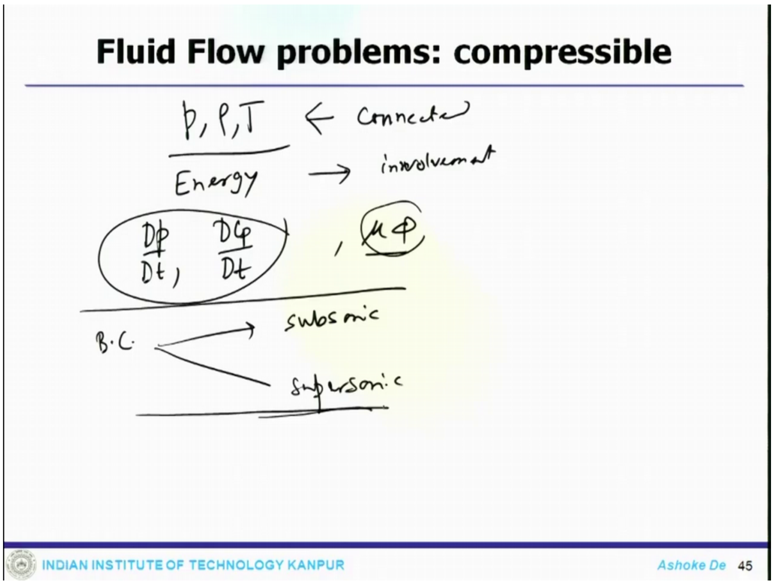
Introduction to Finite Volume Methods-II
Prof. Ashoke De
Department of Aerospace Engineering
Indian Institute of Technology, Kanpur

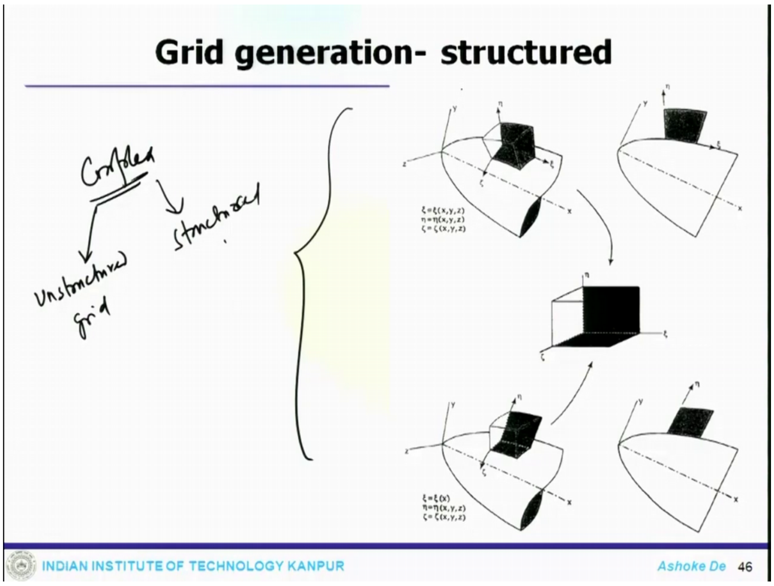
Lecture - 40
Some Advanced Topics-I

So, welcome back to the last lecture of this Finite Volume series and where will continue our discussion where we left in the last lecture.

(Refer Slide Time: 00:23)

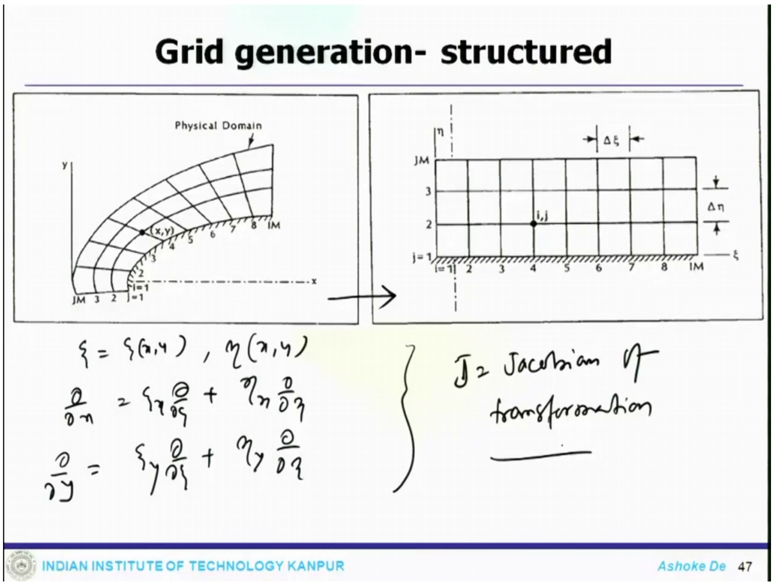


(Refer Slide Time: 00:25)



Now, grid generation if you recall you can actually all your realistic complex grid would be in this nature. When you have a complex grid this kind of a nature either you have option of generating unstructured grid, that is one option or you can generate a structure grid; structure grid then you need to transform this kind of system to xi eta system or the regular coordinate system.

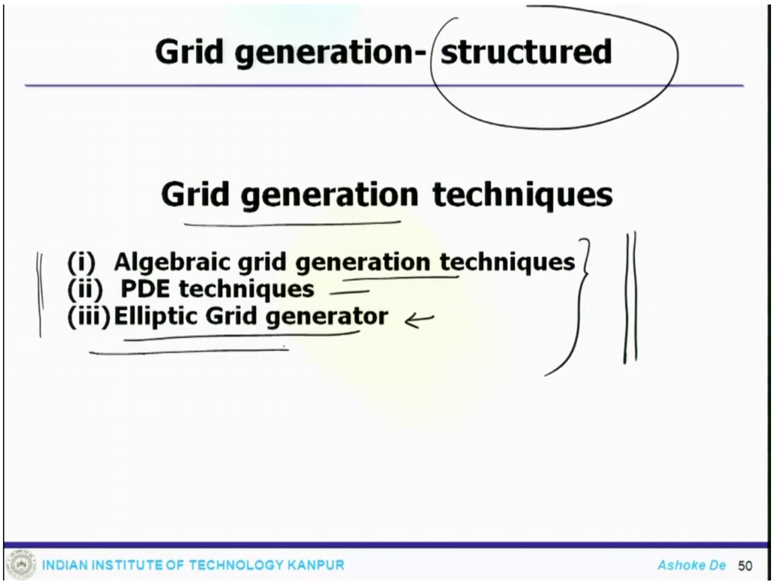
(Refer Slide Time: 00:53)



And how you do that? You have this kind of a physical domain and this is what it transform to computational domain and the weight is done, it is essentially goes to a uniform xi eta direction and as it can be xi x y and eta this would be x y.

So, your del del x would be xi x del del xi plus eta x would be del del eta and similarly del del y would be xi y del del xi plus eta y del del eta and once you transform them then you end up getting a Jacobian of transformation. So, this we have actually discussed when we are talking about gird generation and the transformation or rather talking of the PDEs from there, so this is how one can do.

(Refer Slide Time: 01:52)

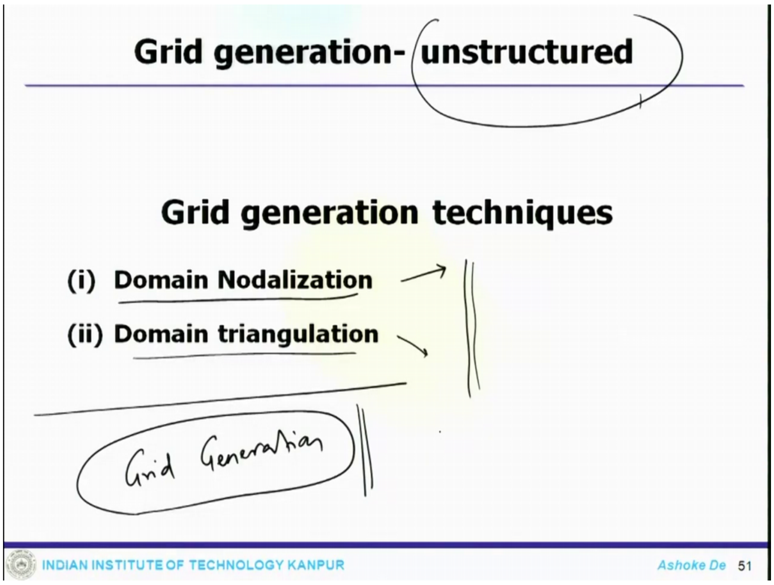


Now, typical gird generations technique if it is a structured system, then either one can use some kind of a algebraic grid generation technique which can take care of this different calculations algebraically and then generate the gird or can solve for the PDE techniques. PDE means you solve for the partial differential equation and then get the well or elliptical gird generator; that means, you solve for some elliptical PDEs.

