this is the field what we used to solve it, we have already solved this part. Today we are looking at what will be the wall stress, shear stress at this point, also this point, stream function, vorticity, velocity potential and the average velocity if known the velocity field. So in the last class we estimate this velocity field from Navier-Stokes equations that we assume it vw is a 0 okay, neglecting the gravity force components. We get this u component that is what we did it last class minus dp by dx okay that is what dp by dx is equal to h square by 2 mu types of 1 minus y square by h square. This is the velocity field we got it applying this Navier-Stokes equations as you remember it.

This is what the velocity field we got it for the pressure gradient. which would have a function of h mu dynamic specificity 1 by h square. The parallel shape okay that is will be the shape will come like this as we have a two fixed parallel plate. So now it is easy so you have to know the first velocity field okay and is a function of dp by dx that means p is varying only x directions if it is that we can compute this part. Now if you look it I am just substituting wall shear stress using the Newton's laws of viscosity we can get tau xy wall fluid kinematics we have done this part mu is equal to del u by del y del v by del x at wall y equal to plus minus h this is what.

So plus h this is minus h okay. So del v is 0 we know it. So you only have a u functions that is what the functions you just substitute here. do the partial derivatives finally you get it this part okay finally you get this part. So the value of the at this point will be the dP by dx into h here will be dP by dx e to the power minus h. So that is what will be the shear stress distributions will come it okay.

This will have a in terms of u max also we can get it as a 2 mu times of u max in it. So simple just substitute it do a partial derivative with respect to y and get it which is a basic definitions as we derive from the flow kinematics okay. Now if I look it as the plane okay it is a 2 dimensional plane flow that is what so it can have the stream flow okay. Stream functions we can get it because it is a 2 dimensional flow okay as the x and y flow is steady and incompressible.

So we know it u equal to definitions okay. So just integrate the u then you will get it the functions u max. So you will get a functions which you will get it this function just to integrate it from setting the psi equal to 0 at the center line. So this is what psi equal to 0. So you can integrate it to finally you will get it these functions.

So that is what is showing it. So you can get a streamline functions okay and that is the value what will be there the value at this point at the both the walls okay that is plus and minus signal will be there. So we are getting the stream functions we are getting the stream function as we are considering the centralized stream functions is equal to 0 we can setting that value then we can be then we can get it what will be the stream functions value at both the fixed walls. Both the fixed walls we can estimate it what is the stream function. So if you look it it is not a difficult only you have to look it what is a relationship between pulse stress which is a xy plane how similar way what is the relationship with the velocity and the stream functions that is that is what we did it very extensively looking to divide this thing.

The same way I look at the verticity okay. So verticity in the zth plane that is what we you have a x and y. So you are looking the verticity which is perpendicular to this plane. So as you will again going back to the fluid kinematics we are looking for curl v okay. So we are looking a curl v in z direction so that what is dv by dx I just writing u dy. So v is 0 so only you know this partial derivative of the y which will comes up to this value okay.

So a non-general vorticity components are there. So now if vorticity components are there as we discuss it we cannot get the velocity potential functions because to get a velocity potential function flow should be irrotational. So that is what is here justified it. You need not to estimate the velocity potential function because potential function does not exist here because flow is not So that is the comprehensive idea of the fluid mechanics you have to understand it we have to estimate the vorticity if vorticity vector could be 0 we could compute the velocity potential functions. But in this case we cannot compute it because flow is not irrotational the velocity potential functions does not exist it. Then you have to compute the average velocity it is a very easy that for a if I considering this discharge which is u dA velocity in area integrations from minus h to plus h divide by this area I will get it integrations over this we will go at it.

B is the dimensions is a perpendicular to this plane that is what is a unit dimension sometimes we can consider it this is what the dy so we will get it the average velocity. So giving these two examples just you want to understand it when you apply Navier Stoke equations to estimate velocity field or you try to look at the shear stress, wall shear stress, the stream functions, the velocity potential function or the average velocity. So as the we have the functions we do partial derivative, we do the integrations, differentiation all these things we apply it to get it from basics relationship we try to get it what will be the average velocity, what will be wall shear stress, what will be the stream functions, what will be the velocity potential function or the vorticity. So that it is a give it is a examples here as compared to integral approach where we simplified many things. But in case of the differential approach, we need to do the integrations and differentiations to get the pressure field, to get wall shear stress, to get stream functions, velocity potential function with a lot of series of approximations.

