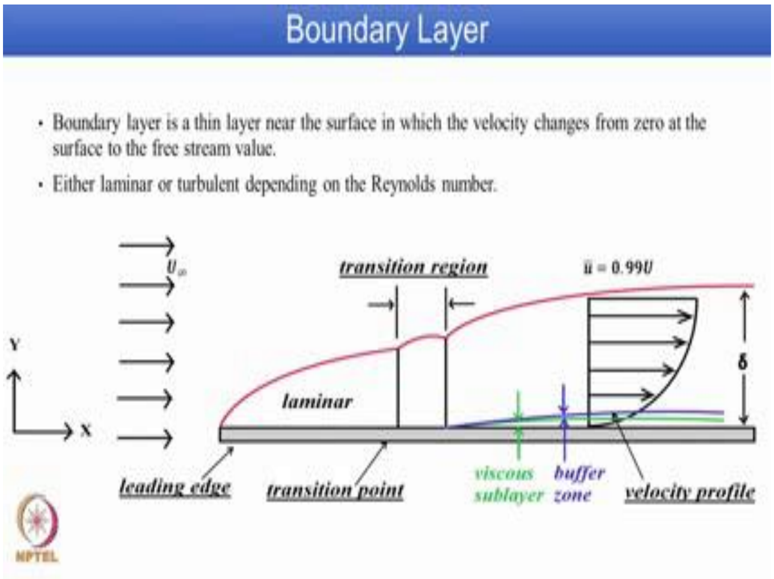
Foundation of Computational Fluid Dynamics
Dr. S. Vengadesan
Department of Applied Mechanics
Indian Institute of Technology, Madras

Lecture - 32

(Refer Slide Time: 00:21)

Greetings and welcome again to this course on CFD. Last class, we had seen deriving governing equations, Reynolds stresses, modelling strategy. In particular, we came to know about standard models, standard K epsilon and standard K Omega model. We also listed different proposal available in the literature in standard K epsilon and standard K omega model to improve the performances one such proposal was low Re model, and today's class in particular going to see about low Re model.

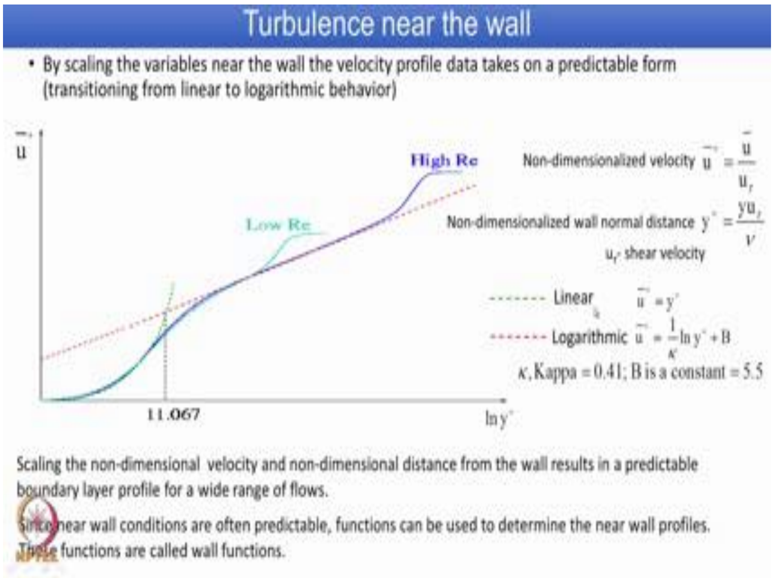
(Refer Slide Time: 00:54)



Before going to the detail, I would like to present what is known as the boundary layer and what is shown here is a flat plate, flow is coming from left to right with the uniform velocity condition U infinity, and this is the leading edge, flow develops over the flat plate, initially you have a flow in laminar then there is a transition point then you have turbulent boundary layer. The turbulent boundary layer has three main region one is viscous sub layer as marked here then buffer layer then log layer and boundary layer thickness is given by a symbol delta and boundary layer is defined as velocity which is equivalent to 99 percent of free stream velocity or incoming velocity 0.99 of U. So, boundary layer is thin region, and near the surface in which velocity changes from zero at the surface as show here to the free stream value as shown here. So, this may be either laminar bound layer or turbulent boundary layer depending on Reynolds number.

