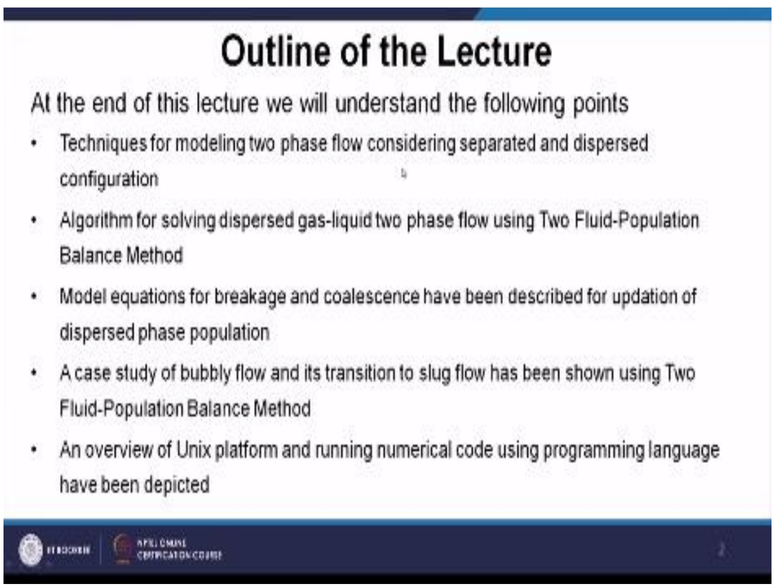
Two phase flow and heat transfer
Dr. Arup Kumar Das
Department of Mechanical and Industrial Engineering
Indian Institute of Technology, Roorkee

Lecture No. 12
Two Fluid and Population Balance Model

Hello, welcome to the twelfth lecture of Two Phase Flow and Heat Transfer. In this lecture we will be discussing about 2 fluid model and population balance method logic for prediction of dispersed 2 phase flow. At the end of this lecture you will be understanding techniques for modeling of 2 phase flow.

(Refer Slide Time: 00:47)

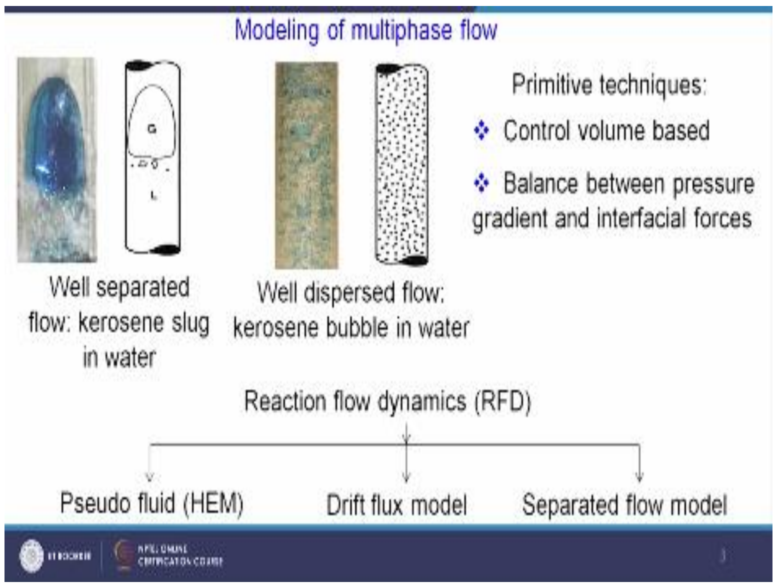


Considering separated and dispersed configuration. Algorithm for solving dispersed gas liquid 2 phase flow using 2 fluid population balance method logic will be also elaborated. I will be telling you that what are the model equations for breakage and coalescence of dispersed phases in case of population balance methodology. A case study of bubbly flow and its subsequence condition to slug flow will be also captured in this lecture using 2 phase 2 fluid population balance methodology.

And at the end I will be giving you a review of UNIX platform okay and running numerical code, how a numerical code can be run using programming language those parts I will be telling you. Okay, let us discuss little bit about modeling methodology for multiphase flow. Now to decide what will be the modeling methodology first you need to understand the interface okay. Now interface can be of 2 different types, 1 will be well separated interface where, interface nature will be clearly identified that means both the phases you can separately identify.

And another one will be where interface nature is not visible clearly. So, distinct identity of any phase cannot be tackled okay. So here I have given you 2 figures, experimental figure is first.

(Refer Slide Time: 02:18)



So you see over here, this is the kerosene Taylor bubble in a slug flow followed by some small kerosene bubbles over here as satellite bubbles okay. And in the second figure, you see we have given lots of small kerosene bubbles well dispersed in the water boundary okay. So if you compare these 2 then you will be finding out that in case of kerosene slug in the first figure we are having well separated flow okay.

And in this side you can find out that here you are having good dispersion between the kerosene bubbles in water okay. Here also you can see the schematic of this same thing we have found out. Here by writing a G and L, I have clearly showed that the gas and liquid phase can be clearly identified. But here we are not in a position to write down G and L separately because everything looks like very very dispersed.

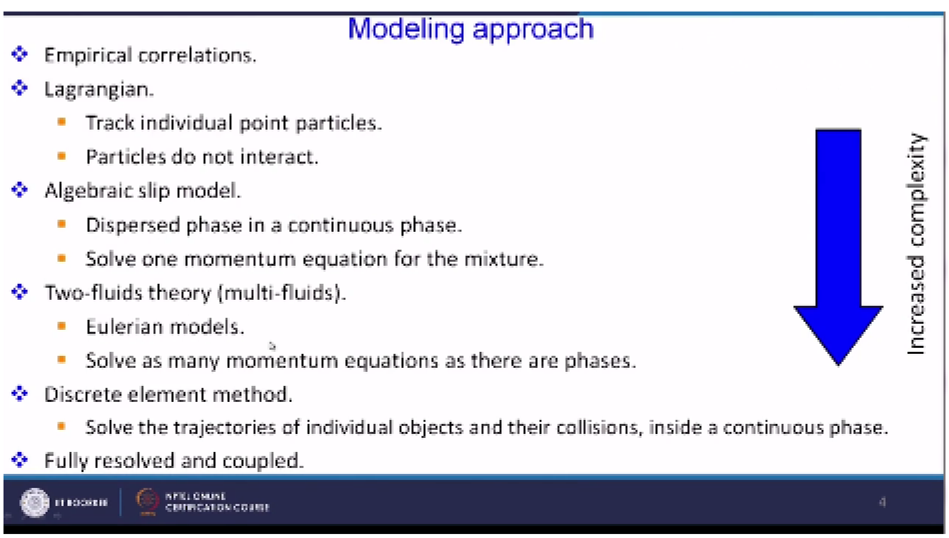
Now what we will be doing, I will be showing you a primitive methodologies or techniques for finding out the modeling methodologies for 2 phase flow. So basically the modeling of multiphase flow was earlier control volume based those some methodologies still are continue in the control volume based frame work but earlier methodologies were all based on control volume based.

There they were doing actually balance between pressure, gradient and interfacial forces. Interfacial forces will be basically laid by surface tension. And some of the methodologies earlier methodologies were like this some first 1 is pseudo fluid which is based on your homogeneous equation methodologies, homogeneous equation model which I have already discussed with you.

You can have drift flux based model also for your 2 phase flow, these are earlier techniques. You can also have separated flow model wherever you know clear and distinct interface is visible okay. So these are some of the earlier methodologies you are having okay. But now let us see with advance print of technology how actually the newer methodologies are evolving.

