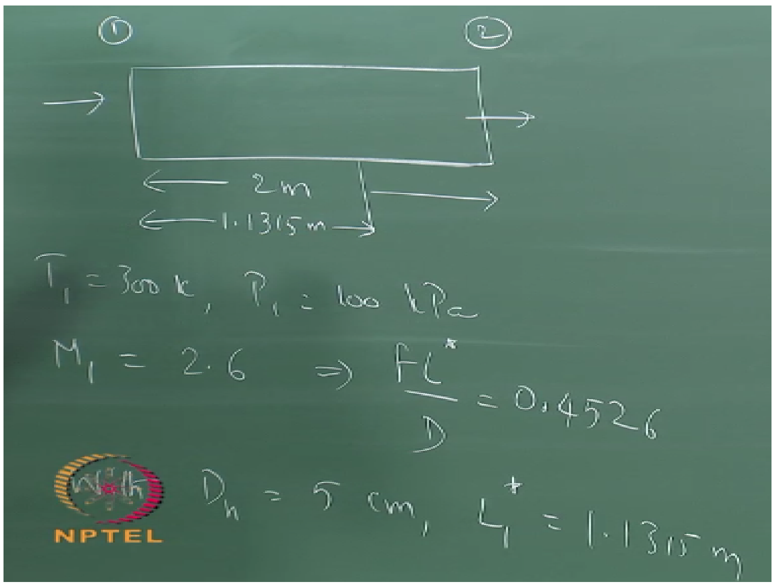
Gas Dynamics and Propulsion

Dr. Babu Viswanathan
Department of Mechanical Engineering
Indian Institute of Technology – Madras

Lecture - 13
Fanno Flow/Quasi One Dimensional Flows

In the previous class, we looked at supersonic flow through a duct.

(Refer Slide Time: 00:15)

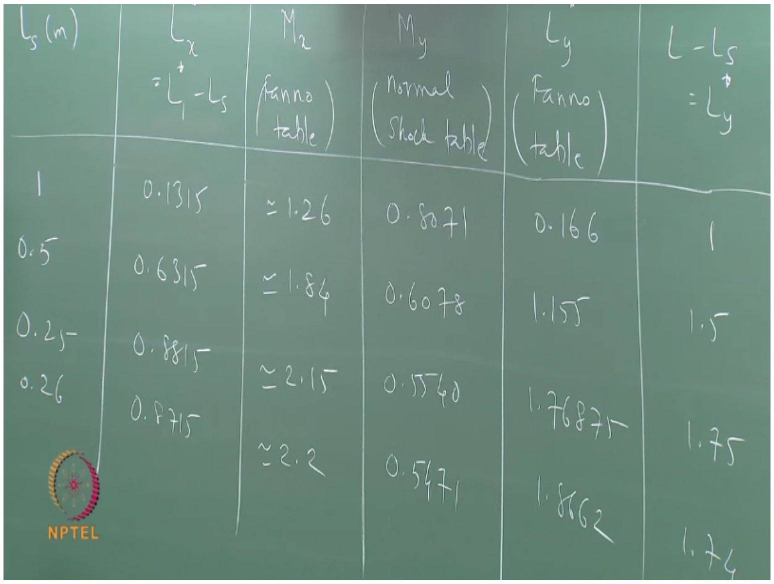


So the duct was square in cross section and at entry, the conditions were given and we were asked to determine the conditions at the exit. The duct length was given to be 2 meters and other inlet conditions were also given. So it was given that T1 = 300 kelvin and the static at the inlet was given to be 100 kilopascal and we calculated M1 in this case to be = 2.6, and from Fanno tables for this value of M1, we obtained FL*/D to be = 0.4526.

And since D here is the hydraulic diameter of the duct, so with Dh, the hydraulic diameter of the duct to be 5 cm. We get L1* to be or L* to be = 1.1315 meters, so this comes out to be 1.1315 meters, which is < this length. So L1* corresponding to this then. So let me label this as L1* because we are going to have multiple values of L*, so we will label this as L1* meaning this is L* corresponding to M1. So this is 1.1315 meter.

So let us say this is 1.1315 meters. So we know that there is going to be a normal shock somewhere in the duct and so we assume a position and then we start the calculation. We are going to do this in a systematic manner. So we create a table.

(Refer Slide Time: 02:29)



So let us say that this is our guess for Ls, which is the distance of the shock. If you remember, yesterday we said that the shock will sit, let us say somewhere here like this and we call this distance Ls and this state was x, this state is y and if you remember, this distance actually we assume the exit mark number to be 1. Let us write that down. We assume M2 to be = 1. So with that assumption, this distance will become Ly*.

So let us put all these things in to table form, then we will go from there. So Lx* as we said yesterday, Lx* is going to be L1*-Ls in meters and we get Mx from Fanno table and then we get My from the normal shock table. Once I get My, I can get Ly* from Fanno table and then I compare Ly* with L-Ls. Notice that Ly* should be = the total length L-Ls. So iteratively, we guess a value for Ls and then we proceed like this.

Once we guess a value for Ls, I know L1*-Ls*, so that is Lx*. For this value of Lx*, I go to the Fanno table, get my Mx, then get my My, then get my Ly*, compare and then keep going like this. If you remember Ly*, the last 2 columns should be equal, so this should be = Ly* until we reach that value, we keep continuing. So begin the iteration, we need to assume the shock to be positioned somewhere.

