Wind Energy

Prof. Ashoke De

Department of Aerospace Engineering, IIT Kanpur

Lecture 59: CFD aspects of wind turbines

Welcome back. so now we will continue our discussion on this wind energy but here mostly what we are pretty much, we have done our what i would say the major theoretical component of the discussion. so now, if you recall from our last few session that we are talking about the design purpose and all these things. so for design what we have discussed is primarily the linearized model or the simplified model like that and those simplified models and all these things those are very very useful for initial design purposes and once we get the initial design data then we can move for more detailed analysis like that but as i have kind of emphasized that the wind turbine design procedure is a complete aeroelastic problem or so what we are trying here is all analytical ways or simplified method which could provide us the initial data and using that data we could come up with the initial design parameters so which could be rotor hub rotor diameter then the blade air fall and all these things so while doing so one of the important factor that has to be considered is that all this simplified model that we have these are these models have some limitations some models can take care of certain phenomena some models cannot take care of certain phenomena and that's what happened while discussing about those particular conditions when we are talking about what you call the talking about that the rotation hints being models and things like that now having said that coming back to the primary thing is that okay what you can do you can have those simplified model and things like that so those simplified model which are good which are useful, which are handy for getting initial design data and all this. But that still doesn't give you the complete picture of that. And also, we have talked about how one can obtain different structural behavior of the whole unit, whether you talk about tower, whether you talk about the rotor blade, whether you talk about the nacelle.

or in combination with the complete unit so structural response also either you can do simplified analysis you can do modal analysis you can do lump parameter analysis you can do multi-body dynamics which are quite involved or you can do complete finally similarly while you are talking about the load calculation on these different components that's the aerodynamic load primarily because we are trying to extract out the wind energy to produce electricity so aerodynamic calculations also we have discussed quite enough about this simplified aerodynamic model how to estimate those forces and moments and things like that now alternatively one can completely do cfd now today's date the computing power has evolved or matured enormously so you have huge computing resources now so your cfd methods or the methodologies those are also quite matured at this stage so cfd can be used to completely analyze those three-dimensional load calculations. So in that context, we are going to talk about the basic CFD perspective, that what CFD does, then how we can apply to those in the turbine on this kind of rotating frame of reference applications. And then Finally, we'll conclude with some alternative concepts and things like that. So that is why we are going to talk about on CFD perspective of this wind turbine.

So here, if you see, I mean, we can start with the role of CFD in a role of CFD in engineering applications. You can see this is an application which is flapping of bird. Then we have next one flow over this morphing wing, noise. these are these are eml application this is wind turbine application this is it so this is electrochemical system gas turbine application so there are different um applications so you can see cfd has been applied in different areas at this moment. I mean, these are only to name a few.

It's not that the list is completely, I mean, but this should give you an idea that now CFD has been used in different engineering application. So when you talk about CFD, it tells you some or raises some questions. so now when you talk about the academies here or academic people can academy yeah play a role in supporting the industry when you talk about design and all these things these are important in industrial application so can we support that so can the academia play a role in supporting the industry so this is where it's very very important that when you have this that how is because academic people can come up with the new technology development and things like that and how they help the industrial design procedure can the company or the industrial people take advantage from scientific expertise because academic science or academic community has the scientific expertise which can be used by them and also academia learned from the people employed in the company because their experience in day-to-day operation their experience in handling a practical problem so it's an win-win strategy of cooperation so if the academia can seek with the industry then it's probably the best possible situation to design something in scientific world where one hand the academia can provide the technological know-how and that can be taken by the industry people for further development and taking to the actually the design procedure and finally do that so now what are the possible answer that obviously the academia can offer competence perceived as an advantage okay so this can be utilized then interdisciplinary approach to solve the issue that means so interdisciplinary approach to solve the issues then capability to redesign with a focus on product innovation and capability to predict the change of demand from society so it's essentially an innovation that we are talking about that this one-to-one hand-seeking would help to kind of go with that now what is computer aided design that uh it's what it does it's kind of an use the computer software to doing the engineering analysis so initially the picture that we have seen that shows that how the computer edit tool which is a common in the all the industry which are in design and manufacture process they are involved so it can so the tools which involve finite element analysis which we talked about when you talk about the structural responses we talk about the dynamics which is also a tool that is also talked about because the multi-body dynamics is important for designing certain component of this optimization procedure there could be some more these are only few listed here and cfd CFD is used for the fluid flow problems, fluid flow and heat transfer problem. But when you go to structural or solid mechanics problem, you need finite element analysis, you need multi-body dynamic and all these things. So any software that has been developed to support this kind of activities are termed or considered the CAE tool.

