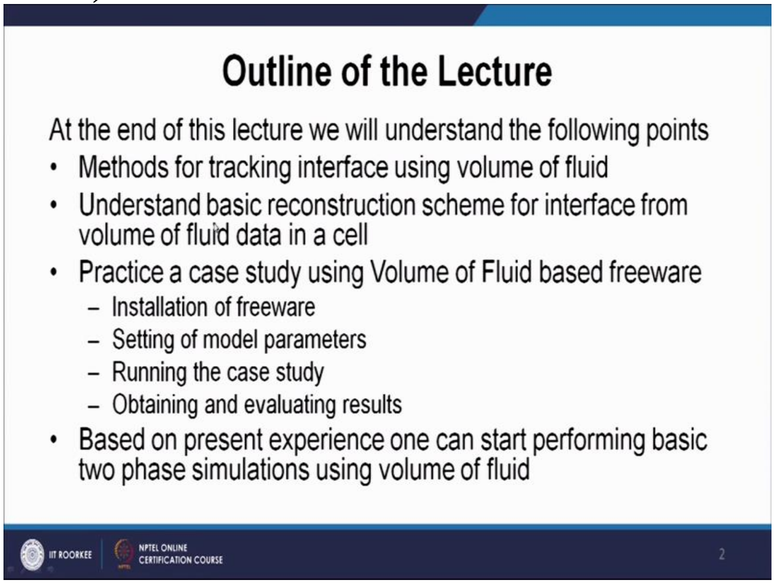
Two Phase Flow and Heat Transfer
Dr. Arup Kumar Das
Department of Mechanical and Industrial Engineering
Indian Institute of Technology, Roorkee

Lecture No: 13
Interface Tracking

Hello, welcome in the thirteenth lecture of Two Phase Flow and Heat Transfer. Today in this lecture, we will be discussing about interface tracking methodologies. If you remember, in our last lecture we have discussed about 2 fluid population balance method where, we have dealt how dispersed 2 phase flow can be handled computationally.

Here I will be showing you how interface can be captured in case of well separated flow okay where, interface is clearly identifiable. Okay, let me see at the end of this lecture, we will be understanding methods for tracking interface.

(Refer Slide Time : 00:59)

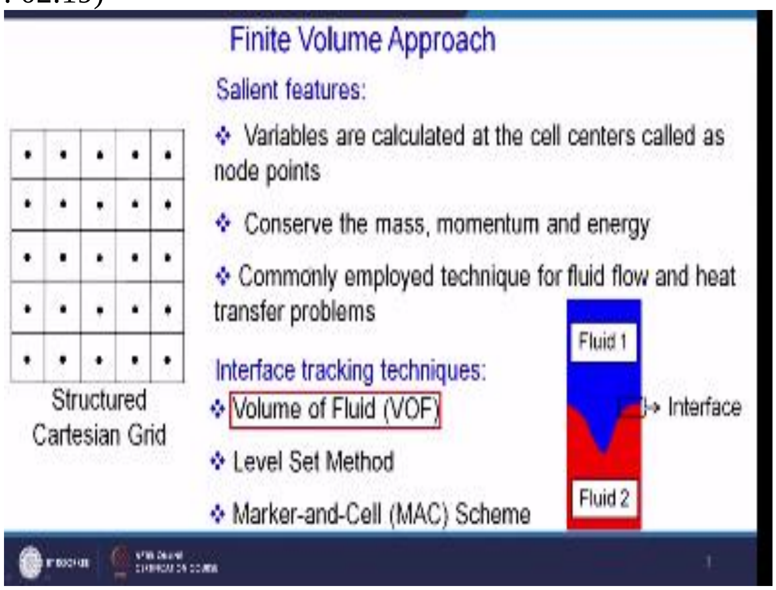


Mainly we will be stressing on volume of fluid methodology. We will be understanding reconstruction schemes of interface from the data of volume of fluid in different cells. We will be practicing a case study using volume of fluid based freeware. So in this we will be understanding how the freeware can be installed.

How we can set model parameters in the freeware, how we can run a case in using this freeware and finally I will be showing you how results can be obtained and evaluated. Based on the present experience of this lecture, you can perform basic 2 phase flow simulations using volume of fluid solver. So to begin with lets us first see that, what is volume of fluid method?

Before going to volume of fluid method we need some grid based framework okay to solve the governing equations basically your mass momentum and energy equations. Here I have taken example of finite volume approach; you can have any other approach also. For example you can have finite difference or finite element methodology but let me give you little bit idea about finite volume approach.

(Refer Slide Time : 02:19)



In this finite volume approach, we discretized the domain let us say this rectangular boxes our domain. We discretized this domain into small, small control volumes okay. And at the center of the control volume or we can say at the CG center of gravity of the control volume, we designate a grid okay node point over here. Majority of the time this nod points are making Cartesian Grid okay.

It will be structured in fashion. So we will be finding out that volume of fluid approach first discretized the domain into finite volumes where, the centroid location is actually considered as the node. Now in this nodes we find out how mass momentum and energy is being conserved. So what we do, we find out the information propagation and integrate over that volume okay.

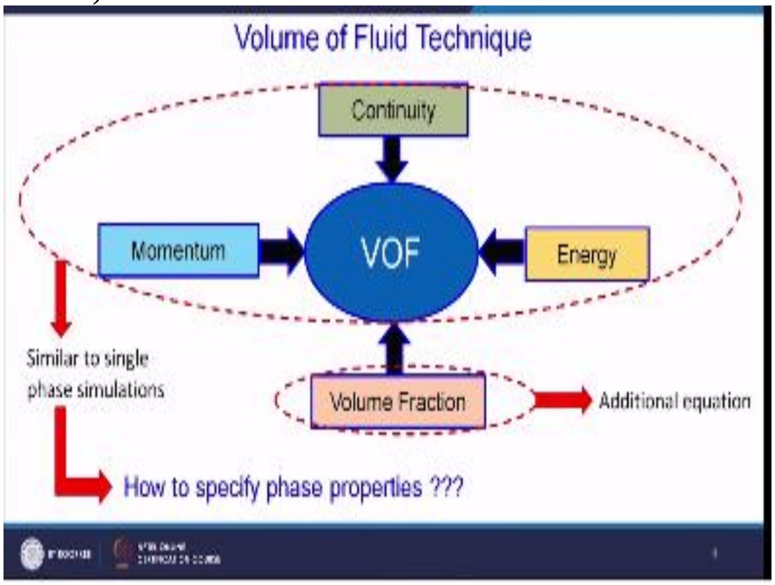
Based on the boundary whatever inflow and outflow of information is coming in or out depending on that we find out, what is the conservation of any information and from there we employ different heat flow and fluid flow problems okay. To take here in single phase, now whenever it comes to 2 phase flow so the important thing what we need to take care is, the interface tracking because apart from mass momentum and energy equation we will be also having 1 interface, which needs to be tracked at every point of time.

