

Adiabatic Two - Phases Flow and Flow Boiling in Microchannel
Prof. Mr. Mrinmoy Dhar
Prof. Mr. Abir Chakravorty
Department of Mechanical Engineering
Department of Chemical Engineering
Indian Institute of Technology, Kharagpur

Lecture-25
Tutorial II

Hello, I am Mrinmoy Dhar; I am a researcher from mechanical engineering department IIT Kharagpur. Now I am going to briefly discuss about CFD and CFD in multiphase flow CFD full form computational fluid dynamics.

(Refer Slide Time: 00:34)

Computational Fluid Dynamics (CFD)

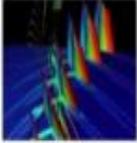
◆ **What is CFD?**
Analysis of problems involving fluid flow - with simultaneous heat flow, mass transfer, chemical reactions, phase change, etc. - by computer based simulation

◆ **Why CFD???**

- Ability of simulating real condition (where performing experiment difficult)
- Gives deeper insight into flow physics
- Helps in better design & improving performance
- Relatively inexpensive



A von-Karman vortex street being shed from a cylinder
(Source: <http://homepages.engineering.auckland.ac.nz/~ame007/90ms/vonkarman.gif>)



Simulation of the Flow in the Transonic Compressor Rotor 37, 2007
(Source: <http://surface3/galleryofimages/>)



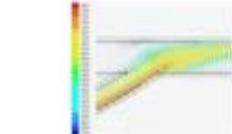
Flow around a cyclist and his bike
(Source: <https://selfmade.files.wordpress.com/2013/01/planeto-bike.jpg>)

So, first question arises in mind what is CFD? CFD is basically the analysis of fluid flow problems, which may simultaneously involve heat transfer, mass transfer, chemical reactions, phase change any transport phenomena, and that would be done by computer based simulations. So, next question comes why CFD? CFD has actually ability of stimulating real conditions; it can elucidate information even when performing experiments are difficult. One may argue that CFD has the strength and that it can generate and data spread all about the composition domain, which gives deeper insight into flow physics and helps in better design, improving performance efficiency, another thing about CFD is that, it is relatively inexpensive; that means, if you are having a one good computer you can go for CFD simulation.

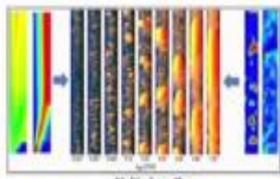
(Refer Slide Time: 01:37)

◆ **Applications of CFD:**

- > Aerodynamics of aircrafts and vehicles
- > Hydrodynamics of ships
- > Biomedical engineering: blood flow through arteries & veins
- > Chemical processing: mixing, separation, polymer molding
- > Multi phase flows
- > Power plant engineering: combustion
- > Electronic engineering: cooling of microcircuits
- > Flows in rivers, oceans, open channels
- > Flows inside turbomachinery
- > Weather prediction
- > Marine engineering



Blood flow through an arterial connection
(Source: <http://www.researchgate.net/publication/318110471>)



Multi phase flow
(Source: <http://www.gutenberg-europe.com/research-areas.html>)

So, next question comes when CFD? Based on this numerous advantage of CFD, CFD has got lots of wide spread applications, starting from physical problems occurring in our daily day life, it has got a lot of applications in industrial applications also, here are few applications common applications as listed aerodynamic of aircrafts and vehicles hydrodynamic of sheets, biomedical engineering for blood flow through arteries and veins, chemical proceeding for mixing, separation polymer molding, in multi phase flows also CFD has got lots of applications. Here we can see that, define flow patterns during multiphase flow obtained by CFD simulations, and here are other applications of CFD.

(Refer Slide Time: 02:28)

CFD Processes:

- ❖ **Pre-processor**
 - Definition of problem geometry: computational domain selection
 - Meshing: division of domain into sub-domains (mesh of cells)
 - Selection of the phenomena to be physically modelled
 - Specification of fluid properties
 - Definition of initial conditions and appropriate boundary conditions
- ❖ **Solver**
 - ↳ (Finite Volume, Finite Element, Finite Difference method)
 - Finite Volume Steps:**
 - Control volume integration: governing equations of the fluid flow integrated over all the control volumes of the domain
 - Discretization: integral equations converted into a system of algebraic equations
 - Solution: algebraic equations solved iteratively
- ❖ **Post-processor**
 - Analysis & Visualization of solution: velocity vectors, Contour plots, Streamline patterns, etc.

Next question how CFD? For CFD simulations we are having many commercial available software like Mc Cfs, Mc fluent Consol star Cd. fellable software's like jurist open form and you can go for developing your own code in house code, but a in one thing common for all the packages are that all 3 contains 3 elements like Pre processor Solver post processor. In Pre processor you have to first define in the problem geometry computational domain selection, next is we have to divide the whole domain into sub domains is meshing; then have you to select the phenomena which will be chemically or physically modeled. Then we have to specify the fluids which are involved in the problem and you have to obviously, define the initial conditions means pressure velocity at initial times and very carefully we have to define boundary conditions means pressure velocity at boundaries means wall inlet outlet.

Once this Pre processing is completed, you have to go for solver. For solver different methods are there, a finite volume, finite elements finite difference finite volume is a most powerful and accurate, much commercial as well as fellable software use this, I am going to discuss about finite volume steps. In finite volume steps it first integrate the governing equations of the fluid flow, over all the control volume of the domain; and then it discretize the integral equations for converting in to a system algebraic equations and this algebraic equations we can solve iteratively and after the solution is completed you go for post processing, by doing CFD simulation we can analyze an dissolution of solution like velocity vectors, analysis contour plots streamline patterns and many others.

(Refer Slide Time: 04:27)

CFD in Multiphase Flow:

Navier-Stokes equation for two-phase flow:

$$\rho_i \left(\frac{\partial U}{\partial t} + \nabla \bullet U U \right) = -\nabla p + \nabla \bullet (\mu_i D) + \rho_i g \quad i=L, G \quad U = (u, v)$$

Deformation rate tensor: $D = \frac{1}{2}(\nabla U + \nabla U^T)$

Continuity equation: $\frac{\partial \rho_i}{\partial t} + \nabla \bullet (\rho_i U) = 0$

If fluid are assumed incompressible: $\nabla \bullet U = 0$

Balance of normal stress at the interface: $[\rho_i] = [\mu_i D \bullet n] + [\rho_i g] \bullet n + \sigma k$

Balance of tangential stress at the interface: $[\mu_i D \bullet t] = [\rho_i g] \bullet t$

n normal to the interface
 σ surface tension coefficient
 k curvature at the interface
 t tangent to the interface

Our focus is on multiphase flow, so let us talk out CFD in multi phase flow, has the Navier stokes equation for 2 phase flow, this same equation will be served over the whole domain, here flow is the density, U velocity field, p pressure, mu kinetic viscosity, g gambit show acceleration. This i stands for different phases and this D is the deformation rate tensor i is the equation for deformation rate tensor, next is the continuity equation, these also gone for old domain, if the fluid is assumed incompetent means ways constant for a particular fluid, that time this continuity equation will become this and this multi phase flow, we have the interface between phases. At the interface balance of stresses are very important; here is the balance of normal stress at the interface, this equation. Here this n term is new, n is the normal to the interface, then sigma is the surface tension coefficient, and k is the curvature at the interface, and for transition stress balance at the interface, this is the equation. At t is the tangent to the interface, these are the equations.

