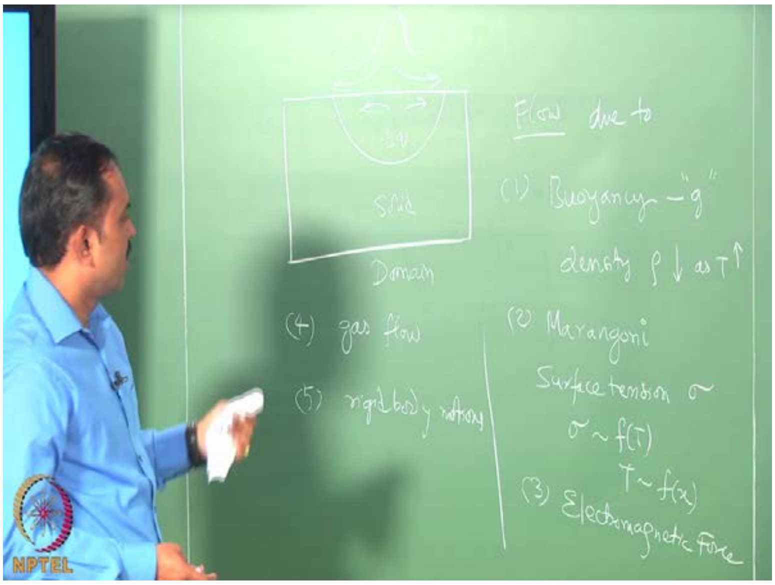
**Analysis and Modeling of Welding**
**Prof. Gandham Phanikumar**
**Department of Metallurgy and Material Science**
**Indian Institute of Technology, Madras**

**Lecture – 11**
**Fluid Flow Modeling – part 1**

Welcome to the part-1 of Fluid Flow Modeling. In this lesson, we will be looking at how to model the fluid flow in the melt pole of a weld meant.

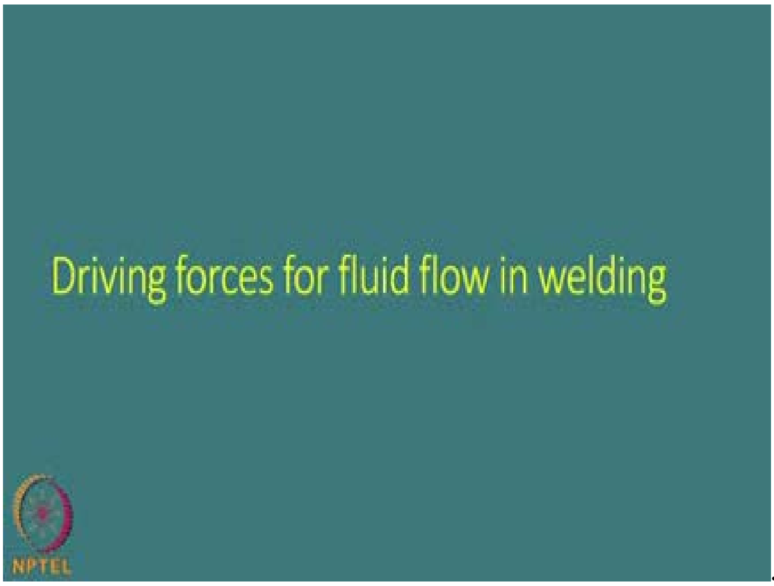
Very often this fluid flow phenomenon is ignored in commercial software mainly because it is very complicated, however as we would see it does effect to the melt pool shape and if you want to do a physically based model to predict the weld pool shape, then a fluid flow is an essential element.

(Refer Slide Time: 00:48)




So, we would start by looking at what we want to call as a domain and this is a domain which is the sample work piece and as we know that upon the heat source that is applied on top of a domain, then you would melt some region and it is in this region that we are looking at the fluid flow phenomenal, and people may draw arrows to indicate how the fluid flow is happening inside the pole and which way does this convection change the melt pole shape will be discussed as we proceed further.

(Refer Slide Time: 01:27)




So, what are the reasons why would the fluid liquid melt pool have any convection? Does not it stay a crescent that is without any convection? So, reasons why flow will happen are the following.

The first reason is due to buoyancy. So, buoyancy as in it is caused by the gravity which means that when we have any fluid which is hot and because the density usually is decreasing as the temperature is increased for most of the fluids, then what would happen is that hotter liquid tends to go up away and colder liquid will try to settle down and this natural way of moving is driven by the gravity, which means that when there is no gravity acting, then definitely the buoyancy conversion is suppressed and this would be one driving force for the liquid to have convection because there are temperature differences within the pool and you can see that the temperature is highest here and it is melting point along this line of fusion zone.