Two-dimensional flow, incompressible, Newtonian fluid. So that is what I am repeating it to tell it that fluid flow problems are not easy. You simplified it for a specific cases to get some solutions. Now let us go for a very interesting topics which is the boundary layer approximations or where we do need the Navier-Stokes equation solvers. like for example as is given here again I will be sketch that same figures for your convenience and better understandings that there are the two chambers are there, two chambers are there.

and these chambers are connected with a pipe okay. The liquid is coming from these chambers and filling is happening in these chambers is the free flow surface. The tank 1 and tank 2, one is supplying tank another is filling tank. The same figures I am just drawing it to make a more attentions to figures. So if you look at this way that we have had 2 tanks the liquids are there and the liquid is coming from tank 1 supplying tank to the receiving tank if it is happening that. See if you look at when fluid flow is coming it that is what we have to understand it.

Fluid flow is coming it okay you are you try to understand is a virtual fluid balls okay that balls are coming it. which is are closer to the boundary at that boundary will have a no slip boundary conditions. That means there is regions very close to this flow where the flow is coming it will have a regions where is a low base low base to maintain the boundary conditions. The no slip boundary conditions you will have a very thin layers which will be making a viscous as well as you have a the layer will have viscous and rotations near to this because near to the wall as is a no slip boundary condition velocity has to be 0. So because of that the fluid particles which are the nearby they will also detract it.

So there will be regions will form it very close to this wall where you will have a viscous dominancy will be there and there will be velocity gradients and that is what will act a shear stress on this wall. Exactly same way as the flow is coming it there will be formations of regions will be viscous dominancy will be there, the vorticity will be there and these the regions we call boundary layers. These the regions we call boundary layers okay close to the boundary. So that is the reasons we define it as a boundary layers is the in the regions where you have a vorticity where you have a viscosity dominance is there.

But this layer is very thin as compared to the flow domains. These are the regions where we have the boundary layers. So this is very close very thin but to just to represent you I just magnified it okay it is not like this thickness. So there will be a boundary layer formations will be presenting in that which is because of presence of these two walls okay. As the fluid particles are coming it not the all the fluid particles here will be active for the flow is going through this. There are the fluid will commit a regions very close to this will have a flow will come it flow will go like this flow will come flow will go like this they will have a flow like this.

So in fluid mechanics you have to understand it how the flow happens at different scales. it is not a simple mathematics or it is so that visualizations conceptual visualization is required. That means the flow at these regions they will not have any rotational okay. These are the regions flow will have a irrotational okay and viscosity will not dominate it.

So that is the reasons we can use Euler equations okay. So if you if you look it there is a two tanks it is a simple examples okay. But in the that two tanks how type of the flows are happening there are the regions where is the boundary layer formations are there viscosity dominate is there vorticity is there. There is a regions where There will be no rotational field will be there. All the streams will come like this which where we can use the Euler equation. But just after these regions okay of this flow jet it will act like a flow jet.

In those regions you will have viscosity dominated as well as the vorticity. So you need to have a Navier-Stokes equations. So NIC equations you need to have. What will happen to this? This will act like fluid at the rest or fluid statics.

Same thing is fluid at the rest. So with this very simple examples if you can look it with a very simple problems that we have a two tanks. One is a supply tank another is a receiving tank. Within that tank flow zones there are the reasons behave like a fluid at the rest that is what will happen it. These regions will remains like a fluid at the rest there is no stress component at all so only the gravity force and the pressure field that is what we discussed that. There is a regions where the flow is irrotational regions where only the streamlines will come like this.

There are the regions close to the wall where they have a boundary layers there will be large vortex vorticity will be there, viscous dominancy will be there. Then you have the regions where you need to have a full Navier-Stokes region. That is what is showing it, full Navier-Stokes equations because there is a jet coming it. As a jet mixing with waters or any liquid, there is a jet, the water jet.

just visualize the flow. This is a water jet. So as the water jet is coining it and this to so if these regions the viscosity will you dominance the all these you have a what is it is only will be there. So those regions we have to solve vision eviscerations. This is the fluid at the rest conditions. So that means in these simple cases we have a four type of conditions.

fluid at the rest which we discussed very beginning it. We also discussed about irrotational flow area which is Euler equations. We also discussed about the Navier-Stokes equations. We approximate it or we know it from computational fluid dynamics we can solve these problems. And other one is the boundary layers very thin layer near to the boundary you will have viscosity dominancy as well as the vorticity. How we can approximate that which will be a form of a solutions of navier stucco equations.

