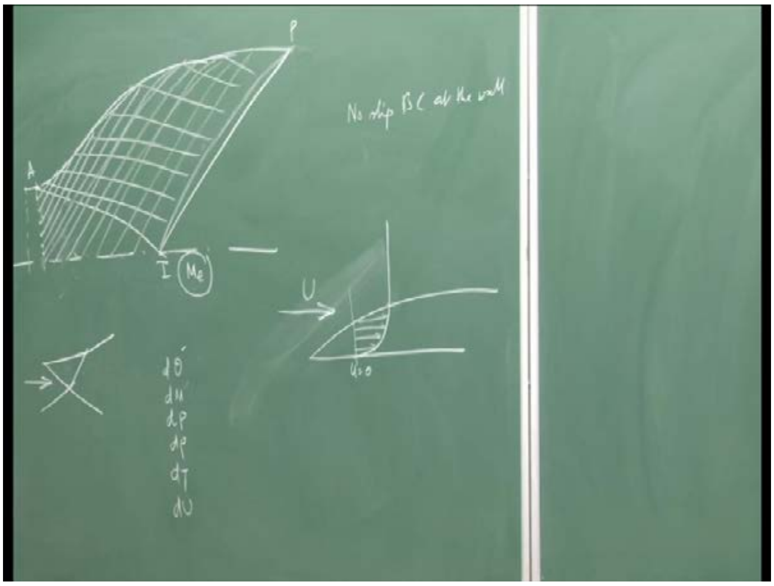
Jet and Rocket Propulsion

Prof. Dr. A. Kushari
Department of Aerospace Engineering
Indian Institute of Technology, Kanpur

Lecture - 30

Good morning. So, since till the last class, we have been discussing the shape nozzle.

(Refer Slide Time: 00:29)



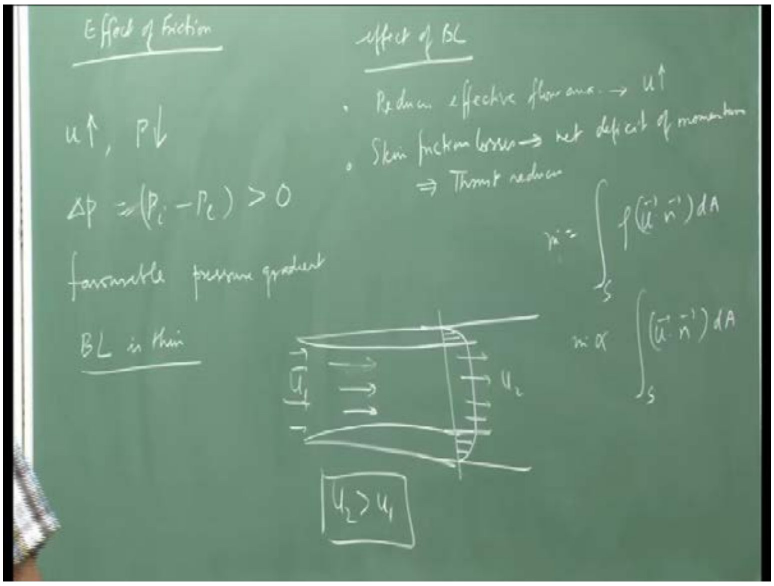
In the last class, what we have done is we have talked about how do we design a shape nozzle using method of characteristics. I have said that first of all the initial condition we choose slightly ahead of the ((Refer Time: 00:43), then initial curvature is fixed. Other we choose that initial curvature, so that we get a required mach number. After that then we make a computational grid consisting of multiple points, and then similarly we have the mach lines emitting from the mark. So, these are all mach lines emitting from different locations. Then first we solve in this domain till we get to the point I, where the mark number is our design exit mach number. Once we get design exit mach number, we drop a mach cone from here best on this mach number and we get the length of the nozzle now p.

Then we have to solve for this domain. So, essentially this is the flow across when the two mark lines cross. First of all, initially we can get this domain by looking at this mach number and how much turning it is given. Then we have crossing of mach numbers mach lines. Just on that we can get the conditions here, and how much turning is required to get parallel uniform flow, thus how we solve for this entire contour. So, by doing this, we can solve for a two-dimensional flow field of course for the entire nozzle. Now, the solution gives us theta d theta d m d P d rho d t d u, etcetera - variation in all the properties. First of all, since d theta is coming from the solution, the wall which is one of the stream lines can be obtained from here. So, the wall contour can be obtained from here.

