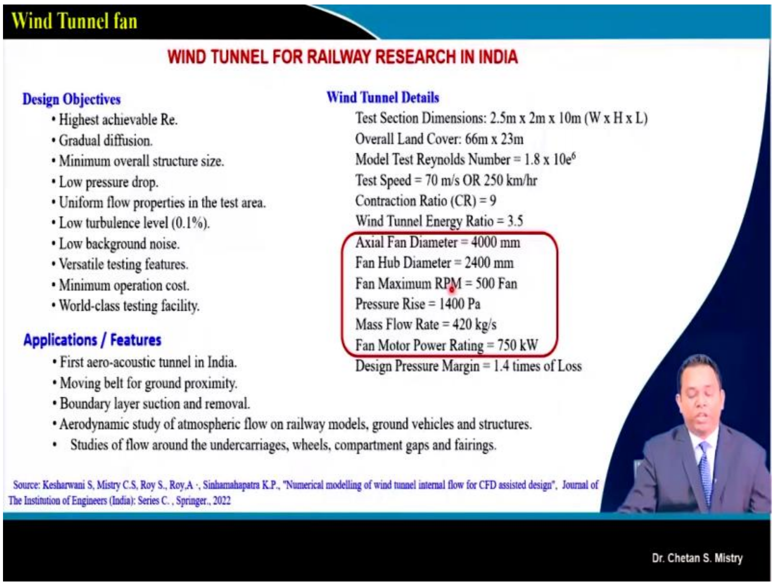
Aerodynamic Design of Axial Flow Compressors and Fans
Professor Chetankumar Sureshbhai Mistry
Department of Aerospace Engineering
Indian Institute of Technology, Kharagpur
Lecture No 62
Design of Industrial Fan (Contd.)

(Refer Slide Time: 00:35)

Hello, and welcome to Lecture 62. We are discussing about the industrial fan design. In last lecture, we were discussing about the compressor design aspects; various aspects we have explored, then future design trend for axial flow compressors and fans. We were discussing about the industrial fans and their application. And this week, as we have discussed, that's what has been dedicated for the design of industrial fans.

(Refer Slide Time: 01:00)



Now, in order to have the design numerical, we have our own design for say wind tunnel fan, and that is the reason why we would like to discuss that design in this lecture. So, this is what is a fan, that's what was been designed for say wind tunnel. This wind tunnel is for railway research in India that's what has been sponsored by say railway research…centre for railway research.

What all are the expectations in sense of design? We are expecting higher achievable Reynolds number, gradual diffusion, minimum overall structure size, lower pressure drop, uniform flow properties in the test section, low turbulence level of the order of 0.1%, low background noise, versatile testing features, minimum operating cost and world class testing facility. With these all design objectives we have defined in certain applications that's what we would like to explore, they are say, this is what will be the first aero acoustic tunnel in India.

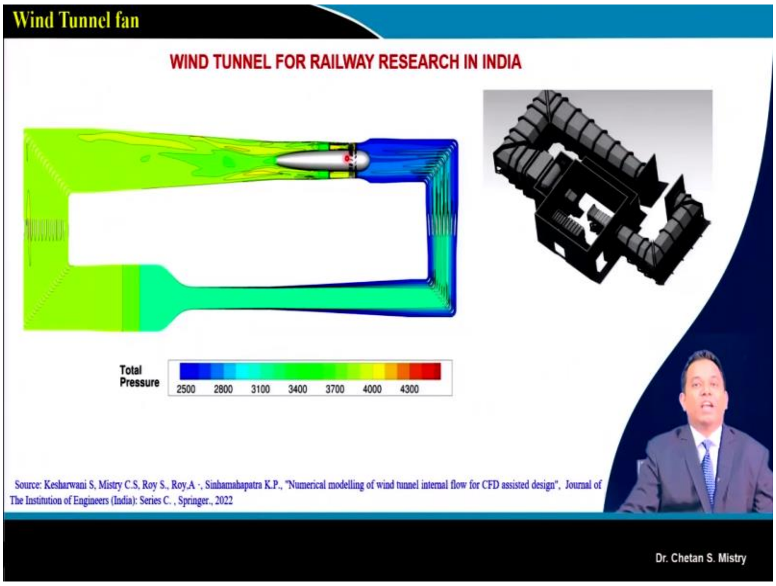
It will be having features like moving belt for the ground proximity, boundary layer suction and removal, aerodynamic study of atmospheric flow on railway models, ground vehicles and structures, studies of flow over undercarriage, wheels, compartment gaps and fairings. So, mainly this tunnel that's what has been required to be designed for say railway research.

Now, in order to have this, these are the dimensions which have been defined with. So, details are our test section, that's what will be of 2.5 m by 2 m by 10 m. Overall length, that will be 66 m by 23 m. Reynolds number that's what is 1.8×10^6, test speed that is 250 km/h, that's what will be the speed of say maximum speed of future train.

Contraction ratio for the nozzle that’s what is 9, wind tunnel energy ratio is 3.5, axial fan diameter that is 4000 mm, fan hub diameter that will be 2400 mm, maximum RPM is 500 RPM, pressure rise it is 1400 Pa, mass flow rate that will be 420 kg/s, power rating for the motor that will be 750 kW and design pressure gain that’s what is 1.4 times of the loss.

So, basically our design targets are this. So, if you look at here, we are looking for fan that need to be designed with a special requirement of casing diameter of around 4 m, hub diameter that will be 2.4 m, the rotational speed for this fan will be maximum by 500 RPM and motor capacity that’s what is 750 kW. So, let us look at what all we have explored here.

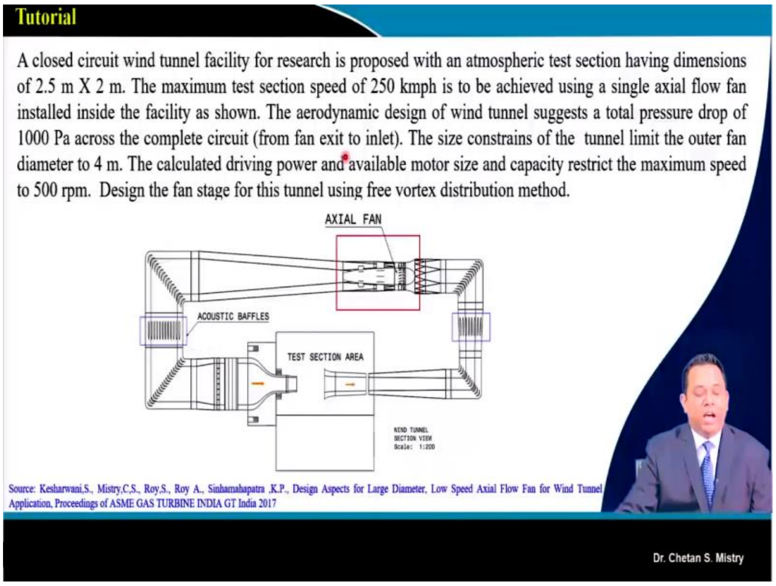
(Refer Slide Time: 04:30)



So, this is what is a final CFD simulation for the wind tunnel that’s what will be developed at IIT, Kharagpur in collaboration with railways. So, here if you look at, this is what is a fan that’s what need to be designed in order to learn this wind tunnel and this is what is with full scale we have done the simulation using CFD. So, more detail, that’s what is available in this paper those who are interested can go through this paper.

And, this is what is say the drawing that’s what will be giving idea. So, this tunnel, that’s what will be used both for say open jet configuration as well as in closed configuration and this is what is our test section, if you look at that’s what is giving more uniform flow as per our expectation. So, now here this is what we are looking for in terms of designing. So, let us discuss about the design of this fan.

(Refer Slide Time: 05:31)

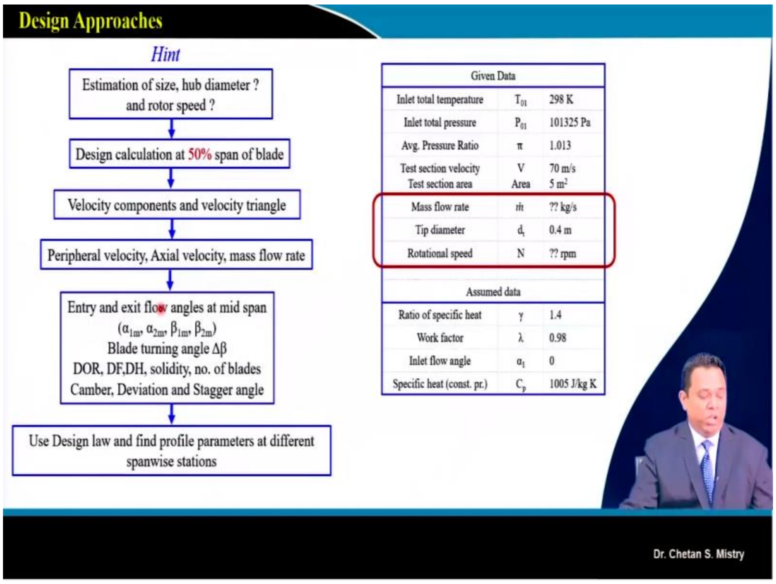


So, the details are closed circuit wind tunnel facility for the research is proposed with an atmospheric test section having dimensions of 2.5 m by 2 m. Maximum test section speed is 250 km/h to be achieved using single axial flow fan installed inside the facility; this is what is a fan we are discussing about. The aerodynamic design of wind tunnel suggests the total pressure drop is 1000 Pa across the complete circuit.