That is the things let us discuss with which is will be lot of approximations because computational fluid dynamics as I said it very beginning is that it has grown last 3 to 4 decades. But how do you solve very challenging fluid mechanics problems in early 90s because that times in World War II we used to have a spacecraft. So without fluid mechanics knowledge how we develop these aeroplanes, how we develop the best challenging fluid mechanics problem solutions. The balancing between them is that the boundary layer approximations that is what I will be talk about here that so we have a lot of problems is comes it and we need to have a take care of equations for the approximations.

okay. So equations of approximations we have to look it it is not like as you can always have a debate it today we have a CFD with us okay computational flow dynamics tools with us what is a necessary to learn boundary layer approximations okay boundary layers But that is what I will justified it with giving some examples that still even if I have a best solver the computational fluid dynamics solvers are there still today we should have a understanding of the boundary layers. So our understanding of a boundary layers with experimental fluid mechanics okay. which we are not covering not but just have a demonstrated few of the cases. Really we can solve many challenging problems using knowledge of the boundary layers and knowledge establishing this empirical relationship from experimental fluid mechanics.

So they are complementary each other that is not that CFD only can solve the problems. CFD plus boundary layer and experimental fluid mechanics they are complementary actually to understand very complex fluid flow okay. So it is basically talking about any biomechanics and all where really we have a complex problems okay. So with this note let us come back to the boundary layer approximations. The basically it is a bridging it very simple one.

It is a bridging between the Euler equations and the Navier-Stokes equations. because Navier-Stokes equation solves with a no-slip boundary conditions but Euler equation does not have a it has to have a slip boundary conditions like for examples if I get it if you just take a simple form of cars okay. The sketch is not that perfect but anyway that does not matter it is moving with velocity v. So let be maybe 100 meter per second okay. So this is So if have the velocity v is I have a car is moving with the velocity v with our understanding it that you will have a opposite like this you will have a free stream velocity and car is standing it okay.

So free stream velocity will be the way B and car is at the rest conditions. That is what just opposite we do with the relative wind testings. If you do that you can this is the free stream velocity. So this is the front side of car. So there will be a boundary layer formations will happen here.

There will be a boundary layer formations will happen here. there will be the boundary layer formations will happen here. At the top you will have the flow will go like this where the Euler equations are valid. Euler equations is valid that is not a big issue. Here the Euler equation is valid But the boundary in Euler equation is a slip conditions but reality it is not a slip conditions it will no slip and there is a formations of boundary layers okay.

Formations of boundary layer. These boundary layers will be change the pressure distributions. These boundary layers will be change the velocity close to this surface. this boundary layer is responsible for what will be the drag force will act, what will be the lift force will act. So there will be a combinations of boundary layers and the Euler equation solvers solve this problem instead of looking for Navier-Stokes equation solver. So we are just replacing or we were just that is what is early before I can say 1940s or 50s almost all the scientists they try to look it alternative way to solve this Navier Stoke equation is that.

near to the boundary you solve this boundary layer equations just far away from these you apply this Euler equations.

The boundary layer approximations always most of the books written in a very complex way. So a lot of mathematics and all but the conceptually it is a very nice concept looking at how the approximation has given it very complex flow to get some sort of solutions instead of the Navier-Stokes equations we are looking a boundary layer equations okay. That is what today I will be discussing on this. We are looking boundary layer equations. We are not looking which is a part of a Navier-Stokes equations only.

That is the flow physics is the same the mass conservation and linear momentum equation. Some form of this Navier-Stokes equations we are trying to look it as a boundary layer equations. as because when you talk about the drag and lift force of a car it all depends upon a very thin layers how it is formatting over the surface okay as the car is moving it. This thin layer is responsible for the shear stress, this thin layer responsible to be the pressure distributions. Combination of these thin layer which is the boundary layers with the oilers if I can solve it with a degree of approximation of Navier Stoke equations that what we can solve it because as you know as we discussed many times we do not have a analytical solutions of Navier Stoke equation for all the problems.

Very few problem simplified problems we have. So now if you look it that is the concept is coming that you need to have a boundary layers is again I am repeating it is very very thin layers could be a millimeter levels could be the centimeter levels okay boundary layers thickness are the very very small except okay I am not talking about atmospheric boundary layers which is much bigger size but the automobile bounded layers flow through any flow past an object is a very very thin in order of the millimetre or the centimetre or so what it instead of looking at the cars the people have considered let you consider a simple plate which is very easy okay. to simplify the problems not looking it complexity at the very beginning looking it considered as a plate and you have a uniform stream flow okay. You have uniform stream flow many of the times we designate as capital U stream flow because of these there will be a boundary layers formations will be there. which will make it a boundary layer formations will be there like this okay.