Then, the variation in all the properties can be obtained. So, this is how we get the solution by making the grid. However, one of another point is that the accuracy of this prediction depends on the finesse of our grid. If we go to a very fine grid, the accuracy will be better. If we go to a course grid, the accuracy will be compromised, but this is a very good initial gaze of our shape. Why I call it initial gaze? Because of the fact that along this, this is a stream line, right. So, we have the velocity is not the wall. It is streamlines. So, we have a velocity, but we know that in reality, all the fluids are discussed. Little bit of viscosity is there. So, at the wall, they have to come to mostly boundary condition. We cannot have this boundary condition in this case. So, therefore, there should be no slip boundary condition at the wall which essentially means that if I look at the wall, let me take any wall not like this.

If the flow is going like this at the wall, the velocity should be 0 and then we have a boundary layer. Across this boundary layer, there is a gradual increase in the flow velocity till it reaches the free stream velocity here. So, if I draw the velocity profile, it should look like this. So, at the edge of the boundary layer only, it reaches the free stream velocity. However, for this method we have considered everything as a free stream flow. We have not considered this boundary layer and that can be to certain extent brining lot of errors, essentially by not considering the boundary layer. We have completely eliminated the screen friction losses, but I have said in the previous lectures that the losses are because of screen friction as well as because of the 3d effect. So, this method will somehow because when went from one-dimensional flow to the multi-dimensional flow, there was some losses that will be eliminated by using this method, but it does not define considered the screen friction losses. So, therefore, screen friction something that needs to be addressed. So, let us now first start from addressing the screen friction.

(Refer Slide Time: 05:45)



Let us look at the effect of friction. First of all, I would like to point out here that from this point to this point the flow is accelerating, right. So, as the flow is accelerating, the velocity is increasing. Then the pressure is decreasing, right. So, this is an accelerated flow. The pressure is decreasing according to Aria's equation which we have derived before as the pressure is decreasing del P within the inlet and the exit is greater than 0. We have higher pressure at the inlet, lower pressure at the exit, right. So, del P which I am defining as d I minus P e is greater than 0. So, what we have is a favorable pressure gradient. This is called a favorable pressure gradient because the pressure gradient is favorable. The chances of flow separation are less first of all.

Secondly, because of the presence of favorable pressure gradient and the velocities are relatively high, not relatively very high at the exit. We can have mach number 6, 7 etcetera, pretty high velocities. So, if the velocities are high and we have favorable pressure gradient, the boundary layer is thin. Boundary layer is quite thin. What do I mean by thin boundary layer is that the extent of this boundary layer is much less compared to the extent of free stream. So, the extent of the boundary layer where this change is occurring when the flow is coming to 0 velocities that is much less compared to the extent of the free stream. That is why we call it as a thin boundary layer. Now, because of this thinness of the boundary layer for all practical purposes, we can assume that the boundary layer is non-existence. So, then essentially instead of going like this, what we can do is we start from this, right.

Now, at the edge of the boundary layer, I have just said that the velocity is equal to the free stream velocity and that is what we are assuming here. So, essentially what we are saying is that this wall is our edge of the boundary layer which is quite thin compared to the free stream. Therefore, the thickness of the boundary layer can be neglected and I can work with this. So, therefore, in other words what we are saying is that the screen friction looses is quite small which is a assumption, but in reality is this assumption going to give us good results. We are assuming boundary layer is thin, but even this thin boundary layer how does it affect. So, let us look at the effect of boundary layer. First of all, what does this boundary layer do? We are saying that across this boundary layer as we can see that the velocity is changing beyond this. We have a uniform flow, but across the boundary layer, the velocity is changing.

So, now this boundary layer effectively what it does is that it reduces effective flow area, right. That is the area in which the flow is uniform is now reduced, right. So, let me take an example of a pipe flow and let us have the pipe is quite large now. So, I have a boundary layer here. A boundary layer here it is not a fully developed flow. So, the boundary layer stores much. Now, if I look at the flow here, say at one location the flow is uniform here, it will be uniform here also, but the question is, are they going to have the same velocity here and here. Let us say I will call it u 1, I call it u 2. Are we going to have the same velocity? The answer is no because of the fact actually this u 1 is here at the entry. Because of the fact that here the velocity has gone down, right. If I draw the velocity profile here close to the wall, the velocity has gone down. In between it is uniform now.

