}{R}\right)^2\right]\frac{\partial T}{\partial z}.$$

This equation, derived by combining the conductive and convective contributions to the energy transfer, is our governing equation. It must be solved with appropriate boundary conditions.

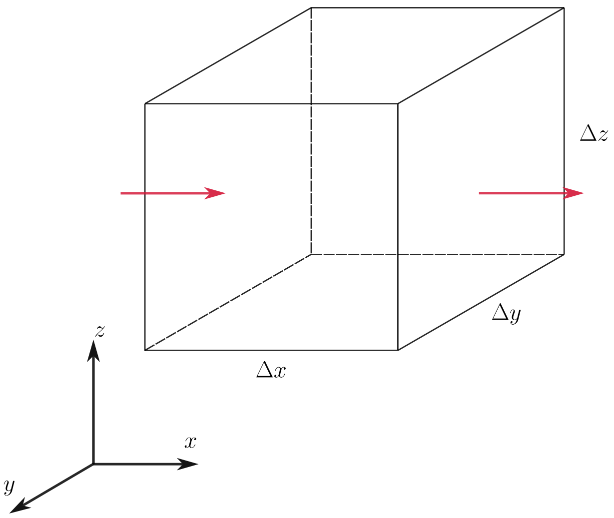
It is important to emphasize that obtaining this governing equation involves a detailed process of identifying the “small” dimension of the assumed control volume (or shell). Because the temperature is a function of both r and z, the problem becomes inherently two-dimensional, which complicates the formulation compared to the one-dimensional case. For a simple geometry such as flow through a heated tube, visualizing this shell correctly is challenging, and one must carefully ensure that all expressions are properly set up to arrive at the final, albeit complicated, governing equation.

Moreover, the repetitiveness of this process in two-dimensional or even three-dimensional problems highlights the need for a more generalized formulation—one that provides a universal energy conservation statement applicable to any geometry, steady or unsteady. Such a general approach would greatly streamline the derivation of governing equations for complex systems.

11 Conceptual Derivation of the Energy Change Equation for a Non-Isothermal System

As before, we assume a stationary volume with respect to x, y, and z directions, and fluid is allowed to enter and exit through the x, y, and z faces.

The fluid also exits through the faces at $x+\Delta x$, $y+\Delta y$, and $z+\Delta z$. This is how the fluid can come in and go out of the control volume.



Thus, if we write the conservation of energy for such a system:

$$\begin{array}{c}\text{Rate of accumulation of}\\ \text{inter. and kine. energy}\end{array} = \begin{array}{c}\text{Rate of inter. and kine.}\\ \text{energy in by convec.}\end{array} - \begin{array}{c}\text{Rate of inter. and kine.}\\ \text{energy out by convec.}\end{array} + \begin{array}{c}\text{Net rate of heat}\\ \text{addition by conduc.}\end{array} + \begin{array}{c}\text{Net rate of work done}\\ \text{by the system on the surrounding}\end{array}$$

the total energy of the system is equal to the algebraic sum of the rate of internal and kinetic energy entering by convection minus the rate of internal and kinetic energy leaving by convection. Since there is a flow, some amount of internal energy—and, due to the flow, some amount of kinetic energy—is entering the control volume, while an equivalent amount is also leaving it. This difference represents the net addition of internal and kinetic energy to the control volume because of convection. I then apply a similar approach for conduction; in this case, I use the term "net" to signify the difference between energy entering and leaving by conduction. Hence, this is the net rate of heat addition by conduction. However, there is an extra term to consider: the net rate of work done by the system on its surroundings. Let us examine this carefully.
The energy conservation equation can be written as the rate of energy input (by convection and conduction) minus the rate of energy output (by convection and conduction) being equal to the net rate of accumulation of internal and kinetic energy in the control volume. The mechanisms responsible for this net addition are a combination of both convection and conduction. Depending on the situation, both mechanisms can be present or only one may be significant. For example, if there is a temperature difference, conduction will always occur; thus, the system will exhibit conduction plus convection—or only conduction if we are dealing with a solid system. Furthermore, if the system performs work (or has work done on it), its internal energy, or total energy, will change. Specifically, if the system does work, its energy decreases; if work is done on the system, its energy