So, these are the different standard available techniques what one can use and once you and these days there are lot of commercial softwares which are available which is dedicated for the grid generation. And as per your requirement of your code you can generate the grid either one of this technique which is adopted, but it is very important to know how those commercial packages actually works.

They behind their grid generating software or their graphical user inter face they actually solve either of this kind of technique, whether it is a algebraic feet or partial differential equation kind of thing or elliptic.

(Refer Slide Time: 03:07)

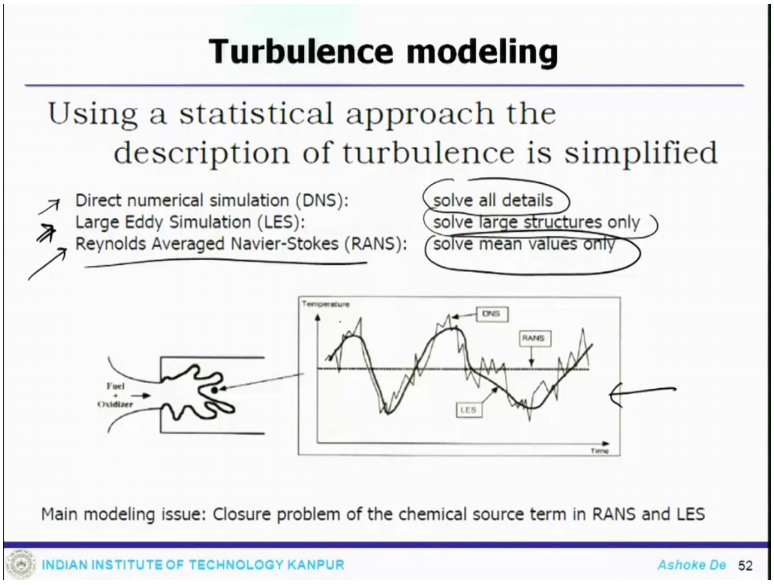


Now the same scenario, gets change when you go down to unstructured grid, here unstructured grid as we have seen throughout this lecture series what kind of element it requires. So, one very common approach is that use domain nodalization approach; that means, you pick a node then calculate the distance and there is a algorithm which is associate with that or you can do a some triangulation kite of approach.

And; that means, you generate some triangular element or do some triangulation and look at the area and then try to generate the grid. So, again your grid generation software when you try to generate unstructured grid it does or takes care one of these algorithm and try to do that.

So, the details of that if you are interested you look at any text book where which is very much dedicated for the grid generation and then find out this. But you have to note that you cannot avoid grid generation, when you are talking about CFD calculation because end of the day your CFD calculation needs an grid which will be the representation of your physical domain.

(Refer Slide Time: 04:26)

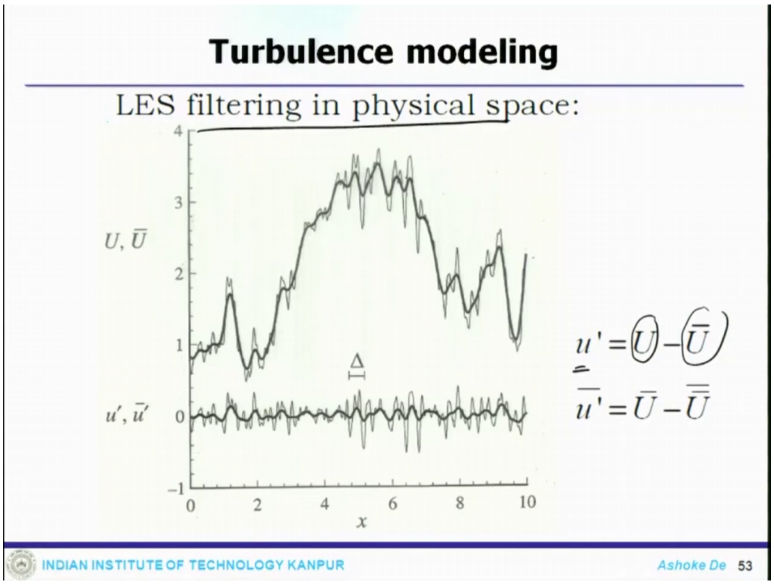


Now, moving ahead most of our practical problems are turbulence in natures and what happen turbulence field that turbulence fields are not like your laminar flow field, so you have different kind of scales. If you look at your instantaneous flow field which is shown here, which essentially is a like of chaotic in nature turbulence in one word you can say that the chaoticness of the fluid particle.

Now, there is a different approaches to model the turbulent flow field, one approach is that you have all sort of small scale which is direct numerical simulation which solve all, that is details; details means it means all the scale or intermediate between DSN.

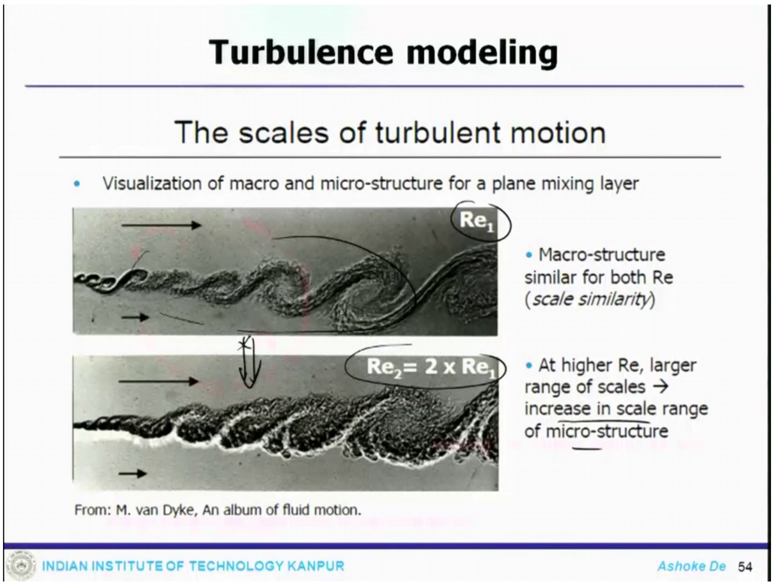
And other case you could have Reynolds Average Navier Stokes equation which can solve we will see how the RANS equations are obtained, solve all the mean quantities and in between that you have LES we solve for the large structures. So, that is how, but theoretically this is how your instantaneous flow field looks like.

(Refer Slide Time: 05:30)



Now, in a LES you actually do some sort of a filtering in physical space and once you do the filtering these gets you back the fluctuating component and instantaneous component which is subtracted from the average quantity.

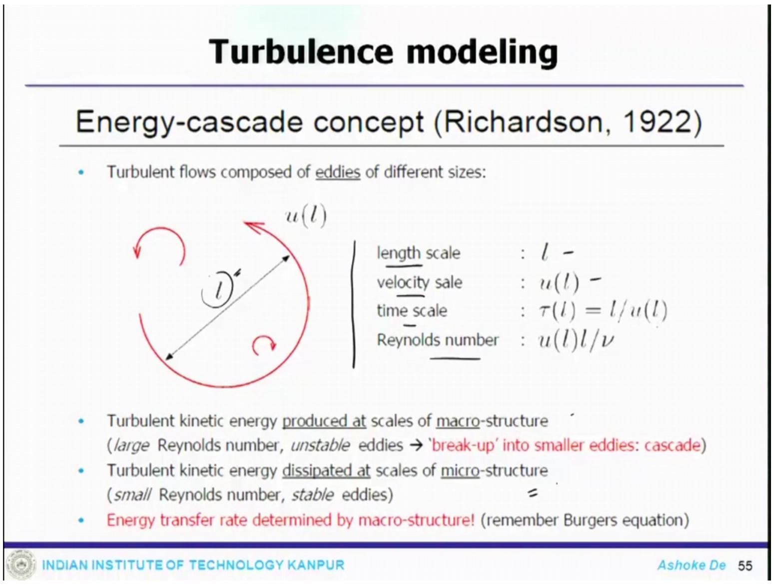
(Refer Slide Time: 05:46)



Now, if you look at the scales this is at the 2 different Reynolds number, this is Reynolds number 1 which is smaller and this is at the double of that you look at that that structures. So, this is a macro structure which are look similar; that means, this structure we are talking.