So, say from inlet to outlet, if we are going through our design calculation based on the standard formula it says the pressure drop in the whole circuit will be of 1000 Pa. The size constraint of the tunnel limits the outer fan diameter to be 4 m, calculated driving power and available motor size and capacity restricts the maximum speed to be 500 RPM and the design of this fan that need to be done by using the free vortex concept.

So, now this is what is a real kind of design that’s what we can say. So, in order to drew this design, we need to think of how do we proceed with. We are having constraints, that’s what has already been defined with. What all are the possibilities for such kind of design. So, let us look at what all we have understood.

(Refer Slide Time: 07:06)

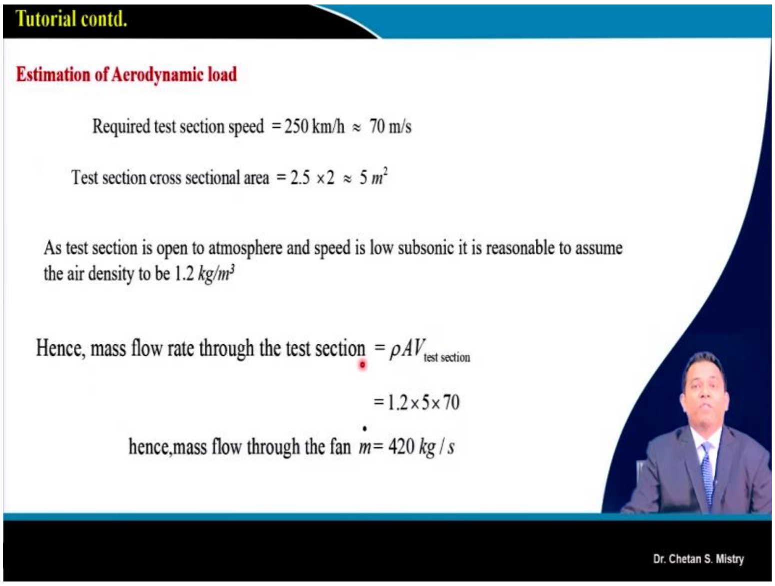


We have our inlet pressure, temperature; we know our pressure ratio, test section velocity, that’s what is say 70 m/s; mass flow rate, total tip diameter, rotational speed, all those things, that’s what we need to decide with. So, mainly here if you look at, this is what is our important parameter for the design of this fan. So, how do we start with is to estimate the size, we need to decide with the hub diameter, we need to decide with what will be the rotational speed, okay.

We will be doing our design at 50 % span, we will be doing our calculation for different velocity components and we will be plotting our velocity triangles for that. Once this is what is known to us we can calculate our cascade parameters, we can test with our say check parameters called say diffusion factor, degree of reaction, de-Haller's factor.

We will be doing our calculation for say cascade geometry, stagger angle, camber angle, incidence angle, deviation angle and once this is what has been done at the mid-section, we will be going with applying say free vortex design method in order to calculate say parameter at different stations or different location along the span.

(Refer Slide Time: 08:30)



Now, in order to do so what all we are looking for is in terms of calculating certain parameters. So here, in this case what it says my required test section speed it is 250 km/h; so that's what is coming roughly 70 m/s. So, this is what is the speed we are looking for at the test section.

$$Required\ test\ section\ speed = 250\frac{km}{h} \approx 70\ m/s$$

The test section dimension it is 2.5 m by 2 m. So, area will be 5 m^2.

$$Test\ section\ cross\ sectional\ area = 2.5 \times 2 = 5\ m^2$$

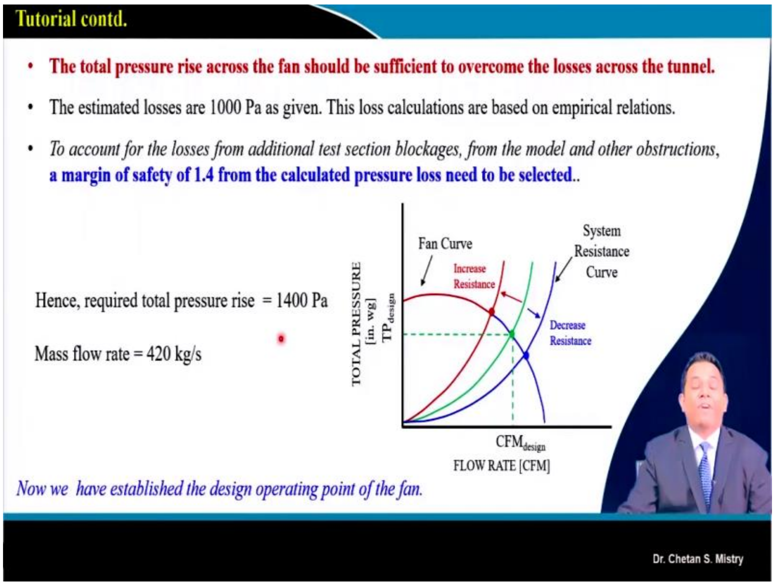
Now, in order to calculate our mass flow rate, we can say that's what is given by density into area into what velocity we are looking for; and that's what is giving the mass flow rate that's what is 420 kg/s, that's what is on higher side what all we have discussed up till now it is not coming in this range. So, understand one thing, here we are looking for handling large mass flow rate of 420 kg/s.

$$Hence, mass\ flow\ rate\ through\ the\ test\ section = \rho AV_{test\ section}$$

$$= 1.2 \times 5 \times 70$$

$$Hence, mass\ flow\ rate\ through\ the\ fan\ \dot{m} = 420\ kg/s$$

(Refer Slide Time: 09:36)



Now, it says the total pressure rise it should be sufficient so that, you know, our fan that will be overcome those loses. Now, the number, that's what is based on correlations…empirical correlation available that is covering say test section, diffuser section, turning, vanes, bends, then honeycomb section, our say screens all those parameters that's what need to be calculated and that calculation it is coming around say 1000 Pa.

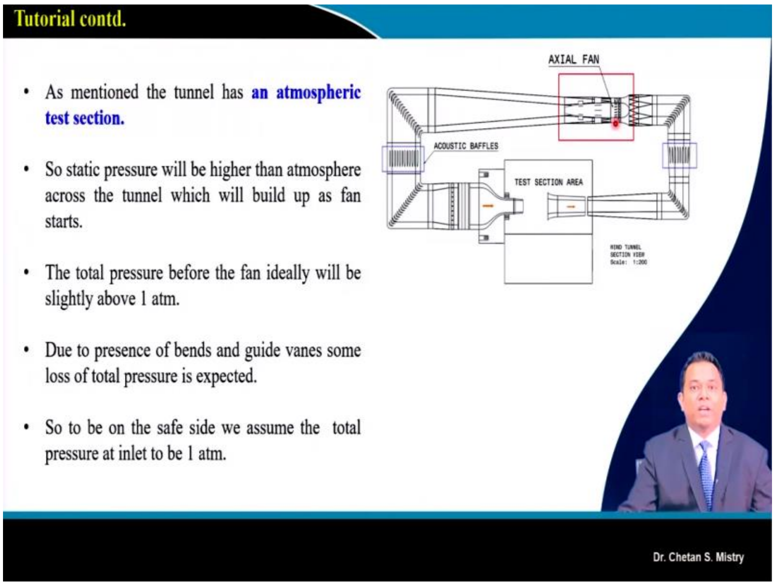
So, here if you look at, this is what is representing the performance curve for the fan. And, this is what is representing my performance of the fan in sense of mass flow rate or we can say volumetric flow rate that's what is defined in sense of CFM versus total pressure rise. Somewhere here, this is what is my design point and here if you look at, this line that's what is representing system resistance curve.

So, this is what is my fan curve and this is what is my system resistance curve. So, we need to realize one thing, you know, with change of my mass flow rate we can plot this system resistance curve and at design point, that's what is meeting with this point here. Now, with increase of resistance my curve…the system curve, that's what will be shifting towards the left-hand side. When we are decreasing our resistance, my system curve, that's what will be moving on right hand side or high mass flow rate side. This is what is very important here, okay.

Now, in order to account for say additional losses because of say our test model blockage, we need to have certain amount of additional pressure losses that need to be counted and thumb rule says we can consider that to be 40% higher. We can say that's what is around say 1400 Pa though it is calculated as 1000 Pa, to be on safe side for the configuration of our test section placement and other losses we are considering that to be 1400 Pa.