So here I have given you a brief history of modeling approaches in the 2-phase flow. So initially we started modeling and deviated from a experiment using empirical correlations. So these empirical correlations are actually found out by lots of experiments okay. Then we can have Lagrangian type of methodology okay.

(Refer Slide Time: 04:59)



Now what is Lagrangian type of methodology, in case of Lagrangian methodology, you will find out the framework is actually fixed on the fluid okay. So that is not fixed in the phase that is fixed on the fluid. Along with the movement of the fluid those can be you know the frame work will be moving okay. So, in those cases you will be finding out that you can track individual particles inside the fluid continue on.

Okay and you will be finding out that as the particles are as the framework is moving with the particle there will be no interaction between the particles okay. So particles do not interact okay. Then came the idea of Algebraic slip model okay. So Algebraic slip model you will be finding out that dispersed phase is there in the continuous phase. So there is a next dispersion between the continuous phase and the secondary phase, primary phase and secondary phase.

And do you solve a single momentum equation okay for the mixture? So these types of models are actually your homogeneous equation model, drift flux models okay. So in case of homogeneous equation model, you consider that informal properties are there based on the void fraction of the phases okay. So here algebraic slip model considers that one. Apart from that Algebraic slip models also considers slips between the phases okay.

Then come the idea of 2 fluid theory so in case of 2 fluid theory, they can separate out between the phases. So that means you will be having phase 1 and phase 2. So it is actually applicable for multi fluids. Majority of this 2 fluid methodologies are actually Eulerian in nature, Eulerian means you will be having a fixed reference frame with respect to space.

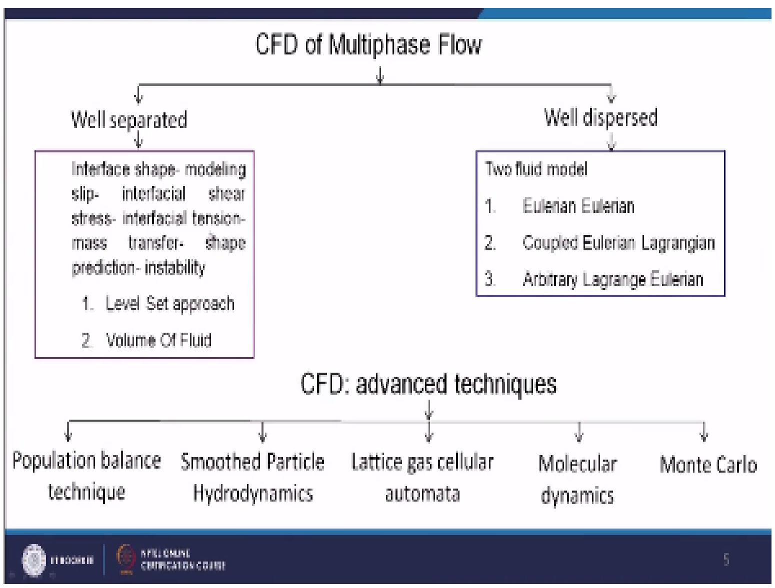
We solve as many momentum equations as there are phases. In case of 2 fluid, 2 phase flow will be having 2 momentum equations, in case of 3 phases you will be having 3 momentum equations so on. Then came the idea of discrete element method okay. So, it solves the trajectory of the individual object okay and find out their collisions okay inside the continuous phase okay.

And this type of methodology is actually very, very well resolved in scale, you will be finding out complexity increases whenever you see with the microscope that means you resolve the scale in lower scale lower level. And the final one, we can go for fully resolved and coupled that means this is little bit towards your direct numerical simulation are (()) 07:30 simulations okay.

So if you see complexity wise, you will be finding out starting from the empirical relation which is very, very easy okay and less complex. And a fully resolved 1 will be very, very complex one. So complexity increases in the downward direction. Next let us talk about that what are the different computational fluid dynamic methodologies abbreviate for multiphase flow.

Basically you will be finding out 2 different types of configurations, already I have scattered well separated and well dispersed. Now for well separated where, interface is clearly visible we need to track the interface. So for tracking the interface we have few methodologies like a level set approach volume of fluid approach.

(Refer Slide Time: 08:11)



So volume of fluid approach I will be discussing in the next lecture okay there we have to tackle the interfacial shear stress, interfacial tension and through the interface how much amount of mass and heat transfer getting place okay. And you know we have to predict the shape and instability of the interface every time. In case of well dispersed where interface is not a problem we need not to track the interface we use methodologies like 2 fluid model okay.

Now this 2 fluid model can be Eulerian, Eulerian in nature. It can be Eulerian Lagrangian in nature okay. Coupled Eulerian Lagrangian Cel or it can be arbitrary Lagrangian Eulerian. So, in case of Cel, actually the continuous phase is treated as Eulerian and dispersed treated as Lagrangian.

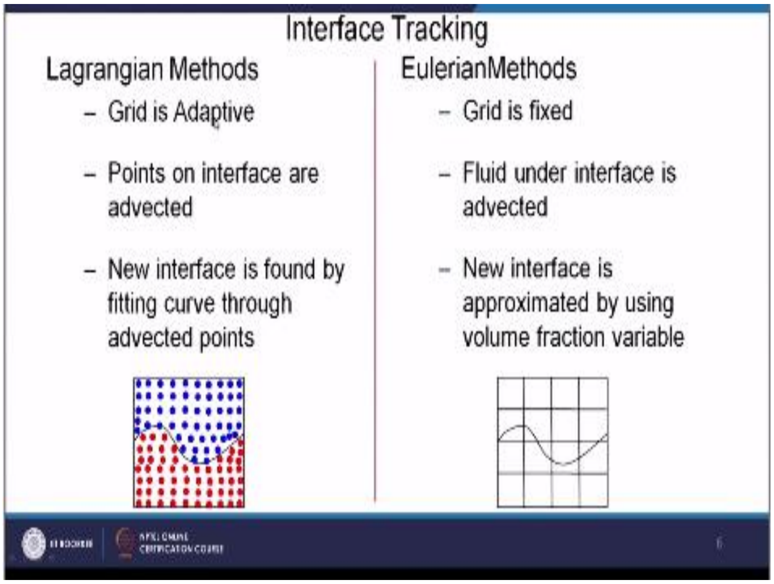
On the other hand arbitrary Eulerian Lagrangian gives you the freedom to choose whatever you want that means you can treat the continuous phase as Lagrangian or Eulerian depending on your domain geometry. Okay, next let us see that what are the advance techniques? Now the advance techniques are coming under computational multi fluid dynamics.

So we are actually going 1 step further from your computational fluid dynamics to computational multi fluid dynamics where, techniques are specially designed to tackle the multi phase flow. So there are methodologies like population balance technique which tackles actually bubble-bubble interaction or drop-drop interaction specifically designs for 2 phase flow applications.

We can also have smoothed particle hydrodynamics where, particle with nature of the continue will be treated and the interface will be automatically coming out from the boundary of the separate colored particles. We can also have Lattice gas cellular automata from where actually Lattice Boltzmann methodology will be introduced. So you will finding out in case of this one, we are have Lattice gas cellular automata you are having more freedom of the information to propagate in directions. So handling 2 phase flow this type of methodology will be very cool.

We can have molecular dynamics, which resolves in the lowest level molecular level of the fluids. So that methodology also we can discuss. And some methodology advance methodologies are like this Monte Carlo and so on okay. So let us see that for interface tracking we can have the 2 different types of methodology still now. Till now we have discussed 2 different types of methodologies as Lagrangian and Eulerian.