(Refer Slide Time: 05:45)

Interface Capturing Method:

- ◊ Volume of Fluid (VOF) Method
- ◊ Level Set Method

Volume of Fluid (VOF) Method

Volume fraction function $\rightarrow f$

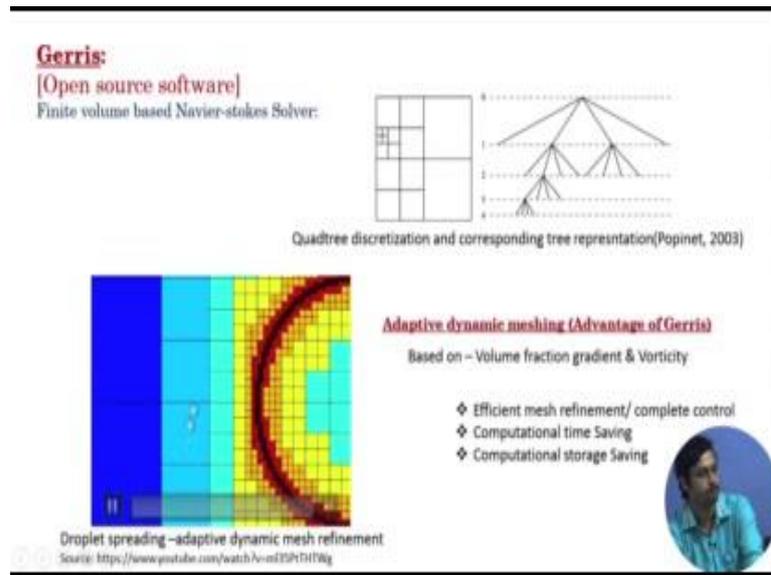
$$f = \frac{\text{Volume of liquid in a computational cell}}{\text{Volume of cell itself}}$$

= 1 for cells containing liquid
= 0 for cells containing gas
 $0 < f < 1$ for cells containing the interface

$$\frac{\partial f}{\partial t} + \nabla \cdot (fU) = 0$$

Another thing go to multi phase simulation is the, interface capturing. For interface capturing there different methods, such as Volume of field method here and Level set method here, I am going to discuss about volume of field method. Volume of field method what you do, we define a volume fraction function f , which is nothing but liquid fraction function. A cell may contain only liquid that times this f will get the volume 1. And cell may also contain only gas, that times this f will become 0. Means there is no liquid and cell may also contain both gas and liquid; that means, this cell is contained by interface, that time this f value will lie between 0 to 1 and this f will follow this advection equation.

(Refer Slide Time: 06:33)



So, let us talk about software which simulate multi phase flow, Gerris is open source software, freely available. And it has some unique advantages, so, I am going to discuss about Gerris. This Gerris is a basically finite volume based Navier stokes Solver; it has the advantage that it uses adaptive dynamic machine, so let us see how it adopts dynamic machine.

Actually Gerris what it does, it usually contains square boxes for 2d simulation and q boxes are 3 d simulation. Suppose this is our compressional domain, it will create number of boxes based on the user. So, here you can see that defilement levels, these are defilement levels, at 0 defilement level, 1 box will be sub divided onto 4 boxes and in the next defilement level, any of the box can again subdivide in to another 4 boxes, here we are seen that initially once box has been divided into 4 boxes, and the next step any box whenever required can divide again into 4 boxes, cells. This way it will continue, this is actually called “quadric discretization” using this quadric discretization, Gerris adapt dynamic machine.

In this way I was talking about whenever requirement, this requirement will be based on volume fraction gradient, whenever in the simulation there will be a gradient of volume impression means near the interface and near the wall, there in this positions, it will go for high dissolution machine. And another criteria is viscosity, when in the simulation there will be the generation of viscosity, that time generation go for high dissolution

machine; this adaptive dynamic machine has advantage that, it is efficient and having compete control and it saves computational time as well as storage. Here we can see that liquid droplet is spreading our net. So, this is the interface, here we can see that around the interface, there is a high dissolution machine. Whereas in the other areas there is course machine, we can also see that and this high recitation machine is evolving with the abolition of the interface, these actually adopting dynamic machine.

(Refer Slide Time: 08:57)

Gerris -

Uses Volume of Fluid (VOF) methodology for tracking interface in two phase flows

Advection equation for volume fraction (f): $\frac{\partial f}{\partial t} + \nabla \cdot (fU) = 0$

$\rho(\bar{f}) = \bar{f} \rho_1 + (1 - \bar{f}) \rho_2$ $\bar{f} \rightarrow$ Filtered variable
 $\mu(\bar{f}) = \bar{f} \mu_1 + (1 - \bar{f}) \mu_2$ Evaluated by averaging eight neighboring values of f

In this Gerris uses the volume of fluid methodology, for interface tracking, for as you know that. volume of fluid we defined volume of fraction function f, this is the equation I would already mentioned this, and this as the equations for generalized fluid density and viscosity; here one term f bar is defined, this is filtered value variable; this is actually used for high density ratios, suppose air water, here the density ratio is almost 1000. So, there we use f for filter variables, evaluated by averaging over the 8 neighboring values of f.

Suppose here we can see, if we are going for the assign the f value of this cell. So, it will go for averaging of this neighboring edge, middles of f, and by that using this f bar it will calculate the generalist fluid density; when cell will be contain only liquid, that time this part will become 1 and it will come get the density of liquid and when cell contain only gas, that time these will part become 0 and it will get the density of gas, same thing for viscosity also.

(Refer Slide Time: 10:11)

Discretized equations:

$$\rho_{n+1} \left[\frac{U_n - U_{n-1}}{\Delta t} + U_{n+1/2} \cdot \nabla U_{n+1/2} \right] = \nabla \cdot \left[\mu_{n+1} (D_n + D_{n+1}) \right] + (\sigma k \delta, n)_{n+1/2}$$

$$\frac{f_{n+1} - f_{n-1}}{\Delta t} + \nabla \cdot (f_n U_n) = 0$$

Subscripts indicate timestep

$$U_{n+1} - U_n - \frac{\Delta t}{\rho_{n+1}} \nabla p_{n+1}$$

$$\nabla \cdot U_{n+1} = 0$$

Poisson equation governing pressure field: $\nabla \cdot \left(\frac{\Delta t}{\rho_{n+1}} \nabla p_{n+1} \right) = \nabla \cdot U_n$

These are the discretized equation, obtained by time splitting projection method. First term is the temporary term, second is the conductive term, then in the right side first term is the discretized discuss force, second term is the discretize surface tension force. Here subscript are the times steps, this star is actually auxiliary stand step and n plus half is the intermediate stand step. The next is the discretized equation for this volume fraction function and a next equation is for pressure, we can see that in the first momentum equation, we have not considered the pressure term, but this pressure term has been considered here, here we have considered pressure for correcting the velocity fluid; and that resulting velocity will satisfy this continuity equation. This last this last two equations, we get Poisson equation which governs pressure field; this is the equations.