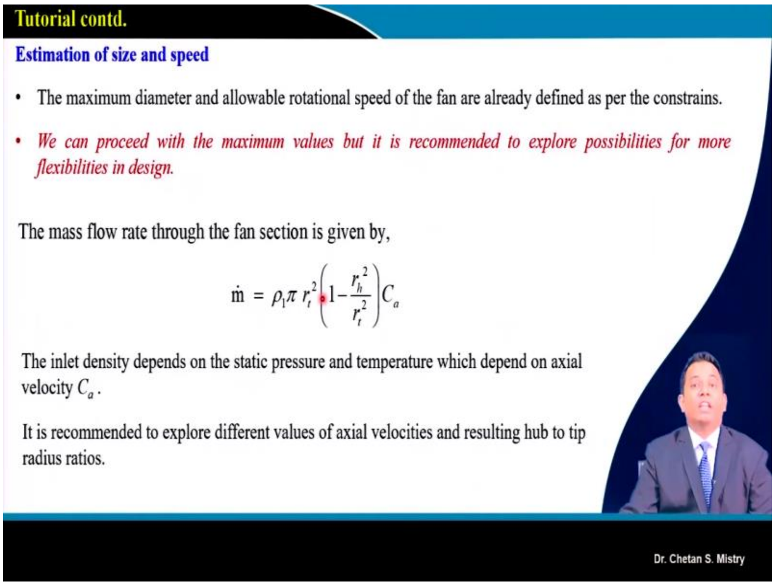
So, now we can say, we are designing our fan for pressure rise of 1400 Pa; mass flow rate that's what is 420 kg/s, okay.

(Refer Slide Time: 12:18)



Now, in order to establish this design, here if you look at, this is what is a kind of configuration. We know this tunnel that's what will be used for both configurations say closed configuration as well as open jet configuration. When we say open jet configuration, under that condition the pressure rise required will be larger. And, that is the reason why, if we configure my inlet pressure at the fan that's what we are assuming to be one atmosphere. That will be coming slightly on higher side, but at the same time because of losses at the entry of fan this pressure will be approximately one atmosphere or what we can say 101325 Pa, okay. Now, we know what is our entry pressure. Now, what we know is what pressure rise we are expecting with.

(Refer Slide Time: 13:12)



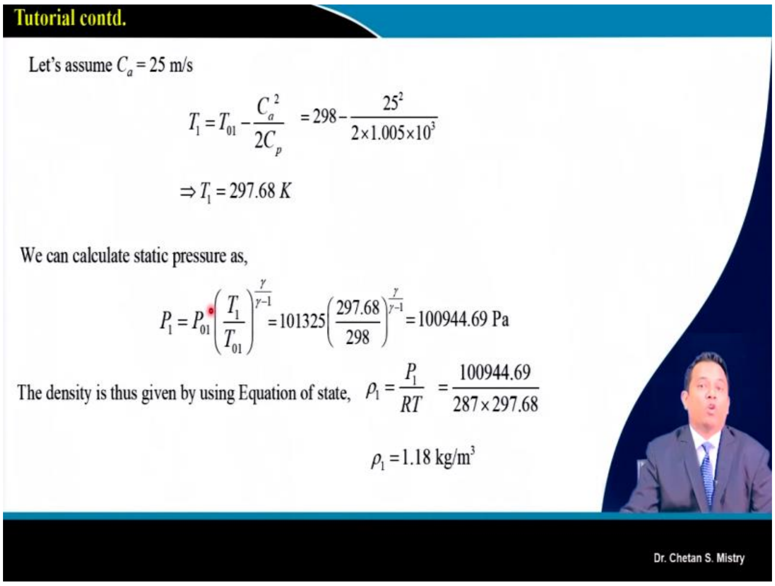
Now, next step for our design it is to decide with the size and the speed. The maximum speed that's what is defined saying like 500 RPM and maximum diameter, they both the things are known to us. So, by that way we can start doing our design, but in order to understand what all are the other possibilities because they are also equally important and that is the reason why this design it has been explored in a different way.

Now, what we have done? We know our mass flow rate, that's what is density into area into our axial velocity. So, the parametric study needs to be done in sense of what need to be hub-to-tip ratio because we are designing our fan, we want to design our fan. We are not having idea about what is our axial velocity. So, it is recommended here to go with the parametric part; so, that's what we have done.

$$The\ mass\ flow\ rate\ through\ the\ fan\ section\ is\ given\ by,$$

$$\dot{m} = \rho_1 \pi r_t^2 \left(1 - \frac{r_h^2}{r_t^2}\right) C_a$$

(Refer Slide Time: 14:16)



Say, initially suppose say we are assuming our axial velocity to be say 25 m/s. If we know our axial velocity, we can calculate what will be the pressure and temperature in sense of static terms. So, static temperature we can calculate it is 297.68 K and static pressure that is coming 100.94 kPa and based on that if we calculate our density, the inlet density that is coming 1.18 kg/m^3.

$$T_1 = T_{01} - \frac{C_a^2}{2C_p} = 298 - \frac{25^2}{2 \times 1.005 \times 10^3}$$

$$\Rightarrow T_1 = 297.68\ K$$

We can calculate static pressure as,

$$P_1 = P_{01}\left(\frac{T_1}{T_{01}}\right)^{\frac{\gamma}{\gamma-1}}$$

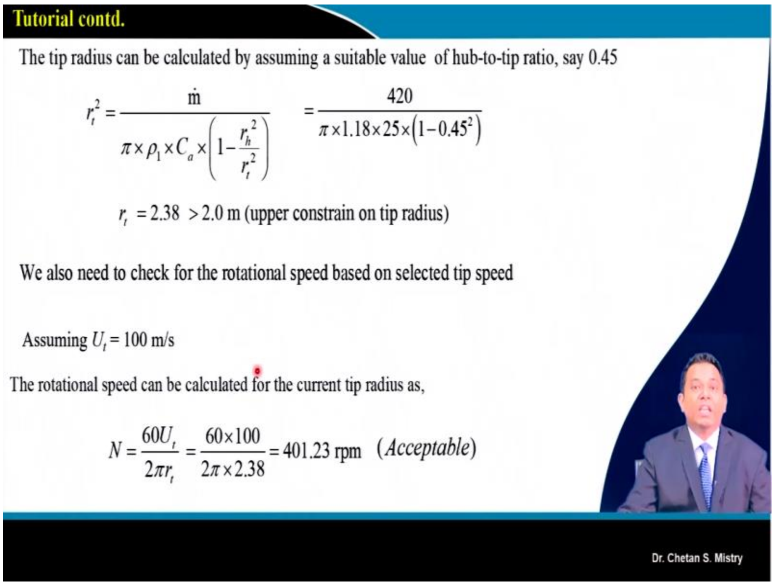
$$= 101325\left(\frac{297.68}{298}\right)^{\frac{1.4}{1.4-1}} = 100944.69\ Pa$$

The density is thus given by using equation of state,

$$\rho_1 = \frac{P_1}{RT} = \frac{100944.69}{287 \times 297.68}$$

$$\rho_1 = 1.18\ kg/m^3$$

(Refer Slide Time: 14:51)



Now, with this if we are putting that in our mass flow rate equation it will give the diameter of the tip and that tip diameter it is coming 2.38 m, that's what is on higher side.

$$r_t^2 = \frac{\dot{m}}{\pi \times \rho_1 \times C_a \times \left(1 - \frac{r_h^2}{r_t^2}\right)}$$

$$= \frac{420}{\pi \times 1.18 \times 25 \times (1 - 0.45^2)}$$

$$r_t = 2.38 > 2.0\ m\ (upper\ constrain\ on\ tip\ radius)$$

We have our constraint with the 2 m diameter and this diameter is coming on the higher side. Let us check with the rotational speed. Suppose, say I am assuming say tip speed…tip peripheral speed to be 100 m/s.

If we are configuring this, it says my rotational speed it is coming 401.23 rpm, that is less than say 500 rpm.

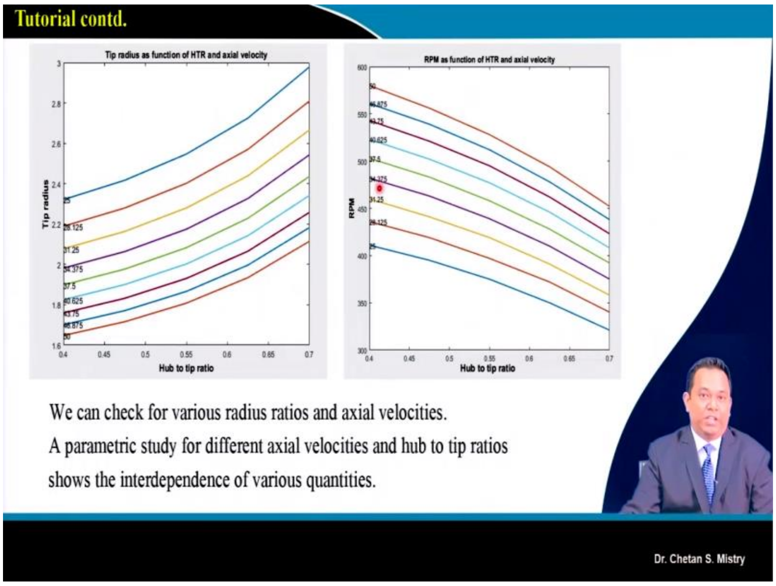
$$N = \frac{60\ U_t}{2\pi r_t} = \frac{60 \times 100}{2\pi \times 2.38} = 401.23\ rpm\ (Acceptable)$$

So, this is what is coming in our acceptable range. Here, for the design of fan there is one more possibility as we have discussed, you can assume your flow coefficient at the mid station;

means C_a/U_{mean} if you are assuming, by that way also you can do your design calculation there is nothing wrong here, okay.