Since we have no idea where it is, the best way is to use bisection method. Bisection methods starts by saying, so the length of the duct is 2 m, so we assume the shock to be right in the middle. It is like opening a dictionary. If you want to look up a word in a dictionary, that is how you do. Either you start looking at the first half or the second half depending upon where the word is. So bisection method works that way. We start right from the middle.

So we assume the shock to be positioned at Ls = 1 meter. This then gives me Lx* to be L1*-Ls, L1* is 1.1315. so L1*-Ls is going to be 0.1315. Now I go to Fanno table corresponding to this value of L*. Remember we have to calculate F* this/Dh, where we use the hydraulic diameter of the duct. So corresponding to that value, I get my mark number Mx to be approximately 1.26. I now go the normal shock table corresponding to this value of M1, I get my M2 to be 0.8071.

Now I go to the Fanno table corresponding to this value of M, Ly* comes out to be 0.166 and L-Ls, what is L-Ls? L is 2 meter. We have assumed Ls to be 1 meter, so L-Ls is 1 meter. So you can see that obviously Ly* is not = L-Ls. That means we have to change our guess. Which way do we position the shock now. We said it is in the middle. Now do we go this way or that way? That is really not known. We have to try both and then see which way the solution proceeds.

Let us assume that the shock is now located at 0.5. If you are lucky, this will be a lucky guess. So we proceed that way. We can also kind of try to draw some inferences from the numbers that you have. For example, you notice that for this choice of the shock location, I get my Ly* to be much less, which means it actually seems that we need to move this a little bit to the left, so that the Ly* will come out to be larger and comparable to L-Ls.

So that kind of gives us a clue that we should actually take the other choice that we should go this way. So let us make it to be 0.5, then this becomes 0.6315. We follow the same procedure corresponding to this value of Lx*, I get my mark number to be approximately 1.84 and My comes out to be 0.6078. This then comes out to be 1.155, but L-Ls is now 2-0.5, so that is 1.5, but the number seem to be getting closer.

So our inferences from these numbers seem to be okay. What we can conclude from these numbers is that this mark number is actually reasonably close to M=1, which is why we are getting this Ly* to be very small. So if you want to increase this Ly*, this mark number should be lesser. If you want this mark number to be lesser, then this mark number should be more. Remember from normal shock, the higher the initial mark number, the lower the final after shock mark number.

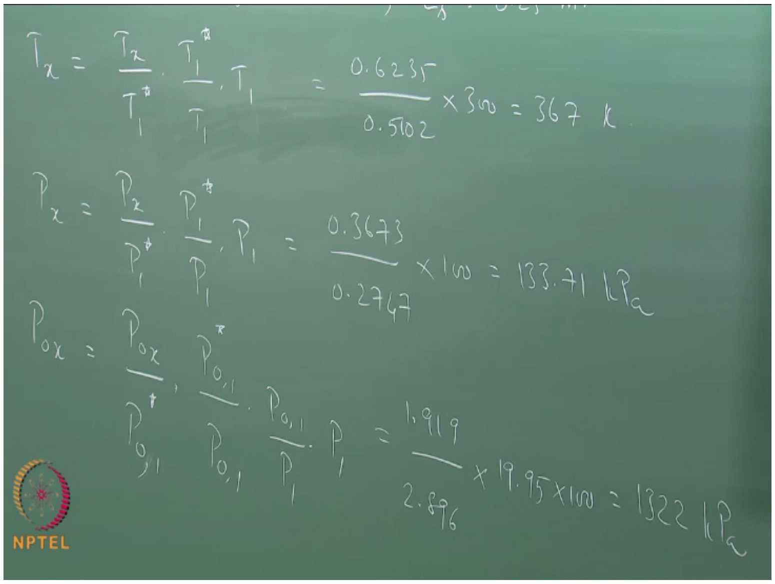
So that means this must occur earlier in the duct. Again you can see that this M is also very close to 1 actually, so that means this is not going to be strong, so we want it to be moved upstream, which is why we made a guess 0.5. So what I am trying to tell you is, I started out by saying that the guess may be a lucky guess, but now I am telling you how to make it an intelligent guess. These numbers are actually trying to tell us something.

There is a physical reason and meaning behind each 1 of these numbers, which we obtained from our knowledge of the theory, from our discussion of the theory. So each 1 of these numbers, we should be able to interpret, then adjust our initial guess. So it need not be a blind or lucky guess. We can actually make an intelligent guess. So now we try to do a little bit more. So these seem to be getting better, which way do we go now? Further to the left.

Because this number is still < this, so same argument. So we go more to the left. Let us make it 0.25 and you go through the sequence, we get this to be 0.8815. This comes out to be approximately 2.15, 0.5540, 1.76875 and L-Ls is 1.75. Now what do these numbers suggest to you? Now they are kind of overshot this value for the first time. So that means we have no bracket at the solution. So the shock is between 0.5 and 0.25. This seems to be reasonably close.

We will try one more value to see whether we can improve this guess. So let us make it 0.26. If you do that and you go through the sequence, you get 0.8715 approximately. Actually, the tables and these numbers are highly non-linear, so sometimes you know we make a small adjustment, we are interpolating in the tables and the behavior is also highly non-linear. So what we can conclude from this exercise is that these 2, we will accept these 2 to be reasonably close to each other. We say that this is acceptable and we take the position of the shock to be 0.25 meters.