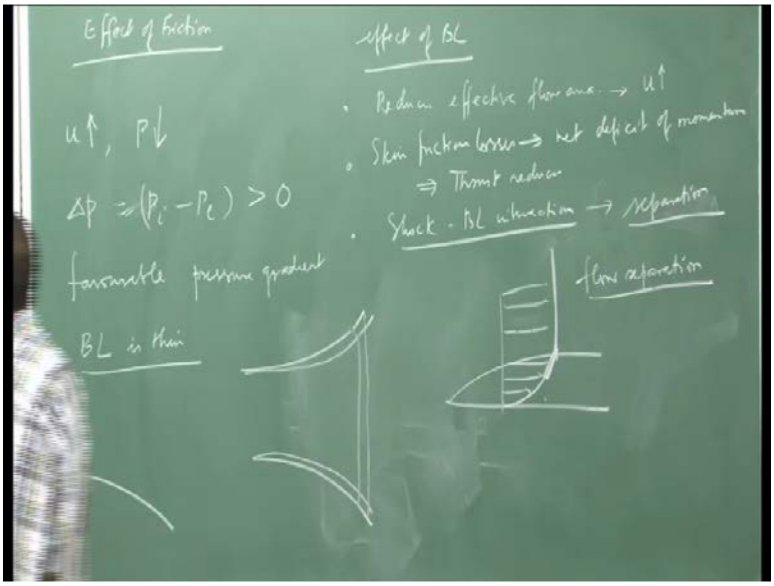
Our mass flow rate m dot is integral rho u dot n d a, right. Over the surface say s over this area, the mass flow rate is given like this. If we consider the density is constant, in that case the integral essentially mass flow rate is proportional to integral u dot n d a, right. Now, let us look at this and this. At this point, the velocity is uniform. It is covering the entire area. So, the effective flow area is more whereas, at this point only through this we have uniform flow. Here the velocity has reduced. Now, if from the continuity equation, there is no accumulation of mass here. So, the mass flow ratio remains constant. So, the mass flow rate here and here should be same in order to achieve that since there is a slowing of velocity here, this has to move faster. Therefore, our u 2 is greater than u 1. So, the boundary layer kind of squeezes the flow and accelerates it. So, same thing is going to happen here, since it is going to reduce the effective flow area. The flow will be accelerated. So, this is called displacement thickness.

So, it seems like effectively the wall is displaced, right. So, this is called the effective, sorry the displacement thickness, where actually flow area or effective flow area is reduced. So, the consequence is that velocity is going to increase. Either we increase the velocity or mass flow rate, but mass flow rate is coming from some source which will not be affected. So, therefore, velocity is going to increase whereas, now yeah because the mass flow rate is essentially dictated by your throat condition, right. So, we have the throat before it. The throat condition and the combustion chamber conditions completely fix the mass flow rate. That will not change. So, the mass flow rate is not changed. Only thing is that the velocity is going to change. Velocity is going to increase as the velocity increases, the mach number increases, right.

Then, every property is going to change because we have defined those things as a function of the mach number, but now your wall is fixed. Now, your wall is not changing. You have already designed it, right. So, now, it is no longer isentropic. The assumption that we have made that was best isentropic assumption. There was a fixed contour; there was a fixed flow property. Now, the flow velocity. So, you have designed for a given mach number or given velocity which has now increased because of the presence of boundary layer. So, it is no longer going to exactly follow your calculations. Therefore, your theta requirement is different, but theta you cannot change. So, the effect, net effect will be, all the flow properties are going to change.

So, this is something that one of the effect of boundary layer. Secondly, if I look at, again this one if I look at these two conditions. The flow is flowing down here, right. Because of that there is a net deficit of the momentum. Momentum goes down, right and we have seen that the rate of change of momentum is the parameter that will dictate how much thrust we will produce. So, now, the momentum is decreasing because of the slowing down of the flow here. So, because of that there is a net deficit of momentum which leads to skin friction losses. Skin friction loss essentially is net deficit of momentum which essentially means that thrust reduces. The thrust is going to decrease. Third, one of the most difficult conditions, another complex conditions in rocket, particularly rocket nozzles is now let us understand what is happening. We have a nozzle designed like this.