Because through that interface only we will be having the mass, momentum mass and energy transfer okay. So here I have shown you a figure, where you can find out 2 fluids are there. So bottom 1 is the red colored fluid may be some sort of heavy fluid or light fluid and here we are having another fluid of different density. So here you can find out that through this interface there will be some sort of momentum or energy transfer okay.

So this interface capturing is very important if the interface is straight. Obviously, the length of the interface will be smaller compared to this curve, curved interface. So the scope of interaction will be reducing whenever the length of interface reduces okay. Now so this capturing this interface or tracking this interface will be very, very important okay. In finite volume methodology we have several options for tracking the interface.

Some of the methodologies are volume of fluid methodology, level set methodology, marker and cell or MAC methodologies okay. Here in this lecture we will be discussing about volume of fluid or VOF technology. Okay, so let me give you brief idea what is volume of fluid methodology. So you see as it is a problem of 2 phase, so obviously mass momentum energy equation will be involved.

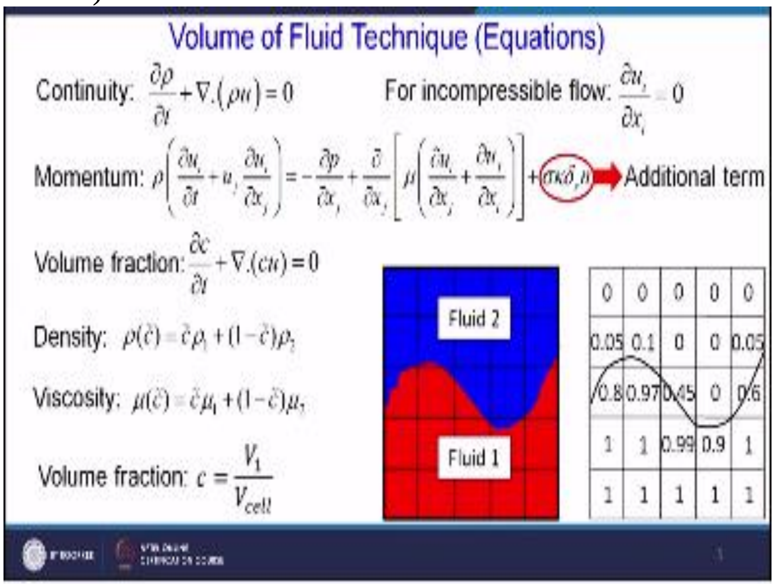
(Refer Slide Time : 05:12)



So here you see we are having continuity equation for mass conservation momentum equation for your momentum conservation and over here energy equation for conservation of energy. Okay all these 3 will be acting similarly as we have dealt in our single phase computational fluid dynamics okay. But only difference will be that here we need to consider that how the phase properties across the interface will be defined.

So for that we include 1 additional equation which is called volume fraction equation. Now this volume fraction equation gives some sort of information to the overall system of equations and finally you will be finding out that all these 3 equations continuity, momentum and energy equation is getting teamed by this volume fraction equation okay. So this volume fraction equation is very, very important in case of 2 phase flow computations.

(Refer Slide Time : 06:08)



Next, let us see that in this volume of fluid methodology what type of equations we solve. As I have told you that we will be having mass and momentum conservation equation. Here also I have shown you the mass and momentum conservation equation. To make it simple I have eliminated the energy part 1, can also add that energy part later on. So here you see I have given the basic continuity equation looks like similar to your single-phase continuity equation having density Rho.

Now what is this Rho, I will be telling you this Rho is actually some average density in the field and this average density will be having some particular value at the bulk and some intermediate value between the bulk properties near the interface okay. So that Rho can be defined actually this Rho will be function of volume fraction of 1 phase inside a cell. Now that volume fraction we actually capture using 1 property called c okay.

So c is the volume fraction. So that means let say we are talking about 2 fluids 1 and 2. Let us say 1 is the first fluid and 2 is the second fluid. So in that case, you will be finding out that we are having c1 = 1 in the bulk of first fluid and c2 = 1 at the bulk of second fluid and the corresponding values of c2 and c1 in the first fluid and second fluid will be becoming 0 okay. So we can define the density based on the volume fraction of any of these 2 fluids.

Let us say this c is nothing but the volume fraction for the first fluid. So this Rho c can be written as c* Rho 1 + (1- c) Rho 2 that means whenever you are in the bulk of first fluid where value of c is 1, you will be getting that Rho gets the value of Rho 1 and whenever you are in the bulk of 2 second fluid then you will be getting that this value of Rho will be taking Rho 2 because c will be at that time becoming 0 okay.

Similarly viscosity can be also defined. So Mu c = c*Mu 1 first fluid viscosity + (1- c) Mu 2 second fluid viscosity okay. Volume fraction, here I have shown that inside a cell let us say this is the cell inside a cell. How much portion of the cell is occupied or volume, how much volume is occupied by some particular phase okay?

Here I have shown that c is the volume fraction for the first fluid okay first fluid means 1th fluid okay. Now depending on the c, you see we can find out the Rho and Mu and we can write down a continuity equation okay. So we will not be having essentially here. 2 continuity equations for both the fluids we will be having single continuity equation but the property is guiding this continuity equation will be changing okay.