(Refer Slide Time: 12:20)

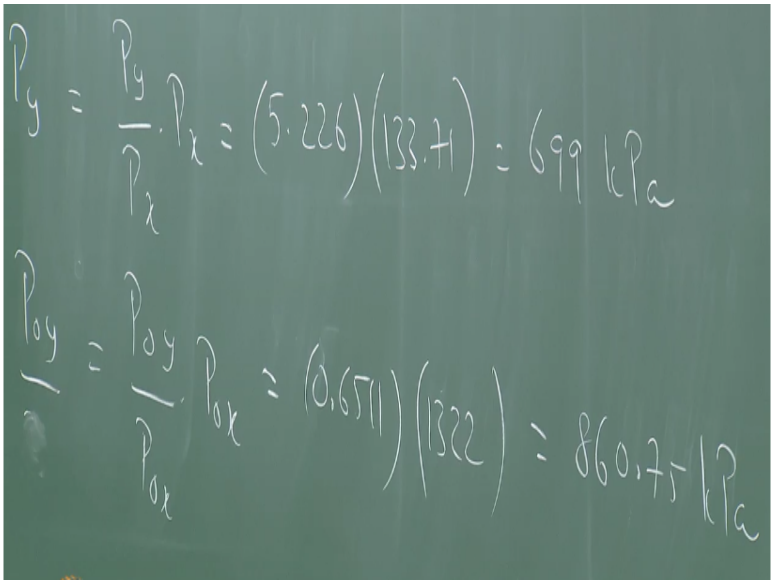


So we will accept this guess and say that Mx=2.15 and we take My to be = 0.5540, Ls to be 0.25 meters. Now we need to determine the exit properties. So we say that Tx = Tx/T1* * T1*/T1*T1. Now since I know Mx, I can calculate these quantities from the Fanno table and I get this to be 0.6235/0.5102*300, so this gives me 367 kelvin for Tx. This is from the Fanno table. I can calculate Px in the same manner. The static pressure just before the shock is.

Once again, if you retrieve these quantities from the Fanno table, you get this to be 0.3673/0.2747*100, which gives me 133.71 kilopascal. There is no change in T0. So we need not worry about it, but there is a change in P0, so I need to calculate P0x in the same manner. P0x = P0x/P01* * P01*/P01 * P01/P1*P1. So this value I can get from Fanno tables. This also I can get from Fanno tables. This I can get from isentropic tables. This is known to me already.

So I can retrieve each 1 of these quantity from different tables and if I plug in the numbers, I get this to be 1.919/2.896*19.95*100, which is =1322 kilopascal. So now, we have to calculate these quantities across the shock wave and then from there we go to the end of the duct to calculate the exit properties. Let us do that.

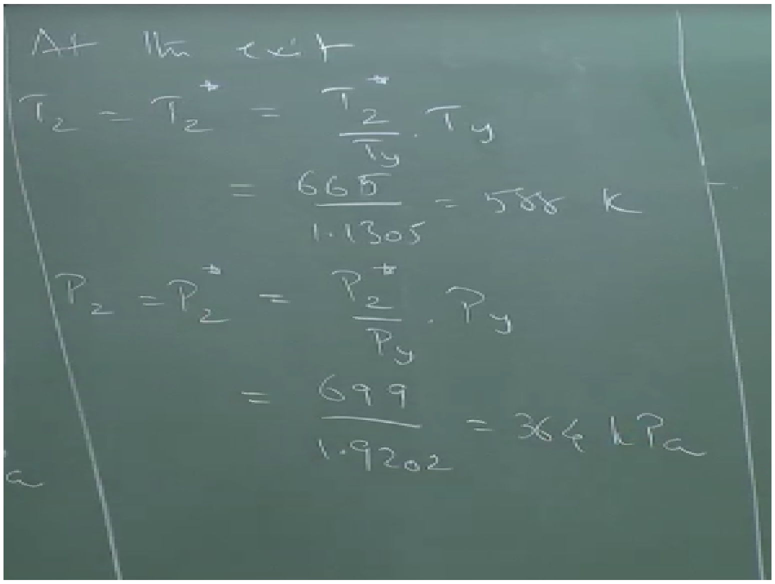
(Refer Slide Time: 16:20)



Flow properties just downstream of the normal shock. So this we get from the normal shock table Ty = Ty/Tx * Tx and from the normal shock table, we get this to be 1.813*367, which is 665 kelvin. Py can be written as Py/Px *Px and from the normal shock table, we get this to be 5.226*133.71, which gives me 699 kilopascal and P0y=P0y/P0x *P0x, so 0.6511*1322 and that is = 860.75 kilopascal. So we have gone from the inlet state.

We have calculated state x using Fanno table, we went across the shock wave, calculated state y using normal shock tables, now we go up to the exit. This is a little bit easy to do because the exit state itself is the sonic state. So we can directly get the values from the table. Let us go ahead and do that.