(Refer Slide Time: 11:25)

```
Example Code of Rayleigh-Taylor Instability
4 3 GfsSimulation GfsBox GfsGEdge {} {
  Time { end = 1 dtmax = 5e-3 }
  Refine 7

  # The tracer T is used to track both phases
  VariableTracerVOF {} T

  # The initial sinusoidal interface (translated by 0.5 along the y-axis)
  InitFraction {} T { 0.05*cos(2.*M_PI*x) + y } { ty = 0.5 }

  AdaptVorticity { istep = 1 } { maxlevel = 7 cmax = 2e-2 }
  AdaptGradient { istep = 1 } { maxlevel = 7 cmax = 1e-2 } T

  # The dynamic viscosity for both phases
  SourceViscosity {} 0.00313

  # This defines the inverse of the density of the fluids as a
  # function of T
  PhysicalParams { alpha = 1./(T*1.225 + (1. - T)*0.1694) }

  # We also need gravity
  Source {} V -9.81
}
```



Simulated Rayleigh-Taylor Instability

- Two fluid of different densities
- A sinusoidal interface separates two fluids
- Heavier fluid is on top
- Mushroom shaped instability develops in time

Source: <http://gerris.daleahbert.upon.fr/gerris/example/example/rt.html#code6>

So, here the all the discretized equation, Gerris based on this quadric discretization; use Gerris multi gate solver and solved it have to be by Gauss Seidal Iteration method.

So, in this Gerris actually offered some lenus machine, for writing the pre percent steps like computer domain selection machine, boundary conditions; these things we have to write syntax for Gerris here in a dot GFI. So, I will you show you how to write this syntax file, for that I have taken one example from Gerris means insight here, this one example of Rayleigh Taylor, instability in Rayleigh Taylor instability what happens 2 fluids are there of different density and they are separated by sinusoidal interface. Here is that at the top there is a heavier fluid, you can see that the simulation of Rayleigh instability, and the heavier as a heavier fluid is at the top. So, for this is unstable condition and this is Mushroom shaped instability will develop in time.

(Refer Slide Time: 12:39)

```
OutputTime { istep = 10 } stderr
OutputBalance { istep = 10 } stderr
OutputProjectionStats { istep = 10 } stderr
OutputDiffusionStats { istep = 10 } stderr
OutputPPM { istep = 2 } { ppm2mpeg > vort.mpg } {
  min = -30 max = 30 v = Vorticity
}
OutputPPM { istep = 2 } { ppm2mpeg > t.mpg } {
  min = 0 max = 1 v = T
}
OutputPPM { start = end } { convert-colors 256 ppm - vort.eps } {
  min = -30 max = 30 v = Vorticity
}
OutputPPM { start = end } { convert-colors 256 ppm - t.eps } {
  min = 0 max = 1 v = T
}
OutputTiming { start = end } stderr
OutputSimulation { step = 0.1 } stdout
EventScript { start = 0 } { echo "Save t-0.eps { format = EPS }" }
EventScript { start = 0.7 step = 0.1 } { echo "Save t-5GhTime.eps { format = EPS }" }
}
GfBox ()
GfBox ()
GfBox ()
GfBox ()
1 2 top
2 3 top
1 4 bottom
```

Source: <http://gerrie.dalambert.opencfd.fr/gerrie/example/example/vt.html#toc6>

So, for writing the syntax of this problem, here 4 boxes are used and they are having 3 connecting edges, at the end of the code there is a written, how the boxes are oriented. See here we can get one 2 top means, box 2 is at the on top of box one, and then two three top means, 3 is top of box two and again written 1foot bottom means box 4 is bottom of box one. So, these are the orientation of boxes selected, at the start of these centers. GFS simulation is Solfer solved, at this is syntax for starting and this time end equal to one means it real time of one second the simulation will complete and ditimacs of the time step, this difference 7 is the already I have mentioned that refinement level, is the 7 refinement level for initial time.

This variable tracer VFD is the VOF of the quotient fraction function, and this is the initial air condition of the interface by defining this t condition at the separe this interface condition has defined by this line and as I mentioned that by Vorticity Titio you can define refinement level here by define based on vortices, it will take them 7th refinement level. Here you can change the refinement level based on your study in (Refer Time: 13:59) a specific problem, next in the based on the gradient of this bear means gradient of this volume fraction, you can choose the refinement level. Here in this case, viscosity of same; this is the syntax writing viscosity then this physical paramus alpha is this is for writing the 1 by density, 1 by row is the 1 by 2 into the density one phase and plus 1 minus 2 into density other phase, this is the syntax for writing the gravity source ton v minus 9.81.

Here minus because this v means v is in the upward direction, but gravity is acting in the downward direction. v minus nine point eight one. These are the steps for generating output simulation files and here nothing to specify about the boundary condition, so I have taken another problem, where I will show the how to implement boundary condition in Gerris.

(Refer Slide Time: 15:11)

```

GfBox {
  left = Boundary {
    BcDirichlet P 0
    BcDirichlet V 0
    BcDirichlet W 0
    BcNeumann U 0
    BcNeumann T 0
  }
  top = Boundary
  bottom = Boundary
}
GfBox {
  top = Boundary
  bottom = Boundary
}
GfBox {
  right = Boundary {
    BcDirichlet P 0
    BcDirichlet V 0
    BcDirichlet W 0
    BcNeumann U 0
    BcNeumann T 0
  }
  top = Boundary
  bottom = Boundary
}
1 2 right
2 3 right

```

impose symmetry conditions on top and bottom boundaries
and inflow/outflow conditions on the left and right boundaries
(so that emitted gravity waves can leave the domain cleanly)

Dirichlet Boundary Condition: value of a given variable on the boundary
Neumann Boundary condition: Value of the derivative normal to the boundary of a given variable

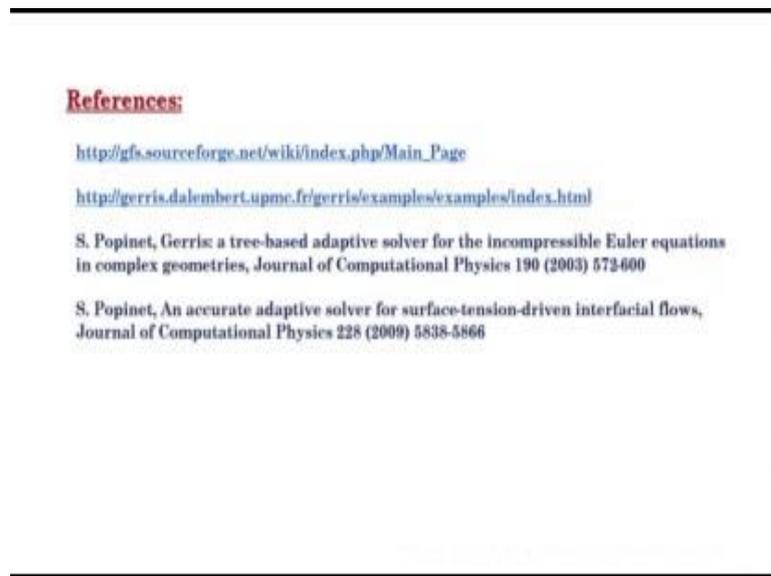
Source: <http://gfs.sourceforge.net/example/example1/disp/disp.gfs.html>

Here in this problem is the A out of around on the astragal, in this problem there are 3 boxes, 3 boxes are oriented like this. Here three boxes are box 2 a is right side a box 1, and then two three write box 3 at the right side of box 2. So, for first box gf box left boundary, Left Boundary condition means, left this is the left boundary for first box, at this position boundary condition BCG is row, here I would like to mention that, in mathematics dericted boundary condition means you are specifying the value of a given variable on the boundary and another condition is just Normal Boundary Condition, the normal boundary condition you are specifying the value of the derivative normal to the boundary of a given variable. Let us see this is $p = 0$ means, here you mention pressure P equal to 0, derict $d_0 d = 0$ these are velocity components $w = 0$ and then this in normal d_0 . If this is the x direction and this is the boundary. So, they had $\text{Del } x$ equal to 0, derivative normal to the boundary; $\text{Del } u \text{ Del } x$ equal to 0.