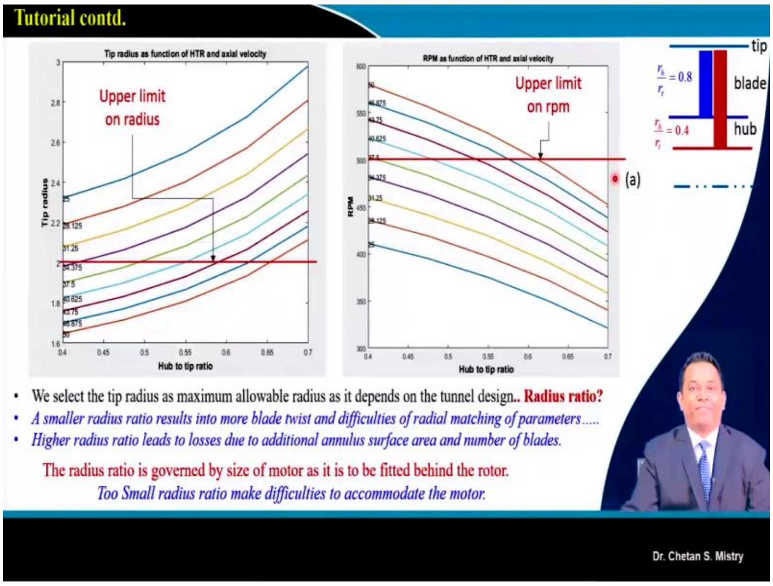
Now, by going through this formulation it is a confusion in sense of putting say diameter as a constraint because that's what is very important, okay. When I am playing with say rotational speed and I am playing with say tip diameter, both are interrelated and that's what is creating the trouble. So, let us try to look at what all can be done.

(Refer Slide Time: 16:22)



What it says? This is what is representing my hub-to-tip ratio because that's what is important parameter, when we are discussing about the fan design. Second parameter, that's what is say tip ratio and here this is what is representing different values of axial velocity. On the other side, this is what has been correlated with say hub-to-tip ratio, my rotational speed and axial velocity.

(Refer Slide Time: 16:49)



Now, what constraint we are having? We have upper constraint with the radius it says my tip radius should not exceed by 2 m, okay. At the same time, we are having constraint with the speed it says it should not be more than say 500 rpm. Now, here in this case we need to decide with what should be our hub-to-tip ratio. So, here if you look at, this is what all we have discussed earlier when I say hub-to-tip ratio of 0.8, my hub-to-tip ratio is 0.4.

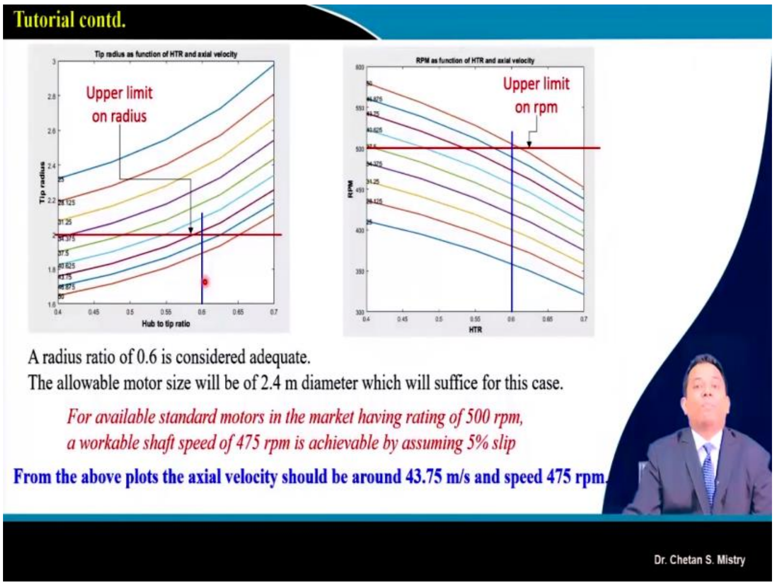
Just realize, here in this case, when we are taking hub-to-tip ratio to be larger, my hub diameter that's what will be coming to be larger. When we say my hub-to-tip ratio to be smaller, my hub diameter is coming smaller. Now, there is a specific meaning here; we need to realize, we need to select the radius ratio that is the reason why this plot it has been given. So, we have two possibilities; either I will be going with say larger diameter ratio or larger radius ratio or we can go with say smaller radius ratio.

If we are going with the smaller radius ratio then here in this case we will be having our blade twist that will be coming to be large that's what is very challenging in sense of matching. We must realize the problem, that's what will be happening near the hub region, okay, and that's what we have discussed again and again in sense of observe parameter as degree of reaction, okay.

There is one more possibility, we can go with say higher radius ratio. If we are going with the higher radius ratio that's what is giving additional annulus surface area and it may require more number of blades. We can understand when my radius ratio, that's what is going to be in higher range, that's what is a problem. Now, the problem is not with the number of blades at this moment or this. Our special requirement is this fan need to be rotate with the motor.

And, that motor that need to be accommodated within our wind tunnel. In order to have that kind of configuration, the diameter of hub should be such that we can put our motor inside, okay.

(Refer Slide Time: 19:20)



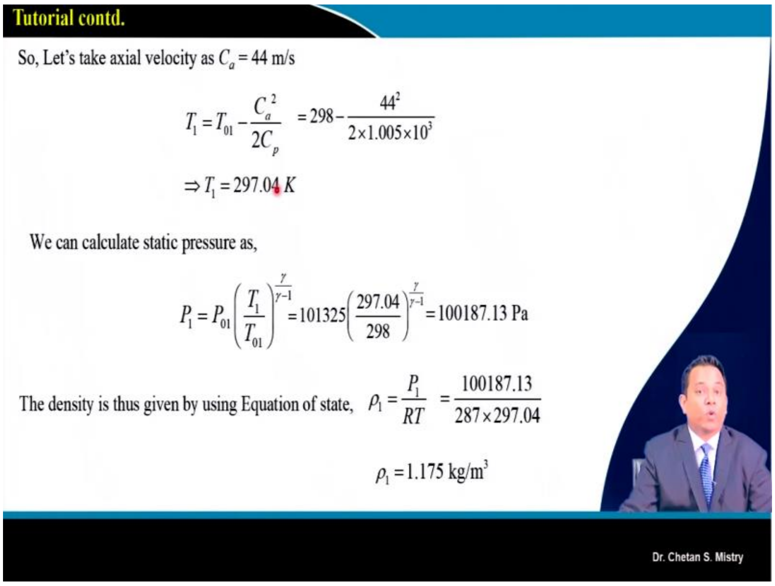
And, looking to this demand, if we go through, we can select the radius ratio roughly to be in the range of 0.6. Suppose say, I am taking this to be 0.6 this is what is giving us idea what need to be the axial velocity. At the same time, as we have discussed, we have constrain with the rotational speed of the motor. Because 750 kW motor, it is a huge power capacity and that's what will be rotating at the lower speed and that speed, that's what is a constraint up to 500 rpm.

So, under that configuration if we will be deciding 0.6 as a radius ratio, we will be having our rotational speed that is coming in the range of 475 rpm and our axial velocity, that's what is coming to be 43.75 m/s. Now, here if we are having say rated speed of 500 rpm if we are configuration say slip, that's what is happening we can say the loss of 5% speed that's what is acceptable.

And, that is the reason why we are saying like 475 rpm, that's what is our required rotational speed. Now, understand one thing here. So, this is what is iterative method with a special requirement. Up till now what all we have discussed, that's what is more in sense of aerodynamics for the selection of say our radius ratio and rotational speed we are defining for compressor as rotational speed of the turbine; or suppose say when we are discussing about the compressor design for industrial application that it will be drive with some additional devices or motors.

So, that's what is being placed maybe outside; and, under that condition we are not having constraints. So, this is what is a constraint design kind of configuration. So, now we have come up with our axial velocity with some range, our rotational speed with some range. Now, let us see how do we proceed with the design.