So Rho will be getting some value in the bulk okay. So different values for different fluids and near the interface it will be getting the intermediate value okay. For incompressible fluid of this continuity equation will be turning out to be delt ui/ delt xi. So for most of the air water related applications we can consider this 1.

Okay, momentum equation as usual from basic fluid mechanics, we can write down like this. Only added portion will be over here for the surface tension portion because as we are dealing with 2 phase. So obviously will be having surface tension involved in that okay. So surface tension will be involved in the interface, which is nothing but given in the form of delta Dirac function delta s.

So delta s is actually the function which controls the value of surface tension will be 0 at the bulk and at the interface only it will be getting the finite value okay. And n, obviously is the perpendicular direction of the interface okay. So that can be calculated from the gradient of this volume fraction c okay. Now as time progresses will be finding out that the interface location is changing.

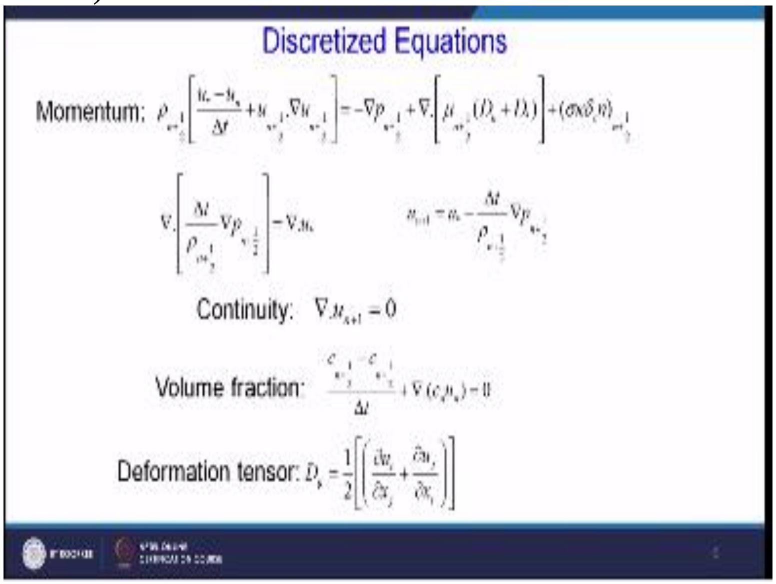
So obviously this value of c will be changing across the time. So you need to get some equation of c, which will be adverting from 1 place to another place along with the time. So we have this governing equation for conservation of volume fraction. So you see delt c /delt t + delta. So this is the special derivative delta c*u, where u is the velocity at that particular cell is equals to 0.

So with this equation we updated the value of volume fraction and calculate the corresponding density and go back to your continuity and momentum equation for solution okay. So altogether we have seen in volume of fluid in place of 4 equations as we have seen in our separated flow. Here we are getting actually 3 equations. 1 equation for continuity and 1 equation for momentum and along with that we are having 1 volume of fluid equation, which is basically your conservation of c equation okay.

Next let me tell you that what are the ways of discretizing these equations and you know using finite volume methodology how we can solve these equations. So there are various schemes for solution of these equations. But today I will be showing you 1 scheme, which is most efficient and which is very good from stability point of view. So it is actually staggered in time solution.

So here what I will be finding out? Let us say we are having 2 time levels, which is n and n+1. n is the present time level, where all the values of u Rho’s and c’s are known to me and we are supposed predict the next time level, which is n +1 okay. So in case of staggered time, we will be finding out apart n+1. We are supposed to find out the property value at n + ½. So, which is half time step?

(Refer Slide Time : 12:16)



So here I have shown you that from the momentum equation. What we do, we find out the values at n + 1/2. Now you see here all the nth tags are actually known to me. Only unknown tag is u* okay. Now this u* is actually some pseudo velocity at n + ½ time level. So what we do in this right hand side, you see everything is based on your nth time step okay.

So we find out the values over here using nth time step and then we get the value of u* but here you see, we are having some components like you know n+ ½ level pressure value and n + ½ level surface tension value. These values are once again calculated from extrapolation nth value okay. So you see, first we get the value of u* and then we put the value over here in the u *, in the right hand side and we get the updated value of p (n + ½).

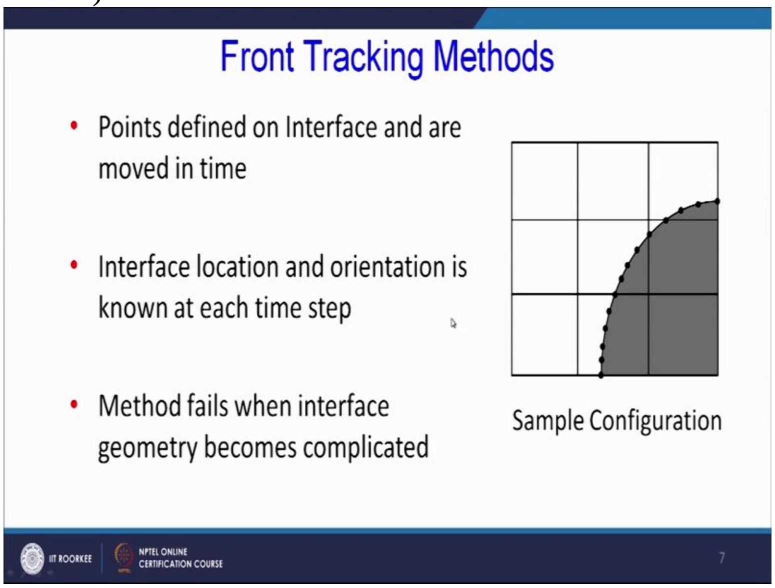
Now, once we get the value of p n+ ½, we can give it back to the momentum equation once again and we can find out the new value of u *. So this procedure continues and you will be finding out after sometime we will be getting some converged result of u * and p (n + 1/2). So once we get u* and p n+ ½ which is converged 1 using this continuity equation, we can find out u n+1 okay. Now remember this u n+1 is velocity as the next time level okay.