Next condition was bcn normal $t = 0$ from, now $\text{del } t \text{ del } x$ equal to 0. So, for other boundaries are top equal to boundary, here it is a cemetery boundary condition in this

problem. So, just to write boundary, in bottom also boundary this are Cemetery boundary conditions. Now going to the second box, in the second box again these 2 are connecting edges and top and boundary, symmetric boundary conditions. So, just to write fully boundary and bottom fully boundary; Now going to the third box, third box right side this boundary condition you to specify, same way using that dedicate and normal boundary conditions here boundary conditions are specified, and (Refer Time: 17:24) and top end bounded, top end bottom there are symmetric boundary conditions.

(Refer Slide Time: 17:36)

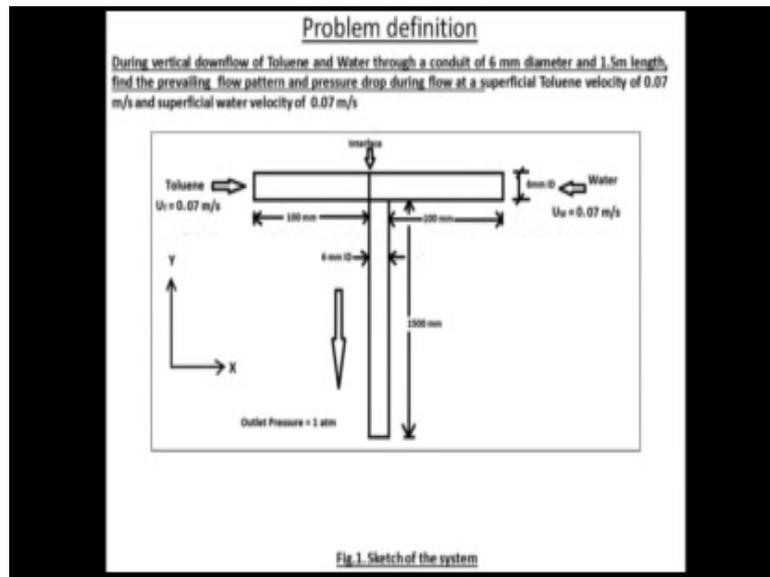


This is the way to implement boundary conditions in the Gerris for further details about implementation of Gerris please visit Gerris main page this the Gerris website and you can go through syntax of the many examples given in this page and for the structures of Gerris you can go through this two channels now mister. Abir Kapoor will demonstrate how to implement a new a multi phasal problem in computer software.

Thank you.

Hello everybody I am Abir Chakravorty I am research scholar in IIT, Kharagpur, Chemical engineering department, working in the field of multi phase flow. Today I am going to illustrate a problem of for two phase flow, in which liquid-liquid two phase flow through a millimeter size channel, the geometry which I will be doing a simulation work will be a t geometry.

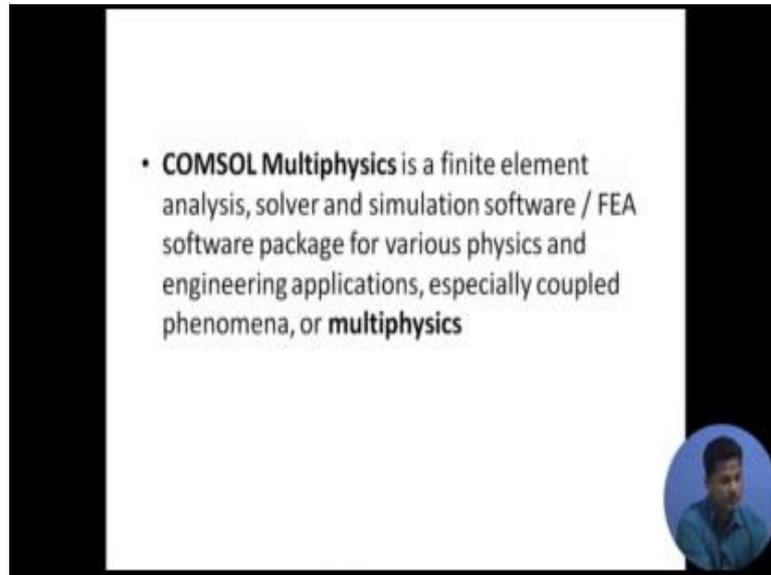
(Refer Slide Time: 18:37)



So, our problem is this is a geometry t shape millimeter channel Size, and we have we are having at two phase flow down flow situations and the (Refer Time: 18:53) which will be using are toluene and water toluene is in a organic liquid water (Refer Time: 18:58) then the thing which will be demonstrating to all of you today is that with given conditions such as the innate superficial velocity of toluene, which is 107 meter per second, and inlet superficial velocity water once the same 0.07 meter per second given and the outlet pressure in down flow case the my outlet will be this, outlet discharged at one atmospheric pressure, we will be calculating what will be my flow pattern acumen the flow pattern in this scenario in this given set of condition, and also extracting some results in the form of graph; which will be pressures the verses the length of the conduit.

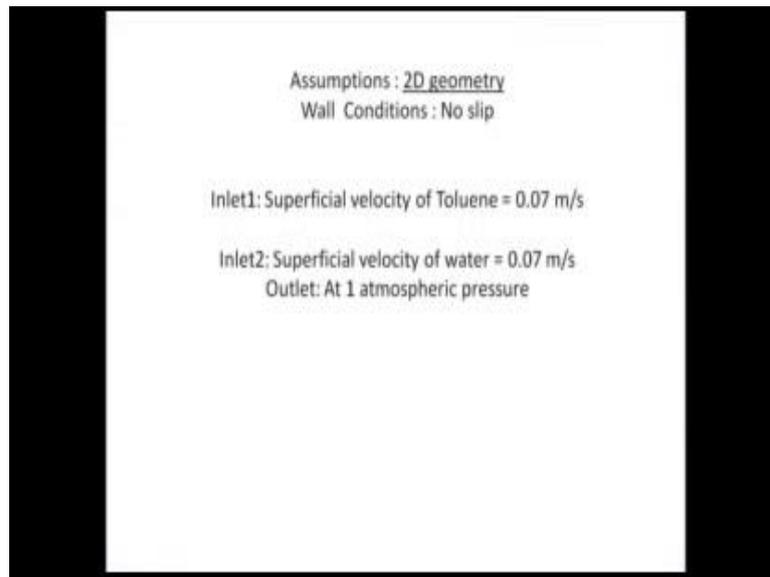
So, here is a down flow, these are two inlets and the 6 mm id channel is selected, and the length of this channel is 1.5 meter, 1500 millimeter with side branches at 100 millimeter. So, my dimensions is y, this is the y and this is x, ys projecting towards the approx the length of the conduit section. So, as we can see this is the down flow situation, the thing which will be doing is that we do not know about what is the flow pattern will be happening in this condition and also what will be the pressure drop across this conduit. So, we will be utilizing the Comsol of software, which is commercially available CFD Software which is Comsol Multiphysics and will be getting our results with help of that, the thing which I will be taking in this question is a I will be assuming at 2D geometry.