Now in this case, Reynolds number we use velocity as a incoming velocity and the lens scale is the distance from the leading edge; obviously, if you are very close to leading edge, the Reynolds number is small; and if you go very far, the Reynolds number is high, the flow actually becomes turbulent. We have seen now flow past a flat plate the different regions that is laminar, transition and turbulent region. In the turbulent region, we have further sub details, we have three the different layers viscous sub layer, buffer layer and then log layer. So in the next slide, I am going to explain in detail about these three different regions in a turbulent boundary layer.

(Refer Slide Time: 03:06)



Before going to the details, we should first know what is the scaling; scaling the variables near the wall. We have lens scale and then velocity scale. In the velocity scale, we have shear velocity u tau define, u and tau is perfection or shear velocity and that is used to scale velocity. And distance from the wall is give as y and nu is molecular viscosity. So, we define non-dimensionalized velocity u bar that is a mean velocity and because it is non-dimensionalized we use superscript plus. So, u bar plus equal to u bar by the velocity scale that is used to non-dimensionalized that is u tau; u bar by u tau is actually u bar plus.

Similarly, you can also non-dimensionalized the distance, so if you non-dimensionalized the distance, it is becomes y plus. It is a non-dimensionalized wall normal distance. So, actual distance is y, we use u tau as a velocity scale and nu is molecular viscosity. So, if we put them together, you get non-dimensionalized wall normal distance which is written as y plus. So, in this plot, we have horizontal axis, which is given logarithmic scale; and vertical axis, which is define the usual linear scale. So this plot is a semi log plot; on one side, we have a linear scale and other side you have a logarithmic scale. We have multiple curves with the different colour. We will explain each one of this very carefully.

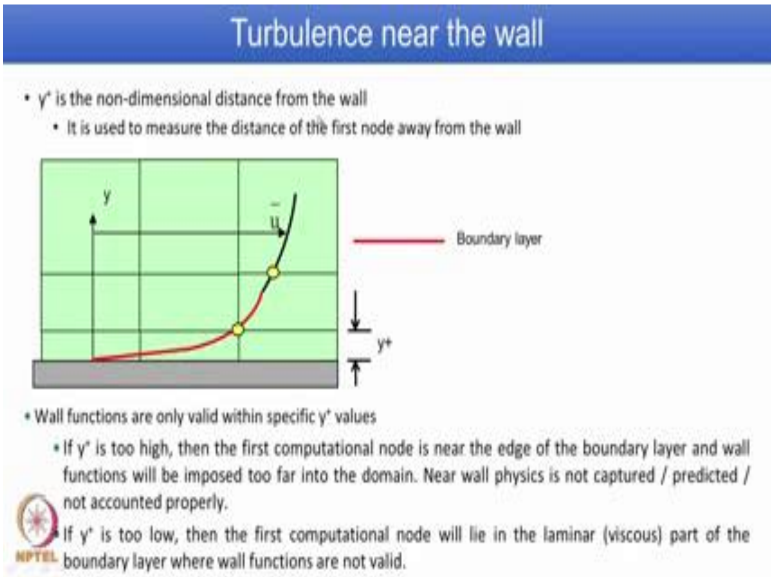
The first one is you have dash line, which is shown in the green colour; and you have actual profile running as shown in blue colour. This blue colour is actual velocity profile then we have one green colour curve which is superimposing over the actual profile for some distance from the wall all the way up to some value of y plus. So, when you say y plus is plotted on the x axis, so this becomes wall that is y plus equal to zero; and this green colour is empirical relationship and that is given by the relationship as u bar plus equal to y plus. So, this green colour line is the empirical relationship and that is a linear relationship, because we are using semi log plot, it appears as a curve. Then the next zone is buffer zone, we will not talk about it right now, we have the log region, third zone is a log region and the expression for the log region is given here that is u bar plus equal to one over kappa ln y plus plus B; kappa is a constant which is given a value 0.41 and B is another constant which is given a value 5.5.

