Course Name: Turbulence Modelling

Professor Name: Dr. Vagesh D. Narasimhamurthy

Department Name: Department of Applied Mechanics

Institute Name: Indian Institute of Technology, Madras

Week - 11

Lecture – Lec63

63. Introduction to Direct Numerical Simulations (DNS) – II

Two methods of testing the mesh, whether it's good or bad now the domain, okay? So your the box that you are using in a CFD calculation this should be sufficient enough to capture the large eddies, ok? So generally, we may think that, let's say if you have, you know, this is a channel flow right where it is shear driven, it's a plane turbulent plane Couette flow, right so if you have a plate like this, and then I have another plate, and this is moving this is fixed, right? So that's the definition of a turbulent plane Couette flow. In this particular problem, we may think that the gap here, let's call it h. We may think this gap is the size of the largest eddy. That is before doing a turbulent flow calculation. But in reality, we see that that is not the largest eddy.

The largest eddy depends on the type of flow. So, in a turbulent plane Couette flow for example, this is again turbulent plane Couette flow look at the this is the x direction like this x and let us call it this is y and other out of plane z and this is the z direction. So, now I am looking into what is called two-point correlation. So, this R_{ii} is nothing but the two-point correlation that we have looked at.

When we did theory, I talked about this two-point correlation. So, that means I have plotted for all the three components u, v, w. So, let us say for example, the $R_{u'u'}$, this is nothing but u', this is the correlation along x direction, two point correlation along x to see how large you need to go before the correlation goes to 0. So, if your probe is inside an eddy, then it is always giving correlation, non-zero correlation will come. But if that eddy will move away out of your probe, then it gives zero correlation, right? So, that shows the length of your eddy as long as it is correlated.

So, if I define $R_{u'u'}$, the correlation of u' along the x direction as u' of your x, y, z, t and sorry, u' of x plus delta x, y, z, t average. And I am normalizing this with the your u prime of x, y, z, t sorry u prime of x, y, z, t. That means its own stress at that point. So, basically, I am changing delta x. So, that means this is the reference probe here.

$$R_{u'u'} = \frac{\overline{u'(x,y,z,t)u'(x+\Delta x,y,z,t)}}{\overline{u'(x,y,z,t)^2}}$$

So, this is x and then I am changing the Δx. So, I go like 5, 10. So, this is $\Delta x/h$. I am going 5, 10, 15. So, it is $20h$.

So, more than $20h$ the correlation is going to 0, but the height is only h. So, that means in the x direction there is a turbulent eddy which is about $20h$ times right. So, you that means your domain has to be twice that of this. You must have at least two such eddies. So, your domain must be double than when this correlation is going to 0.

So, you should keep something like $50h$ or something or $60h$ or even bigger than that. For each flow it is different. So, you need to think about how large the computational domain has to be there and that exercise is called numerical domain verification. And this has to be done for all the three components $u, v, w\ prime$ as well as in the z direction. The size of the structure is not the same.

It is not a sphere, right? It can be elongated. So, and you see here, the correlation is now going to 0. This is the span wise correlation. So, the same $R_{u'u'}$is, let us say, u prime of x y z t, u prime of this is a z direction. So, it is x y z plus delta z t average divided by u' square average, so here delta z changes, and this is z equal to 0.

$$R_{u'u'} = \frac{\overline{u'(x,y,z,t)u'(x,y,z+\Delta z,t)}}{\overline{u'(x,y,z,t)^2}}\ along\ z\ direction$$

As you change Δz, you can see the correlation is about to go to 0 by about $8h$ so this structure is elongated like spaghetti or something there is a structure sitting which is about $25h$ along x direction but about $8h$ along the spanwise direction and in the z direction it is smaller it is not h it is little bit smaller this is this is just an illustration so for each of this flow that you are going to solve you must do this to check whether your box is sufficiently large to capture at least two large eddies. So, the box has to be able to capture this. So if the correlation goes to 0, take twice of that one in that particular direction as the box size minimum. If it's 8, take it as 16 along the z direction. Double it you must have at least two eddies to freely move around because you are going to compute for its correlation the stresses, right? So, $u'u'$ is you are looking at to the single point we discussed about single point data there when I talked about Reynolds stresses this is the two-point they are looking into the separation.

So how to start turbulence in a computer? So this is a bit of a challenge compared to lab because you start with initial conditions in all CFD calculations, right? And if you give a quiescent condition, you set everywhere flow velocities are zero, or if it's uniform, how does a random flow field develop? It will not develop very easily. No matter what you do, your solution will not show that. It may look like laminar flow, even at a Reynolds number, let's say 1 million or so. If you're unlucky, it still shows that flow is like laminar. Then you may be wondering what is happening.

So there are many techniques how to start turbulence, how to generate turbulence in a computer. So one technique is you start with numerical perturbations. You add random number perturbations, a stochastic field superimposed on your initial base flow field. This base flow is let's say your bulk mean velocity. In a pipe flow or a turbulent channel flow you would know what is the bulk mean velocity before you start because your Reynolds number is based on bulk mean velocity.