This means that there is a temperature distribution which implies there is a density distribution and whenever there is a density distribution upon acting by the gravity, you would have buoyancy that is taking place

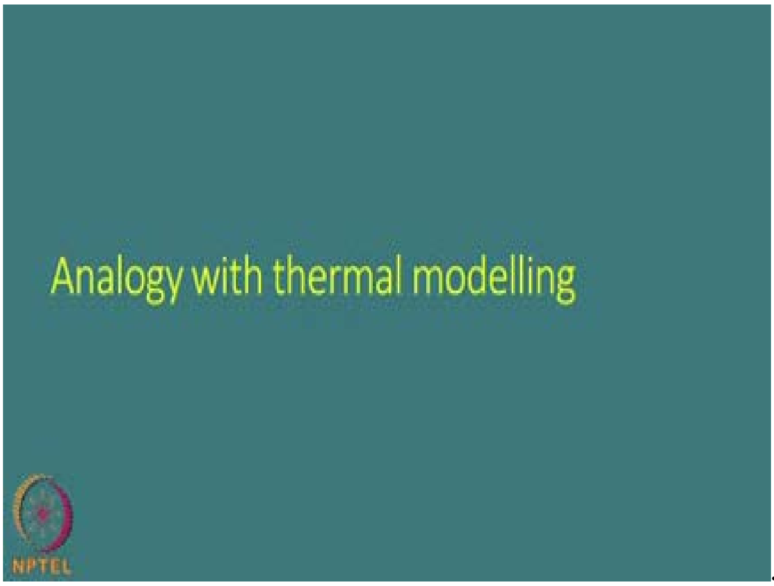
The second reason why the fluid flow will happen is because of surface tension marangoni convection as it is called. So, reason is basically whenever the surface tension is changing with respect to location, this would act. So, the surface tension is actually a function of temperature and we already know that temperature is a function of distance in our case, which means that there is a surface tension changing as a function of distance and changes in surface tension lead to convection which is called as marangoni convection. So, that is a second important reason why that would be a fluid flow within the melt pool.

The third reason would be the electromagnetic force. So, electromagnetic force is because if you have arc melting or electron beam melting, you would have basically current that is going through the sample and part of the sample is actually liquid which can move. So, whenever you have a moving conducting body through which current is going, the current is actually induce magnetic field and coupling the magnetic field and the current direction, you would have electromagnetic force that would be coming in.

There are minor other reasons that would also play a role and those minor reasons are also listed here for example, gas flow. So, on the top of the weld pool, you have the gas rod and that is going to have the shrouding gas going away in that direction. So, while it is moving, it can exert certain amount of stress on the surface of the melt pool causing the melt pool also to move in this direction. Therefore, you could also think of gas flow having some effect of this and you would have for example the convection happening also because of any rigid body. Any rigid body motions that are given to the work piece itself very often that is not done in normal welding setups, but if it is given, then that would also play a role in the liquid getting convicted with this.

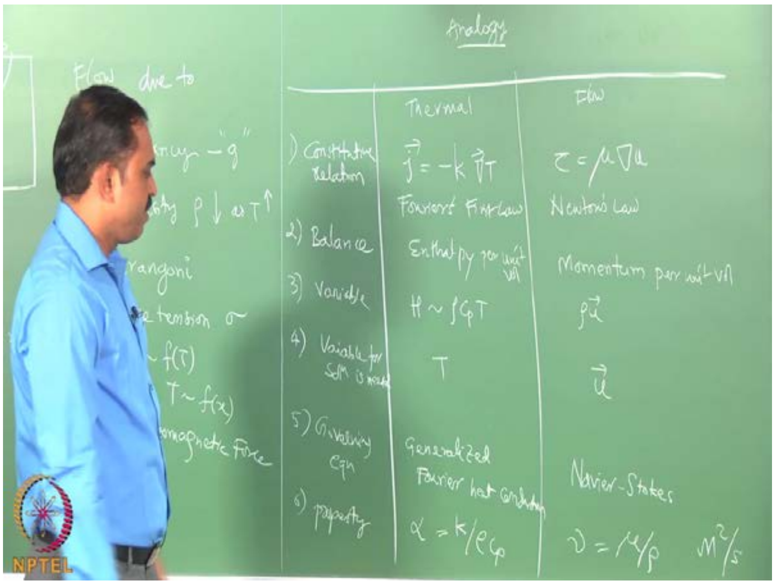
So, these are the various reasons why the liquid would have the convection and to ignore this completely would be not correct because it does not give us the right picture on the physical phenomenon that are taking place, and we would try to keep this into account only because we want to predict the shape of the melt pool from the physical principles. If that is not the reason, then one may ignore and then look at the thermal field only beyond the melt pool only in the solid region. So, these are the driving forces that are taking place.

(Refer Slide Time: 05:53)




So, we can actually understand how fluid flow can be modeled within the melt pool by an analogy to the thermal modeling that we have done earlier. So, I would do the analogy across various terms and that would give an idea on how the equation can be written unlike in thermal modeling, where we have derived the entire set of equations because of the simplicity of those equations. We will not be doing that for fluid flow. I will give those equations directly by analogy, but you are interested in the derivation. I would put up a hand out on the course website where you can look up for further details. So, the analogy with the thermal processing is as follows.

(Refer Slide Time: 06:30)




The first thing we would just divide this into two columns and look at how we can understand the phenomena. So, here we would look at the thermal aspect and here is the fluid flow aspect and first thing is that in the case of thermal modeling, what is the fundamental equation that is giving the relationship between causing the effect namely what is called as a constitutional equation.