And this expression, as you can observe is the semi log expression, because on one side, we have a straight value; on the other side, we have a logarithm expression. And on this semi log plot, this kind of expression will appear like a straight line and that is what is this region, which is marked as a red colour. So, this red colour straight line that is appearing it actually representation this logarithmic expression. So, the actual profile which is blue colour is partly coinciding with the red colour for some part of y plus, you can observe here it is coinciding with the red colour from this value of y plus to this value of y plus. So, then actual your profile deviates from that expression now between this linear region, where the actual velocity profile coincides with the green colour line, so that is a linear region. The another region is a log region that is somewhere here in between you have the region that is what is known as a buffer region.

So, we have three region, starting from the wall linear region, it is otherwise called viscous sub layer region; buffer region which is in between then we have a log region where we have a semi log of expression relating velocity and wall normal distance as shown here. After some point of y plus, it is deviating from the logarithmic expression, so this plot is very, very special because we are using non-dimensionalized velocity and non-dimensionalized wall normal distance. It is special, because it is universal whatever the type of flow because we are non-normalizing with respect to shear velocity u tau with respect to viscosity, this plot becomes universal and that is the speciality of this plot.

Scaling the non-dimensional velocity and non-dimensional distance from the wall will result in a predictable boundary layer profile for wide range of flow, because we are using respect u tau and respect to nu, and you already learn it is possible to split the actual velocity profile into three zones linear velocity profile or viscous sub layer region, buffer layer, logarithmic layer. And there is a specification expression available for two zones and for outside log layer, it is possible to predict velocity profile near the wall. Since the near wall conditions are often predictable, functions can be used to determine near wall profiles, and these functions are called wall functions. For wall functions are used to bridge the gap between wall and the first nodal point. We are going to see in the next slide some detail about it. Important point from this plot is universal which means it is applicable for wide range of flows, because for different flows you get different u bar and u tau, but you can non-dimensionalized, and they all fall in the same plot.

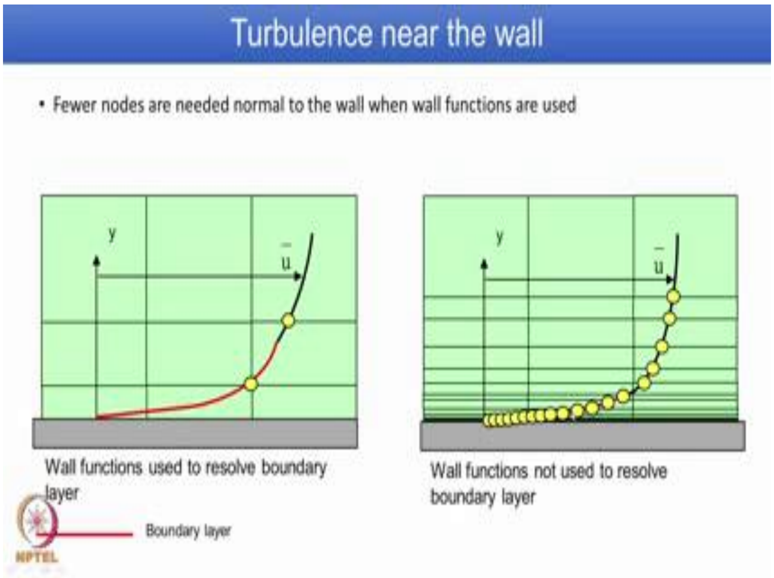
(Refer Slide Time: 10:33)



Y plus is a non-dimensionalized distance from the wall as we said before and it is used to measure the distance from the first node away from the wall; not necessarily, it is used for other point as well. Now here is the description the shaded position is the wall and these are all mesh horizontal lines, horizontal thick lines, and vertical thick lines are supposed with the mesh and here is the boundary layer profile. And if you have the first node that is shown here is a yellow circle that the first node for example, in your computation from the wall then that is give y plus of the first node. Wall functions are only valid within specific y plus values, if y plus is too high that is it happen to have first point somewhere here instead of here or if you have if you happened on the first point quickly far away from this point, or it is very far from the wall.