So, they look quite similar in both the situation, but at the high Reynolds number you have large range scale, so increase in the small range micro structure; that means, we are talking about this structure, these are the microstructure. And turbulent flow field is kind of is a mixture of this macro and micro structure or build in that.

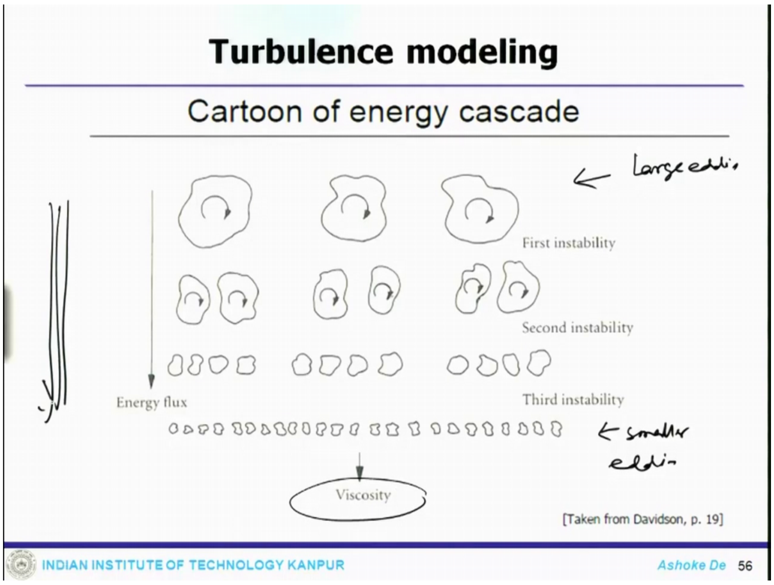
(Refer Slide Time: 06:25)



So, how the energy goes from 1 to n? So, whatever field is there these are the structures are call eddies or vorticites and this eddies are of different sizes. Now that depends what is the maximum characteristics lengths of eddy, this is possible or can be defined based on your geometry; because if you have a channel, the characteristics length or which is the maximum possible size of eddy would be the channel diameter.

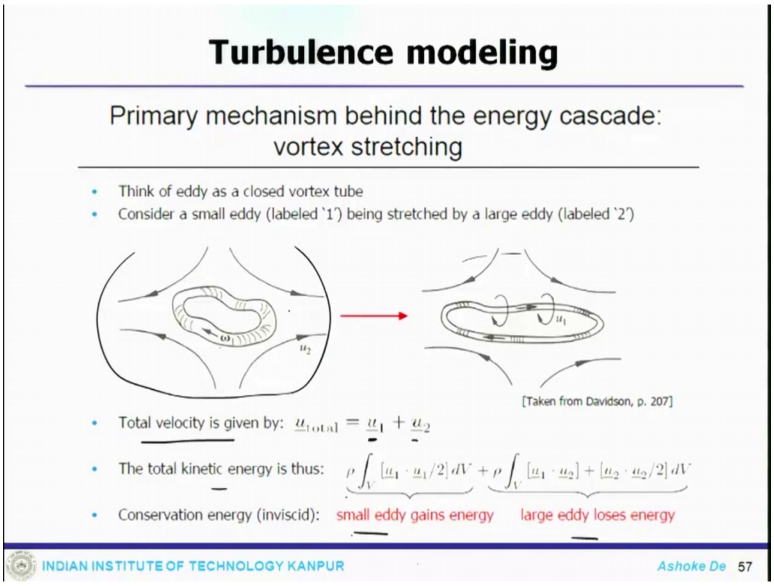
So, what you define one is length scale l, velocity scale u l time scale and the Reynolds number. So, turbulent kinetic energy is essentially produced at the macrostructure, then it breaks up to the smaller eddies and then smaller eddy to smaller eddy and finally, it dissipates.

(Refer Slide Time: 07:17)



So, the energy cascading process is like this, from large structure so this is the large structure or large eddies to go down to the smaller eddies and smaller eddies from their through viscous dissipation it actually dissipates. So, this is the energy cascading process from the large scale to small scale.

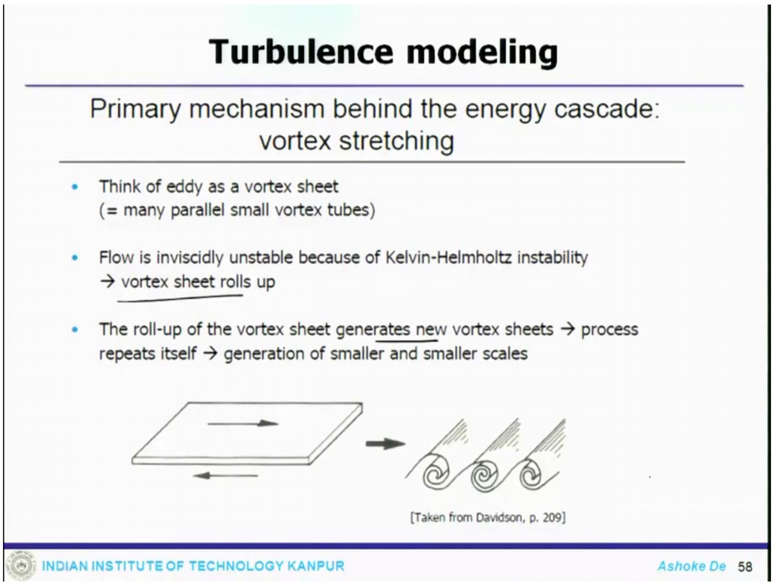
(Refer Slide Time: 07:40)



Now, once you say that what is the primary mechanism? The primary mechanism lies behind this energy cascading process is the vortex stretching mechanism, which is essentially talking about you think about a close vortex tube like this which is shown here. And then, you have a small vortex which is level w 1 and which is stretched at this condition and then this looses.

So, total velocity which is given is u 1 plus u 2 and the kinetic energy is estimated at half rho u 1 square plus u 2 square. So, the energy conversion is that small eddies from this large to this; this gains the energy and this looses the energy.

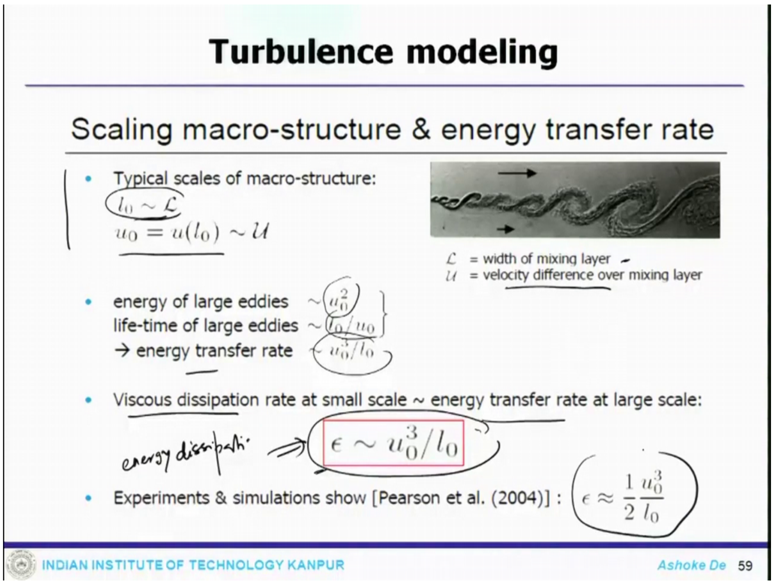
(Refer Slide Time: 08:47)



So, it's essentially the vortex stretching mechanism from this to this the energy cascades. So, once it's happens the smaller one actually gains the energy and larger one actually loses at the energy. So, one can think about is like an vortex sheet. So, many parallel small vortex tubes like that and now see flow is in viscidly unstable because of the vortex sheet rolls up.