(Refer Slide Time: 16:03)



We have a boundary layer coming here like this. Because of this boundary layer, the flow is no longer isentropic. Let us assume that we had designed an ideal nozzle, right but now it is no longer ideal. The conditions are changed. So, it may so happen that we get a over expanded nozzle. If we get an over expanded nozzle, what will happen there will be a shock wave. If the extent of over expansion is such that there is a shock wave that sit at the exit. In the previous classes we have got those conditions. So, let us say now we have a shock wave sitting at the exit or shock wave can go in, right. So, there is a shock wave that is shock wave goes in this. Shock wave interacts with this boundary layer. So, we have a shock boundary layer interaction. This is one scenario. Let us look at the other scenario. Once again let us look at a flat plate. We have a boundary layer. The velocity is going like this, right.

It comes to 0 velocity. Here, this flow is supersonic, this flow is subsonic. How will a supersonic flow turn to a subsonic flow? There is no change in area here, right. There is no work done, nothing. So, the only possibility going through shock wave. So, there is going to be shock wave here, right. Only then the continuity of the flow will be maintained, otherwise how rather shock wave is discontinuous process, but in the molecular level, it is continuous. So, there is going to be a shock wave here close to the wall, right. So, even if there is no shock wave entering from the outside or because of the condition of the over expansion, there will be a shock wave close to the boundary layer because the flow has to become subsonic before it reaches this condition, and it do not have a throat, right. So, there is going to be a shock wave here and now, this shock wave is going to interact with the boundary layer.

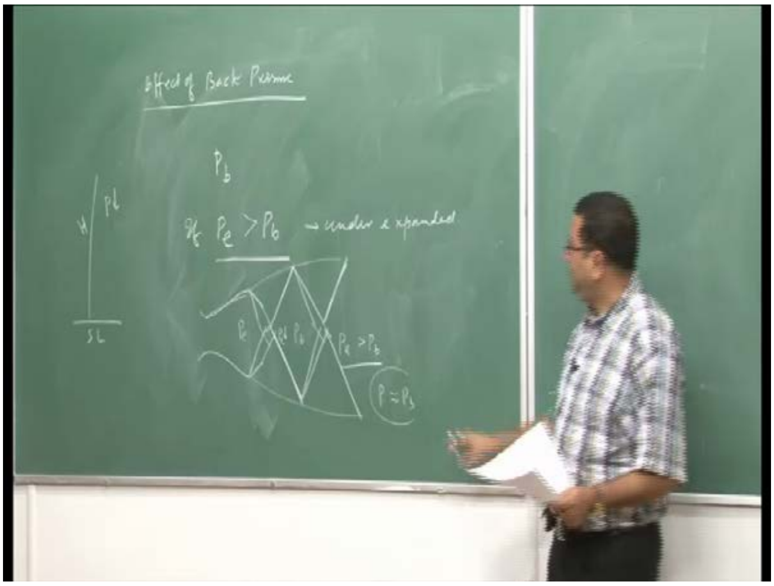
The shock boundary layer interaction is something that is a very complex subject because shock waves inherently are unstable, and they are source of lot of energy. So, essentially it is like local vaporization. It will just push the boundary layer out. It is possible that the shock wave will just push the boundary layer out and we get to that condition, where this boundary layer separates, because of the interaction with shock wave. So, there are two things. First of all the certain ambient conditions, there can be a shock wave entering. That is the different phenomenon, where it is going to interact with the boundary layer and creates some nasty problems. In other cases also because of this flow has to become subsonic. In this load, there can be shock wave and because of that the shock boundary layer interaction can occur which may lead to separation.