(Refer Slide Time: 18:53)



So I can write the exit T2=T2*=T2*/Ty * Ty. Ty is known. T2*/Ty I can get from Fanno table, so this gives me 665/1.1305, which is 588 kelvin is the exit static temperature. Similarly, P2=P2*=P2*/Py * Py and this is 699/1.9202, which gives me 364 kilopascal for the static pressure and stagnation pressure can be calculated in the same way P02=P02* and that is = P02*/P0y * P0y and this if you substitute the numbers, you get this to be 689 kilopascal.

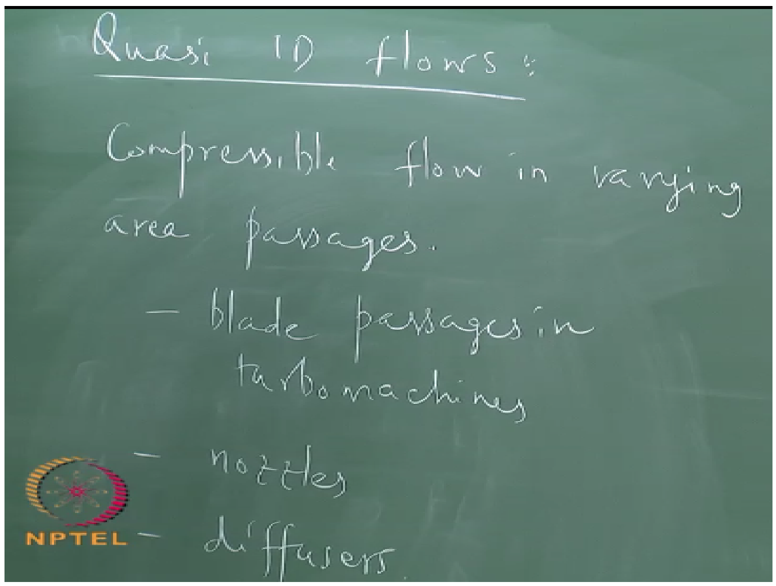
So I have calculated the exit properties also. So you can see how much the values change. The inlet stagnation pressure as you can see has changed so much. It has eventually become 689 kilopascal from quite a high value, so there is loss of stagnation pressure due to friction and also due to the normal shock. With that we come to the conclusion of the discussion on Fanno flows. What we will do next is look at Quasi 1 dimensional flow.

So all the solutions that we have arrived at so far can be categorized as 1D flows. In 1D flow, the underlying implicit assumption in 1D flow is that there is only 1 non-zero velocity component present in the flow, which we took to be velocity along the flow direction or x-direction. We said U was the only component that was present, which means that area term is not present in any of the solutions that we did so far.

All the governing equations had other terms, but area was not present. Even in the case of Fanno flow, we had hydraulic diameter, but not the area itself to be present. So area variation was not considered so far in our calculations. What we are going to do next is look at compressible flow through passages of varying cross sectional area. Now this kind of application or this kind of situation arises in many different practical applications.

So we are not going from 1 dimensional flow to quasi 1 dimensional flow. I will explain in a minute why we call this quasi 1 dimensional flow. So we will see that and this type of flow occurs in compressible flow in varying area passages.

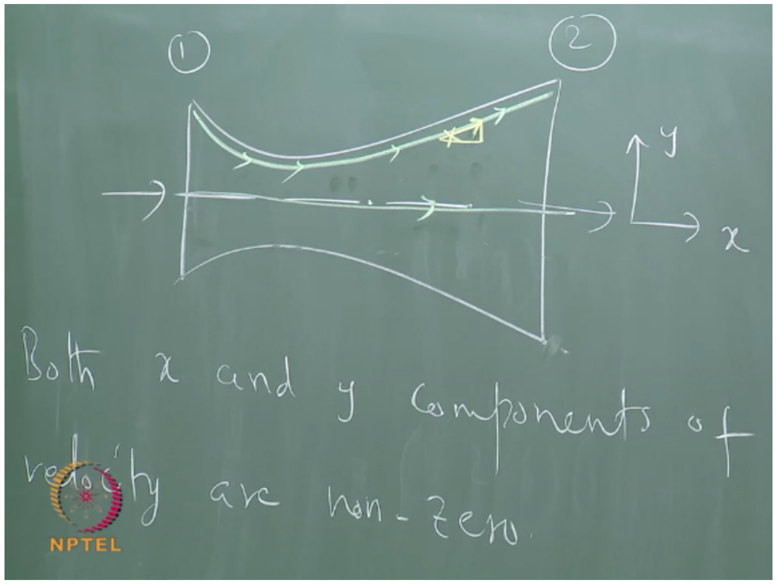
(Refer Slide Time: 24:08)



So we are looking at passages where the cross sectional area changes along the direction of the flow, then we want to account for the effects due to the change in area of cross section and this happens in many different applications, for example blade passages in turbo machinery and propulsion nozzles, diffusers. So this is an extremely important class of flow that we are going to look at and it has very far reaching implications and applications.

Both are very important in this particular class of flows. So let us look at a very simple scenario. Let us say that the passage instead of being constant area.

(Refer Slide Time: 25:16)



Let us say that we have a situation where the area varies along the direction of flow. So we are going to look at flow comes in like this. Let us say this is state 1 just like before, flow leaves here, let us say this is state 2 and as you can see, the passage area varies in some manner along the cross section. Let us say that this is my center line. Now what effect does this changing area have on the flow. That is the first thing that we will look at.