(Refer Slide Time: 21:30)



So, in order to have this design, what we will be doing? We will be taking say axial velocity suppose say 44 m/s. And we will check with whether it is meeting with the requirement of our say radius because this is what is coming 43.75, let us see taking say 44 m/s. If we are taking that, we can say our static temperature, it is coming 297.04 K.

$$So, let's\ take\ axial\ velocity\ as\ C_a = 44\ m/s$$

$$T_1 = T_{01} - \frac{C_a^2}{2C_p} = 298 - \frac{44^2}{2 \times 1.005 \times 10^3}$$

$$\Rightarrow T_1 = 297.04\ K$$

Same way we can calculate our static pressure that's what is 100.18 kPa.

$$We\ can\ calculate\ static\ pressure\ as,$$

$$P_1 = P_{01}\left(\frac{T_1}{T_{01}}\right)^{\frac{\gamma}{\gamma-1}}$$

$$= 101325\left(\frac{297.04}{298}\right)^{\frac{1.4}{1.4-1}} = 100187.13\ Pa$$

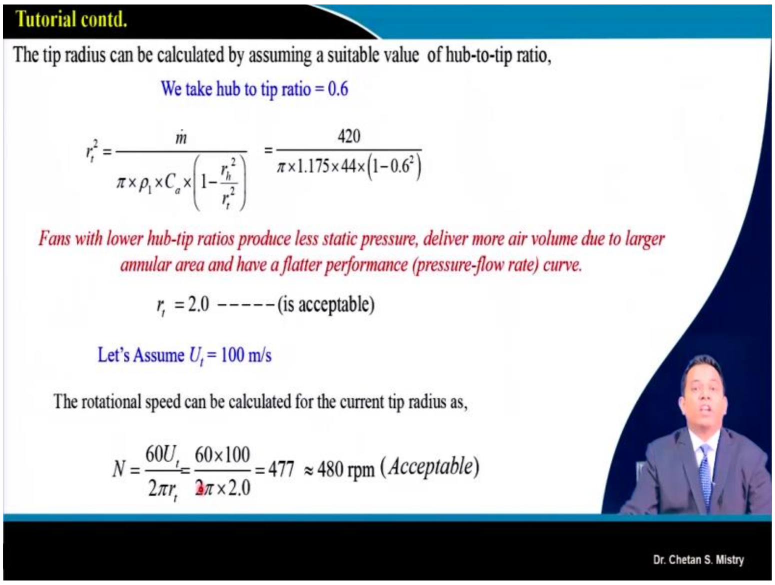
And, our density is coming 1.175 kg/m^3. This is what is in line to what calculation we have done earlier.

$$The\ density\ is\ thus\ given\ by\ using\ Equation\ of\ state,$$

$$\rho_1 = \frac{P_1}{RT} = \frac{100187.13}{287 \times 297.04}$$

$$\rho_1 = 1.175\ kg/m^3$$

(Refer Slide Time: 22:16)



Now, based on that we will be checking with say our radius ratio. Now, for this we are assuming our radius ratio to be 0.6 because we have done a parametric study. It says my radius ratio we can configure as say 0.6. If we are putting this as 0.6, my tip diameter is coming 4 m or we can say my tip radius is coming say 2 m, okay. Now, the hub with lower hub-to-tip ratio, that's what is producing less pressure that's what is our requirement.

$$r_t^2 = \frac{\dot{m}}{\pi \times \rho_1 \times C_a \times \left(1 - \frac{r_h^2}{r_t^2}\right)}$$

$$= \frac{420}{\pi \times 1.175 \times 44 \times (1 - 0.6^2)}$$

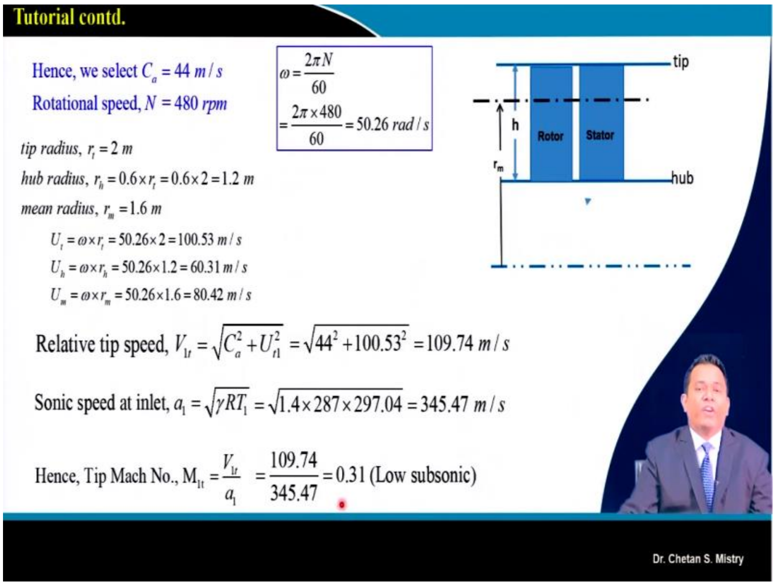
$$r_t = 2.0\, m\ (is\ acceptable)$$

And deliver more amount of mass flow rate with larger annular area and that's what is giving flatter operating characteristic because that is also of our demand. We have seen for this particular fan if we are putting our performance map with pressure variation, with mass flow rate variation along with our system curve it may be possible that we will be having shift of our system curve towards the high mass flow rate or towards the low mass flow rate configuration, okay.

And, that's what is meeting with our requirement. So, let us say suppose we are taking our tip speed to be 100 m/s and for that our rotational speed is coming 480 rpm. Now, we can say this is what is in line to what we are looking for say constraint of 4 m diameter and constraint of our 500 rpm.

$$N = \frac{60U_t}{2\pi r_t} = \frac{60 \times 100}{2\pi \times 2.0} = 477 \approx 480\, rpm$$

(Refer Slide Time: 23:52)



So, let us take this as a data for the next calculation…further calculation. We have selected our axial velocity to be 44 m/s and rotational speed to be 480 rpm. Now, we know our tip radius to be 2 m based on hub-to-tip ratio; we can say our hub radius that is coming 1.2 m and mean radius that's what we can calculate is coming to be 1.6 m, okay. So, based on that we can do our calculation for tip Mach number because we need to check with whether our flow near the tip is going on high subsonic side or is it going on a transonic region.

So, here if we are putting this as a number, it says my tip mach number it is going 0.31. We can say this is what is low subsonic kind of configuration, okay. So, now we have fixed with all our radius, we have fixed with our axial velocity.

$$Hence, we\ select\ C_a = 44\ m/s$$

$$Rotational\ speed, N = 480\ rpm$$

$$mean\ radius, r_m = 1.6\ m$$

$$U_t = \omega \times r_t = 50.26 \times 2 = 100.53\ {}^{m}/_{s}$$

$$U_h = \omega \times r_h = 50.26 \times 1.2 = 60.31\ {}^{m}/_{s}$$

$$U_m = \omega \times r_m = 50.26 \times 1.6 = 80.42\ {}^{m}/_{s}$$

$$Relative\ tip\ speed, V_{1t} = \sqrt{C_a^2 + U_{t1}^2}$$

$$= \sqrt{44^2 + 100.53^2}$$

$$= 109.74\ m/s$$

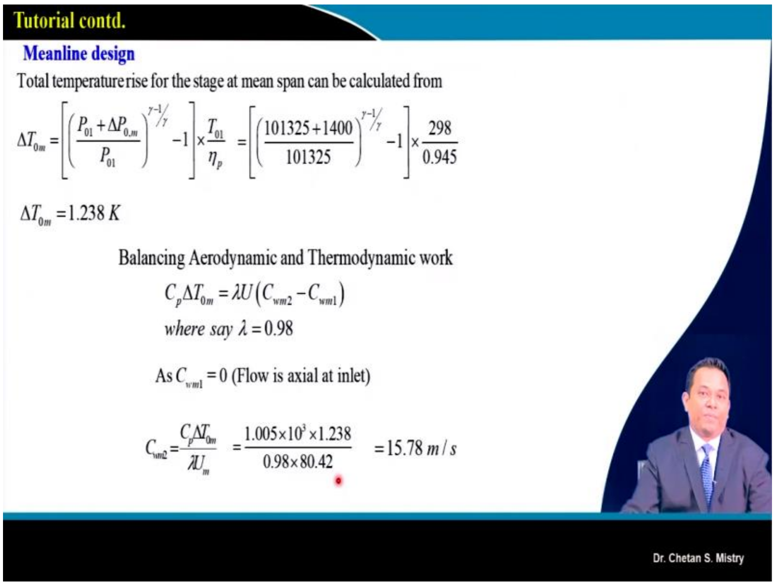
$$Sonic\ speed\ at\ inlet, a_1 = \sqrt{\gamma R T_1}$$

$$= \sqrt{1.4 \times 287 \times 297.04}$$

$$= 345.47\ m/s$$

$$Hence, Tip\ Mach\ no., M_{1t} = \frac{V_{1t}}{a_1} = \frac{109.74}{345.47} = 0.31\ (Low\ Subsonic)$$

(Refer Slide Time: 24:58)



Next, we will be doing our calculation…design calculation at the mid station. Now, for mid station, what we know, we are expecting our pressure rise to be 1400 Pa. So, based on our understanding we can say ΔT_0 at the mid station it is given by this formula. Let us assume this polytropic efficiency to be 94.5%. This is what is slightly on a higher side, we are going aggressively in sense of design, we are looking for higher efficiency and that is the reason we are selecting this to be 94.5%.