(Refer Slide Time: 20:41)



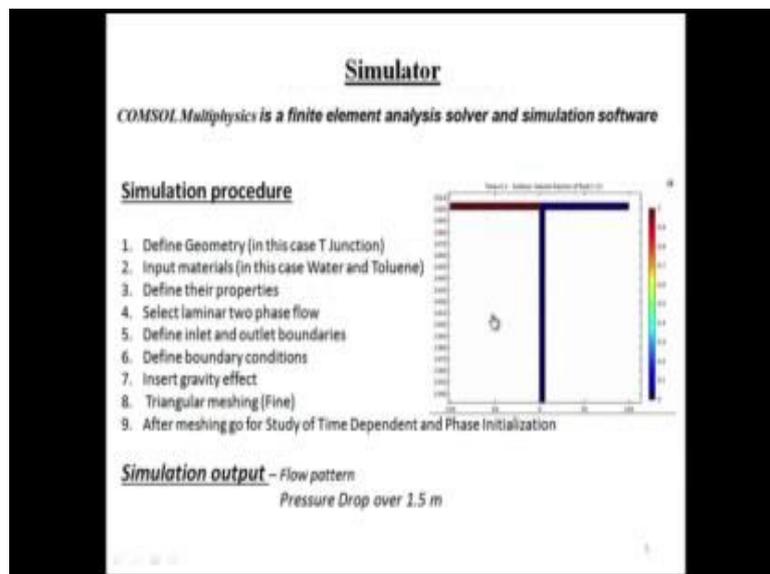
First starting Comsol Multiphysics, I want to say some words about it. So, newly emerged in the market it is a finite element analysis solver, and what is the strong point of the finite elements solver; is that divide the whole entire geometry into different small elements, and it integrates the governing partial differential equation where those elements; after being multiplied by a waited function, generally gallantly method of waited residual is used. The advantage of this is that apart from the others solving mechanisms; this finite element method analysis can handle very easily any type of geometry, whether its distorted geometry or irregular shape geometry, it can tackle with the same reason simplicity as it can tackle with a simplified or straight forward geometry.

(Refer Slide Time: 21:36)



So, my assumptions in this simulation will be a 2D geometry wall conditions, will which I will give is no slip the superficial velocity of toluene will be 0.07 meter per second and superficial velocity of water 0.07 meter per second, the outlet will be at one atmospheric pressure discharge will be happening down flow situation of 2 phase flow.

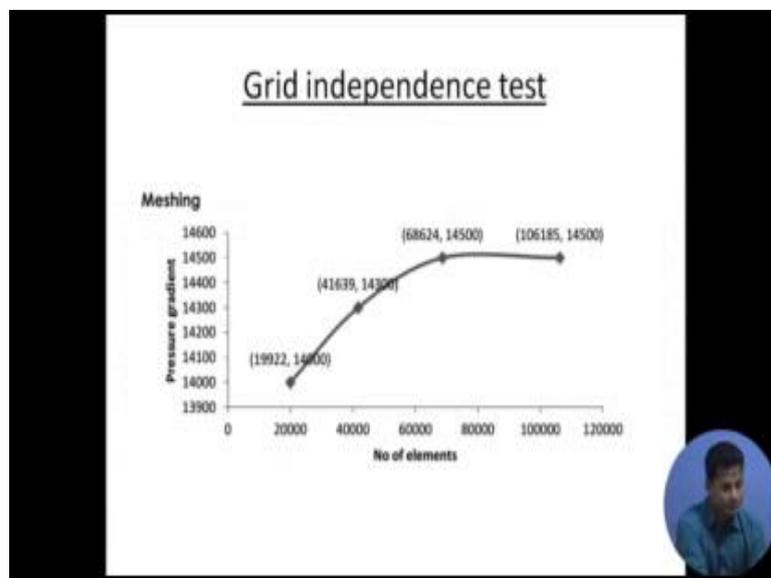
(Refer Slide Time: 21:59)



So, before starting I want to say some steps that are the simulation procedure, what we do first quickly going through it. Firstly, we make a geometry, we make a domain our specify our geometry we unite it after that we give input materials define the input

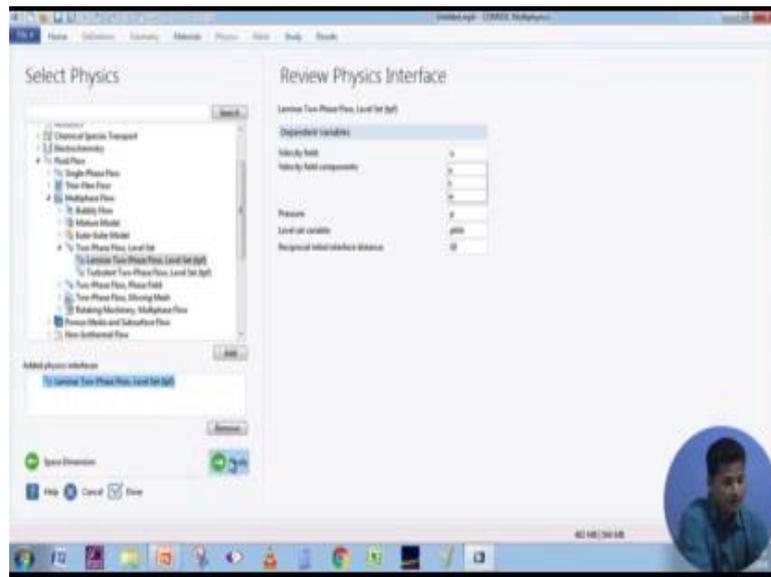
material properties, then select laminar to select on which model we want to study the situation; in our case in taking 0.07 meter per second, the flow was in the laminar region, then we specify the inlet and outlet boundaries give the boundary conditions, then after insert the gravity effect, then do the machine part. One thing important is about the machine is that by default this Comsol Multiphysics in 2D geometry chooses a triangular machine, triangular machine we should select mesh, after getting after doing grid independent study; what does a grid independent study? Actually in this grid independent test we go on reducing the size of elements, as the number a results the number of elements increases.

(Refer Slide Time: 23:01)



And we see after which point the difference in our result has diminished nearly diminished it is based on your criteria about the criteria of your requirement that might be 0.01 percent, 0.02 percent as required by the course problem. So, we see after which number sizing, the effect of the grid or the mesh diminishes and we say that after this mesh it has become grid independent and we select that sizing of mesh.