So let us give little bit of idea what is Eulerian and Lagrangian. So if your grid is adaptive then we can call that it is Lagrangian methodology.
(Refer Slide Time: 10:47)

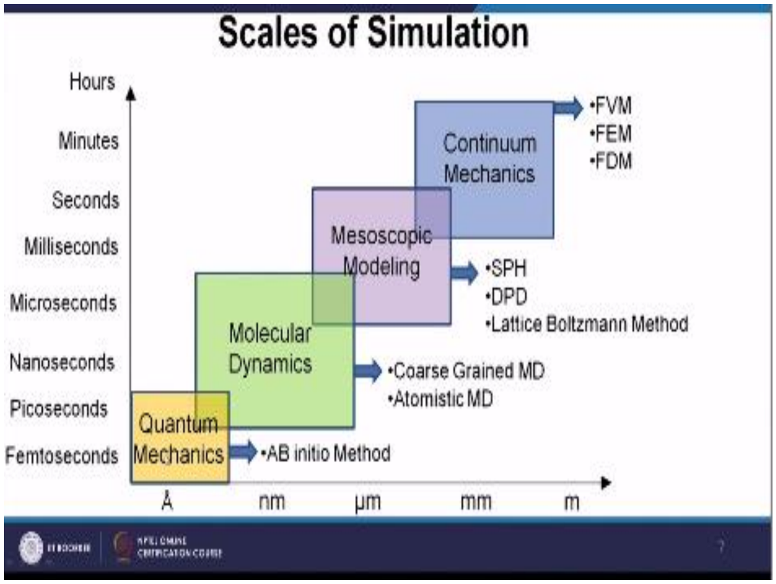


Point of interface are advected so it will be flowing along with the flow okay. New interface is found by fitting curve through advected points. So here you are having some advected points so you have to fit the interface like this okay. So this type of methodologies you will be finding out in smoothed particle hydrodynamics and so on. We can also have Eulerian methodologies okay.

So Eulerian methodologies here this is grids are fixed. So grids will not be moving so grids are fixed you know (()) 01:14 grids we can find out where regular shaped grids will be seen. Fluids under interface is advected. So now here the grids will not be advected rather fluids will be advecting through the grids okay. And you have to find out the interface from the locations of the advected fluid okay.

So here the particles will be advected and the particle boundary giving the interface but here you need to find out that one okay. Next here I have discussed that what are the different scales and methodologies we are having for treating the 2 phase flow simulations. So if you start from the smallest scale, so you will be finding out an angstrom size we are having something called quantum mechanics.

(Refer Slide Time: 11:58)



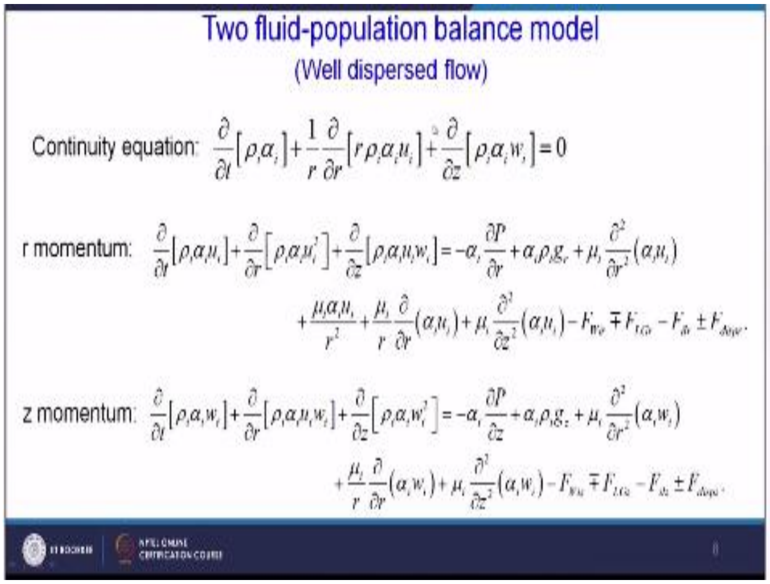
Which is little bit related to physics. And in the higher size in the range of millimeter to meter engineering application will be find out which is having continuum in nature okay. So for this continuum we apply finite volume, finite element and finite different methodologies from computational fluid dynamics.

And now if you go little bit up from quantum mechanics that means in the range of nanometer scale, you will be finding out you are having techniques like molecular dynamic which considers the molecular dynamics properties in nature. And in between you can have some Mesoscopic technique also which takes the advantage of both the macroscopic and microscopic methodologies.

Some of the methodologies like SPH, LBM, I will be discussing in the next 2 lectures. Now in this lecture let us concentrate on the modeling of dispersed, well dispersed flow. So I will be go through the model called 2 fluid population balance method okay. So let us see 2 fluid population balance so that means it will be tackling both the fluids separately. So from the name you can understand that you will be having separate equation sets for the fluids.

So here I have shown you 1 example adiabatic example where, heat transfer is not involved. So you will be having on the continuity and momentum equation. So here I have shown you for a 1 (()) 13:18 problem so you will be only r coordinate and z coordinate. So, first one i the continuity equation. So here you see I have written i over here. So i symbolize the number of phase. So i = 1 means you will be getting continuity equations for a 1 phase, i = 2 will be giving you continuity for another phase.

(Refer Slide Time: 13:44)



And here you see you are having alpha which is void fraction for the higher phase. So alpha 1 + alpha 2 will be always equals to 1. So this is the continuity equation similarly this is our momentum equation okay. And in r, r direction and this is a momentum equation and z direction. These equations I am not going into detailed because we have already derived in a fluid mechanics course.

But the important points I will be highlighting over here is that the source stands in the momentum equation. You see here due to a presence of 2 phase flow we are having some new forces like this. First one is actually a wall friction force. So, frictional force was also earlier there in a single phase. But here we need to tackle separately because we are having liquid contact with the wall as well as the solid the gaseous contact okay.

So here you see Fwi we have kept separately. So, Fwir and Fwiz in the r and z direction respectively. Here you see f L G so f LG is actually the drag force due to the interface. So as you are having the gaseous phase so that gas phase whenever it will be moving through the liquid phase. It will be getting some drag force over there. So this drag force will be calculated over here in r direction and z direction respectively.

The science in the respective equation gas and liquid equations will be different okay. So they will be same in magnitude but opposite in direction. In a similar fashion we are also having interfacial forces. So this interfacial forces are mainly govern by the surface tension because you know across the interface will be getting some line tension and across that you will be having some force in the liquid and gas okay.

They will be also opposite in magnitude. So you see what we have given, we have given minus Fiir and - Fiiz this small i symbolize the interface. Okay, also we will be having the dispersion force so the dispersion force also we have kept over here. That will be same for both the phases that means liquid and gaseous phase but they will be opposite in nature. So we have kept for r direction and z direction respectively.

Next let us discuss about those interaction forces. First you need to calculate the average pressure because you area having 2 different phases your pressures can be different. So average pressures for the dispersed phase we need to calculate. So, average pressure will be depending on the average density.