Total temperature rise for the stage at mean span can be calculated from

$$\Delta T_{0m} = \left[\left(\frac{P_{01} + \Delta P_{0,m}}{P_{01}}\right)^{\frac{\gamma-1}{\gamma}} - 1\right] \times \frac{T_{01}}{\eta_p}$$

$$= \left[\left(\frac{101325 + 1400}{101325}\right)^{\frac{1.4-1}{1.4}} - 1\right] \times \frac{298}{0.945}$$

$$\Delta T_{0m} = 1.238\ K$$

Now, for that if we are putting our energy balance and aerodynamic work, we can get what will be our whirl component distribution at the mid station. Now, entry for this fan we are assuming to be axial that is the reason why C_{w1} at the mid station is 0 and if we are putting that in this equation, it gives me whirl distribution to be 15.78 m/s.

$$Balancing\ Aerodynamic\ and\ Thermodynamic\ work$$

$$C_p \Delta T_{0m} = \lambda U (C_{wm2} - C_{wm1})$$

$$where\ say\ \lambda = 0.98$$

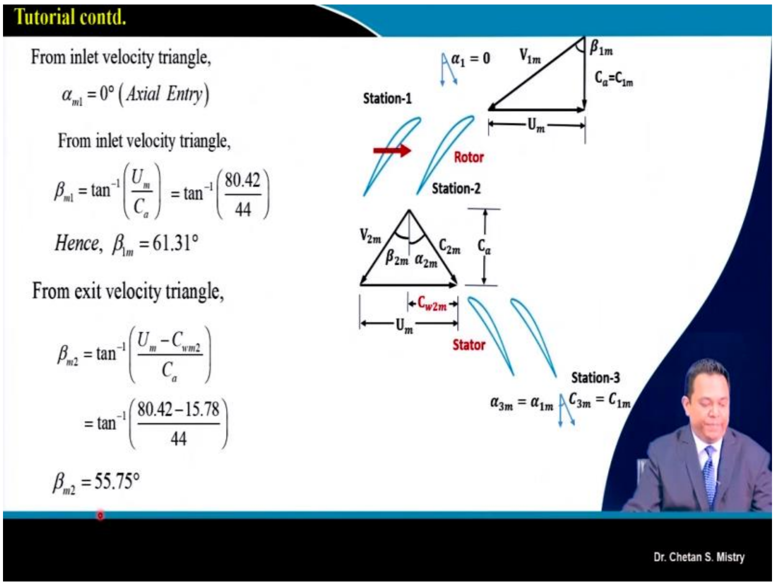
$$As\ C_{wm1} = 0\ (Flow\ is\ axial\ at\ inlet)$$

$$C_{wm2} = \frac{C_p \Delta T_{0m}}{\lambda U_m}$$

$$= \frac{1.005 \times 10^3 \times 1.238}{0.98 \times 80.42}$$

$$= 15.78\ m/s$$

(Refer Slide Time: 26:08)



Now, once we are having mid station calculation, we know our α that is 0, we can say from our inlet velocity triangle we can calculate our β that's what is $\tan^{-1}(U_m/C_a)$. My mid station peripheral speed is known we can calculate our β_1. We can calculate same way β_2. Now, this β_1 is coming 61.31° and β_2 is coming 55.75°.

$$From\ inlet\ velocity\ triangle,$$

$$\alpha_{m1} = 0°\ (Axial\ Entry)$$

$$From\ inlet\ velocity\ triangle,$$

$$\beta_{m1} = \tan^{-1}\left(\frac{U_m}{C_a}\right)$$

$$= \tan^{-1}\left(\frac{80.42}{44}\right)$$

$$Hence, \qquad \beta_{m1} = 61.31°$$

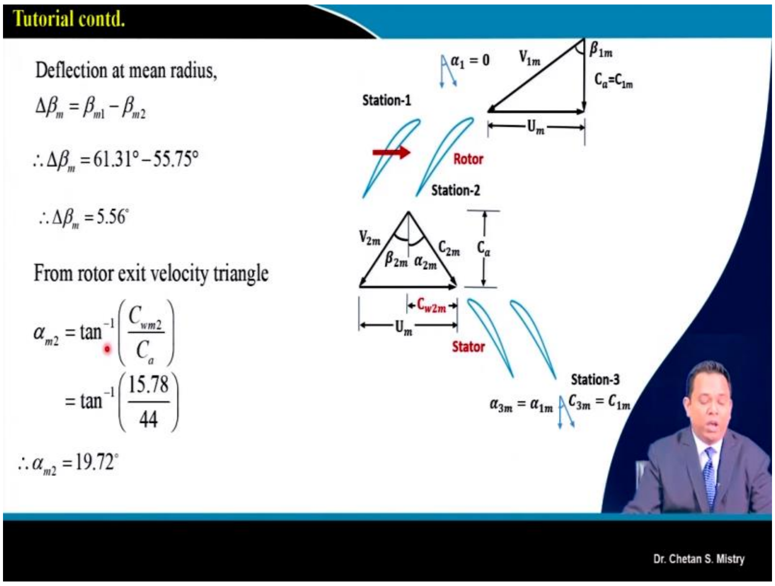
$$From\ exit\ velocity\ triangle,$$

$$\beta_{m2} = \tan^{-1}\left(\frac{U_m - C_{wm2}}{C_a}\right)$$

$$= \tan^{-1}\left(\frac{80.42 - 15.78}{44}\right)$$

$$\beta_{m2} = 55.75°$$

(Refer Slide Time: 26:42)



My $\Delta\beta$ in this case it is coming 5.56°.

$$Deflection\ at\ mean\ radius,$$

$$\Delta\beta_m = \beta_{m1} - \beta_{m2}$$

$$\therefore \Delta\beta = 61.31° - 55.75° = 5.56°$$

Now, we can calculate what will be our exit absolute flow angle and this is what is coming as say 19.72. We know this is what is required for the calculation and design of my stator, okay.

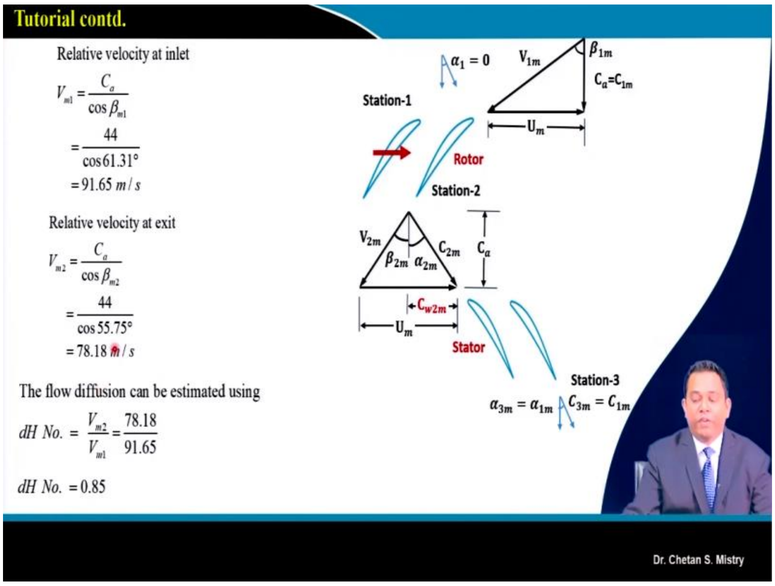
$$From\ rotor\ exit\ velocity\ triangle,$$

$$\alpha_{m2} = \tan^{-1}\left(\frac{C_{wm2}}{C_a}\right)$$

$$= \tan^{-1}\left(\frac{15.78}{44}\right)$$

$$\therefore \alpha_{m2} = 19.72°$$

(Refer Slide Time: 27:04)



Now, based on the velocity triangle since this is what is say axial entry, we can say relative velocity at the entry is given by $C_a/\cos\beta_1$ and this is coming 91.65 m/s.

$$Relative\ velocity\ at\ inlet$$

$$V_{m1} = \frac{C_a}{\cos\beta_{m1}}$$

$$= \frac{44}{\cos 61.31°}$$

$$= 91.65\ m/s$$

Same way, relative velocity at the exit, it is coming as 78.18 m/s.