Now, what is another point? That is important to notice here is that when we are in the boundary layer, the flow is subsonic, right. In the boundary layer, the flow is subsonic and we have a diverging passage in the subsonic flow, a diverging passage. What happens is pressure will increase, right. So, now, we have another problem that we do not have same pressure in the boundary layer and in the free stream, we have a pressure mismatch and then we have a adverse pressure gradient in the boundary layer which may lead to flow separation, right. So, all this can lead to flow separation. So, even though we have a favorable pressure gradient in the free stream, in the boundary layer, we may not have favored all pressure gradient that may lead to flow separation. The shock waves can interact with the boundary layer which will enhance this flow separation because shock wave will make the boundary layer for the subsonic, right.

So, all this lead to a possibility of flow separation. So, even though the free stream is accelerating, there is a very nice clean flow. Boundary layer can bring in lot of problems. So, this is something that needs to be addressed when we are designing a rocket nozzle. So, therefore, what we have to do is once we have this initial contour, now we can do a full envious stock solution and see how the flow behaves with this initial contour and then we can refine our design to take care of these effects, so that another numerical iteration will be required to take care of all these effects which is an essential part otherwise you can get into huge trouble. Now, this is the effect of friction. Let me look at another point. Now, if I look at the exit of the nozzle, I have ambient pressure here P a. This is my back pressure.

Now, I have designed a rocket nozzle for a given condition, but we are not going to operate it at the condition always. Then how does it affect the flow field? How is it going to affect our performance that we want to discuss little bit? So, next topic we are going to discuss is the effect of back pressure.

(Refer Slide Time: 22:15)



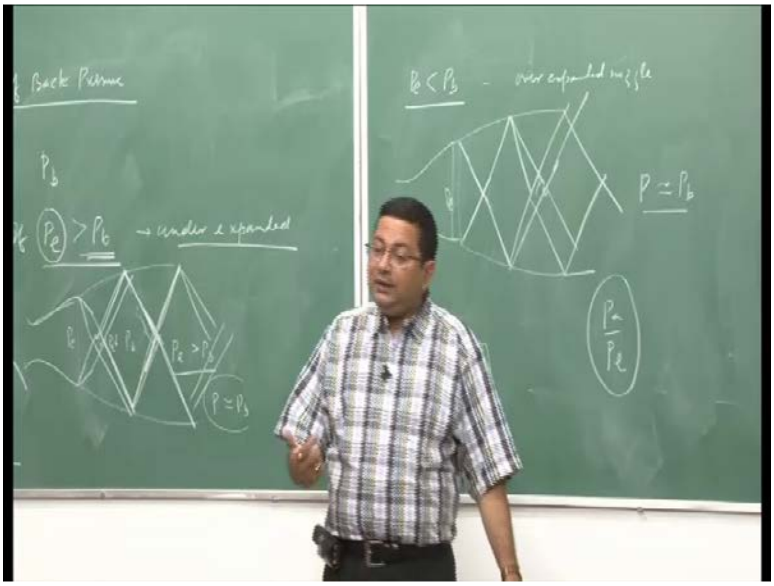
Next is effect of back pressure. First of all, I would like to point out here that this back pressure is not going to affect our flow rate because our nozzle is choked is a converging diverging nozzle. Nozzle is chock. So, flow rate is not going to be effected, but it is going to affect the flow inside the diverging portion. The rockets on the ground at sea level experiences a different back pressure, and as it goes up, the pressure drops. As height increases, the pressure drops and there is a huge variation in pressure, right.

So, if it goes to say about 14-15 kilometers, the pressure will drop to say few hundred from one bar. It will drop to say 5000 pascal or 10000 pascal or so. I do not know exact numbers, but there is a huge variation in pressure. Let me go to 100 kilometer altitude, typically a lower altitude. Then the pressure is even much lower and as we go up almost which is vacuumed which is outer space is almost vacuumed. So, there is a massive variation in pressure. Now, we cannot design for this variation performance of a rocket nozzle. Rocket nozzle is designed for a particular back pressure. Now, the thing is that if we have designed it for a particular back pressure P B and we have to operate at other back pressure, how the performance is going to be affected. Its significance effect for example if our exit condition, this is fixed by our design if this is greater than P B, right.