Now once that rolls up, it generates a new vortex sheets and from this process continuous and this generation of smaller and smaller scales. So, that is how from larger scale to smaller scale this process repeats and it generates the different scale of vorticity.

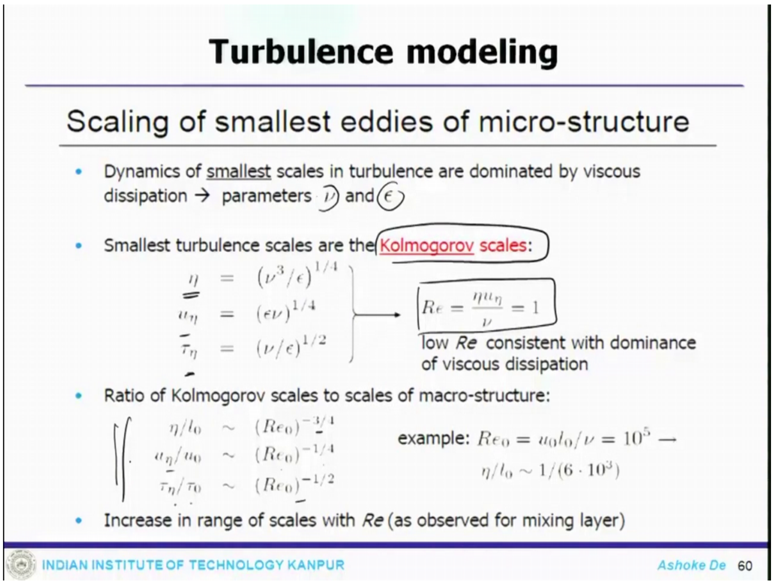
(Refer Slide Time: 09:28)



Now, if you need to scale this energy and the microstructure rate. So, here the picture shows how the large scale eddies are form. So, this is the width of the mixing layer u is the velocity then the macrostructure the length scale would be the characteristic length scale. As one can think if you have a channel, then this could be the order of the diameter of the channel and the velocity at the inlet.

Now, the energy of the large scales are order of u naught square, time scale of the larger is l naught by u naught and energy transfer rate is u naught q by l naught. Now the viscous dissipation rate at small scale must be of the same order energy transfer rate at the large scale, that is how the energy balance is maintained. And epsilon is can be estimated which is known as the energy dissipation or kinetic energy dissipation that can be estimated like this. So, what shows that it could be of this kind of.

(Refer Slide Time: 10:41)

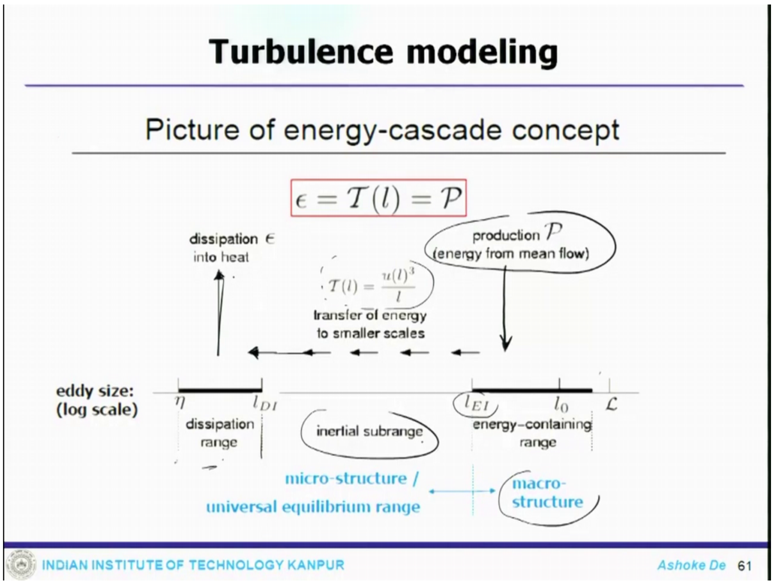


Now, if you scale down the small scale of the microstructure, then the dynamics of the small scale in turbulence are pretty much dominated by the viscous dissipation. So, your kinematic viscosity and epsilon come into the picture and the smallest scales are we are termed as the Kolmogorov scales because Kolmogorov provided that hypothesis.

So, their length scale would be of which we say eta which is estimated as nu cube by epsilon to the power 1 by 4 u eta that is the velocity and time scale and the Reynolds number at the eta scale is eta u eta by nu. So, and the ratio of Kolmogorov scale to the macrostructure if you one has to find out one can find out this length eta by l naught is Re naught to the power minus 3 by 4, u eta by u naught is Re naught to the power minus 1 by 4, tau eta by tau naught is this.

So, increase in the range of scale as with increasing Re. So, if you have a higher Reynolds number case your scales become smaller and that is what you need to essentially your computational grid needs to take care of that small scale.

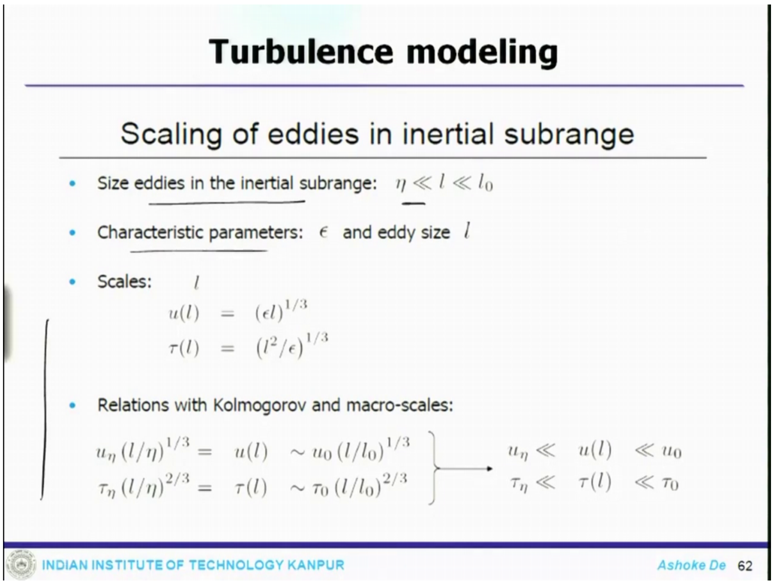
(Refer Slide Time: 12:02)



Now this if you put together in a nicely picture this is how the energy cascading actually takes place. This is the characteristics length scale and then you have a energy containing range or this is your macrostructure, where all the kinetic energy is produced. Then from their you come this is called the inertial subrange, where the energy transfer process follows and you finally, get this length scale and it comes down to the dissipation can.

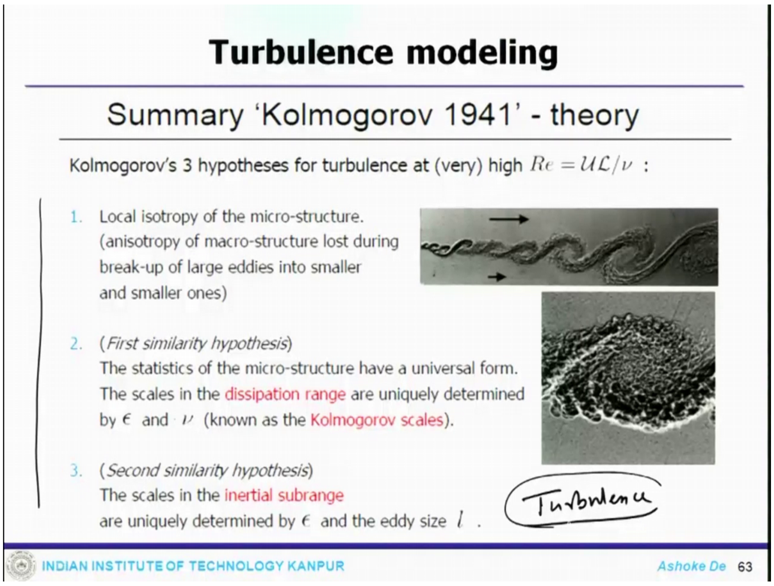
So, the between dissipation and inertial there is a demarcation line and this is between inertial and energy there is a demarcation line. And these are the microstructure or universal equilibrium structure and this goes to the dissipation. So, one hand there is a production energy transfer takes place through the smaller one and then it goes up. So, that is how from small scale to large scale the energy transfer process takes place.