$$Relative\ velocity\ at\ exit$$

$$V_{m2} = \frac{C_a}{\cos\beta_{m2}}$$

$$= \frac{44}{\cos 55.75°}$$

$$= 78.18\ m/s$$

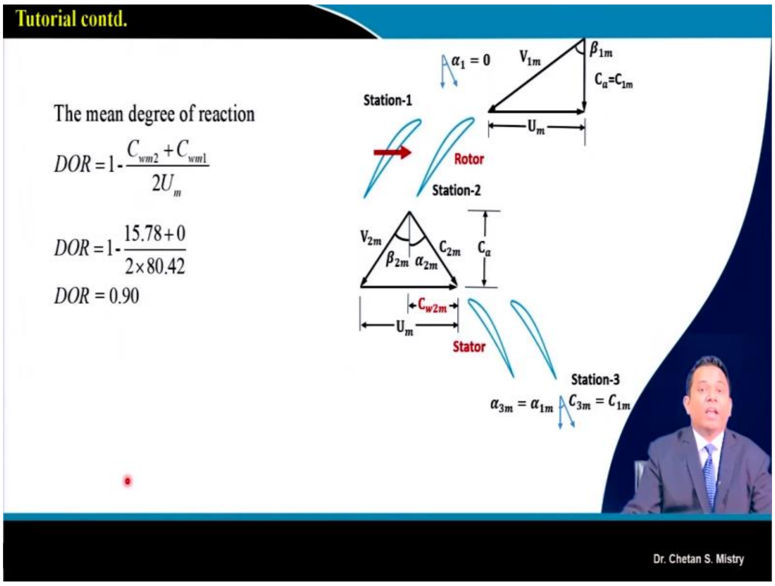
We can calculate our de-Haller's factor and this is what is coming as 0.85, that's what is greater than 0.72 what we are looking for at this moment.

$$The\ flow\ diffusion\ can\ be\ estimated\ using$$

$$dH\ No. = \frac{V_{m2}}{V_{m1}} = \frac{78.18}{91.65}$$

$$dH\ No. = 0.85$$

(Refer Slide Time: 27:41)



The next parameter, that is important for us is to calculate the degree of reaction, at mid station what is our whirl distribution and what is our mean peripheral speed that's what is giving degree of reaction to be in the range of 0.90.

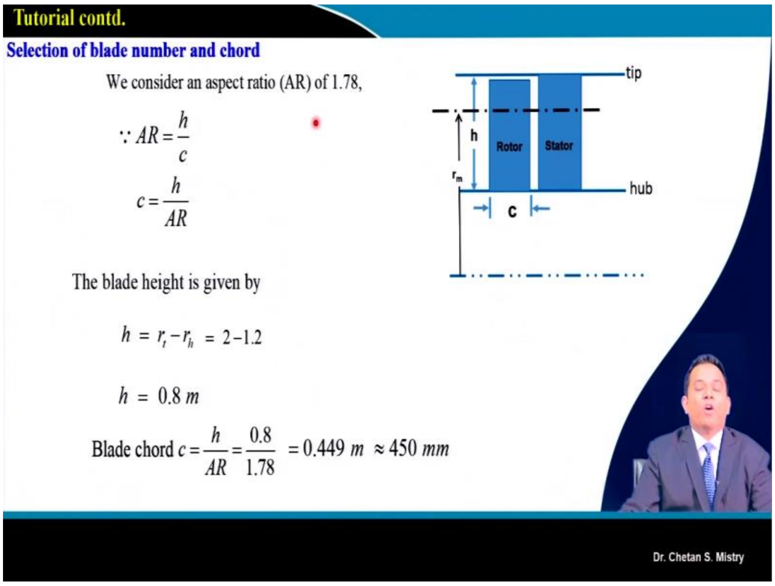
$$The\ mean\ degree\ of\ reaction$$

$$DOR = 1 - \frac{C_{wm2} + C_{wm1}}{2U_m}$$

$$DOR = 1 - \frac{15.78 + 0}{2 \times 80.42}$$

$$DOR = 0.90$$

(Refer Slide Time: 28:01)



Now, after doing all this calculation we need to have the parameter, that's what is called diffusion factor and we are looking for cascade parameters also to be calculated with. Here in this case, say we are looking for what need to be the chord for the rotor. So, in order to do that calculation, we are assuming the aspect ratio as say 1.78. If we are assuming this as 1.78, we will be getting our chord as 450 mm, okay.

$$We\ consider\ an\ aspect\ ratio\ (AR)\ of\ 1.78,$$

$$\because AR = \frac{h}{c}$$

$$c = \frac{h}{AR}$$

$$The\ blade\ height\ is\ given\ by$$

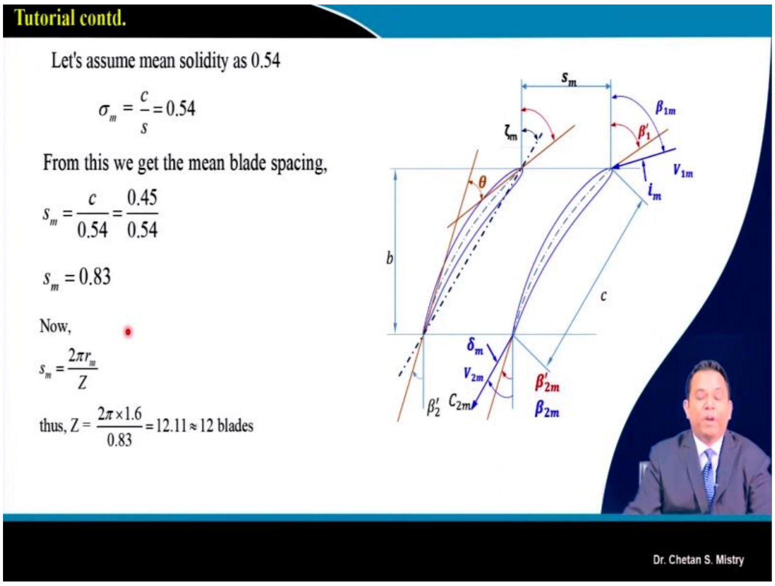
$$h = r_t - r_h = 2 - 1.2$$

$$h = 0.8\ m$$

$$Blade\ chord\ c = \frac{h}{AR} = \frac{0.8}{1.78} = 0.449\ m \approx 450\ mm$$

There is nothing wrong in assuming aspect ratio to be different. This is what is coming with number of iterations and that is the reason why straightway I have kept this aspect ratio as 1.78. We can go with the number of iterations in order to achieve the blade chord.

(Refer Slide Time: 28:56)



Now, once this is what is known to us, next calculation it is for the selection of number of blades. Now, here in this case also we have two possibilities; either we can assume our diffusion factor and based on that we can calculate our solidity and number of blades, okay. Second possibility is you can assume the number of blades and based on those assumed number of blades we can do calculation for the solidity, we can do calculation for the diffusion factor.

Now, here we will be doing this calculation in a different way rather assuming diffusion factor or rather assuming number of blades; let us try with some other approach. So, we will be selecting our solidity that's what is 0.54 at the mid station. Now, this 0.54 if we will be putting in the equation that's what is giving number of blades to be say 12.11 we can say roughly to be 12 blades, okay.

$$Let's\ assume\ mean\ solidity\ as\ 0.54$$

$$\sigma_m = \frac{c}{s} = 0.54$$

$$From\ this\ we\ get\ the\ mean\ blade\ spacing,$$

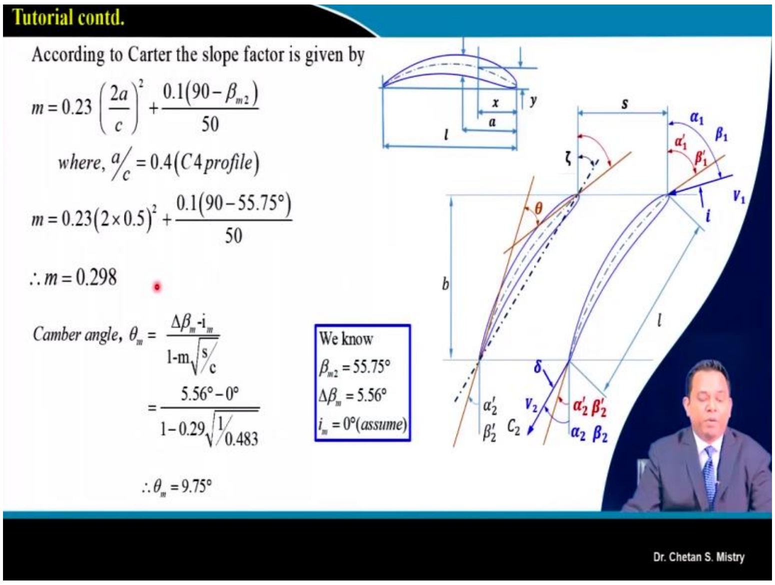
$$s_m = \frac{c}{0.54} = \frac{0.45}{0.54}$$

$$s_m = 0.83$$

$$Now, s_m = \frac{2\pi r_m}{Z}$$

$$Thus, Z = \frac{2\pi \times 1.6}{0.83} = 12.11 \approx 12\ blades$$

(Refer Slide Time: 30:04)



Now, once this number of blades they are being calculated, we can do calculation for say our cascade parameters. So, we will be calculating our Carter's slope factor based on our calculation of β_1 *and* β_2 that is coming 0.298.