If you sketch a few streamlines in this situation, what do the streamlines look like. So if I sketch a streamline along the axis, that is going to be. The streamline will look like this. It is just a horizontal straight line that we can easily see. So I am going to draw an arrow like this. Now if I draw a streamline, which is very close to the wall, remember we are talking about still calorically perfect gas without viscosity.

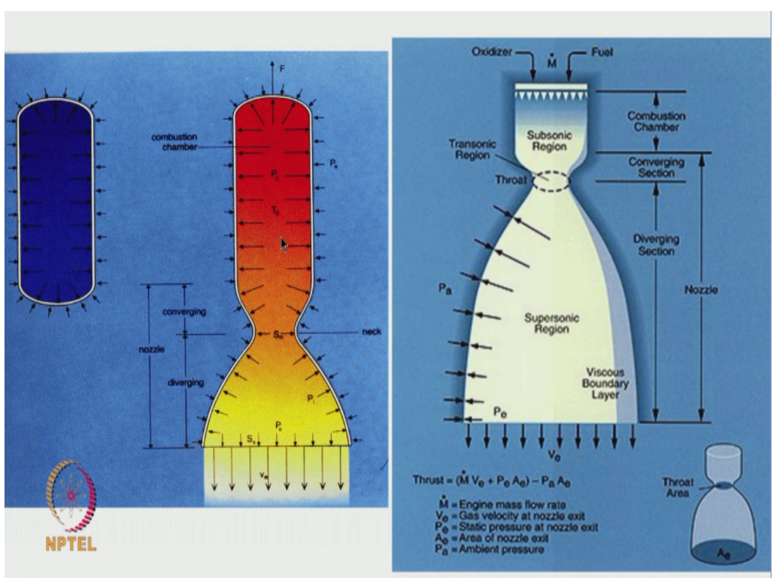
So deceleration of the velocity due to the presence of the wall is not present in this case. So the streamline next to the wall will actually follow the contour of the wall, right. So the streamline next to the wall, if I sketch the streamline, the streamline will look like this. Now you know from your undergraduate fluid mechanics that the velocity vector at any point in the flow field is always tangential to the streamline.

So if I draw a tangent to the streamline at say some point here, then you can see that the velocity component is non-zero not only in the axial direction, but also in the vertical direction. So you can see that tangent to this, will look something like this. A tangent at this point will look like this. So that means it is going to have a component like this and a component like this. So contrary to what we have seen so far, now we are having a situation where 2 velocity components are non-zero.

So here, this velocity component is 0, but as I go towards the wall, the streamline begins to deflect. The deflection increases until I am very close to the wall. So that means the y component or the vertical component of velocity is 0 at the central line and then increases, reaches a maximum near the wall. This is inviscid flow so it reaches a maximum near the wall. Now for the kind of application, this means the 2 velocity components both x and y are non-zero.

If I say that this is my x coordinate direction and this is my y coordinate direction, both x and y components of velocity are non-zero. But it so happens that in many of these applications that we talked about and many of these applications, the actual physical situation is somewhat favourable for us to make an assumption about the nature of flow. So one such thing is shown here.

(Refer Slide Time: 28:49)



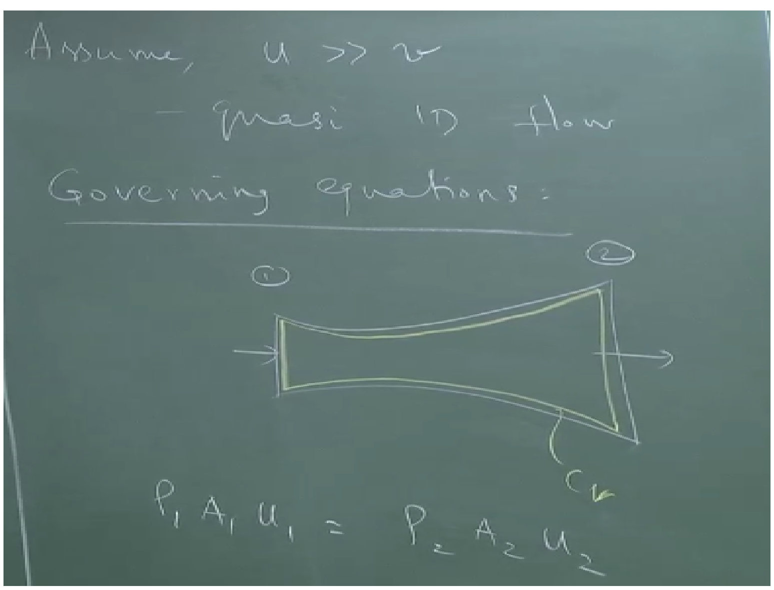
Here I have shown convergent divergent nozzle that is typically used in a rocket engine, as you can see from here. So the flow accelerates as it goes through the nozzle and if you look at the x component of velocity and compare that with the y component of velocity, let us say here or over here as the flow accelerates wherever the y component of velocity is large, the x component of velocity is even larger.

So as the flow accelerates, the x component of velocity increases much more than the y component of velocity. So in fact if you draw a streamline near the wall here, you will notice that the y component of velocity is going to be high near the wall. However, the x component of velocity is even larger in a typical rocket nozzle like this, the x component of velocity at the exit is usually of the order of a few kilometers per second, that is like 2000 or 3000 meters/second.