(Refer Slide Time: 13:06)



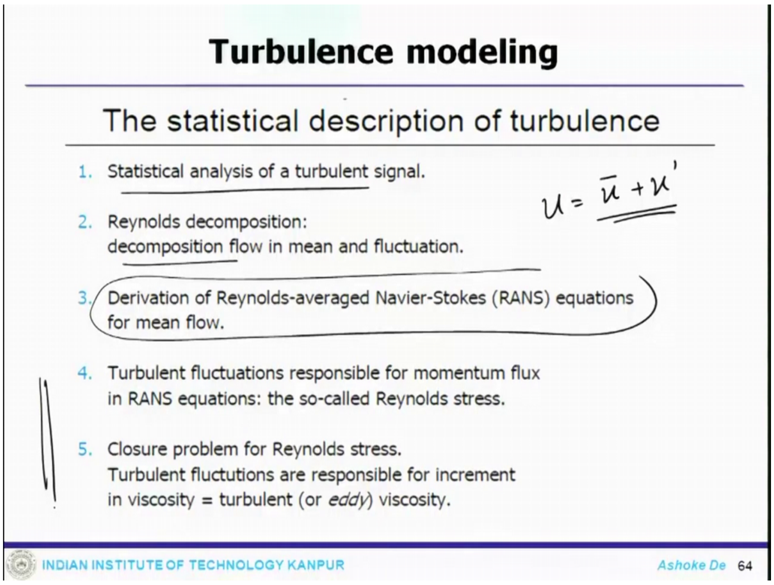
Now if you scale that one scale the inertial range in the like this and you can characterize these things.

(Refer Slide Time: 13:16)



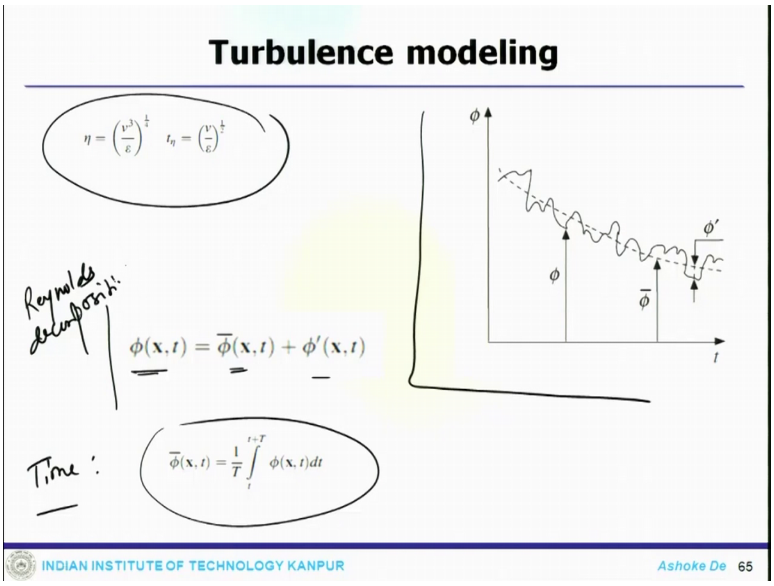
Now, Kolmogorov provided this theory that 3 hypothesis one is that local isotropic of the microstructure, then the statistics of the microstructure have a universal from. So, that is why the scales in the dissipation range are quickly determined. So, any textbook on turbulence you will find these details these are nothing new just to give an idea how turbulence happens before you.

(Refer Slide Time: 13:41)



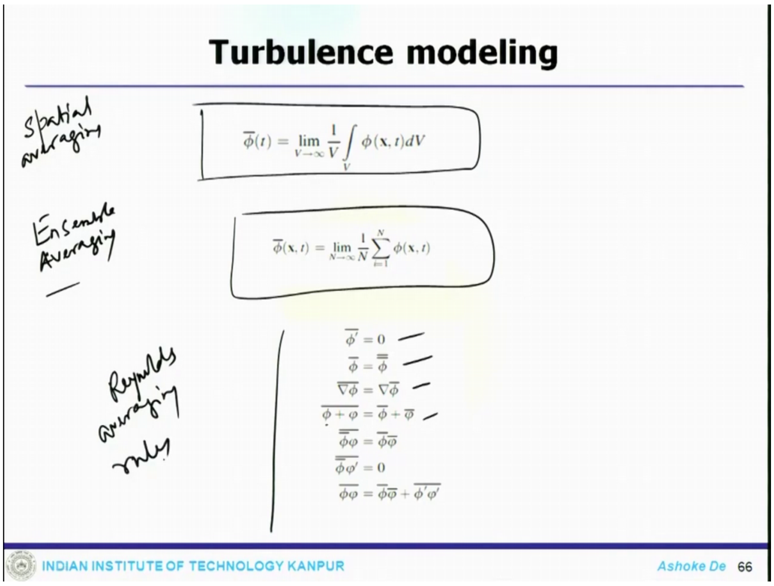
So, how one can actually describe? It has to be described statistically the Reynolds propose some decomposition of the mean flow field you have been instantaneous flow field. So, you can decompose on the mean and fluctuating component. So, which will actually lead to the RANS equation or Reynolds Average Equation, we can see that how it is done and then you get some turbulence model to close that down.

(Refer Slide Time: 14:08)



So, this is how your small scales are defined and this is your instantaneous flow field. So, Reynolds averaging cell any flow variable would be 1 mean and fluctuating component and if you do time averaging, this is time averaging this is how you evaluate that. So, this is the Reynolds decomposition.

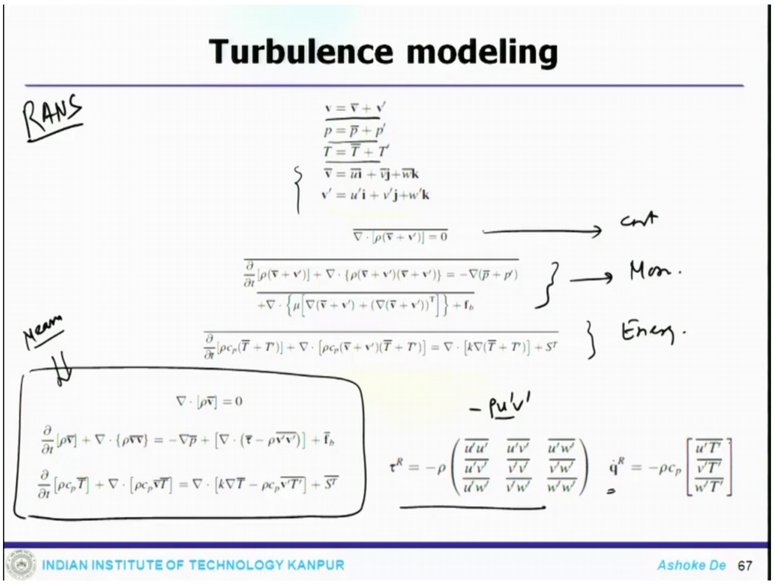
(Refer Slide Time: 14:37)



Then you also can do like spatial averaging, so this is called it's a different ways one can define the averaging and then other way one can do ensemble averaging this is called ensemble averaging. That means, you collect the sample multiple times and divide by that that gives you, there are certain averaging rules for the RANS one is that if you take the average of the fluctuating component that is 0 double bar of that that same.

If you take the delta phi of the average it is delta phi bar, phi plus phi bar it's like that. So, these are the certain Reynolds averaging rules which follows this.