increases. Therefore, when we discuss the net rate of accumulation of energy (both internal and kinetic) within the control volume, it must be equal to the net rate of heat addition by conduction and convection plus an additional term accounting for the rate of work done by the system (which is negative when work is done by the system, and positive when work is done on the system). This equation is nothing but the first law of thermodynamics for an open system. An open system allows fluid to enter and exit the control volume. From this generalized energy equation, we can subtract the mechanical energy equation to obtain the more commonly used thermal energy equation. In other words, the equation presented here is for the total energy, which includes both the thermal (internal) energy and the mechanical (kinetic) energy. By subtracting the mechanical energy component, we obtain the conventional thermal energy balance equation.

Derivation of the Energy Equation for a Non-Isothermal System

So we are going to continue with our treatment of the energy equation, or more precisely the development of an equation of change for a non-isothermal system. In order to do that, what we have written down is the first law of thermodynamics for an open system, where the net accumulation of energy inside the system is expressed as the algebraic sum of the net heat addition to the system (both by convection and conduction) and the effect of work done by the system or on the system. I will now continue evaluating each of these terms, considering that energy enters through the x, y, and z faces of the control volume. For example, the x face has an area equal to $\Delta y, \Delta z$; the y face has an area equal to $\Delta x \, \Delta z$; and the z face has an area equal to $\Delta x \, \Delta y$. Through all these faces, energy is transferred into the control volume both by conduction and by convection. When we talk about convection, we simply have to multiply the face area by the corresponding velocity component. For instance, if we want to determine the total energy that comes in by convection through the x face, we multiply the area of the x face (which is $\Delta y \, \Delta z$) by the velocity component v_x, then multiply by ρ to obtain the mass flow rate, then multiply by C_p and the temperature ΔT. This gives the total amount of convective heat being added to the control volume because of the flow with a component v_x through the x face.
Before proceeding further, let us first evaluate the left-hand side of the energy equation: that is, the rate of accumulation of internal and kinetic energy present in the system. The rate of accumulation of internal energy and kinetic energy (per unit mass of the fluid) must be:

$$\text{Rate of accumulation of inter. and kine. energy} = \Delta x \Delta y \Delta z \frac{\partial}{\partial t}\left(\rho U + \frac{1}{2}\rho v^2\right)$$

where U denotes the internal energy per unit mass of the fuid and v is the magnitude of the local fluid velocity.

$$\begin{aligned}\text{Rate of inter. and kine. energy in-out by convec.} = \Delta y \Delta z &\left\{ v_x \left(\rho U + \frac{1}{2}\rho v^2\right)\Big|_x - v_x \left(\rho U + \frac{1}{2}\rho v^2\right)\Big|_{x+\Delta x} \right\} + \\ + \Delta x \Delta z &\left\{ v_y \left(\rho U + \frac{1}{2}\rho v^2\right)\Big|_y - v_y \left(\rho U + \frac{1}{2}\rho v^2\right)\Big|_{y+\Delta y} \right\} + \\ + \Delta x \Delta y &\left\{ v_z \left(\rho U + \frac{1}{2}\rho v^2\right)\Big|_z - v_z \left(\rho U + \frac{1}{2}\rho v^2\right)\Big|_{z+\Delta z} \right\}.\end{aligned}$$

$$\text{Net Rate of heat addition by conduc.} = \Delta y \Delta z \left\{ q_x^{''}\Big|_x - q_x^{''}\Big|_{x+\Delta x} \right\} + \Delta x \Delta z \left\{ q_y^{''}\Big|_y - q_y^{''}\Big|_{y+\Delta y} \right\} + \Delta x \Delta y \left\{ q_z^{''}\Big|_z - q_z^{''}\Big|_{z+\Delta z} \right\}.$$

The work done can be against two types of forces: volumetric (body) forces—such as gravity—and surface forces. The surface forces, in turn, include pressure forces and viscous forces.