This is an under expanded nozzle that we have discussed. Now, if we have under expanded nozzle, what happens is let us say this is our nozzle exit. We have an under expanded nozzle, this is P e, this is P B and P e is greater than P B. So, now, we have to have some further expansion outside, so that finally the mechanical equilibrium is satisfied. So, let us say that the jet plume let me draw it here. This is my jet plume. This is how it is coming out. Now, we have to further expand for that first thing that will happen is that there is going to be some expansion fan at the edge. Now, this expansion fan will propagate like this. Because of this expansion fan, there is going to be a drop in pressure, right. So, the pressure here is going to drop less than P e, ok but this expansion fan will cross each other and then they reflect back as a compression wave.

There is going to be a compression wave. This compression waves then again reflect back from the edge as an expansion fan, again a compression wave like this. It continues as long as there is a pressure difference. When these two pressures equal to the P B P equal to P B, then this continuation cannot stop. So, essentially what we see is that when we have an under expanded nozzle, first we get an expansion fan followed by a compression wave. Then expansion fan like that there will be series of expansion fan and compression waves. That is how it will finally come to the balance condition for.

(Refer Slide Time: 26:34)



So, therefore, this is one thing that happens for under expanded nozzle. On the other hand, if P e is less than P b, this is called over expanded nozzle. So, once again this is nozzle exit. This is P b P e, this is P e, this is P b and P e is less than P b. So, it was expanded more than that is required, but finally, it has to max this pressure. So, what will happen now is the pressure has to increase. Pressure will increase first to an shock wave. So, again this is my boundary. First, there will be a big shock wave emitting from here when they cross. They will go into here. Yeah, it goes like this. You will get a lambda shock. Actually here, sorry there is a mistake. It continues as an expansion fan. Two expansion fans crossing each other from here. It reflects as a shock wave from the edge shock wave. So, it is the shock wave and then again expansion fan like that, these are called lambda waves is in the shape of like lambda. So, here now we have shock wave. This shock wave now reflects as an expansion fan. Then expansion fan reflects as a shock wave like that. It continues once again the pressure is going to gradually rise till we get a matching that the pressure is equal to P b.

Now, if you look at the plume of rockets operating particular under expansion, this is the shock waves. The flow is going down. You get the plume looks like this, bright regimes, and light regimes. Bright regimes like this. There are some bright regimes, light regimes. Bright regimes were after the shock wave because the flow velocity is low. We can see lot of light there and then light like that. So, it will be like rings, the plume is like rings. This is because of this formation till it finally, and it continues for a while will finally reach the back pressure. So, the bottom line is that the exit, we do not have the back pressure. That will not experiencing the back pressure, some other pressure, right. So, therefore, this angle of this shock wave that is formed here is dictated by the exit velocity m e and also, the pressure required pressure ratio for significantly higher value of P a by P c. P e when this value is very high, then the angle may be as high as 90 degree. That is normal shock wave, right.

So, when the exit pressure is very low compared to the ambient pressure, exit pressure is low compared to the ambient pressure, and we can have a normal shock wave setting here and that condition we had already derived. What will be condition for that, where we can have a normal shock wave setting there and another point is that across the normal shock, across the shock waves, there is going to be lot of shock in these losses. So, when we will have shock waves, the losses increase and we can see the shock wave is present here also, right. Here also we have shock waves, here also we have shock waves. So, finally, the thrust is going to decrease. We have discussed that in detail and that is why when we have ideal expansion, then the thrust is maximum on both side. There is going to be losses.

Now, as we have said that if this pressure is very low, we can have a shock wave setting here. If the pressure is even lower than this, then it will go in, right or in other words, if the ambient pressure is higher, shock will go in. If the ambient pressure is lower, it will have an expansion fan here, right. So, now, once the shock goes in, we will have shock boundary layer interaction that we have discussed. Then there is going to be performance drop because of the losses we induce by the shock waves. All those things can greatly affect the performance of the nozzle. Now, let us come to the practical approach. We have to design the rocket for a particular mission. It has run apart 20 minute, 20 seconds, 30 second whatever it should, we design it for this point or this point or this point.

Where should we put our ideal because we design for the ideal case? If I do it ideal here at the sea level as it goes up the pressure exit, pressure remains like this which is our sea level pressure. Now, the ambient pressure is decreasing. So, we have an under expanded nozzle, continuously under expanded. So, the pressure is dropping, we have continuously under expanded nozzle. So, it will never reach our optimum condition. If we design for this where exit pressure is equal to the pressure here, then from here to here we have a over expanded nozzle. We get into these problems. So, once again that is something we do not want. Again we never get optimum and flow we can have separations and lot of back things happening. So, if we design it for somewhere here, then what happens initially? So, our design pressure is P e. So, this is our P e, initially this pressure is higher.