So the CAE tools are used to analyze the system uh so it encompasses simulation validation optimizations so what is the future the ca system is will will be the major provider for information to help support design team. So this is why we are talking about this CFD perspective. But this is also true for even talking about the structural dynamics or multi-body dynamic system that we talked about it. But yes, when you talk about the CFD, then it is for future and heat transfer related problem. So what is CFD? As I have been saying this, this it's a tool which deals with the fluid dynamics and heat transfer problem so this is what i was talking about um but the cae tool could be also finite and this cfd is an alternative to measurement because if you talk about twin turbine this is such a large system after uh design and all these things so doing measurements in a tunnel and all these things that could be a challenge this is where the cfd could be handy too so this has an application in multiple areas and i will show you some of the example so these are becoming a key component for computer now so how the safety works i mean if you try to see that so when you have a feed flow problem then essentially the tool is going to give you solution to a physical problem so you have that physical problem so essentially which is the problem of fluid mechanics or heat transfer so you understand that and then you try to find out the governing equation so typically in fluid flow problem things are defined by the navier-stokes equation so first thing is that you try to it's a mathematical tool so you have a physical problem let's say flow over aircraft to inflow over wind turbine so how we the what mathematical equations or conservation equation anything in nature they are governed by this basic conservation equation so we try to find out what is the conservation equation that govern that particular system so we identify that particular i mean those governing equations and those are the partial differential equations then we apply different numerical methods even cfd there are multiple ways one can do one can use we'll talk about that as we go along with this discussion so we try to apply those numerical methods then get a mathematical form which is a linear system then that combined with the geometry that actual problem for example flow around the turbine and which we call is a grid and then we convert that to computer program and then program finally get us the simulation results once we analyze that's how the get the so this is what the complete cycle of the clp that how all these processes are done so that means it closes the complete loop so there are primarily three different stages okay three different stages one is the pre-processing stage that is the stage where you try to define the geometrical model the physical model and the boundary conditions uh then you have computing where you do that and then the post processing so if you look at it then you have this pre-processing stage where you define the physical model and then have this grid ready then you do the solution in the using your numerical tool during computation then finally analyze that so this is an iterative process that means once we get our results and the results are not good you go back and make some corrections and then again you redo this process so this is how it happened you first identify the geometry or the physical problem then the geometry you set it up once you set it up and everything then you send it for the simulation once you simulations you get the results to analyze it so this is how the whole process actually works so what cfd is it's a science of predicting fluid flow behavior by solving the governing equations in a numerical framework so this is what it does okay now so by solving this and the result of cfd analysis so that represent valid engineering data that may be used for conceptual studies of new design studies where control experiments are difficult to perform for example in turbine case studies with hazardous operating conditions redesigning the system then analysis represent a valid competent to experiments and all these things so what is this this is essentially analysis and design okay so you have a simulation based design instead of build and test so this is cost effective okay solution provide high fidelity database i mean obviously the any full scale simulations can be done it is not hazarded physics can be completely so what is important here is that that if you compare side by side again while we're talking about any c tool whether it's a finite element or cfd for fluid flow problem please understand one thing that this is not going to replace your measurement or experiment techniques completely but this would reduce your design time and getting a data where your measurement cannot be done okay so simulation can be cheap because it's only computer based simulation that the time is short there is no limitation to the scale you can do complete scale simulation and the complete information is available it's repeatable there is a it is not hazardous these are the advantage and that is why this should go hand to hand with the experiments and should be used as a preliminary design data which can be obtained from the cfd or any cae tool in that way and that can be taken for the further decision making process okay so again coming back to the CFD thing so what so what you can so it's a computers which are built in 1950 there we could do limited floating point operation but today we can do except of floating point operation so the whole idea is that the computing infrastructure has really really gone that much that uh we can do today's uh that kind of computing which was not possible so obviously this was possible because of high performance computing platform so how performance computing platform uses a kind of an aggregated computer system or we