(Refer Slide Time: 23:51)



After this I want to start with the how we can solve this take this problem and solve it with the help of Comsol Multiphysics, as we can say this is a the Comsol desktop environment and there are 2 selections are there, one is module visual window and second one is blank module; blank module we use in this condition when we want to prepare a skeleton of our module, and later we want to implement it with different functionalities you customer made or tailor made functionalities we want to implement those models, for a specific problem we will go on first choosing the module visual window; I click on this module visual window; as we click we can see the different dimension options that is 3D, 3D geometries, 2D access cemetery, 2D 1D and point geometry.

So, 2D access cemetery, 2D access cemetery is nothing, but along an access, geometry is built and the access the reflection of the geometric is taken, along the other access. Is a mirror image replica on the other and access, in this way the geometry is built; I will be choosing simple 2D geometry here.

So, as we can see it has having different modules here, what is that ASBC acoustic chemical PCS transport performing mass transfer calculation, mass transfer stimulations, fluid flow heat transfer is there for structural mathematics, all are present. Our field of reason of interest is on fluid flow, we click on this fluid flow note as and again click on multi phase flow, when I click on this multi phase flow here I find different our models

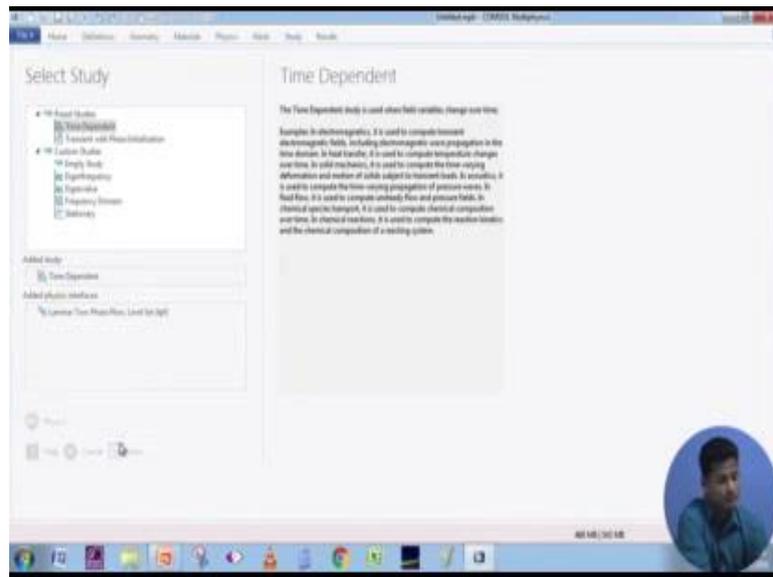
are present here, that is bubbly flow, mixture mode, 1 Yuer . Yuer model, 2 phase flow level set fresh fling moving mash and rotating machinery if I am having a sterol or something like that, moving part in our geometry.

So, this bubbly flow mixture model Yuer. Yuer model are used in those cases where we know, which is our continuous phase and which is our disperse phase; when your certain about our continuous and disperse case or certain about our what might what is my flow pattern can be, then we gone on choosing this models. In bubbly flow model specially useful gas liquid 2 phase flow, then the left ones are level set phase fill, 2 phase flow level set, 2 phase flow phase fill and moving mesh. So, moving mesh is used when we are have doing problem on distorting geometry, the geometry is changing without any material being added or removed in those cases we will use moving mesh.

The phase fill actually this method uses is first solve two transport equations, first to initialize this phase fill vary variable and the second one is to energy solve for free energy and uses the approach of energy minimization to track the interface; the levels at method measure is used in those cases where we want to track our interface with the help of fixed mesh.

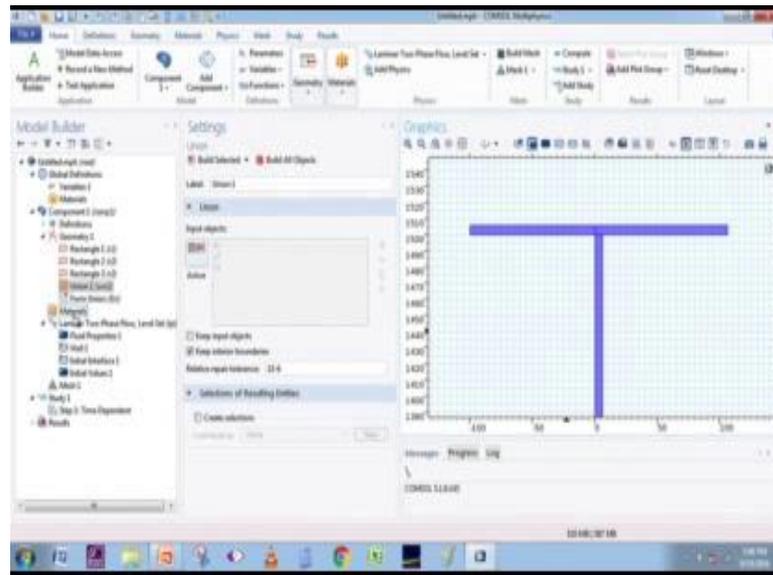
When the whole computation domain can be sub divided into two sub domains and which are separated by interface, in those cases, we will go for level set method and I click on the level set method; I get laminar two phase well and terminal two phase level set as I earlier set based on our Reynolds number calculation, if you put the word those variables that is 0.07 meter per second, then the diameter and the density this positive will get lambda flows. So, I go on selecting the laminar two phase flow level set and click on add.

(Refer Slide Time: 27:39)



As I click on this add, this check boxes our laminar the two phase flow levels get so added, then I click on the study note. Now solver is asking for which study I want to perform, there are preset studies like Time dependent transient with phase initialization, and there are Custom studies that are icon frequency icon value frequency domain Stationary icon frequency, icon value when we want to extract icon value, icon frequency of our linear or linearised modeling in those cases. So, this more frequently used in electromagnetic question or those simulations. So, and there is stationary study, transient with phase initialization and time dependent. Since I do not know what the flow pattern will be or how it will be changing with time.

(Refer Slide Time: 28:28)



I will go on selecting a simple time dependent study, select the stand dependent study and click on done. As we can see this is Comsol Multiphysics environment, now global definitions part, this global definitions know when we want to you specify some variables to a system, we name the variable here in the check box suppose I write here, some typing, something like omega or something we give the expression here and then the unit, and later we want to call this variable in our field of study of that is in our region of study that is two phase flow, I will said want to implement in the wall conditions, that is vibrating wall or something like that or initial values or outlet inlet as the case may be. So, I go on selecting geometry, I click on this geometry; when we will right click we will find different options here, starting from circle, ellipse rectangle square polygon etcetera. Projecting of 3D geometry circular channel, on a 2D surface we it will be rectangle. So, I select a rectangle.