$$According\ to\ Carter\ the\ slope\ factor\ is\ given\ by$$

$$m = 0.23\left(\frac{2a}{c}\right)^2 + \frac{0.1(90-\beta_{m2})}{50}$$

$$where, {}^{a}/_{c} = 0.4\ (C4\ profile)$$

$$m = 0.23(2 \times 0.5)^2 + \frac{0.1(90 - 55.75°)}{50}$$

$$\therefore m = 0.298$$

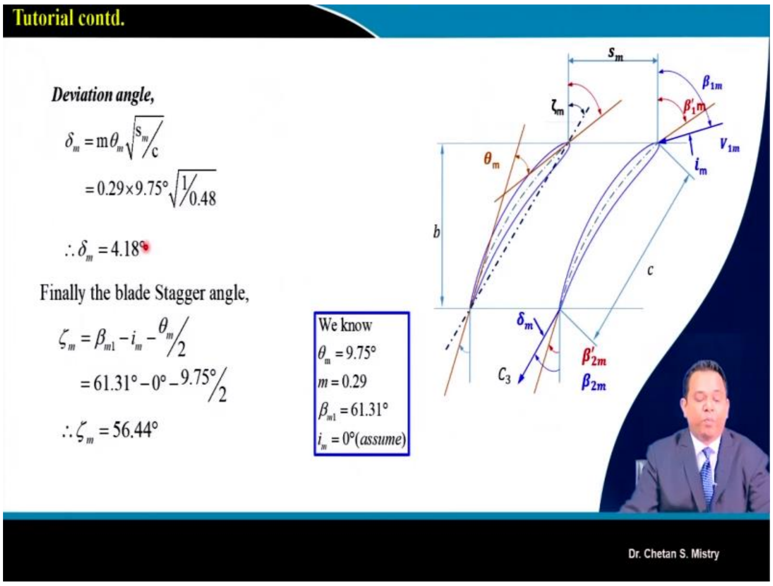
We can calculate our camber angle at the mid station assuming zero incidence angle at the mid station that is giving camber angle of 9.75°.

$$Camber\ angle, \theta_m = \frac{\Delta\beta_m - i_m}{1 - m\sqrt{{}^{s}/_{c}}}$$

$$= \frac{5.56° - 0°}{1 - 0.29\sqrt{{}^{1}/_{0.483}}}$$

$$\therefore \theta_m = 9.75°$$

(Refer Slide Time: 30:37)



Now, deviation angle also we can calculate based on carter's rule, it is coming 4.18. We can calculate our say stagger angle; this stagger angle, it is coming 56.44°.

$$Deviation\ angle,$$

$$\delta_m = m\theta_m\sqrt{{}^{s_m}/_{c}}$$

$$= 0.29 \times 9.75° \sqrt{1/_{0.48}}$$

$$\therefore \delta_m = 4.18°$$

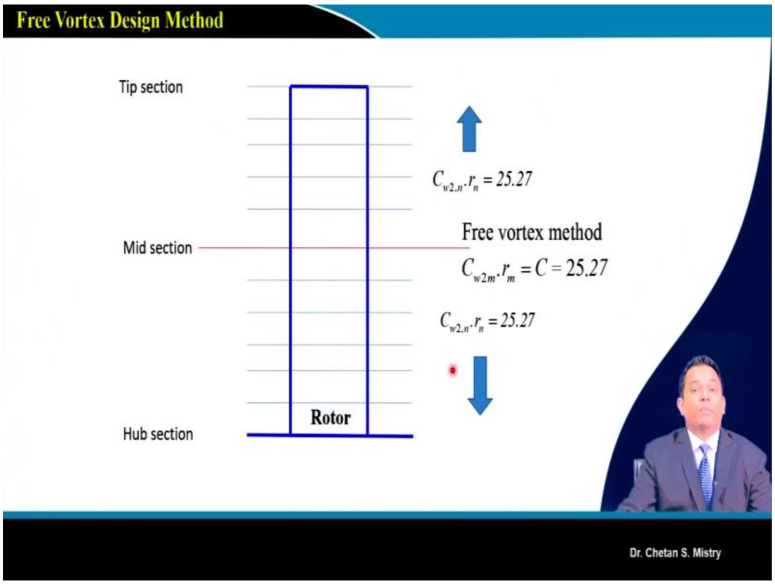
$$Finally\ the\ blade\ stagger\ angle,$$

$$\zeta_m = \beta_{m1} - i_m - {}^{\theta_m}/_2$$

$$= 61.31° - 0° - {}^{9.75°}/_2$$

$$\therefore \zeta_m = 56.44°$$

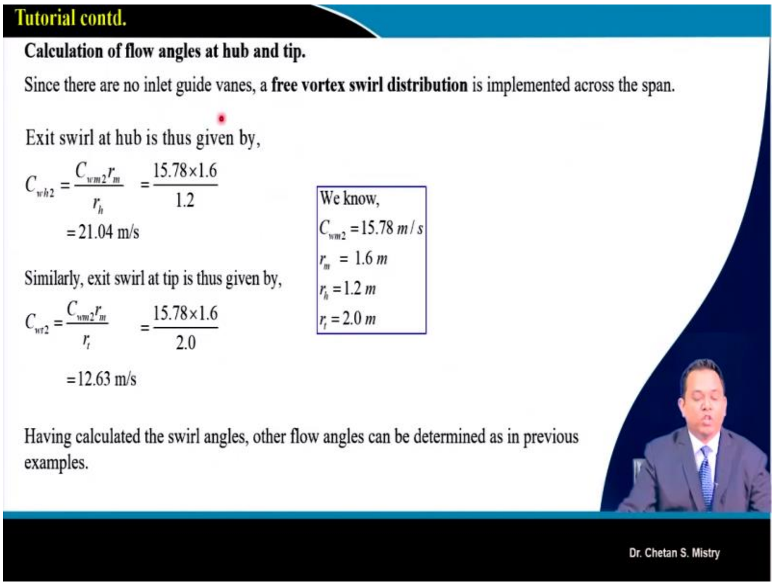
(Refer Slide Time: 30:52)



So, based on all these calculations we are able to calculate the parameters at the mid station. Now, what is our next requirement? It is to calculate the flow angles at different stations. So, what we will be doing? We will be dividing whole span into maybe 10 or 20 number of stations. Here in this case, we are dividing that into more number of stations. We know for free vortex we can calculate our $C_w \cdot r = constant$.

And, that's what we will be applying here with say $C_w \cdot r = constant$ towards say from mid station to tip. Same way, we can say $C_w \cdot r = constant$ from mid station to the hub. That's what will be helping us for calculation of say whirl component at the exit, C_{w2}. Once this C_{w2} that is known to us, we can complete our velocity triangle, we can calculate our flow angles, we can calculate all other parameters which are being required.

(Refer Slide Time: 31:58)



So, here in this case if you are looking at, we will be doing our calculation at the hub, at the tip station. So, we are incorporating our free vortex concept that's what will be giving the C_w at hub as say 21.04 m/s and C_{w2} at the tip is say 12.63 m/s.

$$\text{Exit swirl at hub is thus given by,}$$

$$C_{wh2} = \frac{C_{wm2} r_m}{r_h} = \frac{15.78 \times 1.6}{1.2}$$

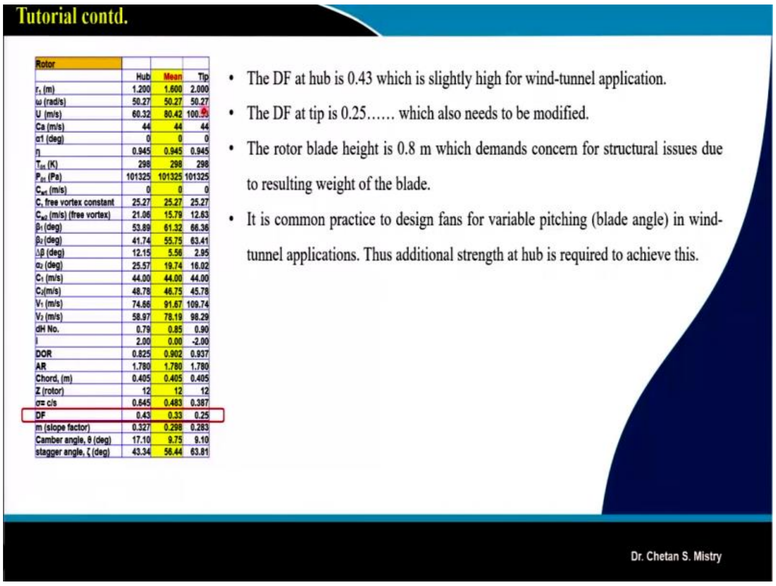
$$= 21.04\ m/s$$

$$\text{Similarly, exit swirl at tip is thus given by,}$$

$$C_{wt2} = \frac{C_{wm2} r_m}{r_t} = \frac{15.78 \times 1.6}{2.0}$$

$$= 12.63\ m/s$$

(Refer Slide Time: 32:21)



And, that’s what will be helping us in sense of making this particular design sheet. So, we are having our hub station, mid station and tip station calculation. Now, once we are doing our calculation for hub, mid and tip station, it is very important to further discuss the modification and the design. So, for present lecture we are stopping here with say having calculation at mid station, having calculation at hub station and tip station.

In next lecture, we will be discussing about what all modification we need to explore in order to achieve desired performance. Thank you, thank you very much.