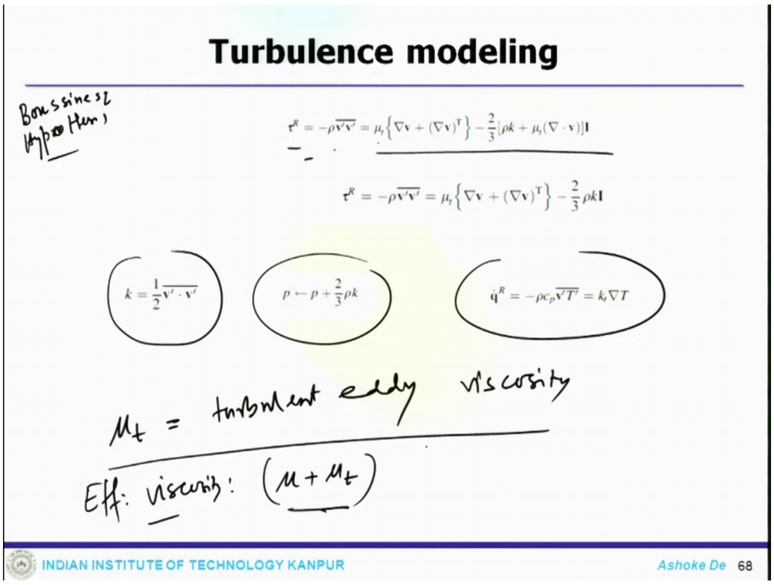
(Refer Slide Time: 15:28)



So, once you try to drive your RANS equation so; that means, you have a flow field which will be average plus fluctuating pressure is p plus p prime temperature is T plus time prime and these are all component. If you put these things back in your continuity, momentum equation and energy equation your continuity equation looks like this; this is your continuity this is your momentum equation.

So, all the variables like pressure velocity, temperature again this is what we are writing for incompressible cases because the density variation is not taken into consideration and this is my energy equation. And after applying those law the Reynolds average form of the equation will look like this, which will only solve for the average quantity or rather mean quantity. So, its solve for or mean profiles and the term which will appear there is a Reynolds stress term in the momentum equation which is a tensor 3 by 3 matrix and same thing and the energy flux which will be a cost term of the u prime T prime, v prime T prime and w prime T prime.

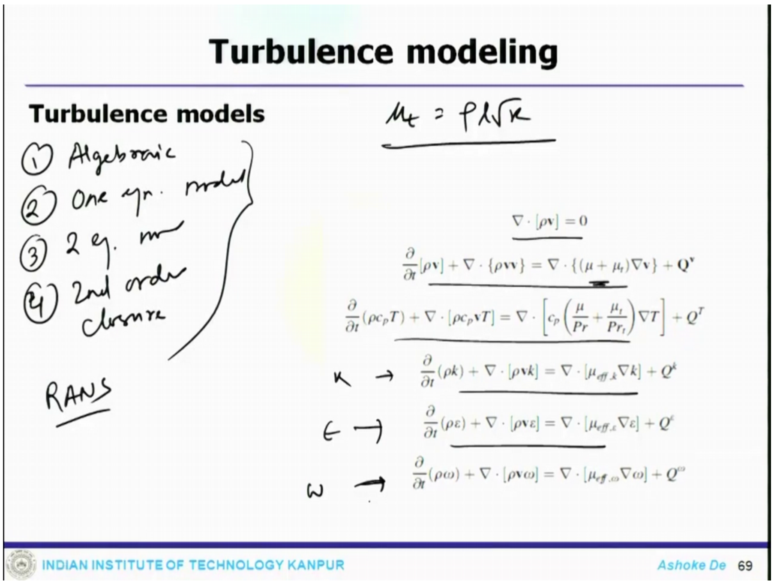
(Refer Slide Time: 16:40)



Now one important hypothesis which is used is the Boussinesq hypothesis to close this terms. So, this Reynolds stress term is correlated like turbulent viscosity and molecular viscosity and your kinetic energy is computed like this and at the same time your pressure Reynolds stress is usually come and with the pressure gradient term. So, the turbulent pressure would be like this and your thermal fluxes are calculated like that. So, this Reynolds stress term is become and quantity which can be correlated with the mu t and this mu t is the term which is called the turbulent eddy viscosity.

So, this turbulent eddy viscosity now everything boils down to the system, if you look at the system here it will be a diffusion term which will have some effective viscosity mu plus mu t; that means, the this is your effective viscosity in your momentum equation. Similarly in your thermal equation there will made effective conductivity now how to calculate this? So, that gives rise to a different kind of turbulence model.

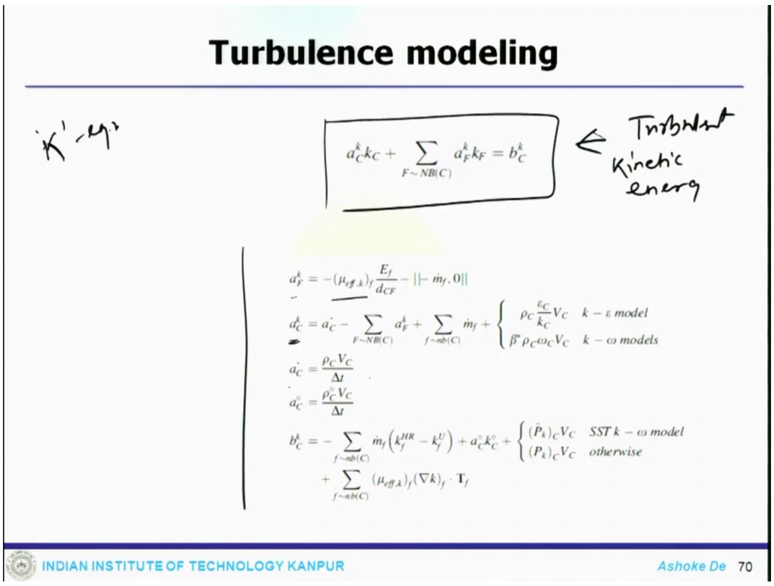
(Refer Slide Time: 18:01)



And if you have mu t it is estimated like this and one can have algebraic turbulence model, one can have one equation model, one can have 2 equation model and then one can have 2nd order closer model. So, that is are different kind of RANS model one can see. So, if you look at the complete picture of the RANS equations.

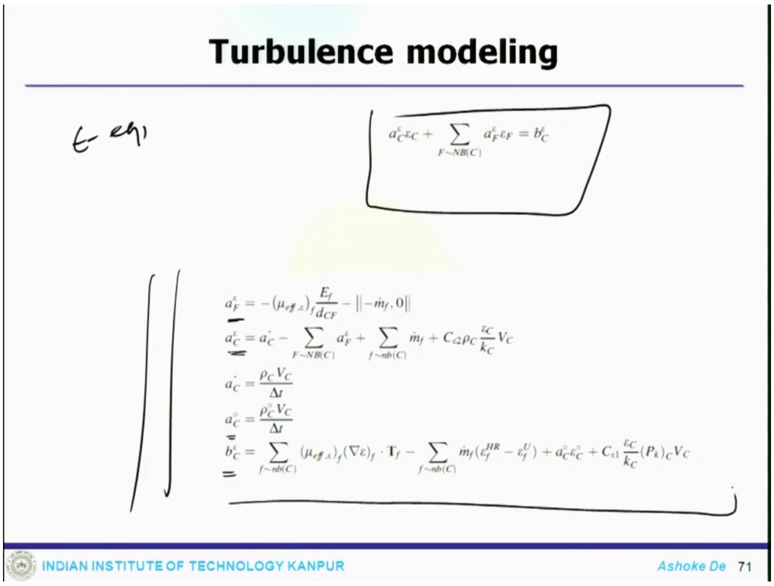
So, there will be continuity equation, momentum equation, energy equation, now this is kinetic energy equation which you can see there is a effective viscosity, epsilon equation which is k equation kinetic energy, epsilon equation and turbulent frequency omega.