So if you look at that this will be symmetry okay. So that is the reasons we always talk about this part as it is a symmetric okay. So we simplify that. When you have a plate, when you have a uniform stream flow, definitely there will be boundary layer formations in both the sides of the plates. But symmetry point of view, many of the books show the boundary layer formations from this okay. and that is what is they consider only symmetric boundary layer formations that is what is biggest problems.

This is only the symmetric boundary layers I am looking it at this point I know this velocity 0 and I have this x and this is the x directions is L is a length of the plate. This is what my boundary layer thickness which is a sigma x which varies with is a thickness. How do we define it? We define the boundary layer thickness if this u is the velocity 99% of this velocity 0.99 u where we have that is the boundary of the boundary layer thickness. One boundary at the plate another boundary is the the point where the velocity is equal to the 99% of the free stream velocity okay that is this is a hypothetical but this is okay.

So 1% we consider it is okay no problems. So that way we consider is a 99% of the velocity that is what we will define as the boundary layers upper boundary and the lower boundary you know it where you have a velocity is 0, u is equal to 0 value, u equal to the 0 value what you consider it. That is what is 0.99 okay always you can divide it why not more okay. So it is considered is okay 90% 99% velocity the free stream velocity if it is that that is what it defining the upper boundary the lower boundary anyway this plate okay that is the simplified case and that is the sigma x is a sorry delta x is the boundary layer thickness which is varying along the x direction.

If you can anticipate it with distance it will be increasing trend. So one things if you look it as I think it that the plate thickness can be length could be a meters okay but the boundary layer thickness again I very very thin in centimeter level or meter levels. So that centimeter meter levels when I look at that what is the condition delta by x will be delta will be much much less the thickness of boundary layers will be much much less as compared to the x value and delta x by value which will be much much less than one value. So it is a geometric problem but that is the thing that is the reasons again I am highlighting think it that the conceptually thinking that many of the times the boundary layers what you show it look at this they write magnified boundary layers because that they know it the boundary layers what is there that we cannot show it any of the textbook things they that is the reason they write it is a magnified boundary layer with compared to this x dimensions this is much much lesser and that is the reasons the delta by x is a much much lesser than one value. That should be the understanding of boundary layers how does it forms okay. That is the reasons if you know it how we make a smooth profiles of aeroplane wing or you talk about any high speed cars okay you can see the smooth profiles that is the reasons that they try to look at the boundary layer thickness formations that is what they try to look at.

That is what is boundary layer is a very thin flow regions where the viscous rotational force are present and boundary layers are solved with these two boundary conditions okay. Outward flow as I said it is in VCD or the root irrotational. We can use the continuity and Euler equations. Bernoulli's equations can give some of the pressure field and we can get the velocity field.

That is what we did it okay. That is not a so big part. Now if you look it the basically what we are looking it if you try to understand it that as I said it Today we have the CFD tools with us. That is not a big issue commercially or freely. Softwares are available to solve these navigation applications. That is not a big issue. It is not now longer to split two domain as boundary layer domains and outer flow regions okay or the boundary regions because CFD can solve it.

But as I said it still we need to have a fluid mechanics knowledge near to the boundary that is what is done by almost 30 to 40 years with a group of scientists and professors in Europe to solve these problems and it is really gives a better understanding of the boundary flow how it happens it. which even if you have the CFD techniques there are the numerical solutions there is a issue but boundary layer can use which is nothing else is a just a approximations of navier stucke equations. That is it again I am telling it is nothing else is we are same Navier Stoke equations but it is approximations near to the boundary layer regions there that is the reasons you have to look it. The boundary layer thickness distance away from the wall at which the velocity component is parallel to wall 99% of the velocity speed outside the boundary layers that the definitions very simple way I said it.

Its looks is that if you have a free stream velocity u you look at the 99% of u where it is 0.99 of u that the reasons will define us the boundary layers thickness which is varies with x. in boundary layers as we have defined the Reynolds flow Reynolds numbers. Here also we define the Reynolds number considering the characteristic length in x direction. That means what we do it here Reynolds numbers when we are defining it this is the x directions okay we are defining the flow Reynolds numbers which is inertia forces by viscous force that part here characteristic length we have given the x so that is what again I could write it that Re x the Reynolds numbers what we consider for a boundary layers okay as v is free stream velocity x is characteristic length from this is the end from this x value by the mu dynamic viscosity that is what is we can write Bx. So as the x is increases the Reynolds numbers is increases that is what if you look it for a particular free stream velocity v.

