Course Name: Turbulence Modelling

Professor Name: Dr. Vagesh D. Narasimhamurthy

Department Name: Department of Applied Mechanics

Institute Name: Indian Institute of Technology, Madras

Week - 8

Lecture – Lec47

47. Boundary and Initial conditions for RANS simulations - II

In addition to that there is one more condition that can be used also. So, which is write as boundary condition option 3. Here what we do is we essentially take this k which is the second equation. I am going to take the second equation and take equation 2 which is k equal to this $\sqrt{\frac{1}{2}\left(\overline{a_1}^2 + \overline{c_1}^2\right)y^2}$ this entire equation now ok.

take square root of this whole thing . So, I get square root of k is $\frac{1}{\sqrt{2}}$ this is square root of I have this a 1 or I can write this as square root + $\overline{c_1}^2$ square root y ok. And then I can differentiate this once with respect to y differentiating with respect to y yields $\frac{\partial(\sqrt{k})}{\partial y}$ equal to $\frac{1}{\sqrt{2}}$ sorry. ah yeah yeah sorry sorry yeah yes this is wrong.

So, this is square root of + $\overline{c_1}^2$ ok right. So, now I have $\sqrt{\frac{1}{2}\left(\overline{a_1}^2 + \overline{c_1}^2\right)}$ So now, what is even square average plus even square average from equation 1? Taking help from equation 1 will give me $\frac{\partial(\sqrt{k})}{\partial y}$ equal to $\frac{1}{\sqrt{2}}$ $\sqrt{\frac{\varepsilon}{\nu}}$, right, $\sqrt{\frac{\varepsilon}{\nu}}$. So, now I can square the whole equation , squaring the whole equation I would get squaring the whole equation I would get($\frac{\partial(\sqrt{k})}{\partial y}$) ^{2} = $\frac{1}{2}$ $\frac{\varepsilon}{\nu}$. So therefore, $\varepsilon = 2\nu \left(\frac{\partial(\sqrt{k})}{\partial y}\right)^2$. So, this is another boundary condition that you can implement.

Again remember k is there k will go to 0 if you implement it on ε on the wall node it should be the first node ε first node above wall it is y^+ is there equal to 1 ok. So, this is another option that you can use it. And in fact this particular condition is also used in somebody asked a question in the last class like why we do not have a damping function also in the ε equation. In one of the modeling option is that this particular term 2 ν ($\frac{\partial(\sqrt{k})}{\partial y}$ that particular term is used to damp ε also in some of the models. So, is this clear? We have boundary conditions.

So, now we can go ahead and solve k- ε model. We have a clear idea of how k and ε models are constructed and then you also have an idea about how the model coefficients arrived, what are the assumptions we made. and of course, the two options wall functions which is a high Reynolds number formulation. High Reynolds number is only locally high Reynolds number that means you are capturing only the away from the wall. So, the the velocity profile will be it will be like this right.

So, you are essentially capturing this zone in the high Reynolds number formulation locally high Reynolds number locally high Reynolds number here. But if you want the near wall zone that is locally low Reynolds number locally low Reynolds number. So, you need LRN formulation to capture this near wall if you do not want that you can go ahead and use the HRN formulation which is the wall function. So, two options you have, but for ω I have not given. So, I can mention here when you what is the boundary condition for ω right.

So, we can see that ω equation for ω boundary condition. Something special about ω is that by default k ω model is valid all the way to the sub layer. So, ω boundary condition you can specify to and it will behave like Lorentz number formulation. So, usually ω many times damping function is not used. So, the k-ω by default is valid in the all the way to the wall and it behaves like LRN ok.

So, only the boundary condition is required for that. So, that boundary condition I can give you the formulation ok. So, note that k ω model is applicable all the way to the linear sub layer that is like a like an LRN model. It behaves like that by default you just need to specify the boundary condition on the for the ω. an ω boundary condition it is derived I can give you the formulation I will not go into specifics of this boundary condition because then I have to discuss every eddy viscosity model.

So, ω here for the first node again will be $\frac{6\nu}{c_{w2}y^2}$ it also has a y^2 behavior. this boundary condition again must be applied ω naught on the wall ok, this should be again first node. So, first node usually placed y+ less than 3 that is in the linear sub layer, first node above the wall y+ less than 3. in the linear sub layer. So, ω model ω at the first node above the wall you can specificize $\frac{6\nu}{c_{w2}y^2}$ again it has $\frac{1}{y^2}$ behavior ok.

So, this will complete the boundary condition part,initial conditions you need to also look at ok. when you start any calculation numerical calculation you need to give initial conditions sometimes even inflow conditions right. So, you need to for a flow you would know the velocities. So, you know the Reynolds number of the flow problem from that you know the inflow velocity you will give it that is not a problem. But you are computing here also k, ε, ω and so on.