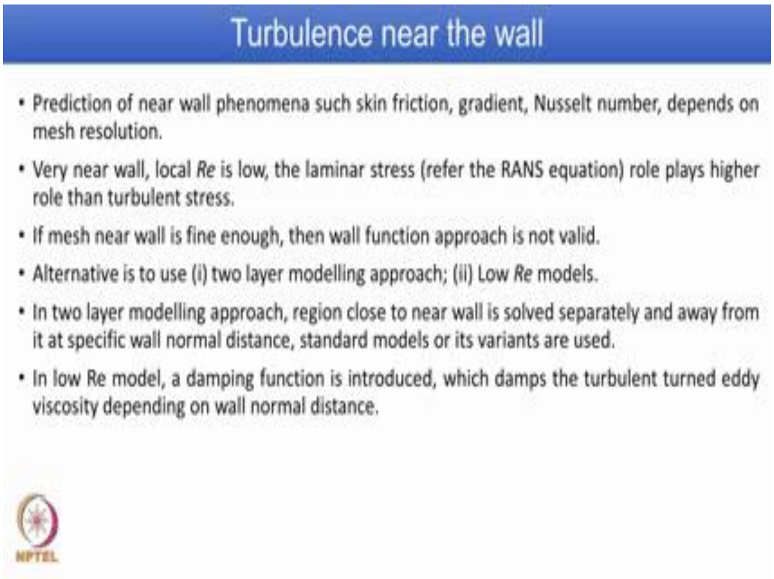
Then the first computational node is near edge of the boundary layer and wall function will be imposed too far in the domain; near wall physics is not captured, predicted or not accounted properly. If y plus value is too low that very close to wall, then the first computational node will lie in the laminar or viscous part of the boundary layer where wall functions are not valid. Now, at this point again left reminder equation we derived mean momentum equation, in the mean momentum equation, we had three terms on the right hand side first term for the pressure, second term for the laminar stress and the third term of the turbulent stress. Depending on the contribution of the laminar stress and turbulent stress then y plus will have the influence. If y plus is very close to the wall, then we have a laminar viscosity playing role there wall function is not suitable; if y plus is very far, you are actually outside the boundary layer there also will not have any influence.

(Refer Slide Time: 12:55)



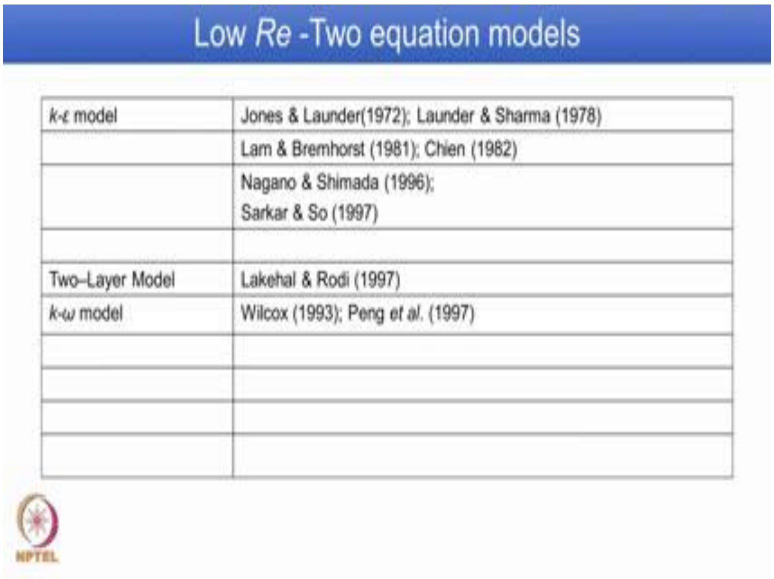
Fewer nodes are needed normal to the wall when all functions are used. So, you can use for points define as shown here very close. And you can relate physics at this point with the wall or the region between first point on the wall, the physics associated the region is represented by what is known as a wall function. If you happen have a luxury of very fine mess near wall as shown here, then wall function cannot be used because you are able to integrate all through the wall layer up to the point on the wall.