Now we turn to the rate of work done. Recall that work is defined as force times displacement; thus, its rate (power) is force times velocity in the direction of the force. In our formulation, the rate of work done is expressed as the product of the force and the velocity component in the same direction.

We now write an expression for the work done for gravity, pressure, and viscous forces. The gravity part is straightforward; the rate of work done against gravity is given by the product of the gravitational force (which is $\rho\, g$ times the volume or mass element) and the corresponding velocity component:

$$\text{Rate of work done against gravitational force} = -\rho \left(\Delta x \Delta y \Delta z\right)\left(v_x g_x + v_y g_y + v_z g_z\right)$$

Note the negative sign used because the work is done against gravity, meaning that the gravitational force and the velocity are in opposite directions.

Next, consider the rate of work done against static pressure. On, for example, the x face, the net force due to static pressure is:

$$\begin{array}{c}\text{Rate of work done}\\\text{against static pressur}\end{array} = \Delta y \Delta z \left(pv_x\Big|_{x+\Delta x} - pv_x\Big|_x \right) + \Delta x \Delta z \left(pv_y\Big|_{y+\Delta y} - pv_y\Big|_y \right) + \Delta x \Delta y \left(pv_z\Big|_{z+\Delta z} - pv_z\Big|_z \right)$$

Finally, the work done against viscous forces is more involved because the viscous stress, τ, is a tensor with nine components:

$$\tau_{xx},\ \tau_{xy},\ \tau_{xz}, \quad \tau_{yx},\ \tau_{yy},\ \tau_{yz}, \quad \tau_{zx},\ \tau_{zy},\ \tau_{zz}.$$

for example τ_{xy} represent the y component of the momentum get transported in the x direction.

The work done against viscous forces is then obtained by computing the force from the viscous stresses on the face area and multiplying by the velocity component in the direction of that force.

$$\begin{aligned}\begin{array}{c}\text{Net rate of work done}\\\text{by the system on the surrounding}\end{array} = \Delta y \Delta z &\left\{ (\tau_{xx}v_x + \tau_{xy}v_y + \tau_{xz}v_z)\Big|_{x+\Delta x} - (\tau_{xx}v_x + \tau_{xy}v_y + \tau_{xz}v_z)\Big|_x \right\} + \\ + \Delta x \Delta z &\left\{ (\tau_{yx}v_x + \tau_{yy}v_y + \tau_{yz}v_z)\Big|_{y+\Delta y} - (\tau_{yx}v_x + \tau_{yy}v_y + \tau_{yz}v_z)\Big|_y \right\} + \\ + \Delta x \Delta y &\left\{ (\tau_{zx}v_x + \tau_{zy}v_y + \tau_{zz}v_z)\Big|_{z+\Delta z} - (\tau_{zx}v_x + \tau_{zy}v_y + \tau_{zz}v_z)\Big|_z \right\}.\end{aligned}$$

At this point, we have laid out the complete treatment for the energy equation of a non-isothermal open system. The equation includes:

- The rate of accumulation of internal and kinetic energy,
- The net heat added by conduction,
- The net heat added (or removed) by convection,
- The rate of work done by (or on) the system.

From this generalized energy equation, by subtracting the mechanical energy equation, one can derive the more commonly used thermal energy balance equation.

Recall that we have already written the generalized total energy balance for an open, non-isothermal system as

$$\Delta V \frac{\partial}{\partial t}\left(\rho U + \frac{1}{2}\rho v^2\right) + \nabla \cdot \left[\mathbf{v}\left(\rho U + \frac{1}{2}\rho v^2\right)\right] = -\nabla \cdot \mathbf{q} - \nabla \cdot (p\,\mathbf{v}) - \rho\,\mathbf{g} \cdot \mathbf{v} + \nabla \cdot (\boldsymbol{\tau} \cdot \mathbf{v}).$$

Here, $\Delta V = \Delta x\,\Delta y\,\Delta z$ is the volume of the control element.