So, the constitutional equation are constitutive relation which is the causing effect relation in the case of thermal field is that j is equal to minus k grad t that is heat flux is causing the temperature gradients or temperature gradients will lead to heat flux and this is a linear constitute relationship which we called by the name Fourier's first law of heat conduction. That is a starting point and similarly there must be a constitute equation for fluid flow and luckily most of the time we talk about metals only for welding and in the case of metals, we have the metallic liquids obeying what is called as a newtons law.

So, the constitutive equation would then be for example, it would be basically newtons law showing you that the gradient of velocity is giving you the shear stress that is acting on the liquid layer or acting upon the action of a shear stress. You would lead to gradients in the velocity. So, it is somewhat similar. The minus sign is not here. One can put it in, but then that would change away the definition of the shear stress. However, we will not do that now and we can see that there is a correlation between the two and the overall equation that we are going to look at that balance is done in the case of thermal flow by looking at enthalpy balance.

So, it was the enthalpy that was balanced for every control volume to look at what is coming in and what is going out. In the case of fluid flow, you would also do imbalance and that is basically momentum balance. What is to be done in the case of fluid flow and the variable that is used for the enthalpy balance is actually h and which we wrote as row c p t for the single domain region. So, either solid or liquid, but not both together and then, able to convert the enthalpy to the temperature variable.

So, similarly in the case of a fluid flow, you would actually have momentum and these are both per unit volume. So, here also it would be per unit volume and then, be row into u because you can see that row into volume will give mass into velocity is momentum. So, row u will become the momentum per unit volume. What is the variable for which the solution is salt, that is basically the temperature variable in this case, with the velocity and these velocities are basically vectors which means that you could look at the three components of the velocities that are of interest for us.

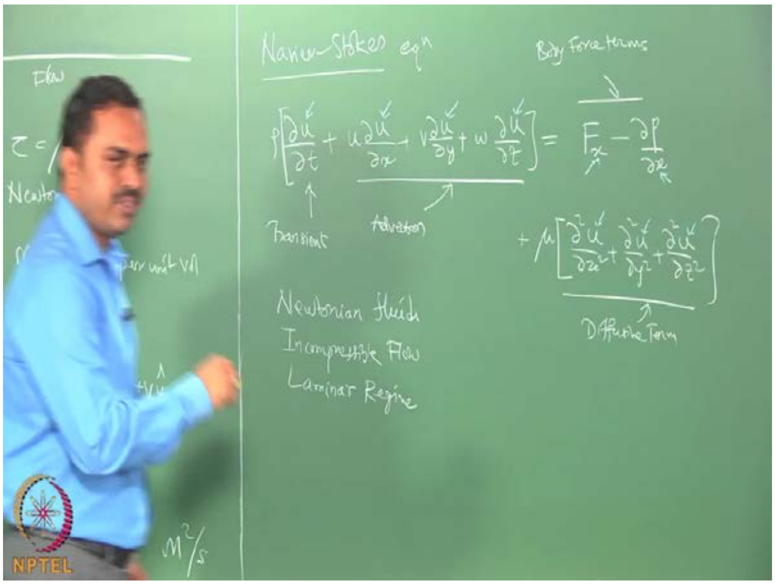
We would have of course then the boundary conditions that will be required to close the entire formulations. So, this is how we can form an analogy which means that the governing equation for fluid flow is going to look somewhat similar to the governing equation for the thermal field and we could then look at each term and see how that is different in the case of the fluid flow and try to understand from where it comes. So, what would be the governing equation in this case of thermal? It is basically generalized Fourier heat conduction equation and in the case of fluid flow, it will be Navier stokes equation and there are some limiting situations for which we wrote this equation in the case of thermal field.

For example, we have a limited generalized Fourier heat conduction equation for a situation where the properties are all location independent and you have a simplistic expression here. So, like that we are able to simplify this situation and the same thing is also applicable for flow. We will look at what limitations are kept, so that the equation is turning out to be simple.

There are some new parameters we discovered as we went about the derivation and that the last one for the compression, I would do some property we came across in thermal would be the thermal diffusivity which we wrote it as k by row cp and in the case of fluid flow also, you would come across that kind of a combination of parameters which you would have it as mu which would be called as a kinematic viscosity and that is equal to the dynamic viscosity divided by the density and both alpha and mu will have the units of meters square per second. So, you could think of the most important parameter for thermal flow.

Thermal field evolution is basically the thermal diffusivity making the thermal field evolve and you can think of the flow, you can think of it as kinematic viscosity making the momentum get diffused are getting distributor. So, like this there is an analogy that we can draw between the two fields, so that the governing equation can be then seen directly as an analogy. So, let us write the t equation itself here.

(Refer Slide Time: 13:24)



So, I will tell you the limiting conditions for which what I wrote is valid, but let me just write it here and we need to write it for three components because this velocity is nothing, but three components u, v, w, which means that you would have one equation for each of these variables u, along x v along y and w along z direction. So, we can write first the for equation u. So, you have the transient term, we have the abjection term. Abjection term is always u, v, w, and then, this letter and this direction of the component should match and you have that coming up and on right hand side, you would expect for example, the diffusive term to be present and there are also source terms we would actually put them here. Fx is a body force term x. I am putting component.