So even in the y component of velocity is 100 meter/second that is still negligible compared to the x component of velocity, which tells you that the effect of area change is to accelerate the flow in the x direction much, much more than the acceleration in the y direction. So in most of these applications, we can actually ignore the y component of velocity, can be ignored when compared to the x component.

This is an approximation and it turns out to be a very good engineering approximation for practical calculation purposes.

(Refer Slide Time: 30:53)



So the assumption is u, which is the x component of velocity to be much, much larger than v, which is the y component of velocity. So our governing equations will have only one component of velocity, but however area will be allowed to change now. Because they are now accounting for area change through changes in x velocity, so which is why we call this, it is neither a 1D flow nor a 2D. it is not a 1D flow because we are allowing the cross sectional area to change.

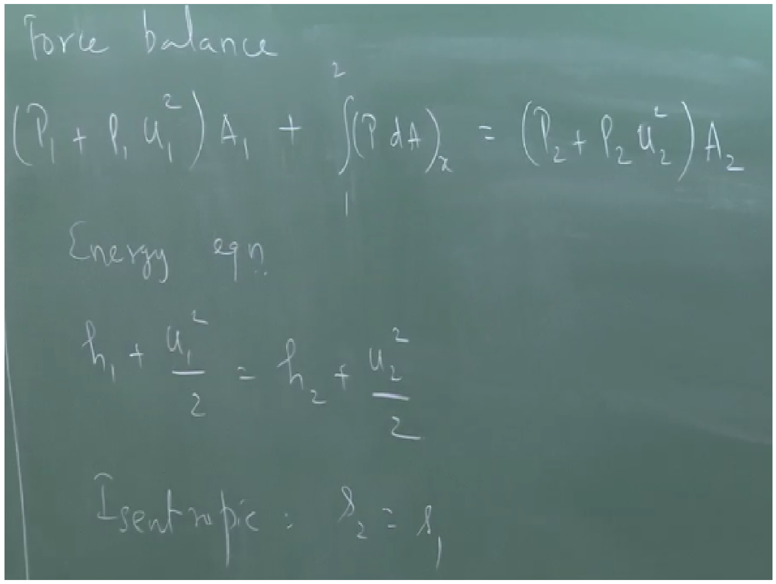
So there are changes in the y direction. So it is not strictly a 1D flow. With this in mind, it is also not a 2D. So it is in between 1D and 2D flow, which is why it is called a quasi 1 dimensional flow. So this concept is extremely important because as we go along and as we move further and further into the chapter, we have a tendency to forget this. Please remember that we are not saying v is 0. We are only saying that u is much large compared to v, v is still non-zero.

That is quasi 1 dimensional assumption. So with this in mind, I will write down the governing equations for quasi 1 dimensional flow in a varying area passage, let us see what that looks like. So if I look at a varying area passage, just like what I did before, let us say that I look at a passage like this. Let us call this as state 1. Let us call this exit state 2. If I take control volume, which is like this. I sketch a control volume.

If this is my control volume, then I can apply mass balance to this and write the mass balance equation as rho1a1u1=rho2a2u2. Notice that for the first time in our governing equations, the area explicitly appears. Now if I do a force balance on this control volume, force balance meaning, rate of change of momentum of the fluid as it goes through the control volume, should be = force exerted on the control volume.

So if I apply this, then I have pressure forces which are acting on different parts of the control volume, fluid comes in and goes out, so there is a change of momentum of the fluid as it comes in and goes out.

(Refer Slide Time: 34:02)



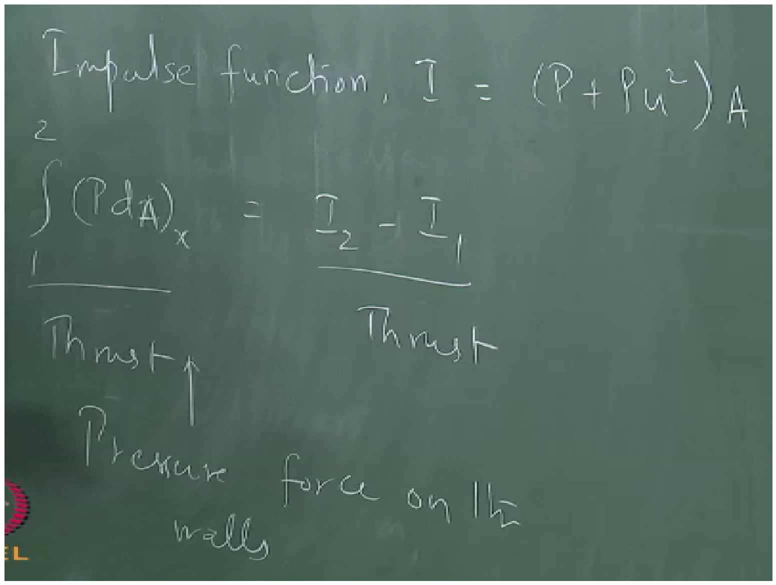
So I can write this force balance as P1+rho1u1 square *a1+integral 1 to 2 Pda. Let us say that we take the x component of this force = P2+rho2u2 square * a2. Notice that the pressure force acts normal to the surface and there is a component of that force which acts in the x direction. So we are writing force balance in the x direction because this is a quasi 1 dimensional flow. We are not worried about the y direction. This is only the force balance in the x direction.