The total energy contains both the internal (thermal) energy and mechanical energy (kinetic and potential). The mechanical energy balance (derived from the momentum equations) accounts for the kinetic energy and the associated work terms due to gravitational, pressure, and viscous forces. By subtracting this mechanical energy balance from the Equation, we isolate the energy balance for the internal energy U. That is, we obtain

$$\frac{\partial(\rho U)}{\partial t} + \nabla \cdot (\rho U \, \mathbf{v}) = -\nabla \cdot \mathbf{q} + p \, (\nabla \cdot \mathbf{v}) + \boldsymbol{\tau} : \nabla \mathbf{v}. \tag{13}$$

In the above, the term $p \, (\nabla \cdot \mathbf{v})$ represents the work done by compression or expansion, and

$$\boldsymbol{\tau} : \nabla \mathbf{v} = \sum_{i,j} \tau_{ij} \frac{\partial v_i}{\partial x_j}$$

accounts for the viscous dissipation.

Assuming that the heat flux obeys Fourier's law,

$$\mathbf{q} = -k \, \nabla T,$$

it follows that

$$-\nabla \cdot \mathbf{q} = \nabla \cdot \Big(k \, \nabla T\Big).$$

Substituting this yields

$$\frac{\partial(\rho U)}{\partial t} + \nabla \cdot (\rho U \, \mathbf{v}) = \nabla \cdot \Big(k \, \nabla T\Big) + p \, (\nabla \cdot \mathbf{v}) + \boldsymbol{\tau} : \nabla \mathbf{v}. \tag{14}$$

For many fluid systems, the internal energy per unit mass can be expressed as a linear function of temperature:

$$U = c_v \, T,$$

where c_v is the specific heat at constant volume (assumed constant). Thus, we have

$$\frac{\partial(\rho c_v \, T)}{\partial t} + \nabla \cdot (\rho c_v \, T \, \mathbf{v}) = \nabla \cdot \Big(k \, \nabla T\Big) + p \, (\nabla \cdot \mathbf{v}) + \boldsymbol{\tau} : \nabla \mathbf{v}.$$

For constant ρ and c_v, the left-hand side of the equation can be rewritten in terms of the material derivative

$$\frac{D}{Dt} = \frac{\partial}{\partial t} + \mathbf{v} \cdot \nabla,$$

yielding

$$\rho\, c_v \frac{DT}{Dt} = k\,\nabla^2 T + p\,(\nabla\cdot\mathbf{v}) + \underbrace{\boldsymbol{\tau} : \nabla\mathbf{v}}_{\Phi},$$

where we define the dissipation function as

$$\Phi = \boldsymbol{\tau} : \nabla\mathbf{v}.$$

Thus, the final form of the thermal energy balance equation is

$$\boxed{\frac{\partial(\rho U)}{\partial t} + \nabla\cdot(\rho U\,\mathbf{v}) = \nabla\cdot\Big(k\,\nabla T\Big) + p\,(\nabla\cdot\mathbf{v}) + \boldsymbol{\tau} : \nabla\mathbf{v},}$$

or equivalently, when expressed in terms of temperature,

$$\boxed{\rho\, c_v \frac{DT}{Dt} = k\,\nabla^2 T + p\,(\nabla\cdot\mathbf{v}) + \Phi.}$$

This completes the derivation of the thermal energy balance equation from the generalized energy equation by subtracting the mechanical energy contributions and applying Fourier's law together with a linear relation for the internal energy.