(Refer Slide Time: 16:01)

Average pressure $P = (\rho_l \alpha_l + \rho_g \alpha_g) R_g T$

Interfacial drag force $\overline{F_{LG}} = \frac{2C_{FL}}{D}\sqrt{\alpha}\rho_g\left(\overline{u_g} - \overline{u_l}\right)\left|\overline{u_g} - \overline{u_l}\right| + \frac{\alpha}{2}\rho_l \overline{u_g}\frac{\partial}{\partial r}\left(\overline{u_g} - \overline{u_l}\right)$

where, $C_{FL} = C_D\sqrt{\alpha}(1-\alpha)^{1.7}\frac{\rho_l}{\rho_g}\frac{D}{2R_B}$

Wall friction force $F_{WL} = \left[1 + \left(Y_c^2 - 1\right)\left(B\left(X_c(1-X_c)\right)^{\frac{2-n}{2}} + X_c^{2-n}\right)\right]\Delta P_{L0}$

where, $X_c = \frac{1}{1 + \frac{1-\alpha}{\alpha}\frac{\rho_l}{\rho_g}\frac{\overline{u_l}}{\overline{u_g}}}$ $\quad Y_c = \left(\frac{f_{G0}\rho_l}{f_{L0}\rho_g}\right)^{0.5}$ and $B = 12500$ and $n = 0.25$

Turbulent dispersion force $F_{disp} = C_{TD}C_D\frac{\gamma_{tg}}{Sc_{tg}}\left[\frac{\nabla\alpha_l}{\alpha_l} - \frac{\nabla\alpha_g}{\alpha_g}\right]$ With $C_{TD} = 0.1$

So this is our average density Rho i alpha i+ Rho g alpha g. So this Rho i alpha i and Rho g alpha g are 2 different phases. And rg is your gas constant and t is your temperature of the mixture. Then first, go to the initial drag force. So in case of initial drag force interfacial drag force which is fLG sorry, interfacial drag force.

So, interfacial drag force can be calculated as 2* Cfl where, Cfl is the coefficient for the drag okay 2Cfl/d root over of alpha. Alpha will be also involved over here because you are having 2 phase. alpha * Rho g then you are having (ug – ul) and then |ug – ul| as mod okay. So depending on the relative velocity this part of this drag force will be actually changing sign. And the other hand side, second term you are having alpha /2 Rho L ug delt /delt r (ug - ul).

So here depending on the space what is the variation of the drag force that will be coming into picture okay. Now Cfl is very important factor over here. Cfl can be written from the drag coefficient. So cd *root over of alpha *(1- alpha) to the power -1.7. They are density ratios and then d / 2 Rb. So Rb is very, very important Rb is nothing but your average bubble diameter okay.

So in the population of the dispersed phase you need to find out the average bubble diameter okay. I will be coming there quickly. Now for the wall friction force if wl, you can calculate in this fashion 1 + yc square -1* b * xc 1 - xc to the power 2- n /2 + xc to the power 2 - n * delt p lo. This delt p lo is nothing but frictional pressure drop due to liquid only consideration okay which we have already discussed.

Now xc and yc are the parameters over here and n and b those are the constants. So xc and yc can be calculated from the knowledge of your fluid properties. They are respective velocities and the value of alphas okay. So xc can be written like this 1/ 1 +(1 – alpha) /alpha Rho l / rho g *ul /ug and yc will be f go okay.

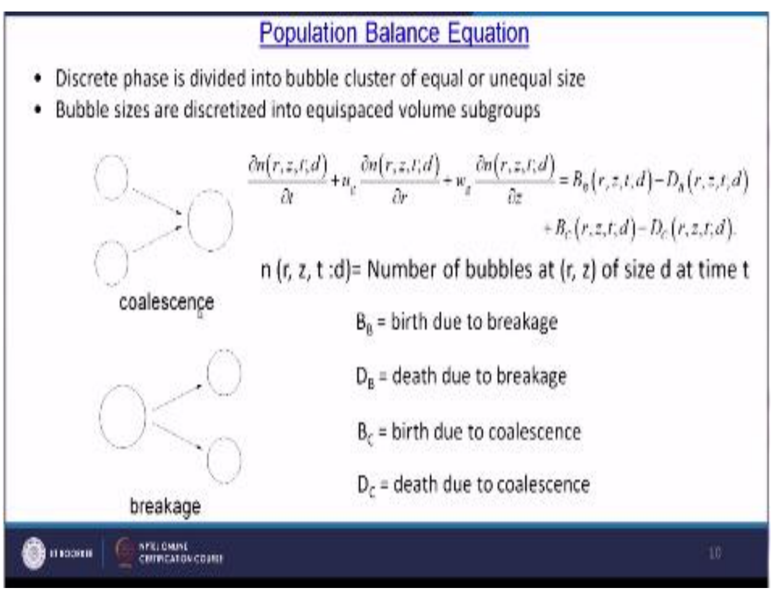
So this is gas only concept friction factor due to gas only concept into Rho l / f lo liquid only concept liquid only friction factor multiplied by Rho g to the power 0.5. The value of b and n can be taken as 12500 and n equals to 0.25. Another important force was there, which is nothing but turbulent dispersion force. So turbulent dispersion force can be written as cTD into cd. So this cTD is the coefficient for the turbulent dispersion okay usually you take the value as 0.1. cd is the drag coefficient okay and then gamma tg / SC.

SC the smith number okay. So, smith number for the turbulent gas and then delt alpha l/ alpha l – (delt alpha g /alpha g). These delts are the special derivatives for the alphas okay. So smith number of the gas you need to find out okay depending on the property and you know the cTD value can be taken as 0 point and you can find out the dispersion force. Once you get all this introduction forces you can put in the momentum equation for finding out the next points.

But importing thing over here is that in all these you will be finding out there is dependants of this average bubble diameter which we need to find out. Okay, so finding out the average bubble diameter actually we go for a technique called population balance methodology. So in case of population balance methodology the dispersion of the phases we consider as a population of different sized bubbles okay.

So let us say we are having from very smaller size to very bigger diameter sized okay. So discrete phase is divided initially okay into bubble cluster are of equal and unequal sizes that means so let say we are having bubbles from very smaller size to a very big diameter size okay.

(Refer Slide Time: 20:09)



So discrete phase is divided initially okay into bubble cluster or equal or unequal sizes that means let us say you are having bubble diameter from 1 millimeter to 10 millimeter. So what you can do from 1 to 2, how many bubbles are there from 2 to 3, how many bubbles are there from 3 to 4, how many bubbles are there and so on from 9 to 10, how many bubbles are there you can count and you can have a population balance kind of thing that in between these sizes we are having this many numbers of bubbles okay.

And then bubble sizes are discretized into equispaced volume subgroups. As I have already stated and you can write down the evaluation equation of the number sizes in this fashion. So dn /dt this n is for a particular size d okay. d is having different subgroups. Starting from the minimum size to maximum size and r, z, t are they are corresponding radial axial and time level okay.

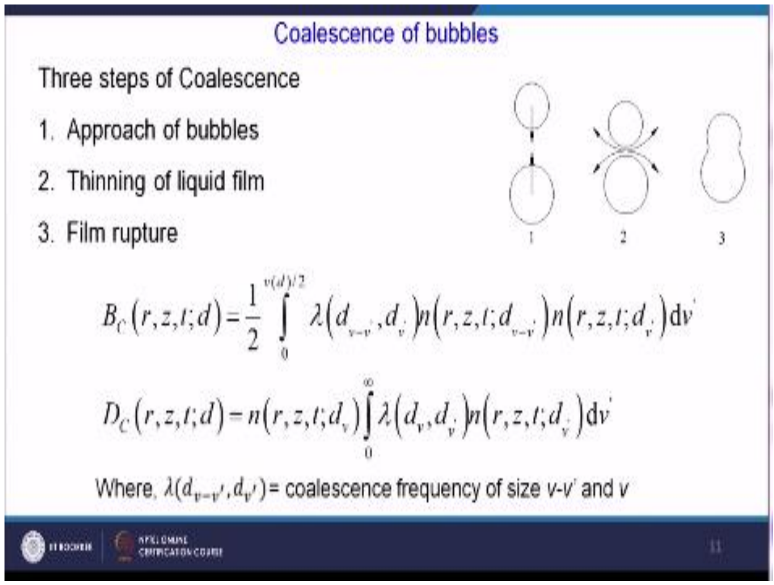
And you will be finding out that this number will be actually into several grades, several control volumes based on the velocities. So ug and wg will be coming into picture over here. So those are actually the continuous phase velocities in the radial and axial directions. And then in the right side, we will be having the source terms this is very important you will be finding out that source terms will be due to breakage and coalescence of bubbles.