So once we have found out this u n + 1 you can quickly cross check whether this solution, whatever you have obtain there is converging or not or satisfying your basic equations or not by checking whether delt dot u n +1 = 0 or not. If you are not converge then once again you need to go back to the scheme and do the iteration to get higher accuracy okay. So once you get the value of u n+1, it is very easy to calculate the value of volume fraction at c n+1/2 okay because all the other values will be depending on the present time step or previous half time step that is history – ½ okay. n-1/2 okay and you can get the updated values of c okay.

So once you get the updated values of c then we have to go for interface reconstruction to calculate the surface tension force. So I will be coming that portion next. Now as we are having some volume fraction field, I will be showing you 1 figure over here okay. Before that let me tell you that this volume fraction whatever you are seeing that volume fraction, will be unique for unique cell. So you will be finding out that some where the volume fraction is 1, somewhere it is 0 and in some cells we are having intermediate that means non 0 and non 1 and intermediate value okay.

So that means all those cells having non 0 and non unity volume fraction value those will be actually occupied by the interface. So let us see now how we can constructing interface in that. So there are various methodology for interface tracking. The first methodology I will be telling you is front tracking method okay. In case of front tracking method, what we do in the previous time step?

(Refer Slide Time : 15:36)

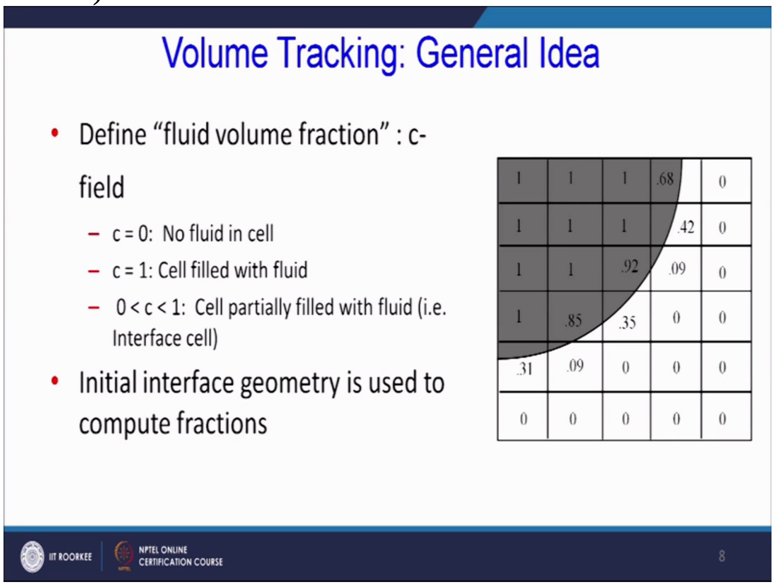


That means at nth time once we know the interfacial configuration on the interface, we put some particles like these okay. Here you see some particles we have given. Sometime we give those particles as uniformly distributed or you can give some heterogeneous distribution also depending on your problem.

So what we do, we give this particles okay. In the next time instant in case of front tracking methodology, we perform the calculation that based on the local velocity how for the particle will be moving okay. So next time instant will be finding out the future position of those particles and line connecting between these particles will be giving you new interfacial position. So this is a very good technology for some fluid flow problems front tracking is very useful. But you will be finding out difficulty for this type of problems whenever you are having strong flow velocities.

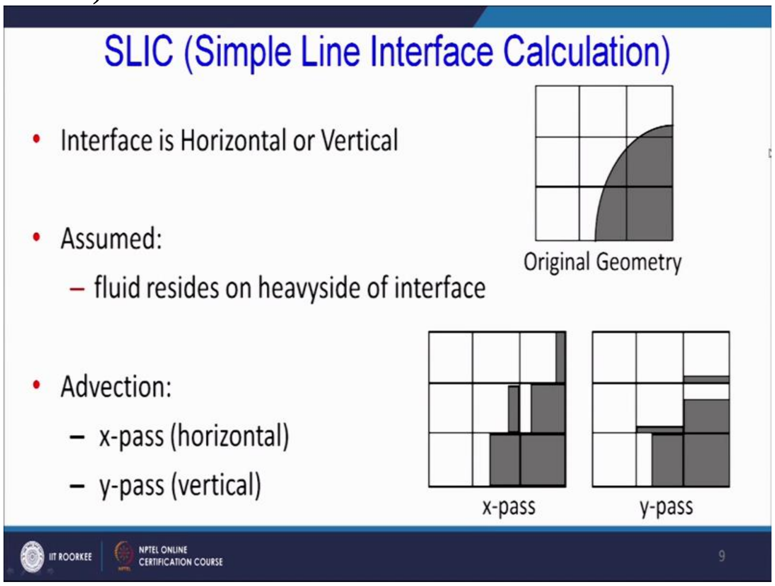
If there is strong flow velocity and fast change of interfacial configuration, you will be finding out that these front tracking particles will be advecting fast okay. And they will be coming far apart from each other and construction of interfacing between will be difficult okay. Next see another methodology which people are using nowadays this is called volume tracking. So in this way in this methodology, what we do, we get the volume fraction value of each cell. For example here I have shown you 1 figure, where you can find out that these cells are having 1 values okay. And other cells are having 0 values and in between we are having fractional value starting from .31 to .68.

(Refer Slide Time : 17:01)



So that means this fractional value tells us that the volume of this cell fractionally is occupied by this shaded colored fluid okay. So we know that through these cells only the interface will be passing. We have to satisfy the volume criteria of all these cells and we have to construct the interface. So this is the methodology, which we follow in volume of fluid. So there are various sub methodologies to construct the interface. This interface whatever I have shown you over here this is actually interfacial structure. So how to construct this, actually interfacial structure for that we are having some sub methodologies okay.

(Refer Slide Time : 18:08)



Some of the sub methodologies are like this first 1 is whatever I will discussing is called simple line interface calculation, which is called SLIC okay. So in case of SLIC what we consider that interface either horizontal or vertical. So it will be always in a cell, the interface will be line in the form of horizontal line or a vertical line okay. So we will not be imitating this curve linear path of the actual interface okay. So what we assume that in this type of SLIC. SLIC calculations we assume that fluid resides on heavy side of the interface okay. So there are 2 methodologies in the slic also. 1 is called x pass, another 1 is called y pass.