Incorporation of Thermodynamic Relations

However, a more careful thermodynamic treatment recognizes that U is a function of both T and ρ, namely

$$U = U(T, \rho),$$

so that its material derivative reads

$$\frac{DU}{Dt} = \left(\frac{\partial U}{\partial T}\right)_\rho \frac{DT}{Dt} + \left(\frac{\partial U}{\partial \rho}\right)_T \frac{D\rho}{Dt}\,.$$

Since

$$\left(\frac{\partial U}{\partial T}\right)_\rho = c_v,$$

and using a standard thermodynamic relation one obtains

$$\left(\frac{\partial U}{\partial \rho}\right)_T = \frac{1}{\rho^2}\Big[p - T\left(\frac{\partial p}{\partial T}\right)_\rho\Big]\,.$$

Employing the continuity equation

$$\frac{D\rho}{Dt} = -\rho \nabla \cdot \mathbf{v} \, ,$$

we deduce

$$\frac{DU}{Dt} = c_v \frac{DT}{Dt} - \frac{1}{\rho} \left[p - T \left(\frac{\partial p}{\partial T} \right)_\rho \right] \nabla \cdot \mathbf{v} \, .$$

Multiplying by ρ yields

$$\rho \frac{DU}{Dt} = \rho \, c_v \frac{DT}{Dt} - \left[p - T \left(\frac{\partial p}{\partial T} \right)_\rho \right] \nabla \cdot \mathbf{v} \, . \tag{A}$$

On the other hand, the internal energy balance (after subtracting the mechanical contributions) is expressed as

$$\rho \frac{DU}{Dt} = \nabla \cdot \Big(k \, \nabla T \Big) + \Phi - p \, \nabla \cdot \mathbf{v} \, , \tag{B}$$

where:

- k is the thermal conductivity,
- $\Phi = \boldsymbol{\tau} : \nabla \mathbf{v}$ is the viscous dissipation term.

Equate (A) and (B):

$$\rho \, c_v \frac{DT}{Dt} - \left[p - T \left(\frac{\partial p}{\partial T} \right)_\rho \right] \nabla \cdot \mathbf{v} = \nabla \cdot \Big(k \, \nabla T \Big) + \Phi - p \, \nabla \cdot \mathbf{v} \, .$$

Adding $p \, \nabla \cdot \mathbf{v}$ to both sides, the pressure terms cancel appropriately:

$$\rho \, c_v \frac{DT}{Dt} = \nabla \cdot \Big(k \, \nabla T \Big) + \Phi - T \left(\frac{\partial p}{\partial T} \right)_\rho \nabla \cdot \mathbf{v} \, .$$

Expressing the Viscous Dissipation Term

For a Newtonian fluid the viscous dissipation Φ can be written as

$$\Phi = \boldsymbol{\tau} : \nabla \mathbf{v} = \mu \, \phi \, ,$$

where μ is the dynamic viscosity and ϕ is a (usually quadratic) function of the local strain rates. A common form (assuming also the usual relation for the second viscosity coefficient, e.g. $\lambda = -\frac{2}{3}\mu$) is

$$\phi = 2\, S_{ij} S_{ij} - \frac{2}{3} (\nabla \cdot \mathbf{v})^2 \ ,$$

with

$$S_{ij} = \frac{1}{2} \left(\frac{\partial v_i}{\partial x_j} + \frac{\partial v_j}{\partial x_i} \right) \ .$$

Final Form of the Thermal Energy Balance Equation

Substituting $\Phi = \mu\, \phi$ into the previous equation, we obtain the final result:

$$\boxed{\rho\, c_v \frac{DT}{Dt} = \nabla \cdot \Big(k\, \nabla T \Big) + \mu\, \phi - T \left(\frac{\partial p}{\partial T} \right)_\rho \nabla \cdot \mathbf{v}\, .}$$

This equation shows explicitly the contributions due to thermal conduction, viscous dissipation, and the thermodynamically "modified" pressure work (which, after subtraction of the mechanical energy, leads to the term $-T \left(\frac{\partial p}{\partial T} \right)_\rho \nabla \cdot \mathbf{v}$).