So breakage means here I have said a single bubble due to turbulence dispersion of the continuous phase can break into 2 small sizes daughter bubbles as well as we can have turbulent dispersion in such a fashion that 2 bubbles will be merging and they will be making a bigger size bubbles okay. Now due to this we can have any number subgroup birth due to breakage death due to breakage.

So breakage we will be having death over here of this sized group and birth over here for this sized group. Similarly for coalescence, we can have birth of this sized group and death of this sized group okay. Next let us see, what are the different models available for coalescence? So we are having over here some figure where, I have shown 2 bubbles are colliding.

So the steps are they will be approaching towards each other and then intermittent film they will be thinning up okay thinning of liquid film and finally the film will rupture and a single bubble will be formed.

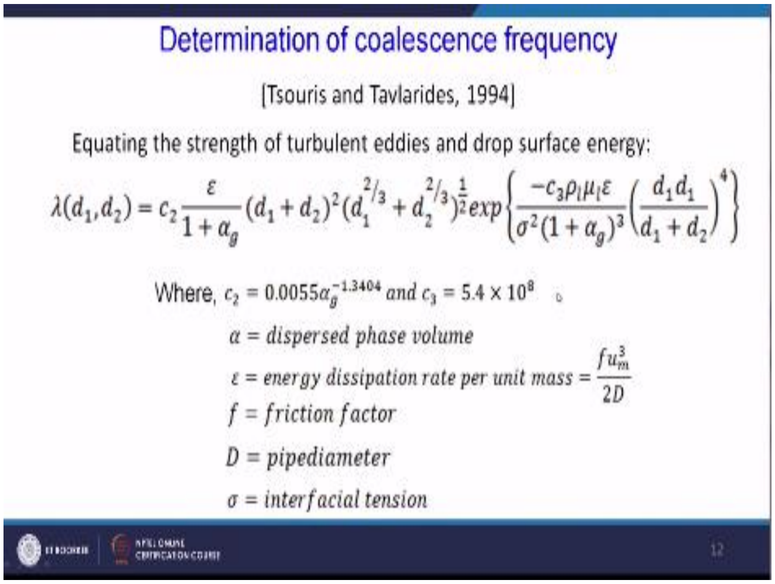
(Refer Slide Time: 22:26)



Now this breakage and this birth and death due to coalescence can be written as bc 1/2 0 2 vd / 2 okay. Lambda n and n dv and now here you see this n is the number density of (v – v) dashed group and n is actually is v dashed group. So these groups are actually this is actually v dashed and this is actually (v – v) dashed. So ultimately if we merged these 2 this will be forming the size group of v.

Okay so this lambda is actually there coalescence frequency. So lambda is a coalescence frequency between sizes of (v – v) dashed and v okay. Similarly for the death you will be finding out this is nothing but n integration of 0 to infinite lambda n based on v dashed dv okay. Now let us find out what is lambda. So far finding out the coalescence frequency we will be following model given by Tsouris and Tavlarides, okay.

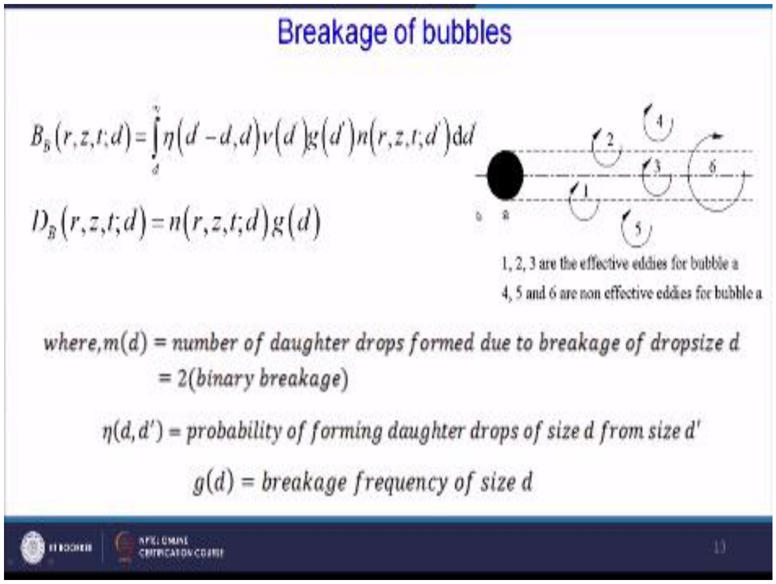
(Refer Slide Time: 23:40)



In 1994 they said that lambda d1, d2 these are the bubble diameters colliding among themselves. c2 * epsilon / (1 +alpha g) where, alpha g is actually the gaseous phase void fraction multiplied by d1 + d2 square d1 to the power 2 /3 + d 2 to the power 2 /3 to the power ½ and e to the power - c3 Rho l Mu l epsilon / sigma square (1 -alpha g) whole cube * d1 d2 / d1 + d2 to the power 4 okay.

Here c2 and c3 those are important coefficients empirical coefficients those can be obtained in this fashion here I have shown in the slide okay. Also the fraction factor is also important over here okay. Friction factor f will be also coming into picture for calculation of the epsilon energy dissipation rate okay. Energy dissipation rate epsilon can be calculated as f * um cube /d2 okay. Now due to breakage you can find out here we have shown the breakage.

(Refer Slide Time: 24:35)

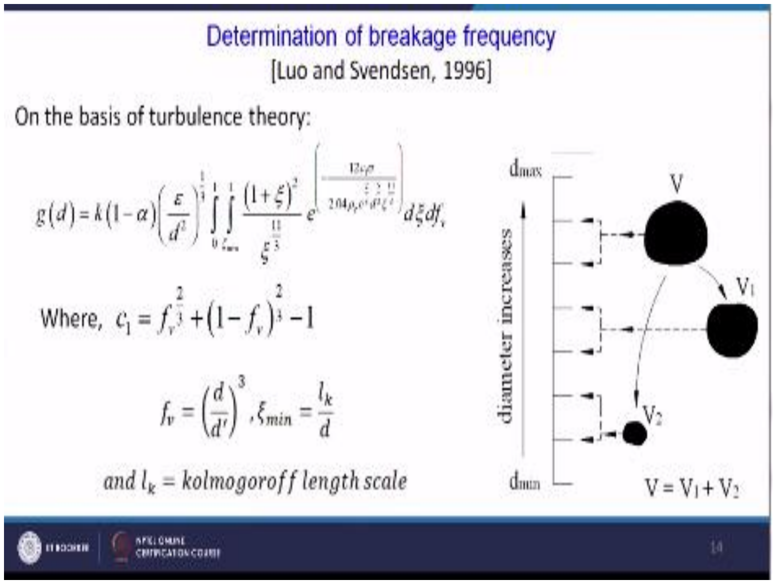


So breakage can be found out as n eta vd gd and nd where vd and gd is actually gd is breakage frequency of size d and you will be finding out n dd that is actually probability of forming daughter drops of size d from size d dashed. So let us say we are having d dashed size over here, from there it is breaking into small bubbles and will be forming d size.

So the what is the probability of forming daughter drop of d size from d dashed size that will be actually given by ndd okay and md which is nothing but the number of bubble that is actually coming as 2 because we have considered over here binary breakage. We can also for tertiary break more than that now depth breakage will be n * gd okay. Now we have to find out what is gd.