(Refer Slide Time: 19:02)



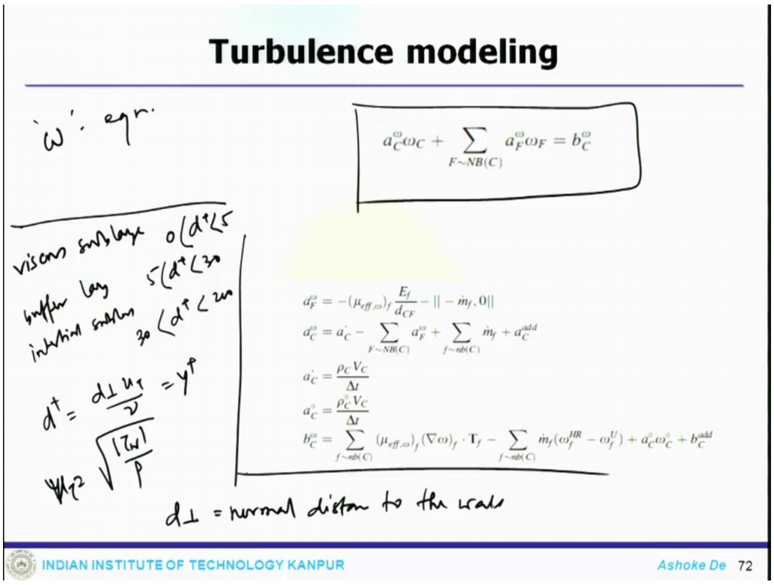
So, these are the set of equations and once you discretize them in your finite volume this is exactly what you k get for the this is for turbulent kinetic energy. So, again the equation looks same, here the coefficient for kinetic energy would be mu effective E f by this; this is the a C then time transient term. So, these are all coefficients which you get for the k equation.

(Refer Slide Time: 19:36)



Now, similarly one will get an equation for epsilon, this is epsilon equation and accordingly your coefficients will be modified as per the epsilon equation. This is your a F, a C a C previous term step and b source term.

(Refer Slide Time: 19:54)

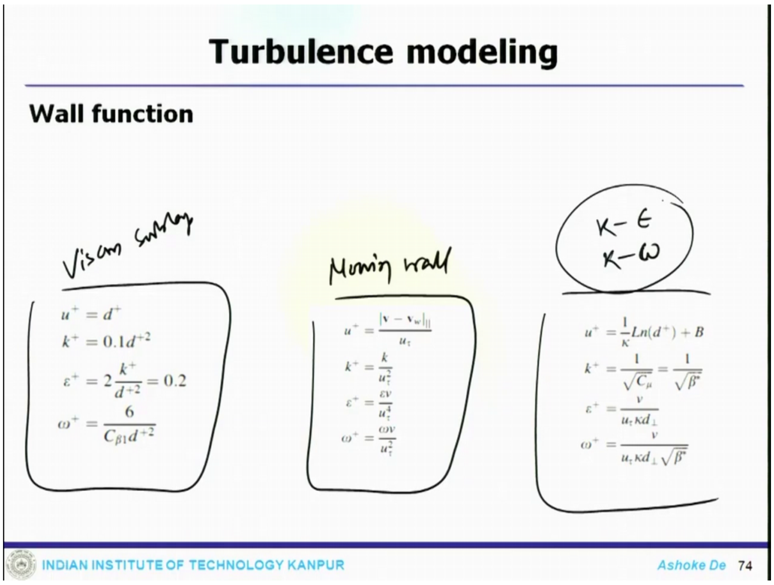


So, this will return all the term one can think about the omega equation 2, so all these equations will appear there. Now one important thing it will happen is that calculation of the wall function. Because all this turbulent equations these are having used and that time and the thing is that near the wall flow field resolution is very important, if you have a domination of the viscosity.

So, to resolve that one has to typically there are three different layers where the layers can be divided. One is called the viscous sublayer and where your distance or the your normal distance is defined d plus less than 5, then your buffer layer buffer layer where d plus lies between 30 to 5 and then inertial sublayer.

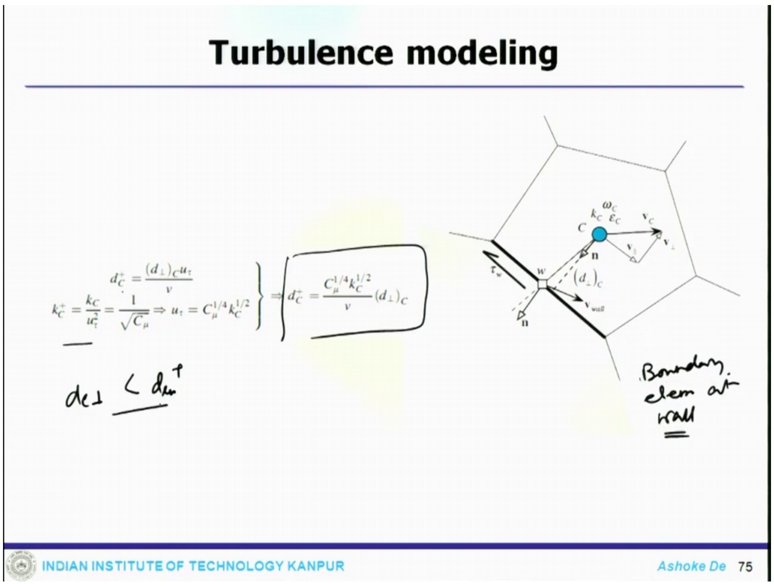
So, where d plus lies between 200 to 30 and d plus is estimated like that d perpendicular distance by u tau by nu which is y plus and u tau is nothing, but tau at the wall by density and d is the normal distance to the wall. So, that is how you can calculate.

(Refer Slide Time: 21:30)



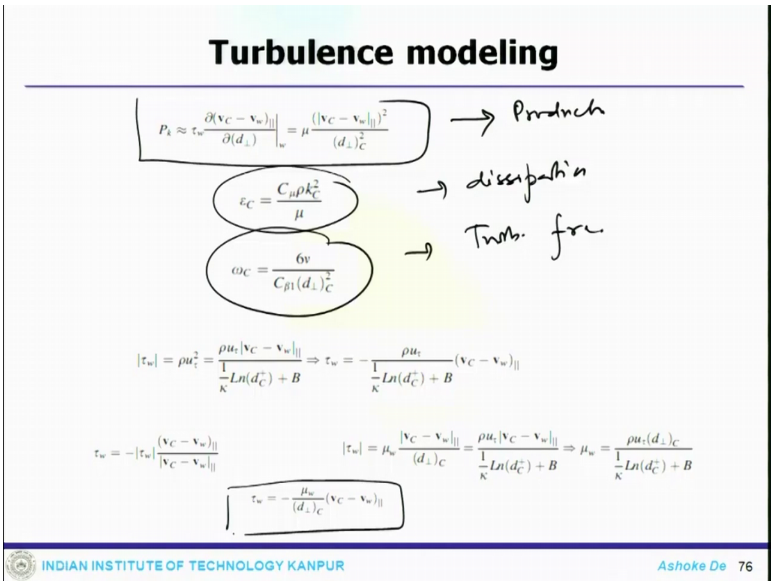
Now, if you go to, so sublayer, then this is how u plus and y plus they are connected and if you have a moving wall, then u plus is connected like that viscous sublayer and for k and epsilon typically this is what done k or omega. So, for 2 equation models this is how it is done.

(Refer Slide Time: 22:01)



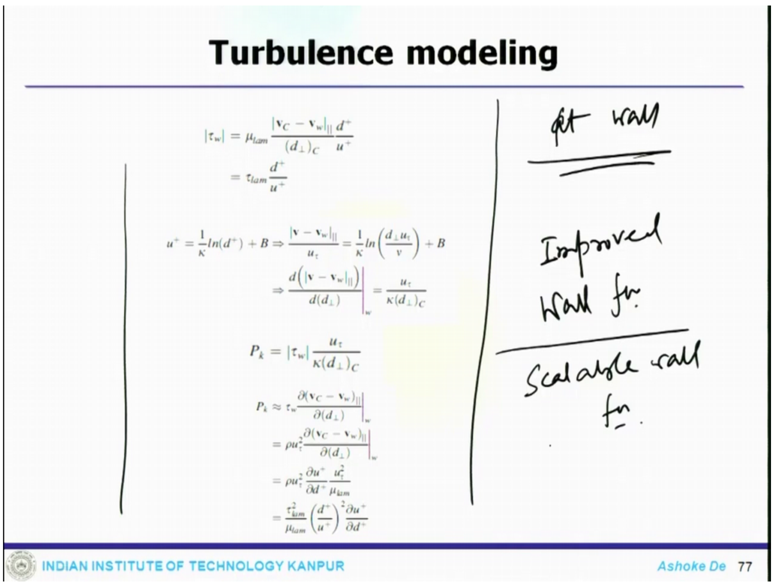
Now, one can see when it comes down to a boundary element this is a boundary element and I mean where close to the wall and you have to boundary element at wall because the normal calculation needs to be done. So, one can calculate the normal distance d c plus like d c u tau by nu and kinetic energy like that; so, which will get you back d c like C mu k c and C mu k c. So, as long as d c perpendicular less than d limiting plus the point lies in the viscous sublayer, otherwise it will be in the initial sublayer.