Western the fluid properties you know the continuity viscosity is a constant but x as it increases the Reynolds numbers is increases. that will have the same conditions that as we have done for the pipe flow. The flow will go through three phases laminar, transitional and the turbulent. Here it will depends upon the threshold the critical Reynolds number based on the boundary layers experiment which was done it almost 100 years back. to find out what is a critical Reynolds numbers to decide which is the reasons follows to laminar boundary layers, which is the reasons follows to for turbulent boundary layers or reasons in between the transitional boundary layers.

So these the critical Reynolds numbers which is estimated from the experimental data conducting plate in a wind tunnels, estimating all these velocity observations, the flow field, they try to find out what is the critical Reynolds numbers to define in the boundary layers itself with the 3 regions the laminar, transitional and the turbulent point road layers. Let me I commit to that figures which very interestingly if you look at that if you have a 3 stream velocity v and this is the plate as we are considering a symmetric boundary layer conditions this is the plate. and the flow is coming it as it will be retracted then flow will be a this is what the sigma x which is the boundary layer thickness which is increasing along this and this is the boundary okay. So you can look it as come to the turbulent and all this will not be smooth as in a turbulence as you know it, it will have a lot of verticity will be there.

and in transitionals both vorticity and decaying of these things will be there. So we should do it. So that way in a plate okay as we moving with x directions there will be regions for laminar boundary layer, transitional boundary layer and the turbulent boundary layer. From the experiments okay basically conducts in wind tunnel. A lot of experimental fluid mechanics did it in almost 50 to 80 years, still we are doing lot of experimental fluid mechanics. here we have not emphasized much. If you look at that from experimental fluid mechanics they found two critical Reynolds numbers to define flow between laminar transitional and transitional to turbulent.

The Reynolds numbers is 1 lakh okay we see easy to remember 1 is to 10 to the power 5. So in case of Reynolds numbers if it is lesser than that critical Reynolds number lesser than 1 lakh 1 is to 10 to the power 5 okay 1 lakh 1 is to 10 to the power 5 if a Reynolds numbers lesser than these this is a critical Reynolds numbers. then flow is laminar which is easy for us okay. We can think it that to get analytical solutions for that that is what we will discuss. So if you have a laminar turbulent laminar boundary layers we can find out these Reynolds numbers because we know the free stream velocity, you know this x value, you know the fluid kinematic properties.

So viscosity, so you can find out whether it does it below this critical number which is 1 lakh, less than 1 lakh. If it is that, that should be laminar boundary. And another critical limit is that for the turbulent boundary layer, turbulent boundary layer is if the Reynolds numbers is greater than 3 millions okay 3 10 to the power 6 just remember it okay 3 10 to the power 6 3 million this is 1 lakh this is 3 million. If more than that you will have a turbulent boundary layers you will have a turbulent boundary layers. Either these turbulent boundary layers we can solve using CFD solvers or we have to work from experiment okay.

Here we do not have any analytical solutions for this. So either you look for CFD or experimental fluid mechanics okay, fluid test okay. Here we in the laminar zones again I will come into basic equations. Here we can do get some sort of analytical solutions okay. So what it happened to these transitional zones which is in between laminar and turbulent zones.

Even in the best CFD substrates today also unable to represent this transitional flow. because when the transitional flow is at there, it is more complex flow patterns happens in these regions. Again I am repeating it. It is very complex flow happens as compared to the turbulent. No doubt the laminar flow is easy to understood it but the transition is more complex as the generation growth decay of the all the patterns happens here. So transitional part is very difficult is even if you today we have a very advanced level of CFD solvers still we cannot solve this transitional case and many of the designer case we try to avoid this transitional zone.

So what we do it but like a in case of aircraft wings things we try to put it a structures such a way that we should try to avoid these transitional zones. Because of this transitional zone there will be acoustic component, there will be other complex components, heat generation component all will be the complexity will come it to avoid that many of the times we put the rings okay some sort of artificial roughness okay you put a trip wires okay. Like if you look at this, this is a trip wire we put it to reduce that transitional zones to make it from laminar to turbulent zone. More emphasis I am giving it please draw the velocity distributions okay. What will be the velocity distribution for laminar, what will be the turbulent and since there is a presence of a trip wire okay is a a artificial resistance we put it so that it initiate early transitions to the turbulence okay that is what we follow it.