A component we are writing for the u component of the velocity and you have the pressure coming also playing a role. So, dou p by dou x and then, you have the remaining equation and that would come as this. So, this is the equation that we are supposed to solve and whatever we discussed with the thermal field analogy is applicable for these terms also. So, we could write those terms as follows we can say that this is the transient term and this is the abjection term and these are the body force or volumetric term and then, this would then be the diffusive term.

So, now that you have seen the equation which is very similar to the thermal field, you could then combine this equation with the thermal field equation and give a name for this class of equations. We could call this class of equations as convective diffusive equation.

Advection or convection term is here convective diffusive equation is what is the category of this kind of equation which means that when you are looking for possible solutions, possible methods to solve, you do not have to necessarily look up what is the way to solve Navier stroke equation if actually just ask for convection diffusion type of equation and then, the solution would be basically applicable for these equations also and let me also then tell you here under what circumstances this is supposed to be valid. This is basically starting point is here which means that it is meant for so-called Newtonian fluids.

All metallic liquids are Newtonian in their behavior which means that they obey this kind of a relationship. So, there is no problem you can go ahead and use that up app approximation. However, if you going to use a heat source for a polymeric liquid to form to join between two polymer blocks for example, then definitely it may not be applicable. You may have to choose a different equation, but that is not a common situation in welding, most of the time you are welding to metallic material. So, this must be valid and it is also for what is called incompressible flow region.

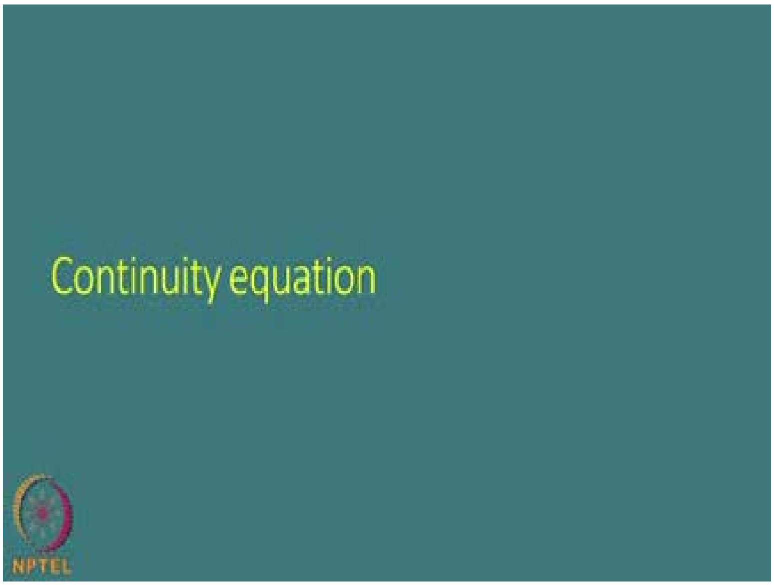
What we mean by in compressible flow region is basically to say that the continuity equation or the mass balance equation is going to be in a simplified form. We will come to that shortly. What is that? Why does that come to be a similar form? We will come to that shortly. It also meant for what is call a laminar. So, laminar region is supposed to be assumed, so that these equations are adequate to tell you that the fluid flow is governed by this equation and turbulence is not been considered because if you introduce turbulence, there you know couple of more equations that you may have to solve. So, these are the situations for which the fluid flow will govern by this kind of a governing equation.

You would write the same equation for v and w also, and when you write for v wherever u is there. So, I would just indicate where you are supposed to replace you can see that here, here, here and here. So, those regions you have to replace u with v to get the equation for velocity along the y direction and when you are doing with v, pay attention to the differentiation here change from x to y and then, when you want to do it for w, then you could again replace u with w and change this x to z here to z and then, all these three also to be w. Otherwise the template of the equation remains the same. So, this is how the governing equation would look like. So, we have seen the meaning of different terms while writing them down, but let me just summarize.

The first term is a transient term which is basically to tell you how the fluid flow will be changing with respect to time which means that if you are going to look at fluid flow in a weld pool at steady state, then this term will be ignored. The second term is advection term. It cannot be ignored. The reason is at advection is already there and that is what we are solving for. So, therefore the coupling between these terms and the velocity components must be there. So, advection term is always there. It is basically telling you how the momentum can be affected by the different components of the velocity itself.