(Refer Slide Time: 13:40)



So the prediction of near wall phenomenon, for example, you are interested in skin friction coefficient or any wall normal velocity gradient, Nusselt number depends on mesh resolution near the wall. So, very near wall local Re is very low and again the laminar stress referred to the Reynolds average Navier-Stokes equation, we have terms corresponding to laminar stress and terms corresponding to turbulent stress. So, laminar stress plays role higher then turbulent stress. If the mesh near wall, the near wall mesh is fine enough then wall function approach is not valid as shown in the figure in the previous slide. You have alternative one you can go for two layer approach, other one you can go for low Re models. So, in the two layer modelling approach, region very close to the near wall is solved separately either empirically or by following one equation model or any other relationship. And away from that region at some specific wall normal distance, you can use any one of the standard models or their variants. If you are very near and there local Re is very low then damping function is introduced which damps the turbulent viscosity depending on wall normal distance.

(Refer Slide Time: 15:25)

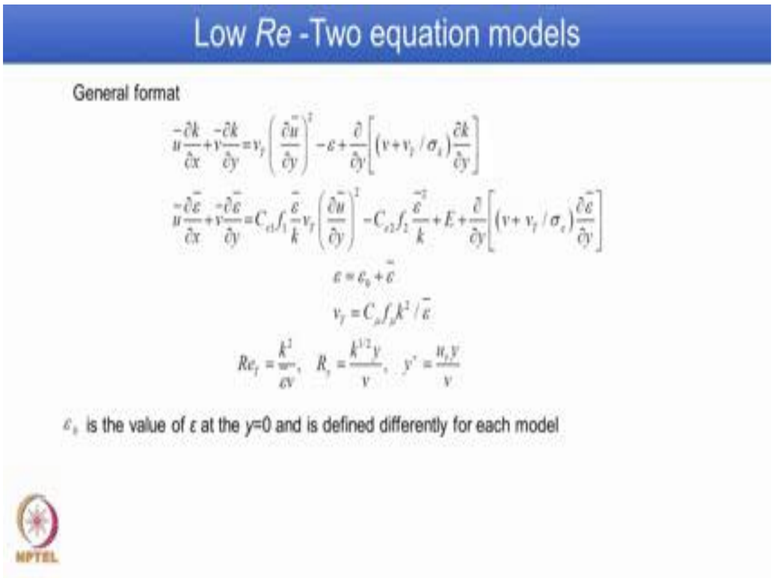


Low *Re* -Two equation models

k-ε model	Jones & Launder(1972); Launder & Sharma (1978)
	Lam & Bremhorst (1981); Chien (1982)
	Nagano & Shimada (1996); Sarkar & So (1997)
Two–Layer Model	Lakehal & Rodi (1997)
k-ω model	Wilcox (1993); Peng *et al.* (1997)

I am showing here list of low Re-two equation model available are proposed in the literature. For every category, for example, K-epsilon model, K omega model, we have list of proposal. I am going to show only a general format of the low Re model and then one or two variance of the low Re model.

(Refer Slide Time: 15:47)



Low *Re* -Two equation models

General format

$$\bar{u}\frac{\partial k}{\partial x}+\bar{v}\frac{\partial k}{\partial y}=\nu_T\left(\frac{\partial \bar{u}}{\partial y}\right)^2-\varepsilon+\frac{\partial}{\partial y}\left[\left(\nu+\nu_T/\sigma_k\right)\frac{\partial k}{\partial y}\right]$$