Energy equation remains the same as before. There is no heat addition or heat removal, so energy equation is h1+u1 square/2=h2+u2 square/2 and it is also a good engineering approximation to assume that the entropy that the flow is isentropic and the entropy remains constant, which means S2=S1. However, we also allow for the possibility that under some operating condition, there may be a normal shock sitting somewhere, anywhere, maybe here, here, anywhere in the passage.

So which means that the entropy remains constant up to the shock, increases across the shock and then remains constant afterwards. So we can handle this kind of situation within the framework of this theory, but for now we will put down S2=S1 with the underlying assumption that we can handle situations like this. You know how to handle changes in properties across the shock wave. So I can calculate changes in property up to here.

And then I can go across the shock wave, do the other things, that is possible. Now the first important thing because we are talking about propulsion in this course, thrust of course is the most important quantity that we are interested in. So we are going to look at the force balance. Rewrite the force balance in a slightly different way to get an insight on thrust. I can rewrite the force balance equation like this.

(Refer Slide Time: 37:11)



So I can rewrite the force balance equation as integral 1 to 2 Pdax=P2+rho2u2 square * a2-P1rho1u1 square * a1. Now this suggests that I define new quantity called impulse or impulse function, i is defined as P+rho * u square * a, where a is the cross sectional area at that point. So I can define this quantity P+rho u square * a, which then allows me to rewrite this equation as integral 1 to 2 P da x component=i2-i1.

Now if you think about a propulsion application, let us say this is a rocket engine or an air craft engine and this is a kind of scenario that we are looking at. Then the net change in the impulse function between the outlet and the inlet is nothing but the thrust that is produced by the engine. So I can identify the thrust that is produced by the engine as being the difference between the outlet and the inlet, but this equation tells me that I can also evaluate thrust by integrating the forces that act on the surface.

So I look at the pressure forces that act on the metal surface and if I take the appropriate component and integrate them, that also gives me the net force that is exerted on the nozzle or on the air frame. So the left hand side is also=thrust. So I now have a way in which I can determine thrust, either I calculate the impulse function and do it this way or I calculate the net force on the walls or metal surfaces and then calculate the thrust. It is easy to do this in most applications.

It is very difficult to do this in most applications. It is actually very instructive to take a closer look and see where this term comes from and what this does. Notice that the impulse function has a pressure term also P*a. There is a pressure force here, but this force is the pressure force that acts on the inlet phase and the exit phase of the control volume. So that pressure force acts on the fluid surface. It does not act on the wall or the solid surface.

Whereas the pressure force that appears on the left hand side of this equation is the pressure force that is exerted on the walls. This is pressure force exerted on the walls. So if you mount the engine on an air frame, this pressure force is what the engine would transfer to the airframe, which will then be realized as thrust by the engine. Let us take a closer look, this concept is very important although this quantity is very difficult to calculate in a practical sense.

It is very important to have an understanding of what this force is and where it comes from. We now turn to this slide, which shows this for a simple situation first and then complicated situation. So on the left you see completely closed pressure vessel and you see the arrows on the outside, which correspond to pressure force exerted by the ambient and you see the pressure force on the inside, which correspond to the pressure force exerted by the high pressure gas, which is inside.

Now, we can see that in this case everything is in equilibrium, so the vessel stays stationary, it does not move. Same thing, I open up this part of the vessel and allow the flow to expand through a nozzle. So I make an opening here attach a nozzle, which is what I have done here, now we can see a pressure imbalance of forces. So you see the ambient pressure that vector remains the same. So here we can see high pressure.

The length of the arrow is proportional to the magnitude of the pressure force and then as the flow expands in the nozzle, which we will show as we go along, the pressure decreases and we can see that the length of this arrows also decreases as we go along the nozzle. We can see that much more clearly here. We can see the length of the arrow decreasing as we go along the nozzle, but the length of the ambient pressure arrow remains the same.

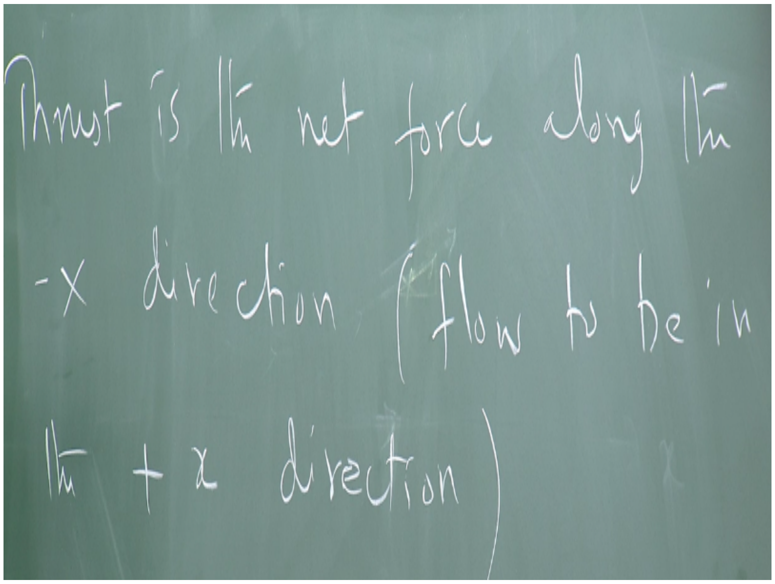
So the net force, if we take a small element here or here, or here, anywhere, if we take a small element of metal or wall, the net force exerted on this wall is actually due to the difference in pressure across. So this – this pressure * the area is the net force that is acting on the metal wall. This is an extremely subtle concept. Notice that I can redraw this picture or this picture by simply subtracting this arrow from this arrow and then removing everything here.