(Refer Slide Time: 22:41)



Similarly, this is the turbulence production term and this is the dissipation term which can be estimate at that cell. And, this is the turbulent frequency term and one can estimate the wall shear stress also.

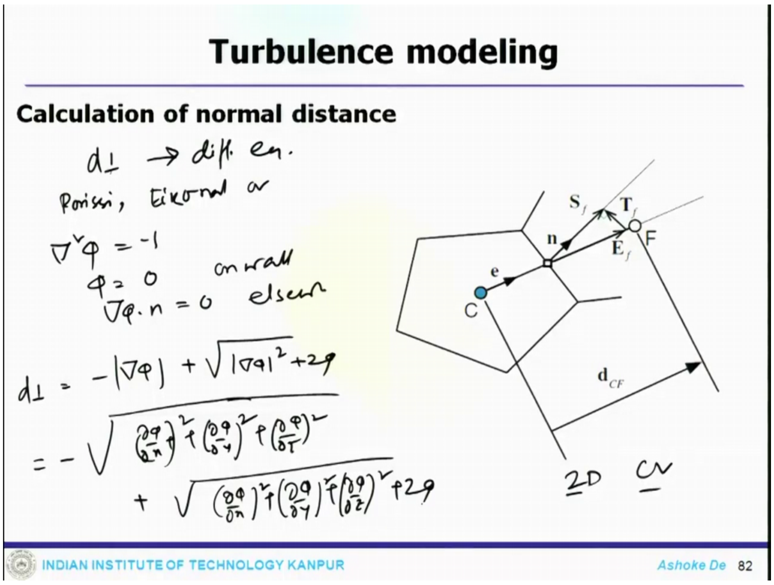
(Refer Slide Time: 23:04)



So, one you estimate all these I mean these are the calculation for that and one can actually get all this calculation for at wall this information can be calculated. Now, the other way one can think about using some sort of an improved wall function because what happens that sometimes the grid resolution is not sufficient.

So, one may use the improved wall function and you can actually derive these equations for improved wall function or thirdly one can think about some scalable wall function. Essentially what you do? You need to calculate the normal distance and once you calculate the normal distance to the wall then you can solve for it. Now, how do you calculate the normal distance? That is the critical path.

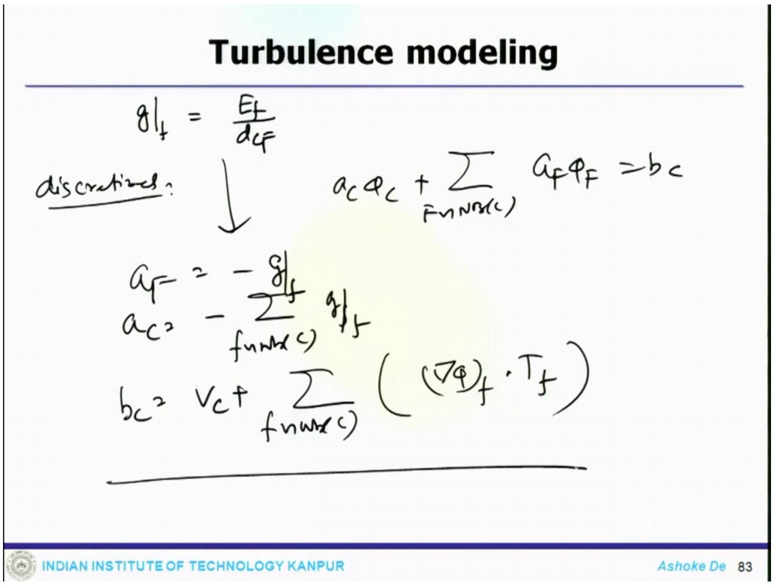
(Refer Slide Time: 24:05)



Let us see a 2 dimensional control volume or 2 dimensional control volume, how do you calculate the normal distance? So, the normal distance can be calculated is just using some sort of an based on solving a differential equation. So, this can be calculate by solving a differential equation and which could be solving a like and Poisson, Eikonel or Hamilton Jacobi kind of equation.

So, if you look at the Poisson equation del to phi equals to minus 1, then which will be boundary condition it's 1 wall and del phi dot n equals to 0 elsewhere. So, the normal distance can be calculated equals to minus del phi magnitude of that plus del phi square plus 2 phi which is minus square root of del phi del x square plus del phi by del y square plus del phi by del z square plus square root of del phi by del x square del phi by del y square and del phi by del z square plus 2 phi.

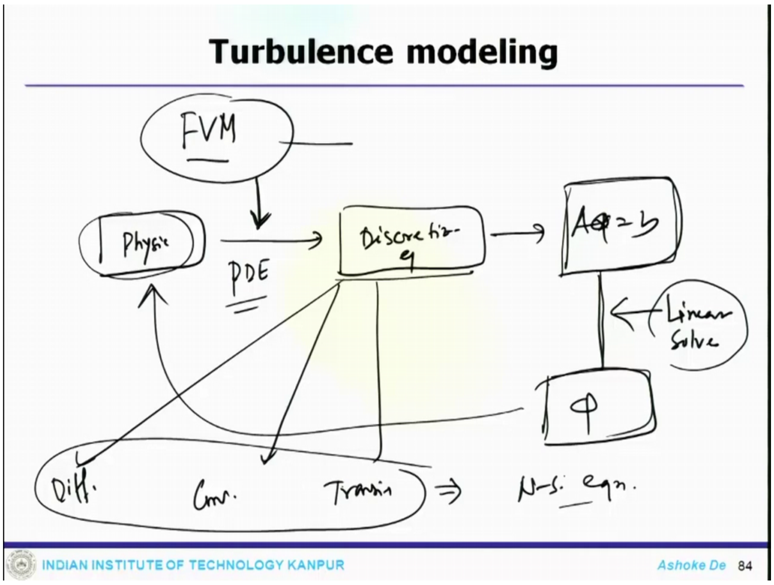
(Refer Slide Time: 25:29)



So, and if you solve the turbulent flow problem and use everything in the in your in your governing equations, you get this geometric diffusion coefficient; geometric diffusion coefficients at the face is E f by d cF. And your discretized equation would get modified like a c phi c plus face n b c a F phi f is equals to b c, here a F equals to minus g f which we got here then a c equals to minus summation over f g f and b c equals to V c plus f which is del phi f dot T f.

So, that is how you can calculate the normal distance. So, essentially that talks about how you implement all this things in your system of equation and if you that also brings down to the to some extend the closer of this lectures.

(Refer Slide Time: 26:43)



And what we have actually talk the our numerical scheme based on finite volume method. And, then we talked about the classifications of the PDEs and all this things. Initially for the finite volume method the prime objective of any CFD is that you have a physical problem. So, that transform to a discretized equation applying this finite volume discretize equation; discretize equation which will nothing, but a linear system a phi equals to b, then apply a linear solver which actually get you a solution and that solution should be representing to physical problem.

So, the global picture what we started off if you close it down you have a physical problem of interest in hand you apply your numerical technique the one which we have discussed here is the finite volume, you get and discretize equation that is essentially leading to a linear system. And then we discussed all different kind of linear solver, like iterative solver, direct solver and which will get you the solution.

Now, of the physical problem which is governed by this PDEs they are actually, then the PDEs are essentially we talked about all different kind of discretion where diffusion separately, convection separately, transient separately. And we have taken care of everything together when you combined everything back to the Navier Stokes equation for fluid flow solver.

And when you go down to the fluid flow solver we have actually used our incompressible solver, then we talked about the pressure velocity coupling and the interpellation then finally, talking about simple and simple Fermi based algorithm. Then we move to the compressible where can we discussed about the discretisation and boundary condition and finally, you test upon the grid generation and the turbulence modeling. And that concludes the discussion on finite volume and I hope it will be helpful to get started with finite turbulent method.

Thank you.