$$\bar{u}\frac{\partial \tilde{\varepsilon}}{\partial x}+\bar{v}\frac{\partial \tilde{\varepsilon}}{\partial y}=C_{\varepsilon 1}f_1\frac{\tilde{\varepsilon}}{k}\nu_T\left(\frac{\partial \bar{u}}{\partial y}\right)^2-C_{\varepsilon 2}f_2\frac{\tilde{\varepsilon}^2}{k}+E+\frac{\partial}{\partial y}\left[\left(\nu+\nu_T/\sigma_\varepsilon\right)\frac{\partial \tilde{\varepsilon}}{\partial y}\right]$$

$$\varepsilon=\varepsilon_0+\tilde{\varepsilon}$$

$$\nu_T=C_\mu f_\mu k^2/\tilde{\varepsilon}$$

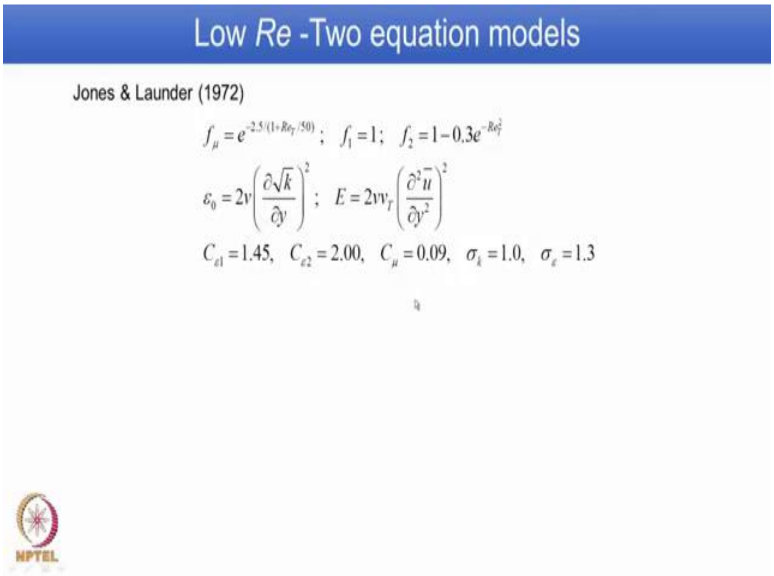
$$Re_T=\frac{k^2}{\tilde{\varepsilon}\nu},\quad R_y=\frac{k^{1/2}y}{\nu},\quad y^+=\frac{u_\tau y}{\nu}$$

ε_0 is the value of ε at the y=0 and is defined differently for each model

So, general format we have here again for k epsilon type turbulent models the equation for k and we have equation for Epsilon, the difference you will notice particularly nu t, nu t is the standard turbulent models is c mu k square over epsilon can be introduce a dumping function called f mu. F mu is a function which damps the turbulent viscosity very near the wall; in other words laminar viscosity we start playing a dominant role. In this expression, we have different definition of the Reynolds number Re t, R y and y plus, y plus is one; y plus we defined as wall normal distance u tau y by nu, it also Reynolds number as I mentioned before.

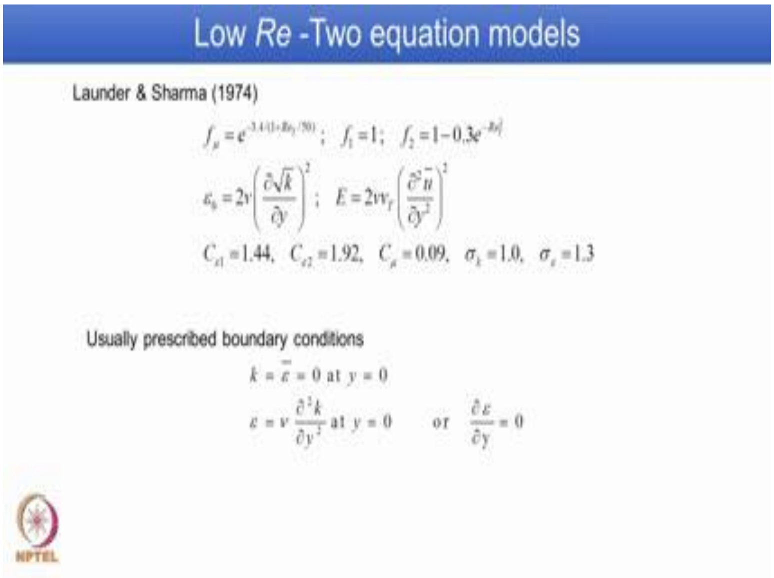
Similarly R y is also a Reynolds number, if you look at the expression root k y by nu, k we know it is the turbulent kinetic energy the unit is meter square per second squared. If we take a square root that will give you velocity scale, so velocity scale distance divided by nu is actually Reynolds number. Similarly, you can define another Reynolds number, so this is called turbulent Reynolds number getting a Reynolds number based on k and epsilon, and this is based on absolute distance and other one based on wall normal normalized distance.