And then redrawing the picture. So if I measure all the pressure with respect to the local ambient, I can subtract the ambient pressure from this that would be the net pressure or net thrust. So this is a notion that we will use when we derive the equation, that if I measure the pressure with respect to the local ambient that will directly give you the net thrust force that is exerted on the metal surface. So you can actually see it is quite interesting to see from this.

Remember if this were to be used in a rocket, in which direction will the thrust force be developed. Thrust force should be developed upwards, but you notice that the convergent portion of the nozzle actually develops net thrust force in the opposite direction, so the convergent portion of the nozzle actually produces drag. It is only the divergent portion of the nozzle that produces thrust because after I subtract this ambient pressure from these arrows.

Now I have a substantial component in the flow direction, which produces thrust.

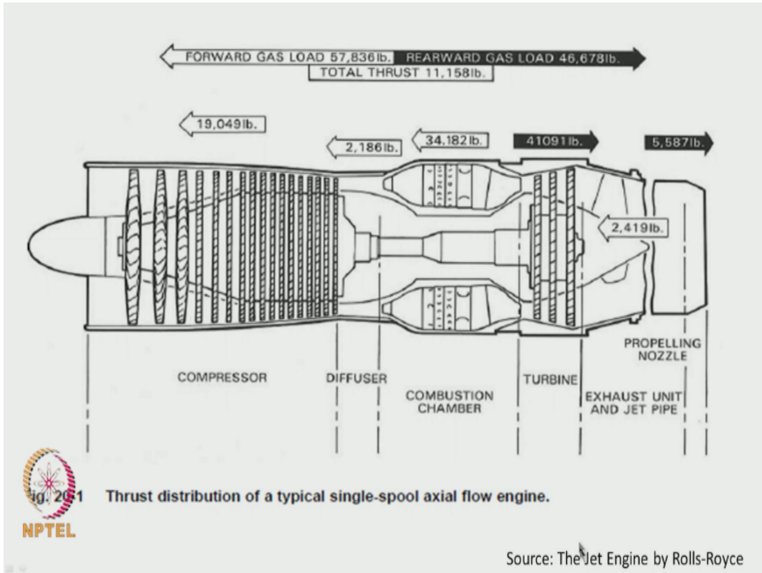
(Refer Slide Time: 44:23)



Notice that thrust is the force along the negative x direction assuming flow to be in the positive x direction. So if the flow is in the positive x direction, thrust acts in the positive direction. When I do i2-i1 and I get a positive value that to me tells that I am producing thrust. If I get a negative value to me tells that I am actually producing drag. So you can see that the unbalanced pressure forces produce the thrust as a result of which the vehicle moves or the airframe moves.

I can calculate the force either by integrating the pressure force on the surfaces. I take this integral evaluate it along each one of the surface. So I can evaluate the integral along each one of the surface, calculate the net force, if it is possible or I can calculate this as i2-i1. How difficult is this to do for aircraft engine? We are talking about aircraft engines in this course. How difficult is this to do for an aircraft engine?

(Refer Slide Time: 45:48)



Can you imagine calculating the pressure force exerted on the walls for an air craft engine that looks like this. It is very difficult to do, but if you have to do it, it is a very instructive thing to do because you now realize. Here is a very interesting slide from the jet engine book by Rolls Royce. You see here a very interesting fact that emerges from this. Notice that positive thrust is produced by components like compressor, the diffuser and the turbine.

So that produces positive thrust. Whereas components like the turbine and the propulsion nozzle actually produce the negative thrust or drag, which is a very unsettling thing to think about because the nozzle is called a propulsion nozzle. It is supposed to produce thrust not drag. So you have to bear that in mind that this is a very subtle concept and you can see that the exhaust cone here also produces positive thrust.

So you get a net positive force in this direction as you can see from here from all these arrows from these components and you get drag from these components on this side, but if you calculate i2 here – i1 here, you will get the thrust to be correct in the negative x direction and a positive number, which is = as you can see from here 11,158 pounds. You can calculate it using the pressure method or you can also calculate it using the impulse function method.

Both will give the same answer, but the pressure method gives us more insights into which components of the engine are producing thrust and which components are producing drag. Just because these components are producing drag, for example just because these are producing drag, we cannot do away with them. The turbine has to be present to drag the compressor and the nozzle has to be present to increase the momentum of the fluid, to convert the enthalpy of the fluid into velocity.

That is a different perspective, this is a different perspective. So you must understand both. In the actual course, we will use this formula extensively and exclusively, because it is impossible to calculate this quantity in a practical situation, but you must understand that both will give you the same answer. So this is how impulse function is related to thrust. So thrust = i2-i1, we will expand this formula in the next class and continue.