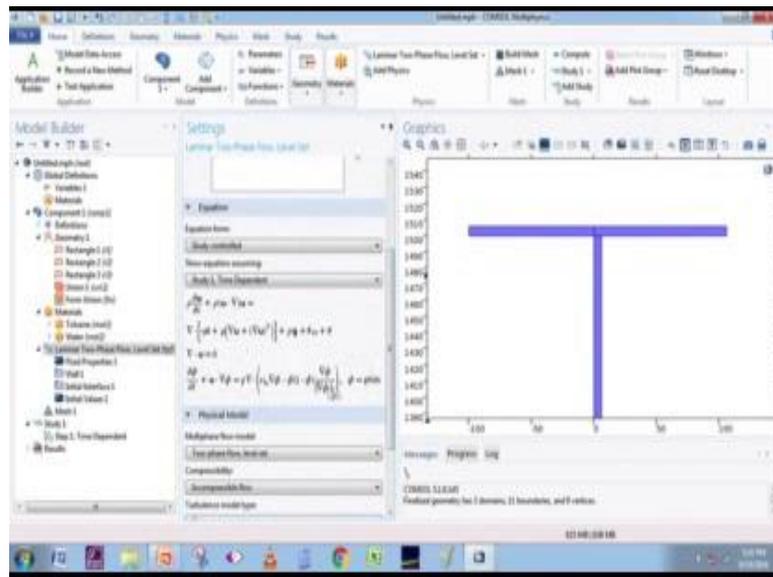
Now, this width will be this rectangles, this will be the rectangle that our geometry id internal diameter of the tube. So, I select this us width 6 and height I gave you as 1500 and select on Vildol objects. So, as we can see these geometry, there is a rectangle has been built in our geometry box; graphics box. You are seeing and we go again on making two other geometries, click on rectangle to and we give this us width 6 and height 100 this is basically the this part the question, the earlier which I which have shown to all of you this section what is geometry, this is based 100 millimeter none our 1500, that has build geometry the side wings the left arm and the right arm a 100mm this id that is

weight is 6 mm also the same here, down flow situation. So, the rectangle two this, now I want to orient this rectangle in horizontal direction. So, I want a rotation here, I select clock wise 90 degree rotation, and want to position my this rectangular 00 access, I select and then select on click on Vildol objects, and I click on zoom extends, this pallet the comprising of zoom extends, zoom box is zoom extends is used to view the whole geometry at a glance, zoom box is used. So, our specifically viewing our geometry the particular section of our geometry I want this section in above that is at about 1500mm. So, in rectangle two, I choose my y coordinate as 1500 mm and select on Vildol objects and again click zoom extends to view the whole geometry at a glance.

Now, I again right click on geometry an add my x third rectangle, third rectangle I give my vuda 6, height as 100, then my positioning as 1506, rotation will be counter clock as 90 degree, and I select Vildol objects.

See I click on zoom box here; we have finished building our geometry. Now one thing important to note is that, that we need to unite the geometries, why we union is important is that, we do not unite the computation will stop at the very boundary of the individual domain it will not progress, since we want to see how the simulation is going, how the flow pattern, which flow pattern will be occurring in this situation through the entire geometry we need to unite all these three rectangles here. I click on word click on this geometry, again right click and we will find this Boolean and patrician operations, click on union then after that we select all the three rectangles and click on Vildol objects, now our complete geometry has been unified now the second step is adding material store geometry.

(Refer Slide Time: 33:11)



When we right click on this material part there are two different options coming add material blank material switch material link, these are documents basically documentation part, blank material is used when we want to customize our material and add it in to a library. Add material uses the Comsol Multiphysics library and brings the our material of interest, I click on this add material ah box appears here with add material written, on it I type in here I first which is toluene, this is an organic fluid organic way then I click on search. This process takes time, access Comsol Multiphysics environment uses, two types of mechanism to solve a particular problem it goes for direct solution of a problem and it goes iterative method of solving a particular problem.

The advantage of direct method is that its quick and circulate it builds a set of matrixes of our governing partial differential equations and tries to solves it in one go note, there are three different direct solvers available in this Comsol Multiphysics; one is Padiso which stands for parallel direct solver, Mums for massively multi frontal parallel direct solver, then Pools past subject oriented parallel director matrix oriented solver. This three actually the mums this solver, can do cluster computing you can divides its work into many notes, as the case may be such as for the solution and in iterative solver we have GMRES is the best, GMRES is stands for Generalize Minimal Residual Approach.

So, in GMRES it is actually likes the gauss ideal iteration technique, which assumes solution then goes on it iterating and stops until it meets the required convergence

criteria. So, as we see it has arrived, now click on tolean then liquids and double click on right click on this tolean and select add to selection, the tolean gets added; now our second material of interest that is water I am typing, water here click on search and double click on water, now you want to specify which domain is continuing, which fluid.

So, initially in this condition, our first it our whole test geometry was initially was filled with water and from the left domain toluene inlet was provided and thus left domain was completely filled with toluene, as we have learnt from our theoretical classes that, the flow patterns gets affected depending on the initial wetting of the phases. So, we need to be very sure about the wetting properties and need to check them.

So, then initialize these two domains with water, as was the experimental condition it should done, and this left end side domain with toluene. As I click on this domain particular domain, in the check box the domain number gets added; this means that it has it has been selected, now in we go on click in the material properties, then choose to non solid then again same for the water portion we click on material type non solid, now the material domain selection has been done now we go to laminar two phase levels it study, our study. Equation form which we study control all the domains are have been selected this is the basically Navier stokes equation which we will be solving in the continuity equation, and this is the level set equation which initialize the level set variable and solve for the solve for the interface the whole computational domain. So, in this question I click on fluid properties, in fluid properties the one thing very important to notice that for awarding any kind of confusion we should keep the material numbering and fluid numbering the same or else it will create a lot of problem; you might get confused.

So, I click on this fluid one and check it as toluene material one or say for fluid to I select water, then after that we need to enter here what is the it will my Liquid liquid interface, from experimental conditions I select here, means benzene water other options are there, adequate air condition I select user define. Since measure by experimentally our interfacial tension between toluene and water was 0.037, I type in here in the interfacial tension box 0 0.037 Newton per meter.

Then after that, I go for selecting my next part, very important part is that to add inlet outlet interface to our geometry. So, when I lightly click on laminar two phase flow; I get

these options, inlet outlet initial values gravity etcetera. I click on inlet, inlet what gets added since our geometry is has have a two inlets and one outlet, I need to select another inlet and outlet, then after that again I right click on this laminar two phase flow ,and add select the gravity, then initial values two.

What are initial values that, initial which fluid was present initially in this two domains, what understood domains water was present. So, I select this fluid initial domain as fluid to which was water, then initial values one, but the default the other two values get over ridded and one has been one get selected here its already its fluid one, after that we need to specify our inlet where the solver Multiphysics solver will take in its inlet, I click on the zoom box and then I click of this inlet here, the inlet demarcation gets selected in the check box; after that I need to specify my normal inflow velocity here.

Here we can choose velocity field if you are having different complex special components in a velocity or you can go for normal inflow velocity, I select here normal inflow velocity and select click type in as 0.07 meter per second. My inlet what will be complete consisting fluid one, no fraction of two will be present; I let it remain as 0 volume fraction of fluid two.