So from turbulent qd we can find out gd is actually k * (1 – alpha) epsilon/ d square to be power 1/ 3 and then double integration over this term based on zeta okay now zeta is actually your lk / d okay.

(Refer Slide Time: 25:36)

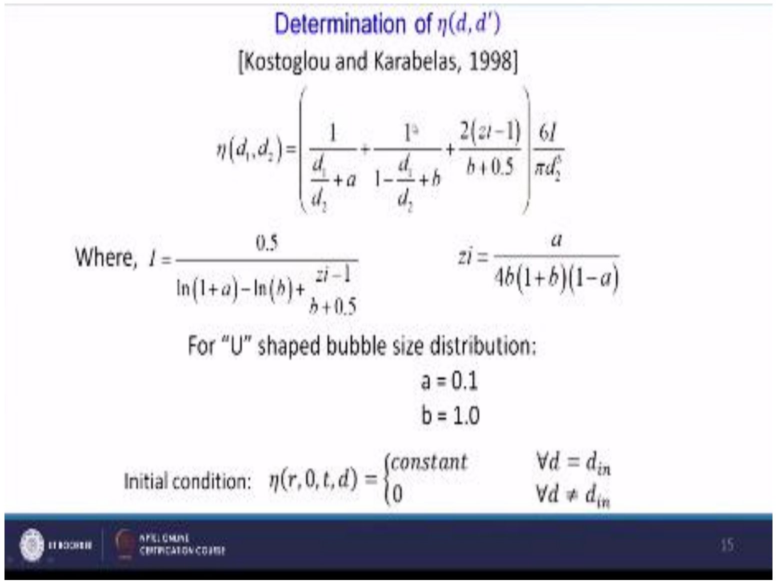


So this is the value of zeta mean over here okay from here to 1 it will be varying okay. Now here you will see you are having c1 as constant over here. What is e to the power expression so that will be fv to the power 2 by 3+ 1 - fv to the power 2 /3 +1where fv is nothing but the volume ratio of the daughter droplet in the mother droplet okay.

lk is the kolmogoroff length over here in the zeta minimum calculation. Here I have shown whenever breaking up. So it can happen that the daughter droplets are falling in between so what we will need to do from here by balancing the volume and the number you have to distribute in the neighboring groups okay.

So it is coming over here, in between so we have to distribute in the neighboring groups here and here by keeping the number and volume constant so some type of configured equations are also necessary over here. Next the value of eta ed.

(Refer Slide Time: 26:49)

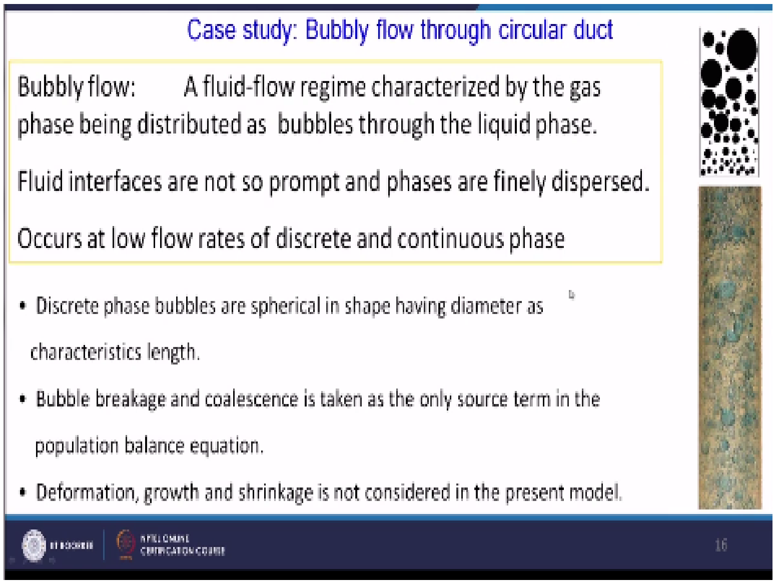


So eta ed can be written as like this which is function of diameter obviously and I which is parameter over here I can be written as value of a and b and zeta can be written once again the value of a and b. Now this, a and b value on the distribution of the bubble size okay. You size the tube actually consider that there is some parabolic distribution so for parabolic distribution we consider a = 0.1 and b = 1 okay.

Then for initial condition we considered that we are having equilibrium size everywhere of the bubble okay and which is the inlet size and apart from that we are having no other sizes okay. So constant and then we have advanced what in time and using population balance and volume of fluid method and find out the where it is forming the bigger size bubble than the general diameter okay.

So that is the initialization of the slug flow or in a bigger size bubbles. Let see when case study. So we are having shown the bubble flow through circular duct.

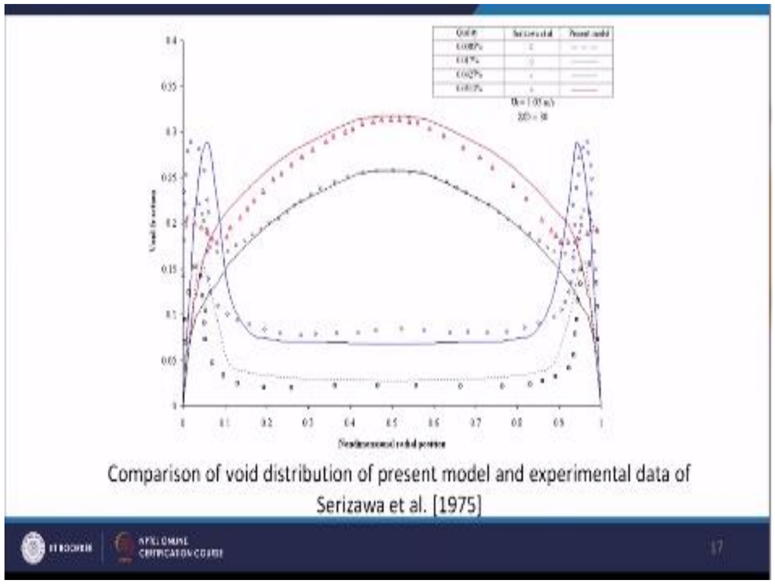
(Refer Slide Time: 27:54)



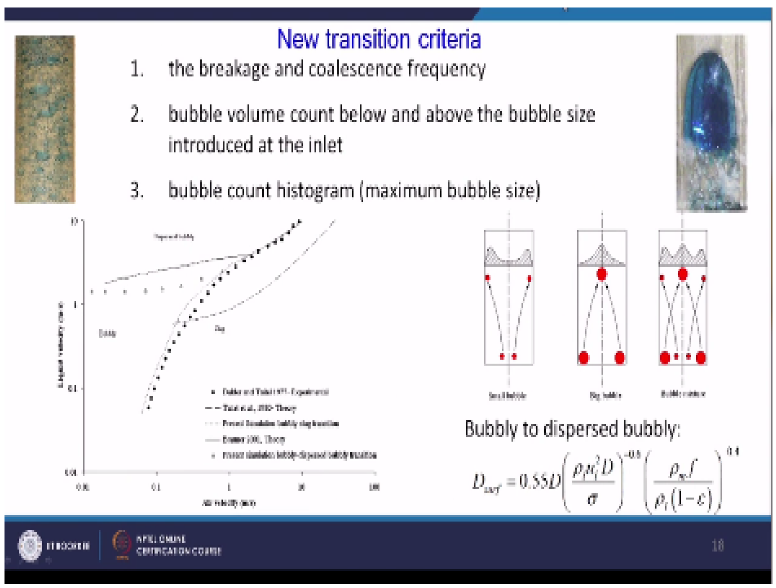
So circular duct we are having bubbly flow. So here the discrete phase we have using population methodology and deformation growth and shrinkage we have not considered in the present model okay. So breakage and coalescence we have considered using your population balance methodology. So let us see over here as we will some results.