call it a supercomputer so it's a there are multiple cores of the processors are available so your performance can go up and you can get simulation done in a quick turnaround time so why do you need those hpc because your lab computer cannot do those things you want to increase your complexity with existing calculation which your lab computer laptop cannot do you need to specialize resources which are not available in your lab computer so this is where the hpc environment comes into the picture and you can see this is a super computer with high level computational capacity compared to general purpose computer so that means you have those which are this was first compute hpc was introduced in 1960 which is a cray system which massively parallel computation can be done so that means what you do that you kind of distribute your task instead of it just you can think about i have one laptop or one desktop here it's a multiple desktop which are connected to each other so the task what your desktop and laptop can do now you are distributing that whole task in multiple laptop and desktop to do that and that is through this parallel uh processing and all these things so you can solve those great challenges you can solve those large scale problem you can solve a huge problem and as i said that we can do i mean now zeta flop calculations not even except of zeta flop calculations today we can do that that means the supercomputers are so powerful that you can do this much of calculation but while doing that the key ingredient here is the parallelism that means your software the ca tool that has to be and that must be able to execute over this multiple laptops or desktops or multiple this hpc environment so the cfd or c tool development is another challenging area where scientists and engineers are working they are developing everyday different tools that's what i started that today the ce tools are quite mature they're quite powerful they're hugely scalable over this computer so that you need to do this tasking over multiple computers rather than doing it over a single computer single laptop or single desktop so that's what you can solve this problem this is an idea about how this is a list from november 2023 i think every six months they release this data so one can go this particular site and get this data out that this is a country who is here obviously united states dominates most of the supercomputer and this is performance here also here the united states kind of dominating and it is an spectrum of application if you see it is quite uh mesmerizing okay so it has been i mean it is everywhere okay kind of uh so that's what one has to see that this is everywhere so these are some of the top computer based on their performance but you can go to that top 500 side and get this data it is not something hidden or not available of such thing so these are readily available and this is how performance development so this is what is projected how the floating point calculations would go and this is the projected performance so that means the computers are going to be quite powerful so that means the performance improves by factor of 10 every four years so this is where that means you can think about the kind of calculations that you can do today maybe four years down the line you could be able to do 10 times higher i mean complicated problem or that kind of problem of that nature so this is where the this is where the computing platform and all these things this is where as per 2023 november 2023 from this side the indian supercomputer list so this is in the cdac this is also sedaq this is indian history of meteorology and so obviously this year list could have been a different list so now if you see the application the application is everywhere okay so whether you talk about aerospace application whether you talk about appliances automotive industry so that means pretty much every industry that's what i started with that nice picture of showing different kind of example that where are this rule of cfd or ce tool in different areas its complete spectrum is quite huge okay so different ids a different i mean sector they are heavily used so that's how your application of cfd is now the cfd solvers there are commercial servers which are available there are in-house capability there are academic solvers there are open source codes i mean these days there are plentiful open source codes which are available that one can use and do that how this problem is solved we have talked about it so the physical problem that you want to solve from the continuum real problem simplified mathematical problem so prob so this is your pre-processing uh then you model setting then finally you solve this problem so it's just so it's a tool that require you to switch your brain and your fantasy essentially what it does you identify the problem their governing equations convert to a mathematical problem do the programming set up instruction to the computer to solve it whether it's a single computer or over this hpc environment get the simulation result done and then analyze it and use it for design purposes that's how it works so just to problem identification that is pre-processing stage where you identify the problem let's say this problem this is how you generate the grid setting up where your partial differential equations then your computer program which are going to run over this hpc finally the result once you analysis analysis done so you have cost time ability to simulate these are quite advantageous obviously physical models numerical error boundary conditions so one has to be very very careful while doing this then they should not commit to such mistake which could lead to a problem of that okay now so when you talk about fluid fluids are having liquid and gas their density could be uh constant for incompressible and variable for that, then viscosity is there.