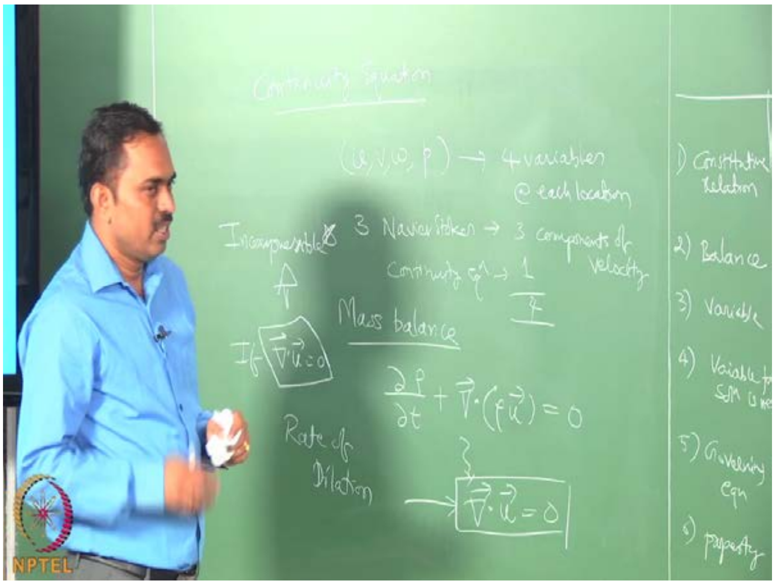
The body force term means basically how the gravity direction or electromagnetic force or the thermal buoyancy or solutal buoyancy etcetera will be affecting the flow. So, this body force is essentially to take into account volumetric effects on the flow pressure gradient term is always be there and the reason being that it is coming from the stress by separating of the hydro static components. This will always be there and this is a diffusive term and this again cannot be neglected because in metallic liquids, a viscosity is quite high and therefore this term will not be neglected. It basically tells you how the momentum is getting diffused by the action of the viscosity and how the viscosity is dissipating the momentum across the entire melt pool.

(Refer Slide Time: 20:34)




So, this is how the Navier strokes equation is going to be telling us the governing phenomena.

(Refer Slide Time: 20:38)




So, here we look at what we need beyond the Navier strokes equation to solve for the fluid flow in the weld pool. We need what is called continuity equation. The reason why we need one more equation is as follows how many variables are there which are required at every location in the melt pool. Basically you have got u, v, w, and the pressure. So, there are basically four variables which are at each location in the pool and how many equations are there for us to solve at each location.

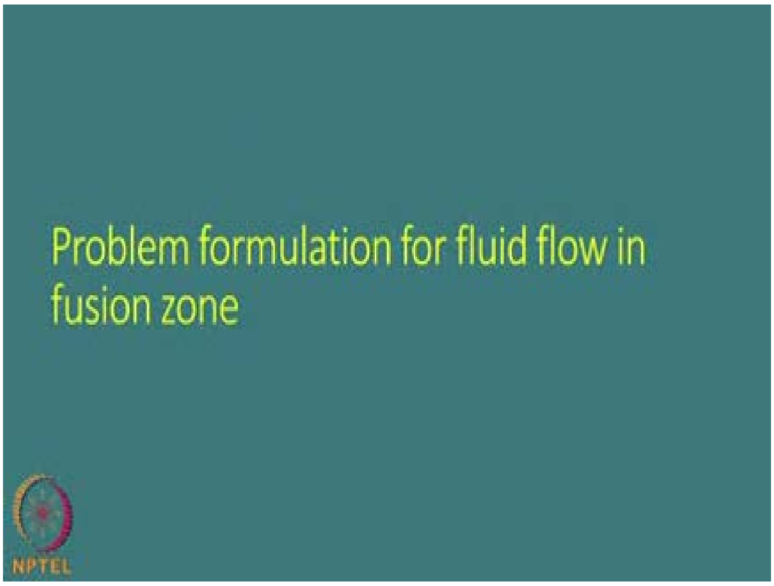
You have basically three Navier strokes equation for the three components of the velocity and therefore, we need one more and that one equation basically is the continuity equation, so that you have four variables that are unknown at any location, four equations that are available at every location and therefore the problem becomes closed. So, that is the reason why we need it.

What does it actually mean? Continuity equation, the meaning of the continuity equation is nothing, but mass balance and that can be written in this situation like this and you could for example expand this and look at that. You could split the two variables of the two differentiations and collect the terms and you would then see that you could simplify it to have what is called dilation to be zero as an assumption. So, this is called as rate of dilation.

So, if this rate of dilation which is assumed that implies that the material is taken as in compressible, so it is not necessary to take this into account. You could actually use this as a mass balance equation itself and then, use the equation in connection with Navier strokes equations. Its only that if you take this assumption, then the continuity equation itself become simpler and this quantity can also make Navier equation look very simple like the way I have written and in case you have not assumed this to be 0, then not only the full equation is valid, the Navier strokes equations will have additional terms.