So here you see we have first shown that different types of void friction. So usually we will be finding out whenever we are having bubbly flow inside a channel there are 2 different type of void fractions these are called wall peak and core peak majority of the bubbles are coming at the core that is core peak and the majority of the bubbles are going to the wall is actually called wall peak.

So here I have shown the present volume fluid model with present population models can predicts the wall peak and core peak and those are actually matching value the experiments okay.
(Refer Slide Time: 28:42)



(Refer Slide Time: 28:45)



Then using this model 1 new transition criteria also. So you see whenever we are getting a bigger size bubble that is very much flow into slap bubble shape and go to the slug flow regime. So far that we have given new transition criteria so that can be constructed so depending on the breakage and coalescence frequency.

I have said in the previous peak flow sites we can count the volume okay below and above the bubble size whatever we have given as inlet. At the initial condition we have given some bubble size. So we can count always what is the rate of formation of the above and below size bubble and we can predict that whether it will be increasing the size or not and reach towards the slap flow or not.

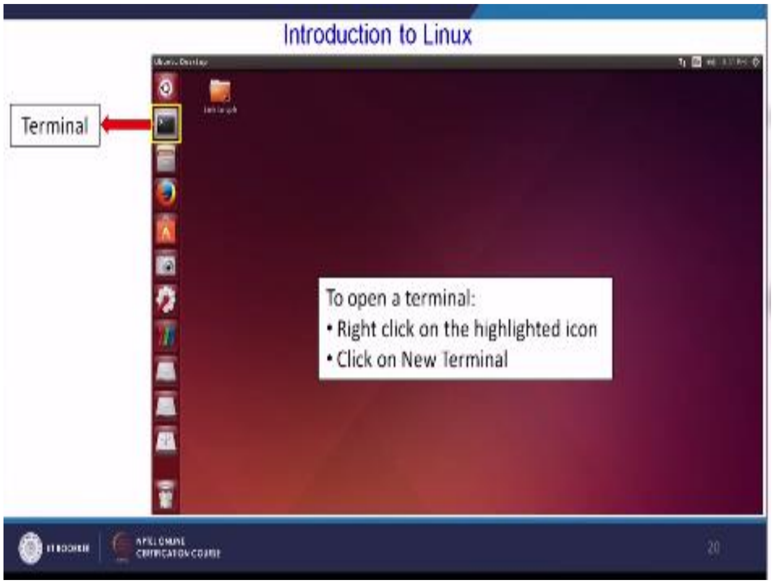
Finally, we can see the bubble count histogram also and find out what is the maximum size bubble to get the population. So if you do so then we can predict this bubble flow to slug flow okay. So in this fashion over here you see, bubble flow to slug flow you can find out the based on the liquid velocity and air velocity differently liquid velocity and air velocity one can plot a flow regime okay which will be separating bubble flow and plot flow.

So here you see bubbly flow and then air it is slug flow. So using this present model then you can predict the experimental very nicely. We can also go for bubble to disperse bubble flow conversion using in this present model. So in that case we used disperse diameter as the criteria for the lower sized bubble possible in the bubbly flow and higher size bubbly flow possible for the this dispersed bubble.

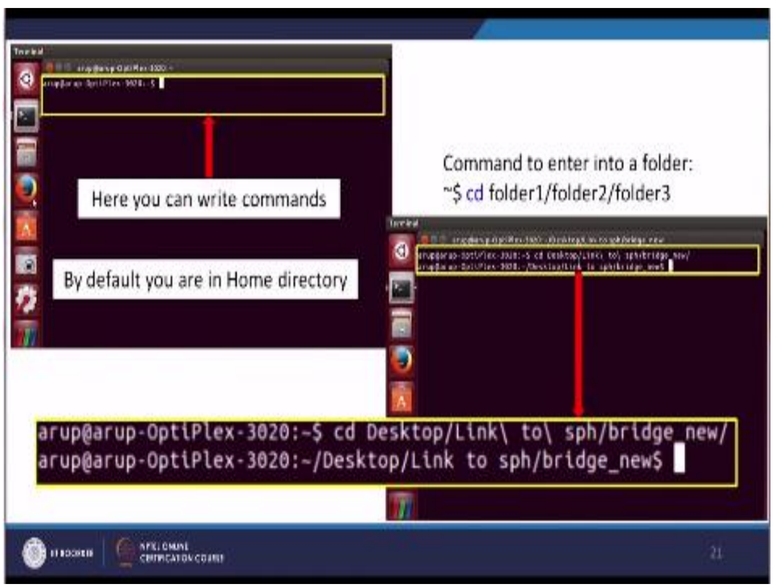
So this is the criteria which we take and dependant on the tube diameter liquid velocities properties okay. Mean properties we take this one and this one we calculate. If it is below this diameter the bubble sizes is average bubble size is below this diameter then this dispersed flow above this bubbly flow okay. Next let me tell you that what this two fluid population balance model is doing actually this two fluid population balance modeling.

Always we need to go for numerical coding because no free where is available which can accommodate this two fluid population balance method. So we have to go for writing our own code you can use any software like FORTAN, C ++, MATLAB okay. Most of this codes run in UNIX platform. So let we give the summary of the UNIX platform next.

(Refer Slide Time: 31:05)

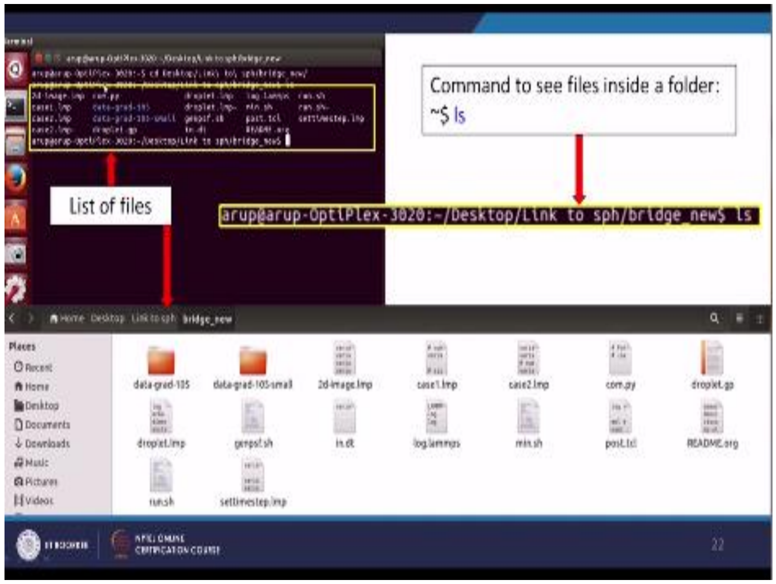


(Refer Slide Time: 31:07)



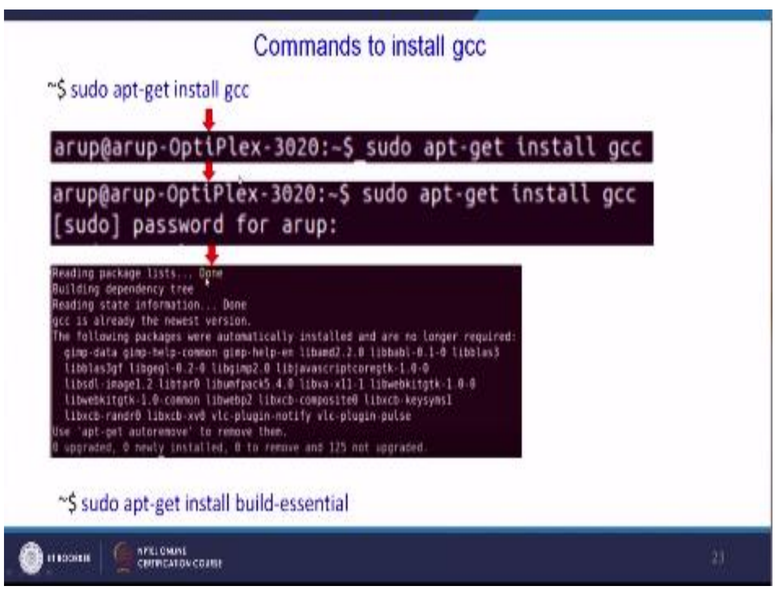
So here this is the terminal you can find out in Linux best server and here this will be the command prompt in the terminal where, you can write the command here. You see I have shown how you can change the directory using cd. So from one directory to another directory how you can change. So for space you can give this kind of character okay font space then space okay.