(Refer Slide Time: 17:30)



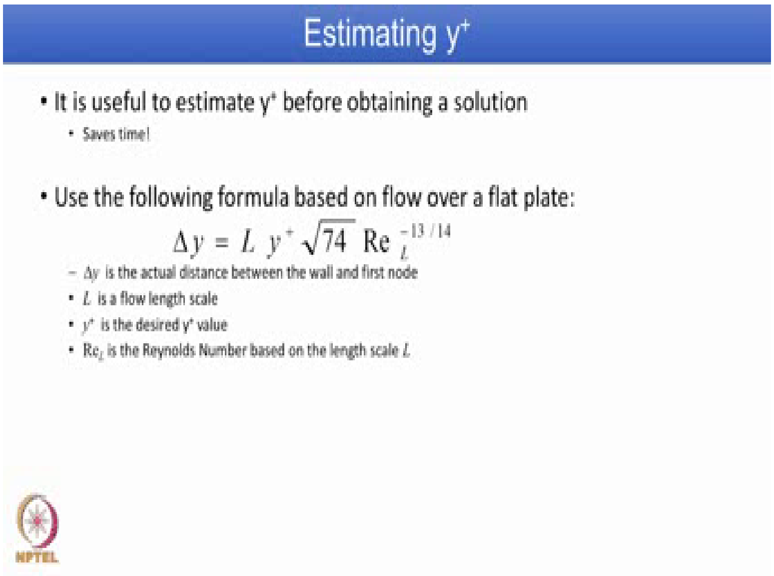
So, different proposal that listed in the table the difference here is only the way function are expressed and how different constant are obtained. So, Jones and Launder is one such proposal some around 1972, an expression for this is available.

(Refer Slide Time: 17:48)



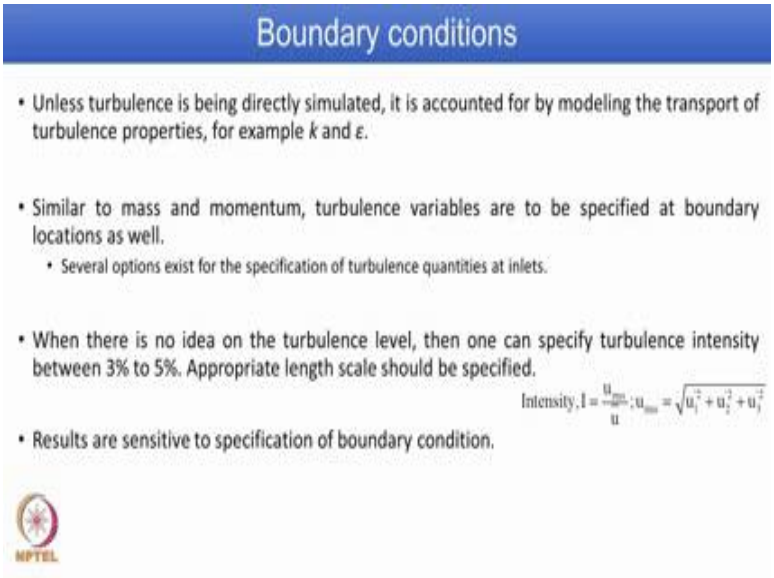
Similarly, another proposal Launder and Sharma again expression for functions and different quantities are available here along with the prescribed boundary condition. These are all well documented in the literature available in all standard books and it is difficult to go through all the derivation in detail in this class.