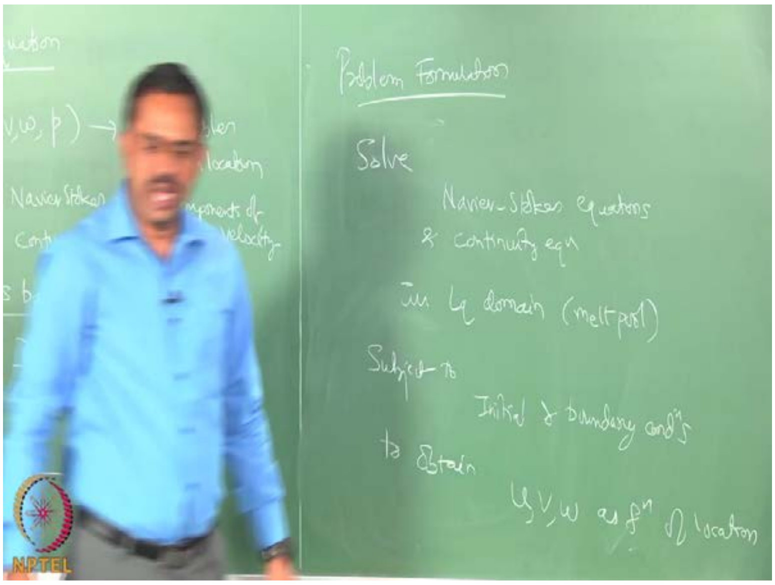
So, that is the only difference, otherwise you can say that continuity equation plus the three Navier strokes equation, four equations are required to solve for the four available at each location in the melt pool and that is when the problem for the fluid flow is well posed for the entire melt pool.

(Refer Slide Time: 24:01)




So, the problem formulation can be now written, I would just do it by writing in the middle.

(Refer Slide Time: 24:10)




So, here is how we write the problem formulation. So, solve Navier strokes equation and continuity equation in the liquid domain or in the melt pool subject to initial and boundary conditions. So, this is the problem formulation to arrive at the melt pool convection and why are we doing this solution is to obtain u, v, w, as a function of location. So, once you have u, v, w, as a function of location within the melt pool, then if you plot them, you can get how the weld pool convection is taking place and you can get it as a function of time at each time step how does the liquid convection is evolving within the melt pool can be looked at.

(Refer Slide Time: 25:40)

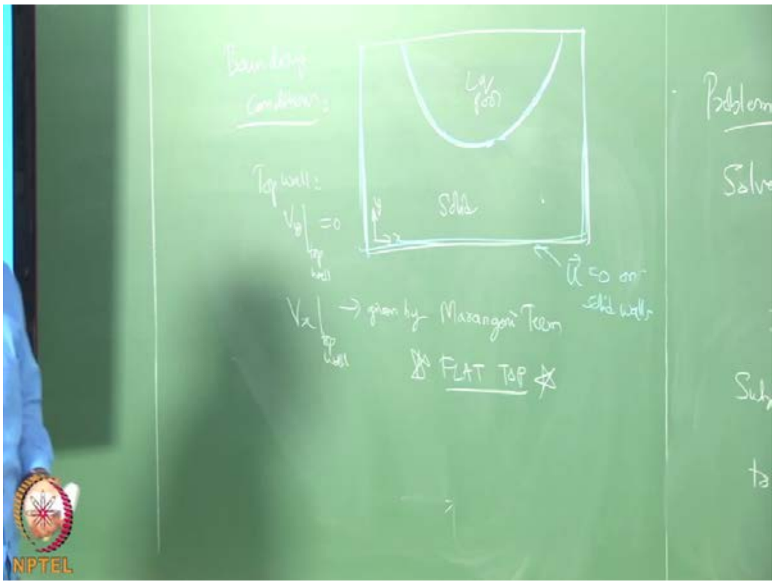

So, what would be the meaning of initial conditions because we would have to look at these conditions also very often? The initial conditions are such all the velocities are zero and pressure is equal everywhere. So, that is a very simple initial condition and the reason being that you have the initial part of the domain is basically solid and time proceeds, this is a solid that is melting slowly and therefore, at the time of initiation of a melting, you can assume that the velocities are zero to start with and the pressure also is equal everywhere.

So, that way the initial conditions are quite straight forward unlike in the thermal field, where the initial condition can be slightly different when you have a preheating condition. That is possible in the case of liquid pool in such a condition is not relevant.

(Refer Slide Time: 26:45)


What kind of boundary conditions are special conditions are necessary that we would do now and let me just erase the left side of the board.

(Refer Slide Time: 26:56)




So, these are the boundary conditions that have to be applied. So, if you take here top wall and you would see that the liquid should not escape out. So, you basically have the vertical component to be 0 and if you want to have the axis like this, then v y vertical velocity at the top wall is 0 and there is no clarity on how the horizontal component can be there and that I would tell you.

Now, here the horizontal component to be x at the top wall should be given by a phenomenon that changes the velocity in the horizontal direction on the surface of the liquid pool and you already seen that on the surface. You have got surface tension driven flow and therefore, this horizontal velocity should be given by what is called the marangoni convection. So, v x and v z as a function of location on the surface at the top wall should be given by this condition, v y will be 0 and that is a boundary condition on the top.

We are making an assumption in writing the conditions like this. The assumption we are making unknowingly is that the surface profile of the liquid pole is flat which means that we need to pay attention. So, it is very important that we are making that assumption. What kind of situations we will not to be able to use an assumption is basically when you have surface undulations in the melt pool which will lead to the surface rippling effect. If you want to reproduce the rippling effect, then this condition is not valid and we must change that we look at those effect later on, but we must remember we are making that assumption at this junction.