So, we have a bulk mean flow field and superpose some numerical perturbations. So at high Reynolds number turbulence, let's say a million or so, the nonlinear terms in Navier-Stokes will take care of generating turbulence. You don't have to worry about it. Usually at high Reynolds numbers, you don't have to worry. But DNS is usually done at low Reynolds numbers.

Maybe like 10,000 or maybe if you are lucky 50,000 or so Reynolds numbers, very low Reynolds number. So at low Reynolds number turbulence, this is a big problem. Turbulence is not easy to generate. That's because viscous term will diffuse. Whatever perturbations you are adding, the numerical oscillations that you give it as initial flow field, the viscous term diffuses it.

You are running it in time, you run for several time steps and then we see that flow looks like laminar flow. You give a very random flow field looking like turbulent and you run your Navier-Stokes simulation for some time steps and you see that it looks like laminar because viscosity is killing all the perturbation. You are getting a converged solution but not the correct one. So, one way of doing is you reduce, you artificially reduce the viscosity that means you take the Reynolds numbers artificially to very large value. Let us say you are interested in 10^4 as your final Reynolds number, you take it to 10^6.

In the beginning run it okay that means your mesh size is now much larger than the Kolmogorov there will be lot of error this error is what you want some numerical error is accumulating if you run continuously it will crash you must save flow fields in between, just before let's say it crashes you will have some flow field which will have lot of perturbations looks like a lot of random noise use that as the initial conditions to reset the

Reynolds number to 10^4 we can artificially take it to 10^6 save that flow field take it as initial condition for the actual 10^4 Reynolds number case. Then run further, and hopefully, turbulence will sustain once it is sustaining; it will sustain because turbulence, I told you in the absence of you giving power, it will dissipate as long as the bulk mean is acting, turbulence will be there, it will sustain, ok? So this technique can be used to reduce viscosity and run till the solution starts diverging, reset the viscosity to the original value, and continue. Another technique, if this is very expensive, it looks like trial and error. Reduce it, increase, reduce, increase. One can also use synthetic turbulence.

There are many techniques like the divergence-free synthetic eddy method. So here, some stochastic field is generated using some; you need to have some kind of an idea of what should be the Reynolds stresses in a given flow. Okay if you can assume what is the Reynolds stress in that flow and you can give the turbulence intensity also then you can generate some synthetic turbulence this can also help so there are other techniques is one of the example there is also you have digital Fourier method okay using Fourier modes digital Fourier modes so there are many techniques All will look into generating a synthetic turbulent flow field in the beginning to kick start turbulence. Again, if it is low Reynolds number turbulence, again it may also diffuse.

It depends. So that is also possible. Another technique is to do an a priori DNS flow field. For example, let us say you are simulating what is called or let us say an aircraft diffuser. You would have seen aircraft engineers would know what I am talking about. Let's say I have a diffuser and you want inflow outflow like this.

You want a turbulent inflow here. It should not be a laminar inflow. You cannot expect a laminar inflow coming in to generate turbulence downstream. The inflow is turbulent and this is your numerical domain. So what people do is they take this particular cross-section here.

You can do an a priori, let's say a channel. You can do a DNS calculation of this and use this data as an initial condition, not the initial condition, sorry inflow condition. One channel flow DNS is done that data is actually given as inflow to the So, this kind of an a priori DNS is an expensive method. You need two different DNS calculations. Otherwise, you cannot go ahead in, all this will cost money.

DNS itself will cost money. It's an expensive technique. These are different techniques. I'll show you data for one of the technique quickly. How it looks like the first one where you are artificially increasing Reynolds number and then reducing to the actual one. So, let us say this is a turbulent channel flow, where I have the wall here, two walls and then you can see this particular case, I solve the actual Reynolds number using perturbation and it looks like this, almost no turbulence.

This is the first simulation, this is the final converged solution that I am getting is like this. Without artificially increasing nothing. Let's say this Reynolds number is let's say 10,000. Example Re 10^4.

I solve this. This is what I get. No turbulence being generated here. So what I do? I increase to let's say 10^6. and solve this suddenly I see that flow field is looking nice okay obviously there are errors now I can save this as my initial flow field so before the code crashes I save this as my initial flow field I use this as my initial condition for Re equal to 10^4 case so if I am using this as the condition now I get this as the final converged solution. This is 10^4 converged. So now you see the difference between this case and the other one.

So turbulence has sustained. I run it for a very long time to see if it is sustaining. So you can do this technique. I will come to that one. So I'm showing you the same one for the three components u, v, w three velocity components you see here it looks like laminar flow. This is when I let's say Re is 10^4.

I'm just solving without taking the flow to artificially high Reynolds number, just 10^4 with some perturbations I do well it looks like turbulent your v and w it is looking like turbulent But look at the values here compared to this. It looks colorful but it's practically zero. Your v, w instantaneous is zero here.