So, you need to specify initial conditions and or inflow conditions for that a problem for that kind of problem right. So, we will see how we do that ok. We will quickly go through that inflow conditions initial and or inflow conditions inflow conditions initial conditions are definitely required for all CFD calculation, inflow conditions also for certain type of flow ok. If you have a flow over a bluff body or if you are looking into you know a jet right, if you have a jet flow coming out from an orifice then you need this k and ε at the orifice right. The u inflow you would know, this you would know from your flow problem right flow dependent flow dependent, but what will be k inflow ε inflow right this will be user dependent here.

So, you have to be very very careful. so how to give that so I can give you just kind of a guidelines for here. So, to specify initial conditions or inflow conditions first we need for the k right initial condition for k or inflow condition k. So, k is of course, we know sum of the 3 stresses. So, if I assume isotropy assume statistically isotropic.

Statistically isotropic will make k half of one component is enough to define in a statistically isotropic flow. Whether such a flow exists or not is not a question here, it is only we need to find out an initial condition. There can be in a generic flow, there can be local regions where flow can be statistically isotropic not everywhere. So, k is now like this and of course, I can make this as half of square of the u_{rms}, u_{rms} is essentially this. right square root of u prime square average like this.

So, this makes half of u_{rms} squared. So, now k I can obtain from u_{rms}, but u rms you do not know what it is right. So, for that all I can do is sorry this is nobody corrected me here this should be 3 half right. If these 3 are equal to each other statistically isotropic then it is 3 times of this sum of 3.

So, it is 3 half here. So, 3 half of this and I can write this again as 3 /2 u_{rms} I can specify it as what is called i and your $(u_{reference})^2$ where u reference is your reference velocity. It could be $u_{infinity}$ or a jet inflow velocity whatever is your reference velocity that you know from the problem that you can specify. i is the one which is user dependent right u reference is your reference velocity. So, this is flow dependent. So, this is completely fine, but I is the problem, I is what we call turbulence intensity.

So, this turbulence intensity is defined as $u_{rms} / u_{reference}$. So this is completely user dependent, user defined. You have to be very careful here because you are specifying the value here. What is I? So for example, if you take a typical wind tunnel, they would usually say what is the operational condition? They can say it operates from 1 percent to 10 percent turbulence intensity. right the manufacturer would tell you that this can operate in between this much turbulence intensity.

So, there is no perfect guidelines here what you should use it you have to be very careful what I value will specify here. But it is left to you in that is what I said in statistical modeling of turbulence lot of user say is there you choose whether you want k- ε or k- ω or k -ω s s t or whatever you are going to figure it out. So, there is a lot of user choice in trans modeling. So, this is completely your choice you have to be very careful here what you will give it user dependent value and so you can take a note here. So, choose i carefully say 1 percent to 10 percent or follow best practice guidelines, best practice guidelines or literature.

carefully is the word you have to underline I carefully means this value I here. So, now ε also has to be figured out after k. So, this is the formulation you can use it to get k sorry to get k here I can use this formulation. Now to get ε, ε we have written when we did wall function as purely dimensional arguments a turbulent velocity scale and a turbulent length scale $\frac{u^3}{L}$ if you recall right. So, here I am going to use this turbulent velocity scale as u_{rms}.

cube by L, L is again user dependent again this is you have to be careful here ok. So, here this can be many times this is written simply as I said this I $\frac{U reference^3}{L}$ this can be like this because you are already giving $U_{reference}$ as input I is chosen L has to be also chosen. where L is also user defined. So, this is L is a numerical length scale, no physical meaning you have to remember this. no physical meaning in a turbulent flow because turbulent flows are characterized by multi scales.

So, this is purely a numerical length scale user defined in many codes this is not how it is done in some of the commercial codes you would see it slightly different. So, what ε they would do is this also an acceptable form I can say this is acceptable but in some of the codes what they do is they take this u_{rms} using the equation here $3/2\ (u_{rms})^2$. So, they would do this urms is nothing but they would take it as $\sqrt{\frac{2}{3}}$ if you look at here k is nothing but $3/2\ (u_{rms})^2$. So, u_{rms} will be $\sqrt{\frac{2}{3}}$ k. So, in some commercial codes they would use this formulation.

So, $\sqrt{\frac{2}{3}}$ $\frac{k^3}{L}$k cube by L and this would become basically $\sqrt{\frac{2}{3}}$ and then $\frac{3}{2}$ f. right $\frac{3}{2}$ and then $\frac{k^{\frac{3}{2}}}{L}$ and this $(\frac{2}{3})$^$\frac{3}{2}$ will essentially become $c_{\mu}^{(1/4)}$ and $\frac{k^{\frac{3}{2}}}{L}$. So, in some commercial codes you would see this formulation. again L and I are chosen. So, essentially you need you have you reference coming from your flow no problem, but be careful about choosing I turbulence intensity what percent it is and then a length scale L these two will control your flow you have to be careful about choosing L and then I.

In some commercial code this Cμ is also dropped or it is taken a different value. So, you can take a note here in some CFD codes $c_{\mu}^{(1/4)}$ is dropped or $c_{\mu}^{(3/4)}$ is used. different codes you will see some combination regardless of this L and I has to be chosen and you have to be little careful about what you choose ok.