What are the other walls? It is the side walls and to know what to other boundary conditions should be kept, we must actually remember what the other boundary that we have. So, if you are going to solve this only liquid pool, then it may be assumed as if this is a boundary. If this was the boundary, then boundary conditions and that is very simple because from the solid which means that you basically no slip condition that can be valid or in case you have got this surface moving back and forth, then we would also like to take other phenomenon to account as such as the change in the density, leading to flow at that location and so on.

So, basically rigid wall condition can be applied, however as we have discussed in the thermal modeling lecture already, its very nice if you are able to have the entire domain in one set of equations and not separate, the two because where you separate that boundary itself is the result that we want to obtain. So, therefore it is necessary to imagine that the boundary is actually around here. If this was the boundary, then what is the boundary condition here is very simple. It is all solid. So, the velocity is, all the velocities are zero on the solid wall. So, that is quite straight forward. We can apply that boundary condition.

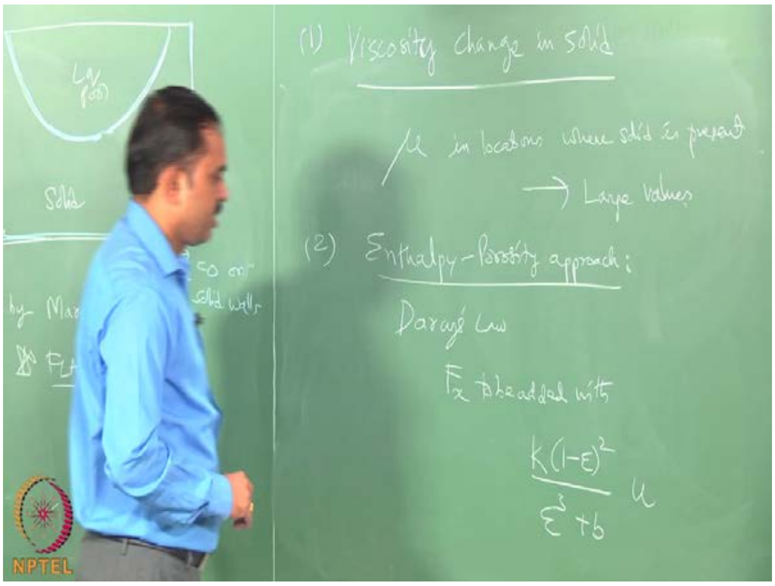
Only one tricky thing is that the equation you wrote is valid only for the liquid pool, but then you are applying a boundary condition for the solid part also. How do you separate these two regions? That would be question that we will answer shortly in this lecture itself.

So, this is the need for the single domain we mentioned. We have this interior separation between the liquid pool and the solid base material or work piece and this boundary has to be determined as a solution and therefore, that cannot be taken as a given condition, a priory and therefore we need the entire domain in a single piece and that is requirement for us and the way you can arrive it is by two methodologies.

(Refer Slide Time: 31:25)


So, we look at each of those two shortly and I will just give them to you now to the way you can ensure that the equations you wrote are valid in the entire domain, but are correctly written only for the liquid pool and they are giving you zero as the answer for velocity everywhere in the solid is by two methods.

(Refer Slide Time: 31:46)




The first method of getting that is what is called the viscosity change in the solid. So, essentially you can say that mu in locations where the solid is there, you make it take large values. So, in other words we are trying to define the solid as a very viscous liquid. That is all.

So, viscosity is so high that liquid is actually not moving and that is as good as solid. So, that is one way by which you can interpret the rest of the region which means that you will have velocity is here like about let us say 10 to the power of minus 8 meter per second which is practically 0 and in the liquid pool, you may have more meaningful velocities and that is how we can say that this is a solid and that is a liquid and artificially you can take viscosity would be very high and you can use that as location dependent parameter mu which means it can go in and become location dependent and that you can change.

The advantage of doing that is that there is no special formulation required. Equation can be used as it is, but the disadvantage is that it is going to be numerically causing difficulties. The reason is that velocities are going to take such a large range of values known small values here negligible almost 0 to values, very close to 1 meter per second in the top of the melt pool. So, such a large range of velocities are going to take place and the transition is happening in a small region. So, there will be numerical instabilities in the solution procedure. So, one must watch out through it.

A very simplistic way of handling you could actually land up in numerical problems, but this is one method that could be used, so that the Navier strokes equation that we have written can be applicable for the entire domain in the solid as well as the liquid. How do you know that you are in the liquid you can use the temperature as a condition? So, whenever the temperature is below the melting point, you can say that viscosity is then very large. You can use it as a way to update the properties as the temperature is evolving. In other words, we are looking at coupling between the flow and the temperature field to solve the flow field itself.

The other second method is a little bit more elegant method and it is supposed to give you the result with little less difficulty in the numerical implementation. That is basically the enthalpy method. So, it is called the enthalpy porosity approach. So, the motivation for this approach has come from the porous medium flow and porous medium flow. As you would recollect, you could also look it up separately. It is ground by what is called the Darcy's law and the Darcy's law basically tells you how the fluid flow is affected by the pressure drop across a porous medium and how the porosity will affect the velocity flow through the porous medium. Now, that can be applied to the welding situation in the following manner.