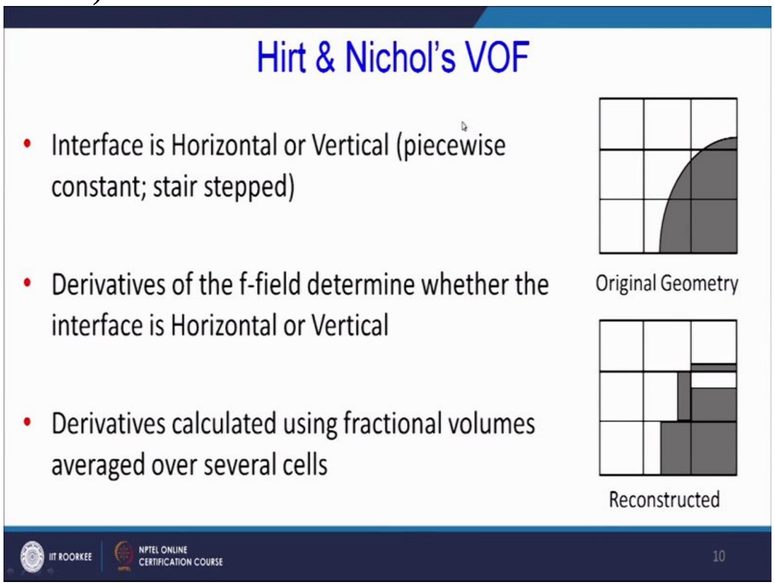
x-pass also considers that you are having interface perpendicular to horizontal line on the other hand y-pass considers that the interface will be always perpendicular to the vertical line okay.So here I have given you 1 schematic representation how using x-pass interface can be calculated or tracked and how using y-pass interface can be captured. So you see over here the volume fraction of this center cell is somewhere in between you know .2. So here you see by giving a vertical line we have given 20 percent of the shaded zone and here using a horizontal line in y-pass we have given 20 percent of the shaded zone okay.

So you can find out the volume fraction are same but as you go for y-pass and x-pass depending on the orientation of the interface, you will be getting different interfacial configurations okay. The same method is actually followed for the other cells also over here and here okay. Now if you are having well resolved interface then only this simple line interfacial calculation whether x-pass or y-pass will be giving you the accurate prediction of the interface. If the cells are not, well resolve then probably you will be finding out difficulty okay.

Apart from that, after this SLIC method, which I have not shown you over here there is another methodology called polynomial line interfacial calculation, which is called PLIC that is having provision for fitting the interface with some inclined line. Here I have shown in SLIC that we will be having always horizontal or vertical lines but in PLIC you will be having provision for setting up 1 inclined line. So in those cases from in that case from the neighboring cells, you will need to fit the value of the inclined line as the slope and intercept that means $y = nx + c$.

Where m and c will be found out from the neighboring cells volume fraction okay. So in those cases you will be finding out interface calculation will be little bit more near to the actual interface but complexity of calculation once again increases okay.

(Refer Slide Time : 21:08)



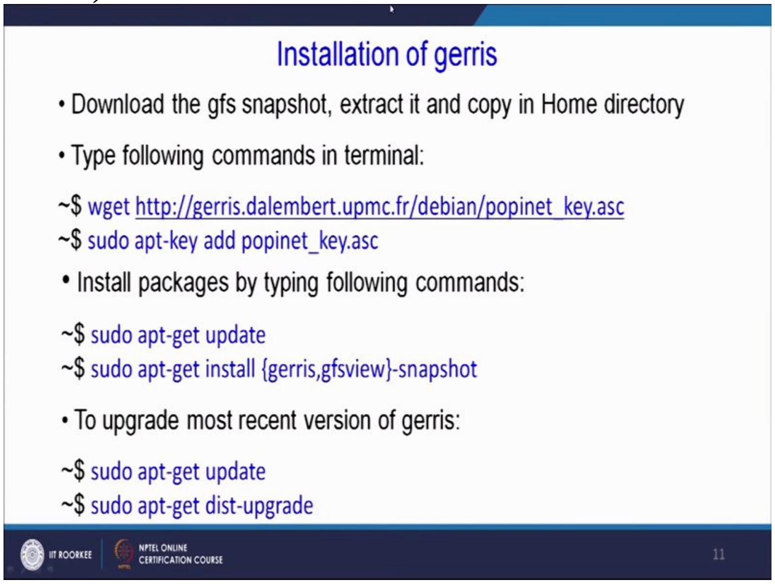
Next I will be showing you another 1 which is very famous, which is called Hirt and Nichols volume of fluid okay. In this Hirt and Nichols volume of fluid they considered that we will be having piece wise constant and stair stepped interface okay. Stair stepped interface means we will be having some sort of once again the horizontal or vertical lines.

But they have chosen mixture between the x-pass and y-pass. So depending on which 1 is suiting best with the neighboring cells interface they can swap in between x-pass and y-pass. So in the in the SLIC methodology, I have showed you that there will be same procedure followed throughout that but here in Hirt and Nichols VOF you will be finding out they are swapping in between depending on the neighboring cells interfacial direction okay.

So here I have shown you once again the similar type of figure. You see this was the original interface but using Hirt and Nichols you can replicate like this, which is more near to this 1 compare to your SLIC calculation x-pass and y-pass okay. Next I will show you okay. Before going to this slide, let me tell you that some advanced methodologies are also available. In case of volume of fluid, you will be finding out that a methodology is like height function and some advanced techniques are also available. Nowadays in volume of fluid which will be giving you idea of calculation of exact interface okay.

So we are not going to detail of those but here I will showing you 1 freeware, which is actually based on height function method okay and gives a better accuracy of the interface okay. So here I will be discussing about freeware Gerris okay. Now Gerris flow solver is a freeware and using this we can do 2 phase simulations involving volume of methodology. So let us first see how Gerris flow solver can be downloaded. So, to download Gerris flow solver, we have to go to for the website of Gerris which is nothing but http://gerris.dalembert.upmc.fr okay.