(Refer Slide Time: 31:25)



Then if you want to see whatever folders you have in the directory you can use the command ls. So you give the command ls, it will be displaying all the files and folders inside that. So you here I have compared what we have in the folder and what it has listed down okay.

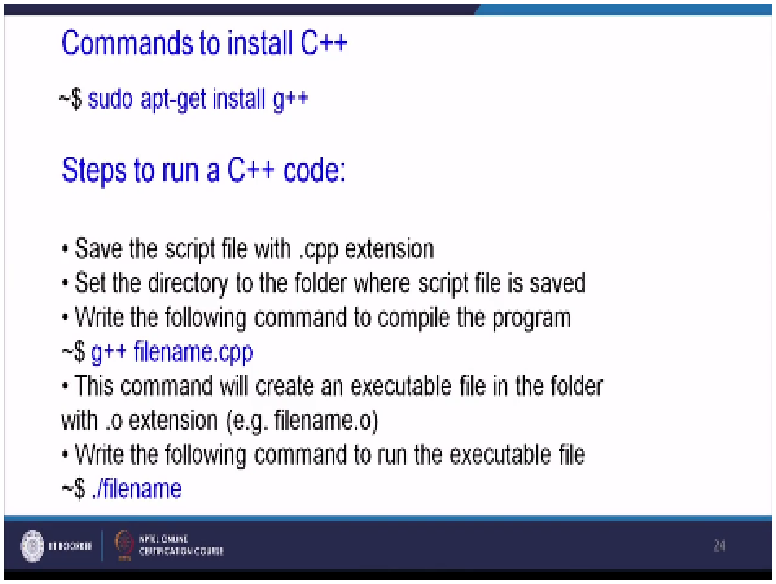
(Refer Slide Time: 31:39)



Then if you want to install gcc that means a complier for C coding so what we need to do sudo apt get install gcc this command you need to type and then automatically it will be asking you for password. So user password once you give that it will be giving you the update that whether it has been install properly or not okay.

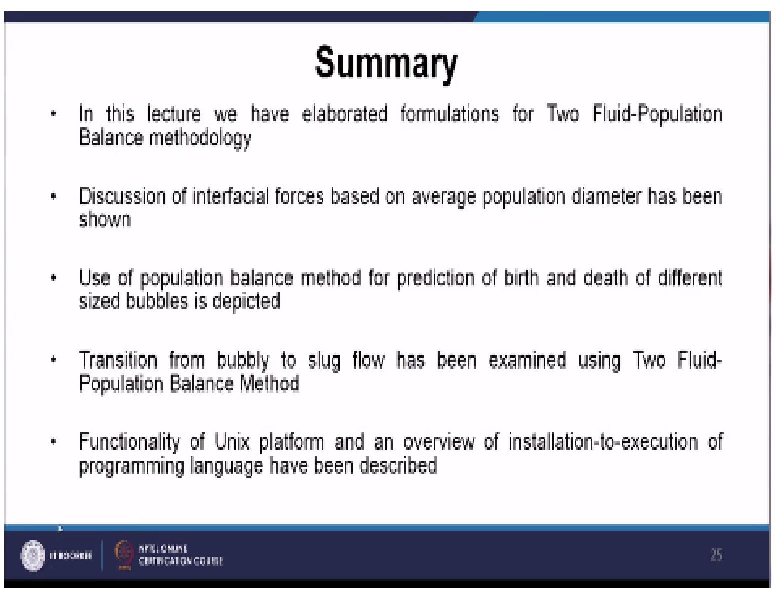
So if necessary what you can do, you can write down pseudo apt get install build essential before running this command so that you are not having a difficulty in resolving the dependencies.

(Refer Slide Time: 32:14)



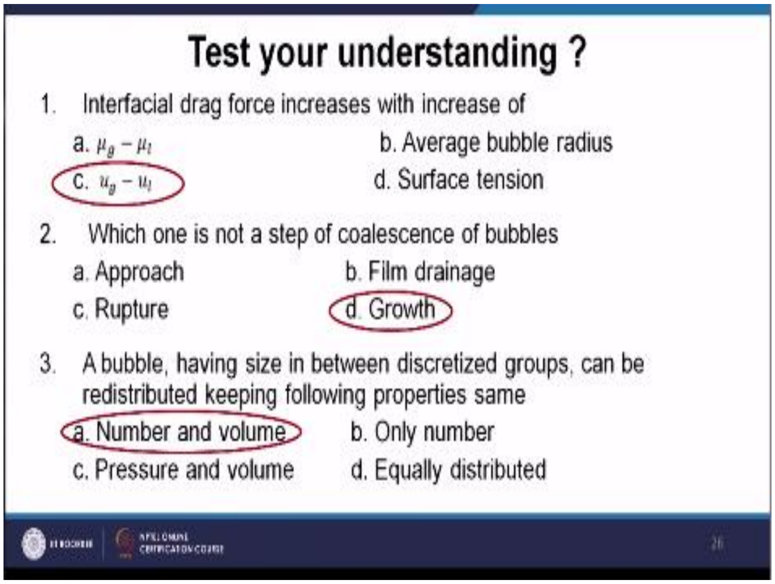
So these are the commands for installing C++. So C already I have shown this C++ so pseudo apt get install C++ okay. To set up to urn 1 c ++ code what we need to do, you create a file and follow the language of C++ and write down the code over there depending on the methodology and save it as some filename dot CPP. And then to run that you have to give the command in terminal as c ++ filename .CPP and to the execute the filename you have to write down dot slash filename okay then the code will be running okay.

(Refer Slide Time: 32:53)



To summarize in this lecture we have elaborated formulations of two fluid population balance methodology. We have discussed interfacial forces based on average population diameter. We have shown you how using population balance methodology birth and death of different sized bubbles can be tackled. Transition from bubbly to slug flow using two fluid population balance method.

We have shown you and finally some functionality of UNIX platform and how to run a coding open source platform that we have shown you over here. Let us test your understanding at the end of this lecture as usual we are having 3 questions.
(Refer Slide Time: 33:27)



First one interfacial drag force increases with increase of. We are having 4 options Mu g - Mu l, difference between the viscosities average bubble radius, ug- ul and surface tension. So obviously correct answer is c because it depends on ug –ul. Second question which one is not a step of coalescence of bubbles. We are having approach film, drainage, rupture and growth.

Obviously growth is the correct answer because there is no growth in case of the coalescence of bubble. Third question, a bubble having size in between discretized groups can be redistributed keeping the following properties same okay. We are having 4 options number and volume only, number, pressure and volume and equally distributed.

So already I have told you that you have to redistribute using the number and volume same. So the correct answer is number and volume. So with this I will be ending this lecture. Thank you.