And this eddies from the tripwires cause enhanced local mixing and create the disturbance that very quickly lead to a turbulent boundary layers that is what we follow it most of the designing to try to avoid these transitional zones because that is the reasons we put the trip wire to facilitate as quickly as possible from laminar to the turbulent zone. Now if you look at another very interesting part to show you that many of the times we believe it the turbulent the boundary layer formations happens near to the plate okay that is what we presented at some of the books but not necessarily boundary layers happens in many places. Let me I show you to that as we have given this place that this is the boundary layer and you have a these outer layers and these and if you look it these two case of experiment cases that you have a Reynolds numbers is 10 to the power 2 it is okay and 10 to the power 4 okay. So it is almost 100 times the higher the Reynolds numbers okay higher the Reynolds numbers we have Either you can increase the velocity because anyway if you increase the velocity you will have this. So if you look it as the higher the Reynolds numbers these are the experimental finding also analytically also we can get it that trend as higher the Reynolds numbers okay the Reynolds numbers as increased by 100 times okay from this figure to this figure.

the thickness will going to decrease it. So higher the Reynolds number, thinner the boundary layers. That is what you have to remember it or conceptually we try to understand it that when you have a higher Reynolds numbers. that means the ratio between the inertia viscous force okay that is the numbers we are announced is that you can understand it the thickness will reduce it. The viscous force is lesser as compared to the inertia forces. So that is what experimentally also we can see it that when you change the Reynolds number it is almost 100 times okay. the thickness will be decrease it and here showing this again I am emphasizing that just look at the velocity distributions because of the presence of different Reynolds numbers you will give a different velocity distributions profiles.

And beyond these outer regions the velocity will be more less the constants because it has reached to the 0.99 of u 90% of the free stream velocity. So you will not have a much gradient. So this is the the relationship between the Reynolds numbers and the boundary layer thickness. So higher the Reynolds numbers thinner the boundary layers.

As I said it that boundary layer formations happens in many places like for examples you have a jet okay maybe you have a smoke. some jet chemical jet you are injecting it okay just like 2 tanks there is a water jet. So we have a water jet this is the fluid another fluid jet is happening it. So if you have a water jet is happening so you can see that the velocity distributions will be like this and velocity distributions will be 0. So if you look at this this is the boundary layers symmetrically if I look it this is the boundary layers even if of case of a jet flow. If I have a two liquids one is having a jet so as we this jet will disperse it that is what is I can see it that this is and this is what the boundary layer formations.

This is the fluid and this is the fluid may be rest and this jet part this is what is showing me the boundary layers. So and this is the symmetric line for me. This is the symmetric lines and this is the distributions showing me the how the velocity fields are happening. So you can understand it for the jet problems also we can make it as equivalent flow past a flat plates or I just show it I have the cars. So top of the cars I can represent a different regions of the boundary layers I can try to solve it. So, that is the reasons boundary layer the tools to solve many of the problems it is not the tools as presented some of the test book it is just for a flatbed that is not there that is what my point you try to understand the boundary layer concept and that concept you can even if you solve the jet problems you can use the same concept how boundary layers we can form it.

Similarly the wake formations like a boot is moving it as you know it just for your understanding I am putting it to that a wake formations a boot is moving with velocity v then you having the wake formations you can see that any ship is moving it and from the top you can see that wake formations. So that wake formations also can be considered it as you see that this is the velocity and this is the 99% velocity points as a constants then velocity distributions like this, like this. So this is also can be considered as the boundary layer concept. to solve how the wake formations growing are happening it. Just if a move boat is moving the downstream of boat there will be wake formations and that wake formations you can say as equivalent as similar to the boundary layer formations that is what the wake formations we can do it not only that if the tool velocity fields are mixing it okay.

There is two fields are there. One is moving with v2 velocity, another is v1 velocity and this is the interface layers. So if you see that because of these two velocity zones are mixing v1 and v2, v2 is greater than v1 value. So you will have these distributions which will be similar distributions. This is also the same way what we are talking about the boundary layer passing on a flat plate. So boundary layer approximations and the techniques to solve this approximate the Navier Stoke equations not only the flow past a flat plate also we can solve z problems we can solve the wax problems or mixing layer problems. So, with this let I conclude today lectures and tomorrow we will talk about how we can derive this boundary layer equations and some of the approximate solutions. Thank you. Thank you.