So these are some of the physical or these are the transport properties or properties of fluid. What happened in the fluid mechanics? you have inviscid flow you have viscous flow which could be laminar and turbulent then again laminar can go to internal flow like pipe external flow airfoil so our wind energy wind turbine belongs to the external flow so this is where our wind turbine belong to that turbulent external flow then you have compressible incompressible but most of that time we are dealing with incompressible situation here.

- Fluid = Liquid + Gas
- Density ρ

$$\rho = \begin{cases} \text{const} & \text{incompressible} \\ \text{variable} & \text{compressible} \end{cases}$$

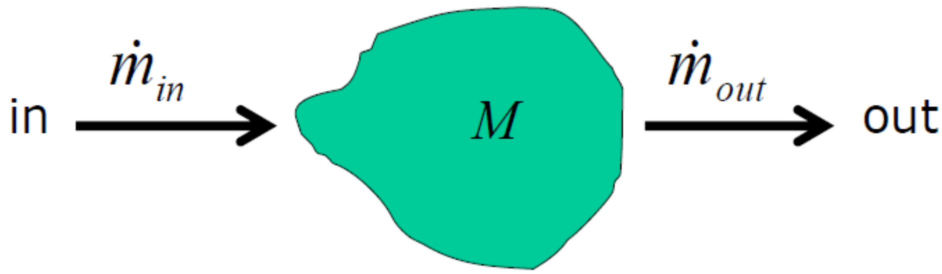
- Viscosity μ:

resistance to flow of a fluid

$$\mu = \left(\frac{Ns}{m^3}\right) = (Poise)$$

Substance	Air(18°C)	Water(20°C)	Honey(20°C)
Density(kg/m^3)	1.275	1000	1446
Viscosity(P)	1.82e-4	1.002e-2	190

So TFD course typically designed for, I mean, it could be, I mean, that is why the problem definition is very, very important that what we are going to use now, there could be viscous or inviscid situation. There could be laminar or turbulent situation. incompressible versus compressible single versus multi-species so these are all these numbers which are there these are called non-dimensional number which are non-dimensional numbers which are used so obviously one can have a quite generic cfd code but also which can use for multiple purposes but there could be some specific purpose codes which are also available uh because every problem the nature of the problem is different so dealing of those problems are also slightly different that's what happens now when you talk about navier-stoke these are the two great scientists who came up with this navier-stokes equation so where you have this conservation loss so first basic conservation law is the mass conservation then you have mass momentum and energy obviously when you talk about the wind turbine then you have also angular momentum conservation that is required and when you deal with the reacting system then you have a species conservation and all these things so continuity equation talks about the mass conservation equation so this is a complete continuity equation one can use but this is the simplification for incompressible so for incompressible flow for example for wind turbine mostly we will be dealing with incompressible situation so here this simplified in continuity equation would be good enough to use then you have momentum conservation equation so the this is a local time derivative term this is convective term this is pressure gradient term this is or surface force this is diffusion term mass force term so these are now you can again simplify for incompressible flow and this could be used for incompressible situation this kind of uh equation.