So, we have a over expanded nozzle initially, but as we go up this pressure is dropping P b is dropping. So, we are initially over expanded. Let us over expand it as we go up the thrust is increasing because P b is dropping, we are coming close to ideal at one point. We did ideal beyond this. Now, the pressure is further decreasing and we have this condition. So, we have like this, right. So, we have an increase and then decrease in thrust. Now, depending on where we want to, depending on the mission requirement, we design it like this. So, this is the best possible scenario for everything, else we have a continuously dropping thrust as we go up. Therefore, for all practical applications, the nozzles are not designed for either c level or at the maximum height. It is designed for somewhere in between to have the ideal condition, so that it grows across the optimum condition while operating, ok.

That is one way is passive in the design state itself. We can make sure that we get the optimum condition, but are there any other ways? Yes. What we can do is, we can have a variable ideal nozzle, right. So, we can change the exit area and we can play with the variation in pressure. So, the area keeps on changing, we can maintain the pressure. So, another thing is that we design in such a way that we get advantage of not having to deal with all these things, shock waves and so on. So, that is the other designs which are preferred, particularly for higher stages of rockets which are plug nozzles or expansion deflection nozzles. So, let me discus little bit, not much about those nozzles.

(Refer Slide Time: 34:24)

So, first is plug nozzles. What did we say? Then our design philosophy is that we design the rocket at a higher altitude. So, during the launch, then it is over expanded, right. So, we have initial over expansion and we know that this over expansion is going to bring in some losses. We want to minimize this loss for that we can design it; we can make some changes to offset thrust laws associated with over expansion. What we do is instead of having a confined jet; we actually play with the jet boundary. This is a plug flow comes here goes uniformly outside from here, and this is called a plug. At the exit of the nozzle if I look from the front, we have a body like this. This is called a plug. Of course, this body is contour and this contour can also be obtained using method of characteristic. So, the flow is coming here it is a uniform flow coming here now what we have is a jet boundary going like this. So, the we have jet boundary on both the sides like this. So, what is happening is that the flow coming from the combustor or in the nozzle is going through an annular passage now.

(Refer Slide Time: 36:46)

It is no longer the straight open passage. It is going through the annular passage and what we have is, we have two corners here, here, and here and these two corners are not at the same location. Now, when I drew this, there was a jet boundary, right here at the edge. It is equal to the atmospheric pressure, it was equal to P a. So, at the corner it will take the atmospheric pressure. So, here also, then actually like this, it takes the atmospheric pressure. Because of the presence of this plug and this plug can be designed in such a way that there is a gradual expansion. So, therefore, this shock formation and yeah, initial shock formation, all can be eliminated. This is typically what is used for supersonic in takes for aircrafts. If you look at a fighter aircraft or as a supersonic missile, we have a nose coming out is like a plug.

Now, that plug essentially has two purposes. First of all, whatever shock wave is there, it is terminated from here and that shock wave can be controlled by proper designing. So, we can have a weaker shock that will reduce the losses. Secondly, the distance here will determine how much will be the reflected shock wave, and that can also be then controlled with the design. Thirdly, this plug can be moved up and down also. So, we can change the area, we can change the shock location depending on our requirement. So, therefore, the losses induced by the shock wave can be reduced to a certain extent by providing this plug, and that offsets, the thrust loss because of the over expansion during the initial stage of this rocket operation. So, that is one of the ways.

Another possibility is instead of putting this contour plug; we can have something like a plug body. So, that is called expansion deflection nozzle. This is although essentially in the plug. Your plug was protruding outside the nozzle in expansion deflection. It does not protrude outside, it is all inside. So, you have inside body like this. So, flow once again comes annually to this and then it flows like this and we have the edge here inside. So, here we have atmospheric condition, here we have the jet, right. So, now, depending on this location, we create the shock wave whatever it is required and this is the ambient. So, by that time it comes to the end here, all the bad phenomenon occurs here and finally, we want to come to the exit. It has almost matched the ambient pressure. So, all the shock formation, all the expansion waves, everything will be occurring here because this is my jet boundary and then therefore all the effects are confined inside, but not directly effecting the flow field. This is safe what is happening in this passage.