(Refer Slide Time : 23:09)



So this Gerris website repository is actually maintained by University of Pierre and Marie Curie. So from there you can download okay. You we require actually 3 snapshots rather here I will be showing you 2 snapshots Gerris and gfsview. Gerris is doing the basic calculation whatever procedure I have told you and gfsview is actually gives you the visualizations options okay. So what you can do in your command prompt or terminal. You can write down this command w get and then the website id over here okay.

And then you can write down sudoaptkey add popinet _key.asc. This will be enabling you to download the Gerris software okay. You will repository so you will be finding out the key being downloaded and then in the command prompt you can give the update sudo apt, get update, do not forget to change your repository to the universal repository okay before doing this updation. Now once you updated, you have updated then you can type sudo apt get install Gerris, gfsview-snapshot.

This will be actually downloading and installing present stable version of Gerris as well as gfsview okay. Now to upgrade to the most recent version okay, what you can do, you can go through 2 steps. sudo apt get update and sudo apt get dist upgrade. So it will be upgrading to the newest version okay whatever available okay. Next I will be telling you something about geris flow solver. Gerris flow solver is actually giving you freedom of discretizing the domain into a ______ or _____ (25:11) format okay.

So what is ______ or ___format? If the domain is having, you know rectangular parallel piped form then it will be discretizing in the domain using this cube okay. So where we are having 6 faces okay. All the faces are having directional nature okay. So it is something like front, back, left, right and top and bottom okay and then what we will be doing this boxes whatever we have created this boxes, will be actually discretized further.

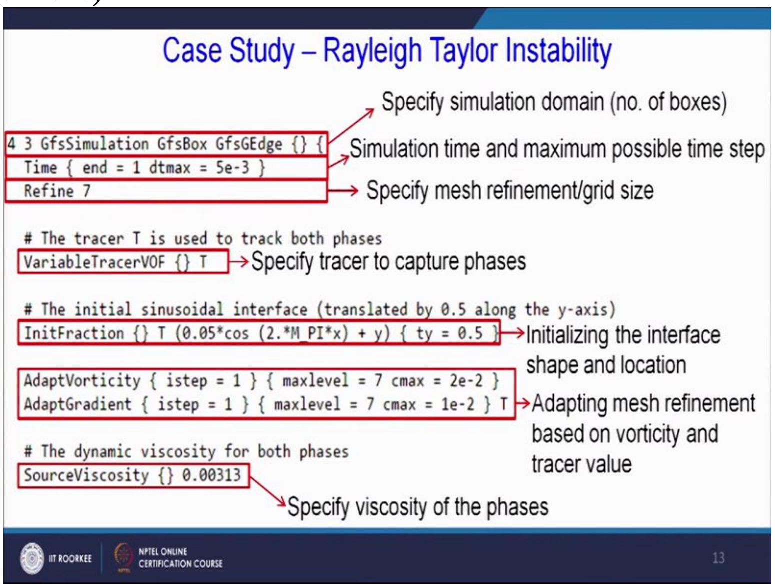
Okay, what type of discretization we can get? Let us say I have given you an example in 2d. This will be followed in 3d also. Let us say this is a front face of this box whatever I have shown in 2d.
So this is called as level 0. So if you go for 1 level of refinement. So this cell this box will be actually discretized into 4 sub boxes okay. In case of 3d, it will be 8 sub boxes. And level 2 the small sub box will be further divided into 4 okay.

You will be finding out 16 sub boxes over here and continuing like this. You can increase the number of cells in your domain okay. Now Gerris gives you some option of adaptive meshing. Adaptive meshing means wherever necessary, there you will be making your grids final and wherever not necessary you can go for cores meshing okay. So depending on some criteria there are choice of criteria.

So you can select based on your interface location. You can select based on any other property. So here I have shown you 1 example of adaptive meshing. So here you see this was the zone of interest for the present problem. So, what we have done, we have actually refine this okay and elsewhere we are having cores meshing. So this essentially actually saves your computational time okay. Next, let me show you 1 case study using Gerris and this case study will be dealing about Rayleigh Taylor instability.

So we know Rayleigh Taylor instability is nothing but if you are having 2 stratified flow and in the upper layer we are having heavier fluid then you will be finding out that it is forming a finger in the in the lower fluid. And then depending on the surface tension the finger will be forming several structures okay. So, that we have solved over here involving 2 phase volume of fluid method okay.

(Refer Slide Time : 27:46)



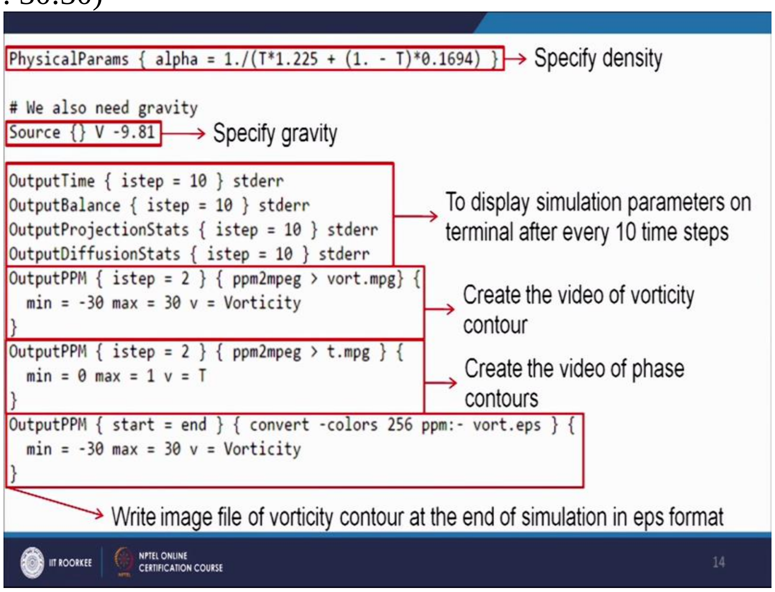
So first I will be showing you some important features of the script. What we need to write down, so here you see first we have written 4 3 gfssimulation, gfsbox, gfsgedge. Now this 4 3 means we are having 4 boxes. Already I have told you what is the meaning of a single box. So which will be nothing but a cube in 3d and a square in 2d. So we are having over here 4 boxes okay. So 4 boxes, how they will be placed I will be showing you at the end.