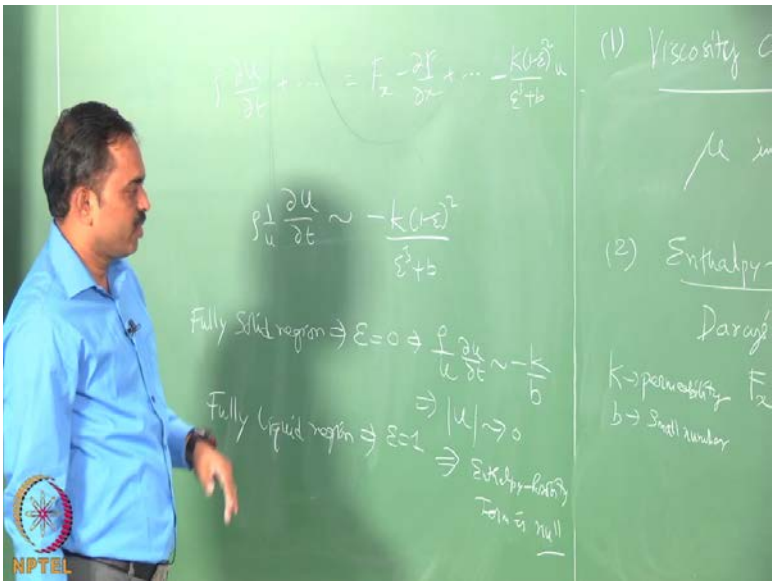
You can actually treat the region between the solid and liquid here as a Mash zone and in the Mash zone, you can say that it is almost like a porous medium approach and you could use one relationship and you could imagine the solid as if the porosity is been 0 in which case the velocity will then automatically be negligible.

So, you could actually do that which means that the body force term, this body force term we are looking at would have an additional term that will come because of the porosity approach that we are taking and that additional term would then be looking like this. If you are looking at x velocity, you would have one additional component would look like this. So, this formulation can be related to the so-called Kozeny Carman expression that is used for the porous medium approach, the 1 minus epsilon square and epsilon cube. This is quite familiar for those who are familiar with the porosity approach.

The constants are as follows. The k if the permeability and the meaning of permeability is the ability of the porous medium to allow the liquid to go through it which means that if you take this permeability to be very high which means that the velocity will die to 0, very quickly when the region is changed from liquid to solid and if you take permeability k value to be very small number, then that actually spread over away more range and is very small. It is a small number to ensure the there is no numerical difficulty.

Now, you can inspect this term to see whether it will help you in making the velocity go to zero in the solid, so that the equation is valid or not. So, you can do it by inspecting this term plugged into the Navier strokes equation and ignoring all other terms. So, I would demonstrate that now.

(Refer Slide Time: 37:20)




So, how we could write the entire equation? I am putting dots to avoid having to write all the other terms which means that you could see that this guy is going to go as minus k 1 minus epsilon square u by epsilon cube plus b and u. I will put into the right hand side as follows which means that the relative change in u with time is given by minus k into this. Now, what happens in the solid region epsilon is 0 and what happens when epsilon is 0 on the right hand side. It will be 1 minus 0 that is 1 square and 0 cube plus b which means that this expression is going to look like this. It goes as minus k by b.

Now, I already told you that b is a small number for numerical reasons and k is permeability which can take as large numbers which means that it is small large number divided small number which means it is a very large number, typically 10 to the power of 8 or 10 to the power of 10 etcetera and with the minus sign which means that this term is going to hence essentially drive the magnitude of the velocity going to 0 because relative change in u dou u by u relative change in u with respect to time is negative which means that this is going to ensure that u magnitude is going to approach 0 as you literate, ok.

So, this formulation will ensure that in the solid when epsilon is equal to 0, the velocity will be 0. What happens in the fully liquid region epsilon is 1. Now, what happens when epsilon is 1 minus epsilon that is 1 minus 1 square that is 0? So, it means that this term is dropped. This means that that term, the so-called enthalpy porosity term is null. It is 0 which means that you have retrieved the equation which is meant for the fully liquid region.

In other words, this term is not going to do anything to the equation in the fully liquid region, so that the liquid flow is completely governed by the Navier strokes equation with those conditions and in the solid is going to give you the velocity is going to be 0 and for in between, it will be going to give you a very smooth changer of velocities which means that it is going to give you a velocity field that is going to go from the full liquid field to a 0 value from liquid to solid region as you progress with iterations. So, this means that by using this kind of a term to add to the body force term, you will be able to achieve the purpose of single domain approach for this formulation. So, this is how you could also write.

So, there are two different approaches as you can see. This second one is quite elegant and you can plug it in which means that when you look at publications which look at the fluid flow in the weld pool and the Navier strokes equation is written. Watch out for this kind of term on the right hand side, but that means they are using single domain approach with enthalpy porosity formulation and if this term is absent and no such term is actually there at all, then the equation is only for liquid region and then, you could ask a question how are they handling it as a single domain. What are they doing with the solid region? So, you should pay attention to that aspect.

So, with that we will just close the first part of the fluid flow modeling and we will continue in the second part.