So, this is something that expansion, this is called expansion deflection. So, the flow here also expands along a corner and because of this expansion, it can expand to the actual atmospheric pressure particularly this is the actual atmospheric pressure, right. So, it expands to the actual atmospheric pressure. Therefore, the over expansion is eliminated. So, if you eliminate the over expansion, the shock induce losses. Once again you have to reiterate will be reduced and because of that the performance will be better. So, for the over expanded case, we were having a continuous drop like this. Now, that can be eliminated. We can have a uniform or an increase thrust like this. So, therefore, this is a better way of putting a nozzle or designing a nozzle rather than having a state contour because a state contour essentially we have to be very careful, and as we have said particularly, this over expansion case is more dangerous because of the fact that I can have shock waves going in, and that will have huge losses because there will be shock induce losses, there will be shock boundary layer interaction leading to flow separation.

So, there can be very, I will say devastating phenomenon occurring in the flow which is something we do not want to have. That is why all these, both of this plug nozzle as well as the expansion deflection nozzles are primarily for the over expansion case. Under expansion is still much safer. So, this is what we wanted to discuss about the nozzle flows. With this we come to an end of our discussion on nozzle flows.

So, if I can just conclude what we have said is that we started with defining the performance properties of a rocket, we defined the thrust coefficient, we defined the characteristic velocity, and we have seen that the thrust coefficient essentially is a function of nozzle design. So, we discuss the nozzle in detail starting from the area valuation, area role, then how does the performance effect, then over expansion, ideal expansion, under expansion. Then we want the shape nozzle, we discussed conical nozzle, we discussed shape nozzle, we discussed how do we design a shape nozzle using method of characteristics, then little bit of we discussed what is a plug nozzle, what is an expansion deflection nozzle.

We have also discussed how the friction or bond relay affects the performance, and we discussed how does back pressure affects the performance. So, essentially all these parts was focused on the thrust coefficient maximization, right. So, the entire nozzle descriptions or discussions was focused on that another parameter which we had discussed during the performance of chemical rocket is characteristic velocity and we have shown that the I s P which is the final important parameter for us is the product of these two, or thrust is the product of these two, thrust coefficient and characteristic velocity.

So, now, I think we should have a fairly good handle on the thrust coefficient. So, next what we are going to talk about is the characteristic velocity. We have shown that the characteristic velocity depends on chamber pressure and chamber temperature. So, now, next thing what we are going to talk about chamber pressure also is something as I have said, it comes from the pump and all we want to maximize it, but there is a limit. We do not want to go beyond that. So, next thing what we talk about is the temperature. Another parameter that appears again and again in all our derivations, where gamma coefficients, other ratios or specific heats that depends on the composition. So, the temperature as well as the value of gamma and r, both of them essentially is the output from the chemical reaction or combustion that is occurring inside our combustor. We have talked about designing of combustors also by the way, but at that time we have assumed that the value of gamma is known t t c naught is known.

Next thing what we are going to do now is to see how we get those values, how do we get the composition, how do we get the final temperature. So, the next topic we are going to start from the next class essentially is the combustion part. We will not go into combustion dynamics or kinetics; we start with combustion chemical equilibrium. So, we will see that if we take two type of fuel and oxidizer, when they react what is going to be the final product. How we get that final product? We can because we consider equilibrium chemistry. Secondly, once we have the final product, how do we get the temperature? These are the two things that we are going to focus on. Once we had done with that, then we talk little bit about the propellants, but again in the propellants, I will not go into much of detail. So, after the propellants are done, then that completes our discussion on chemical rockets. I would like to tell you that there is a very good book by Professor Rama Murthy on rock propulsion that describes particularly the chemical rockets, solid propellants, liquid propellants etcetera in a very nice way.

If you read that, at the same time you can also follow his lectures in NPTEL on rocket propulsions, where he even talks about various components of rockets, how they are designed, how they are chosen. So, in this course, we are primarily focusing on the theoretical aspects of rocket and not going into the component details. So, we will focus only on the nozzle and the basic combustion characteristics, I will stop here. Now, in the next class, we will start discussing combustion.

Thank you.