Then we are having time end. time = 1 and dtmax is some sort of idea what can be your maximum time step okay. So maximum time step we have chosen a 5 *10 to the power of -3. Now refinement I have kept as 7. What is refinement level, already in the next previous slide I have shown you here we are choosing the refinement level 7 okay. Now for volume fraction whatever I have shown you in the equation as c, we use a tracer of volume of fluid as t over here, capital T. So T = 0 and 1 will be in the bulk and in between values will be at the interface okay.

Then we can place the initial configuration of the interface. So here what I have done using a C++ formulations. We have written init fraction t and we have given here some sort of expression. If the expression comes out to be positive then t will be getting 1 and if the expression comes out to be negative then t will be getting 0 values okay. Depending on the value of x and y, which is the spacial coordinate you will be finding out t is getting some different values of 0 and 1 okay. Then you have the provision for you know translating the interface. Also interface means the intermediate cells, intermediate values of t okay that you can translate in this fashion okay.

Then I told you that we are having provision for adaptation grid adaptation here. Also we are doing some sort of adaptation you see we are adapting based on vorticity. So vorticity of the field will be calculating and wherever vorticity is more they are will be adapting in the finer cells. And wherever vorticity is less, will be going for the cores mesh. Similarly we can do vorticity based on gradient of some parameter. Here the gradient we have taken for t. for t is nothing but your volume fraction okay. Here you see for adaptation we have written the maximum level will be going to 7 and then we can reduce further depending on the cost of computation okay.

(Refer Slide Time : 30:30)



So, then we have given the source viscosity. So, viscous term will be added over here in Gerris source. So source viscosity is this value. So this in this problem we have considered both the fluids are having same viscosity okay. Next slide you see over here, we have given the densities of the fluid depending on the value of t. So we have given alpha which is nothing but 1/ Rho. So what we have given alpha = 1/ T* 1.225 + (1 – T) .1694. We considered that the lighter fluid is having density .1694 and heavier fluid is having 1.225.

You see over here if T=1 then you will be finding out alpha becomes 1 /1.225 and T= 0 essentially gives you 1 / .1694 okay. So in this way we have defined the densities. As it is

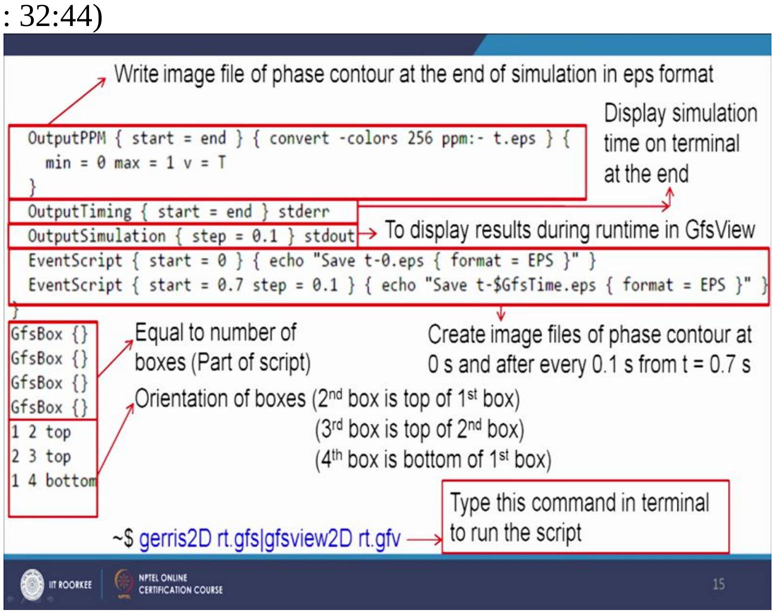
Rayleigh Taylor instability, so gravity needs to come otherwise the finger will not be forming. So we have given the gravitational acceleration as the source term over here.

Source and the gravity will be acting in the downward side. So, v is the y coordinate velocity of the y coordinate and with a value of – 9.81. So which is the downward direction. Then here there are having some loops for output. So we can have the output files a different time level okay. We can have output time, output balance of different residuals we can have output projection stats.

So, statistics of statistical nature of the values different values. We can have output diffusion stats a different time level. We can write down those things okay. Now here in this 2 blocks, I have shown you how you can see or create the results okay. For example here we have first made some snapshot of the vorticity contour. So you see the field we have kept as v = vorticity and vorticity limit we have kept as -30 to 30 and after every time, 2 time interval we have created the vorticity image. And using ppm to .mpeg.

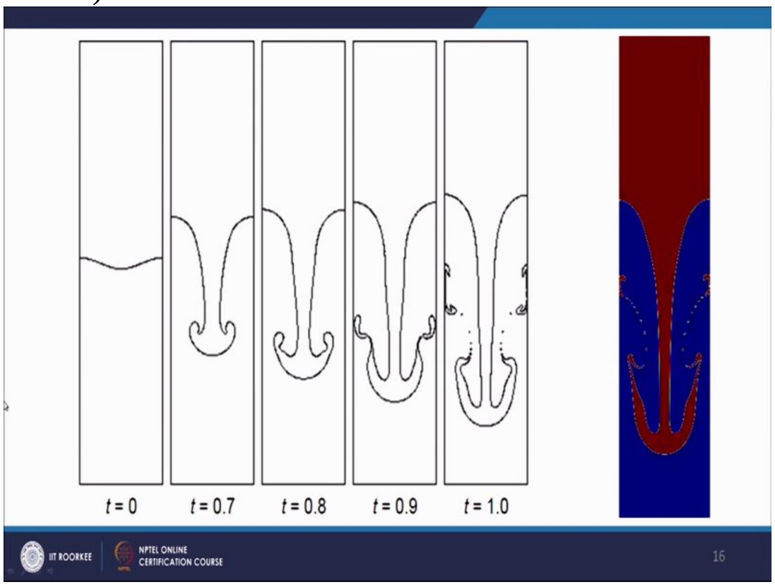
So which is nothing but you snapshot to the movie, we have created a movie called vor .mpg in a similar fashion for the volume fraction we have created the movie t.mpg okay. And at the end we are also writing the value of the vorticity over here in a separate file okay. Next I will be showing you that how to create, how to create the boxes as I have shown you over here. Here you see we had 4 boxes. So gfsbox, gfsbox 4 times I have written.