$$\boxed{\frac{dM}{dt} = \dot{m}_{in} - \dot{m}_{out}}$$

$$\dot{m}_{in} = \dot{m}_{out}$$

$$\boxed{\frac{dM}{dt} = 0}$$

Mass
Momentum
Energy

Navier-Stokes Equation I

- Mass Conservation→Continuity Equation

$$\boxed{\frac{D\rho}{Dt} + \rho\frac{\partial U_i}{\partial x_i} = 0}$$ Compressible

$$\rho = const, \frac{D\rho}{Dt} = 0$$

$$\boxed{\frac{\partial U_i}{\partial x_i} = 0}$$ Incompressible

Wind turbine

Navier-Stokes Equation II

➢ Momentum Conservation→Momentum Equation

$$\underbrace{\rho\frac{\partial U_j}{\partial t}}_{I}+\underbrace{\rho U_i\frac{\partial U_j}{\partial x_i}}_{II}=-\underbrace{\frac{\partial P}{\partial x_j}}_{III}-\underbrace{\frac{\partial \tau_{ij}}{\partial x_i}}_{IV}+\underbrace{\rho g_j}_{V}$$

$$\tau_{ij}=-\mu\left(\frac{\partial U_j}{\partial x_i}+\frac{\partial U_i}{\partial x_j}\right)+\frac{2}{3}\delta_{ij}\mu\frac{\partial U_k}{\partial x_k}$$

I : Local change with time

II : Momentum convection

III: Surface force

IV: Molecular-dependent momentum exchange(diffusion)

V: Mass force

Navier-Stokes Equation III

➢ Momentum Equation for Incompressible Fluid

$$\frac{\partial \tau_{ij}}{\partial x_i}=-\mu\frac{\partial}{\partial x_i}\left(\frac{\partial U_j}{\partial x_i}+\frac{\partial U_i}{\partial x_j}\right)+\frac{2}{3}\delta_{ij}\mu\frac{\partial}{\partial x_i}\frac{\partial U_k}{\partial x_k}$$

$$\frac{\partial U_i}{\partial x_i}=0$$

$$\longrightarrow \frac{\partial \tau_{ij}}{\partial x_i}=-\mu\frac{\partial^2 U_j}{\partial x_i^2}-\mu\frac{\partial}{\partial x_j}\frac{\partial U_i}{\partial x_i}=-\mu\frac{\partial^2 U_j}{\partial x_i^2}$$

$$\boxed{\rho\frac{\partial U_j}{\partial t}+\rho U_i\frac{\partial U_j}{\partial x_i}=-\frac{\partial P}{\partial x_j}-\mu\frac{\partial^2 U_j}{\partial x_i^2}+\rho g_j}$$

okay so now if you look at the energy equation this is written in terms of temperature but the energy equations can be written in multiple form so there are different forms of energy equation could be written in terms of enthalpy could be written in internal energy be written in terms of total energy so there are different form and this is written in terms of temperature where again you have time derivative term convection term pressure term uh heat flux term and mechanical energy and so these are the different term which are associated with this so when you talk about wind turbine so you need one more additional additional conservation equation which is angular momentum then because this is a rotating frame of system so anything rotating so it is not only wind turbine or the hydro turbine or the palm compressor so you need angular momentum conservation also so and if you recall towards the beginning of that we are talking about the wind turbine vx and where the angular momentum is getting conserved so this is something is also required so that means you need basic conservation's equation to solve this continuity or mass conservation momentum conservation energy conservation and angular momentum now we'll stop the discussion here and continue the further discussion in next session and conclude.

Navier-Stokes Equation IV

- Energy Conservation → Energy Equation

$$\underbrace{\rho c_{\mu}\frac{\partial T}{\partial t}}_{I}+\underbrace{\rho c_{\mu}U_i\frac{\partial T}{\partial x_i}}_{II}=\underbrace{-P\frac{\partial U_i}{\partial x_i}}_{III}+\underbrace{\lambda\frac{\partial^2 T}{\partial x_i^2}}_{IV}-\underbrace{\tau_{ij}\frac{\partial U_j}{\partial x_i}}_{V}$$

I : Local energy change with time
II: Convective term
III: Pressure work
IV: Heat flux(diffusion)
V: Irreversible transfer of mechanical energy into heat

energy (forms)
→ enthalpy
→ internal energy
→ Total energy

Wind turbine → Additional eq. (Angular mom.)

Thank you