(Refer Slide Time: 18:12)



Another important aspect is whether we can have some idea of y plus before starting the calculation. We mention y plus is a factor where you decide whether you want to have a wall function or you want to have a two layer approach or whether you can go for low Re model. So, it is; obviously, beneficial if you know some idea about y plus before starting a solution and that saves lot of time. Some empirical formula is available for example, what is listed here is a flow over a flat plate delta y is the first normal distance from the wall. So, delta y equal to l y plus square root of 74 R e L to the power of minus 13 by 14. Now this will decide if you fix you are y Plus for example, you want to have first y plus value as 5, then using this expression you can get delta y and delta y is the distance on the wall of the first node.

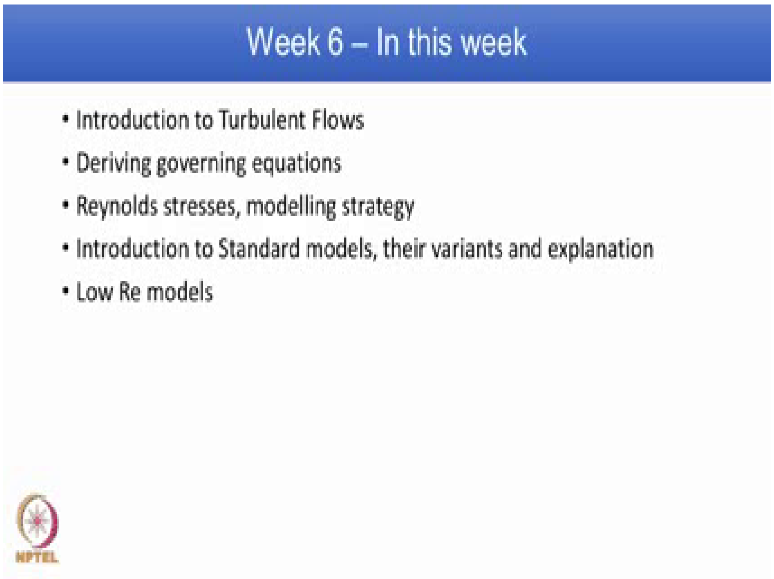
(Refer Slide Time: 19:26)



Just like we have boundary conditions for primary variable u, v, w and pressure, turbulence equations k and epsilon also required boundary conditions. And several option exist for specification of turbulence quantities preferable mostly at inlet, and you follows similar expression at other locations; unless turbulence is being directly stimulated that is there is a class of turbulence stimulation called direct numerical simulation ,where there is no modelling approach, Navier-Stokes equation and continuity question is solved as they are available. The interpretation of the variable is each one of them represents instantaneous quantity. If you happen to use the instantaneous Navier-Stokes and continuity equation as they are then you are actually doing direct numerical simulation, you have only three or four variables, three velocity component and one pressure component. We prescribed boundary condition for them and there is no need for knowing about the boundary condition for k and epsilon.

There is another point what is known as intensity turbulent intensity and that is shown here. Intensity is i u rms by u bar, where u r m s are is equal to square root of u 1 prime squared plus u 2 squared plus u 3 prime square you take a square root, they will give the intensity. So, intensity normally is given either in experiment or appropriate condition. If it is not given that one can take the value between three percent to five percent. In addition to intensity, on also need to specify a lens scale, and from both you can get kinetic energy estimate at the inlet. And results are very sensitive to the specification of this k, epsilon, omega at these boundary condition location.

(Refer Slide Time: 21:56)



So, in this week, we had seen important topic introduction to turbulent flows, driving governing equation for turbulent flows, what is known as the Reynolds stress and how to close Reynolds stress terms appearing in the mean momentum equation. Introduction to standard models, different variants of the standard models and explanation associated with them. We also learnt little bit about what is known as low R e model and then some information about boundary condition. With this, I close class for this week.

Thank you very much, we will see you next week until then have fun.