So tiny. In reality that should not be. This is an instantaneous velocity. I'm not looking into a mean. This is u, v, w instantaneous. So when I do Re let's say 10^6. I don't remember the exact Reynolds number what I did.

But just giving a usual practice that I do. Let's say increase it by 10 hundred times. Then I get this. Now I see the difference. So v and w now is at least about you know what is it like 10 percent.

Yeah, it's about 10%. Looks like reasonable to me compared to having 0, right? Turbulence is 3D. So now, at least now I see something has happened here. The other two flow fields got generated. Now I use this as initial condition. I use this, reset it to initial condition for Re, let's say, 4.

I get this. So this is Re, this is what I wanted finally, now I see it sustains, fully 3D flow field, anisotropic. So some amount of trial and error is there, some experience is required to do this. You should be careful about sensitivity to initial condition. most because I'm talking about initial condition right in many flows this is not a problem I already told you initial condition only affects the convergence how fast or slow convergence you can get by your guesswork right here you are not guessing you are doing something else for initial condition But you have to be very careful in some problems where hydrodynamic instabilities, thermal instabilities are present you can see that by changing initial condition I am getting a completely a different solution non-unique solution is possible here, right and it sustains this is how the flow looks like I run for eternity no problem this is what I get this is the final solution for these two initial conditions so to be careful in especially eddy resolved techniques this is not so much important for, let us say, RANS because you are solving only for the mean flow such eddies itself are not there right both will look the same if you are solving RANS the mean velocity both will look below you will not see any eddy there a boring flow field will be there. But here in eddy result techniques you have to be careful initial condition.

And statistical sampling that is the question Geetarth asked how long you have to run? I already told you about statistical stationarity. You must make sure that you are running enough running for a long time such that the flow has come to a statistical stationary state that I already discussed how to check for that one. So, look for statistical stationarity and also look at your statistics are not changing as you increase the number of samples. Let us say statistics coming from 1000 samples and 2000 samples are same that means your statistics are no longer changing with samples n. is fine but before that you must make sure that the flow has come to a statistical stationary state and you should make sure that your samples are uncorrelated this is what i meant right so spatial correlation we looked at it for the domain size two point correlation in x, y, z to make sure the box is big fine but you should also look at auto correlation how much time I have to wait, how many seconds I have to wait before I take one sample.

You should not take all samples at every time step. You should give some time before you take the next sample. So, that the temporal correlation is also going to 0 right. The temporal let us say the temporal correlation for any phi this is already discussed. I am just showing you your x, y, z t and phi prime x, y, z t plus delta t at the same location you change spacing is changing in time now average by divided by I have phi prime of x, y, z t square average. So autocorrelation is also have to look at that will give you how long you have to wait before you take each sample.

$$R_{\phi'\phi'} = \frac{\overline{\phi'(x,y,z,t)\phi'(x,y,z,t+\Delta t)}}{\overline{\phi'(x,y,z,t)^2}}$$

Sampling frequency, this is same even in experiments. It is nothing to do with just a calculation here. This particular slide is useful also for experimentalists. How many samples have to take? When to start to take the sample? And what should be the sampling frequency? Three important questions. And this is just a illustration to show that sometimes what is long or how long you have to wait is will become a big problem so what we define here is called large eddy turnover time that means what is the time taken for the largest eddy to turn one roll and here i am showing you the data six figures each one separated by 100 large eddy turnover time that means for 100 eddies have turned and the flow looks like this but after that it looks like this and again for another 100 it is looking like this and again it is changing look at that the flow is different now right so this is like perfectly anti-phase this side is big this size is big and these two are small but now you see here this is biased this has become a bigger wave this has become a smaller so the the flow is going at an angle here in between the gap same thing is happening here biased now it has become in phase so compare these two the red one and these two are rolling up not like here big and small so this both are big and big here and these two are in phase shedding so basically this is just an example but you have to be careful that sometimes when you run longer especially when you have instabilities in the flow it can suddenly show you a new result a new flow profile you have to be careful when you perform statistics If I am averaging now at 600 larger eddy turnover time, I will get a viewpoint which is now time averaged between all these 6 flow fields which is not representing neither one nor any of these things.

It will show something else. You have to be also aware of this. And finally, this slide, PIV is an experimental technique. It is a state-of-the-art experimental technique today while measuring turbulent flows. And usually, you can say that we use experimental data to validate CFD.

but today you can see this is a website called pivchallenge.org where when you produce an experimental technique that is the manufacturer of this pivs they have to validate that their experimental setup is correct before people start to buy it right that is their own validation system so now experiment is validating you can see one of the case d here 3d piv images based on time resolved dns So, DNS data is used today to make sure experimental technique is correct. So, there is some relevance for this. It is not just turbulence that is being seen in a computer. It is helping the modeling community definitely, physicists as well as even experimentalists. So, with this I stop. If you have any questions, I can take.