(Refer Slide Time : 32:44)



So this signifies we are having 4 boxes. Now how the boxes are placed over each other. So we are having 4 boxes. So first, second box will be placed on top of first, third box will be placed on top of second box and fourth box will be placed at the bottom of your first box. So the orientations of the boxes are given over here okay. Some results I have shown you over here you see we have started with this kind of interface. If you remember the unit fraction whatever we have given that was having a cosine nature.

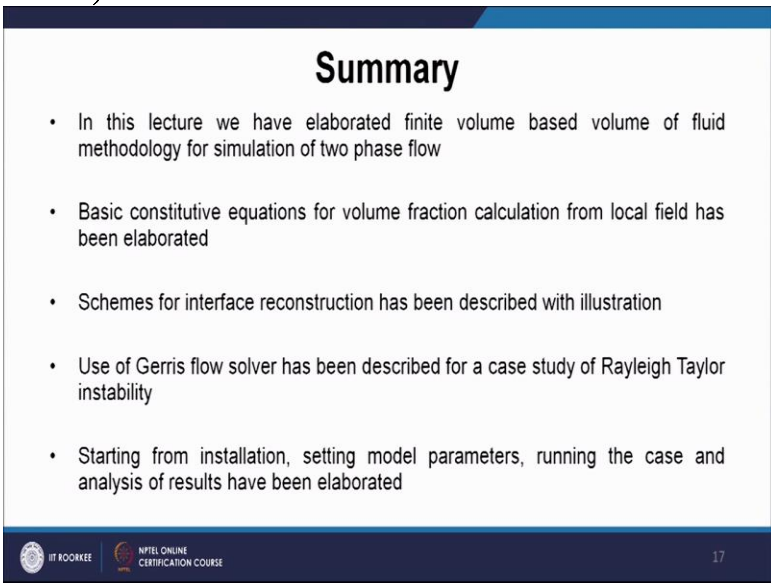
(Refer Slide Time : 33:27)



So at the beginning, we will be having a cosine nature of the interface and due to the gravity and density difference will be finding out the finger is forming and due to surface tension, you will be finding out this type of structure we are getting as time progresses. Here I have shown you a movie, which you can find out in real time how the finger is forming and interface is being captured okay.

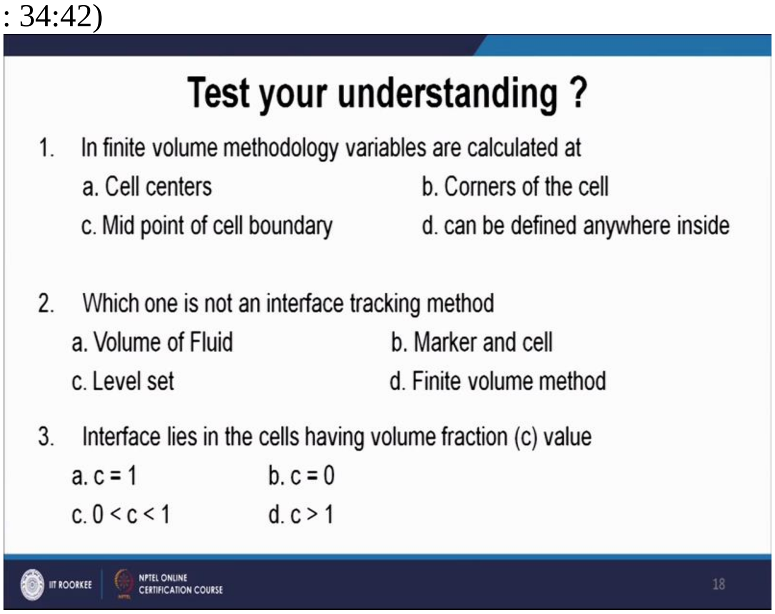
Even very tiny droplets also we can capture in this type of situation. You see how many, how small droplets we are capturing over here using volume of fluid okay. So at the end of this lecture, let us summarize. So in this lecture what we have done, we have elaborated finite volume based volume of fluid methodology for simulation of 2 phase flow.

(Refer Slide Time : 34:04)



Basic constitutive equations for volume fraction calculation, we have dealt with .We have given you a different schemes for interface reconstruction with illustration, we have explained and we have also shown you freeware Gerris flow solver and we have studied a case study of Rayleigh Taylor instability. Starting from the installation, setting up the model parameters and running the case I have shown you over here okay.

(Refer Slide Time : 34:42)



So at the end of this lecture, let us once again test your understanding, we are having 3 questions here. So first question, in finite volume methodology variables are calculated at. 4 options we are having, cell centers, corners of the cell, midpoint of cell boundary and can be defined anywhere inside the cell okay.

So already I have told you finite volume the centroid or center of gravity is the node point. So, correct answer is cell centers okay. Next which 1 is not an interface tracking methodology? 4 methodologies I have named over here. Volume of fluid, Marker and cell, Level set and finite volume methodology. So here you see part d is actually discretization methodology of the domain volume of finite volume methodology. So definitely that is not the correct answer okay.

Then interface lies in the cells having volume fraction c value. So which we have seen that we are having several options for volume fraction c. So first option is $c = 1$, $c = 0$, $0 < c < 1$ and last option is $c > 1$. We know that last option is nowhere possible and this a and b those are actually for the bulk. So obviously the correct answer is in between 0 and 1 okay. So with this I end this lecture, thank you.