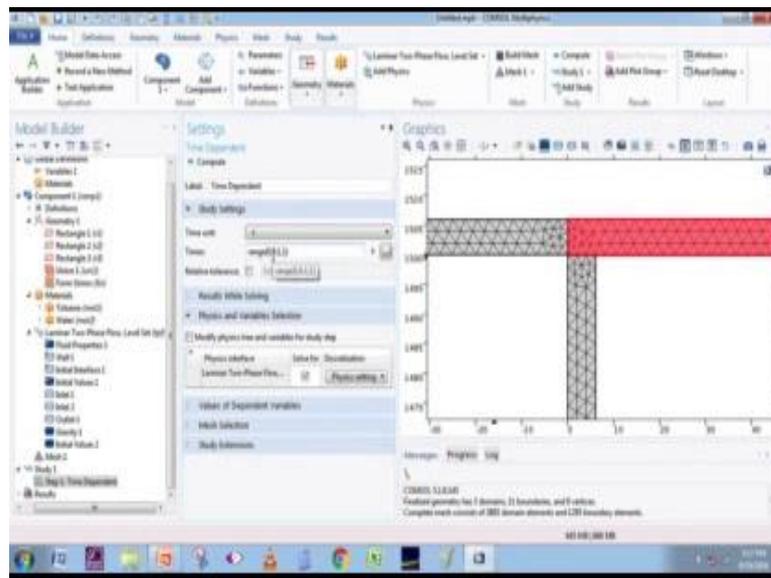
In the inlet two will completely comprise of fluid two. So, I type in here one my normal inflow velocity 0.07 meter per second need to specify the domain click was zoom box again enlarge it select this is point domain, now one thing is important is to specify the interface right click on that the interface is already present, here click on zoom box again and this magnify this my interface here, as was experimentally observed initialize actually then after initial interface has been selected, we have specified our initial values inlet two inlet one has been specified.

Now we need to specify our outlet conditions, it this the discharge will get atmospheric equation again click on zoom extends and go for imaging my outlet, I select this is my outlet the it automatically gets select it gets added and I click on it there its pressure outlet, what is that is there are various options velocity pressure laminar outlet pressure outlet and I type in as 101325 paschal, during the experimental condition no back flow was observed. So, I click in this check box that is applies back flow, after that we need to gravity affect towards system, again zoom extends select and then zoom box.

Now, in these three domains my gravity force will be active. So, I click on to three geometries and three geometries get selected, these all the three geometries were selected; now this is the gravity vector x axis is this one, and y axis is the curve which is 1500 mm length axis, this is y axis.

So, in this axis my gravity is acting in this direction going downward that is minus g has constant, g is initial stored in the Multiphysics environment as 9.81 meter per second square. So, then after that gravity affect has been added, now machine part. As we as I explained earlier that this machine was selection of mesh size is very important you first, go for a grade independent test then after the after that mesh size, we can be certain about our mesh size. There are two different options when I click on this, this is physics control mesh, user control mesh user control mesh we gave a mesh size in physics control utilize the multi physics library for its different machine criteria and my select it is my machine has final.

(Refer Slide Time: 43:03)



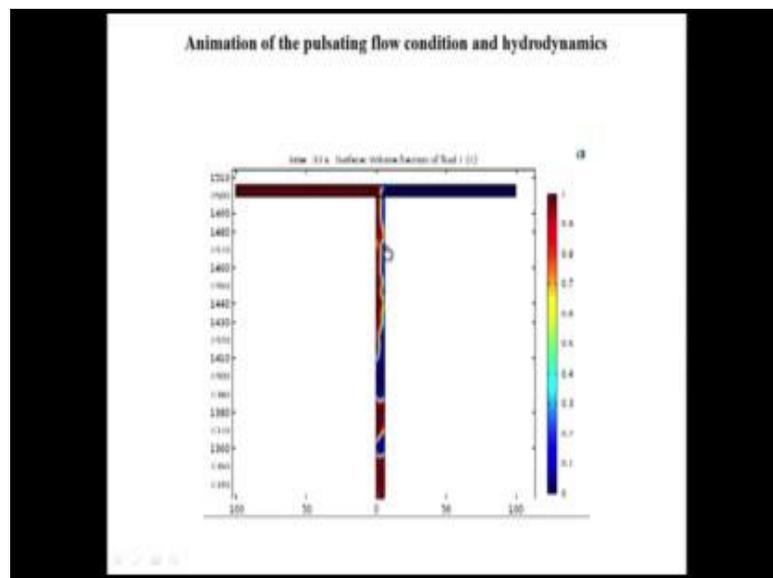
I click on Vildol, as we can see meshing has been done more dense mesh is present at the interface and less dense has the bulk, after the machine gets completed we directly we go to our study part; was we have since we have selected time dependent study, to specify time unit when I click on this box, there are different time units coming nano second, macro second, mile second, minute second, I select click on second as per the question there is a range; range is signifies that from what on which time we want to our

stimulation and were to when it what times takes to. So, I give my solver that we start from 0 second then go with point one time stepping calculated 0.01 then 0.02, 0.03 like in this manner you end at one second. Results while solving we click on plot.

So, if you want to see the plot the how the volume fraction are the flow pattern is changing during the computation click on this check box here, then after that we go for compute, in this situation actually there is a thing that, it uses to as I earlier said that it uses two different type of solver, that is direct and iterative method of solver solution, is best to go for a three d complicated geometry best to go for a iterative method of solution; rather than going for direct method solution, since for direct method solution your system might get hanged or it will slow down your system and it will not be so the progress will takes some time it cross huge ram for computing for going for direct solution.

GMRES is the best iterative method for going for complicated geometries. So, as I have started the simulation previously I have done some simulation which is which I am going to present to I before all of you the same in the same condition.

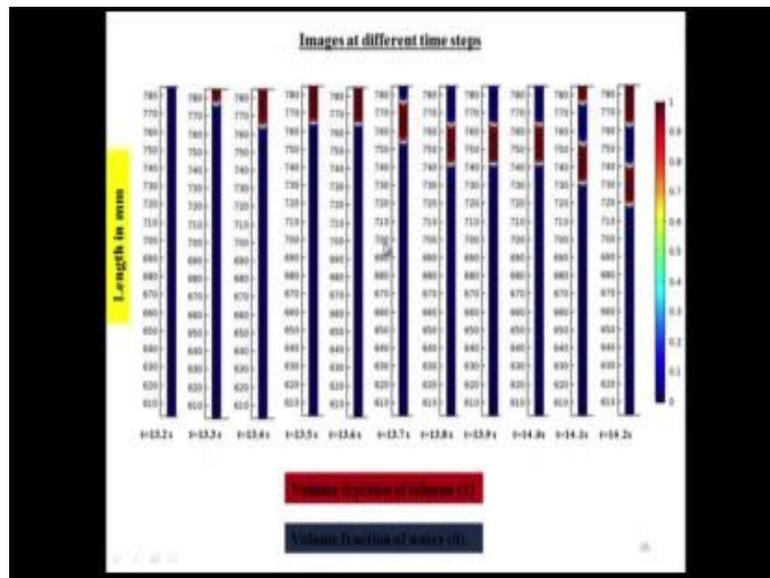
(Refer Slide Time: 45:19)



This was my the simulation, which have previously done with at the same conditions, with the known velocity superficial velocity we were not aware of the what flow pattern can prevail in this situation at atmospheric pressure discharge and now we are seeing here that theses slug flow, slug flow is accruing the red one is what toluene, the blue one

is for water. So, it also not verified experimental and we saw that the same slug flow accruing in the real time experimentation, and in this way we can also extract the animation.

(Refer Slide Time: 46:02)



And we can also extract the snap shots of the images of slug at different time steps, these are the images of a slug at different time steps, that is 13.2, 13. Seconds how this likes this progressing with respect to time.

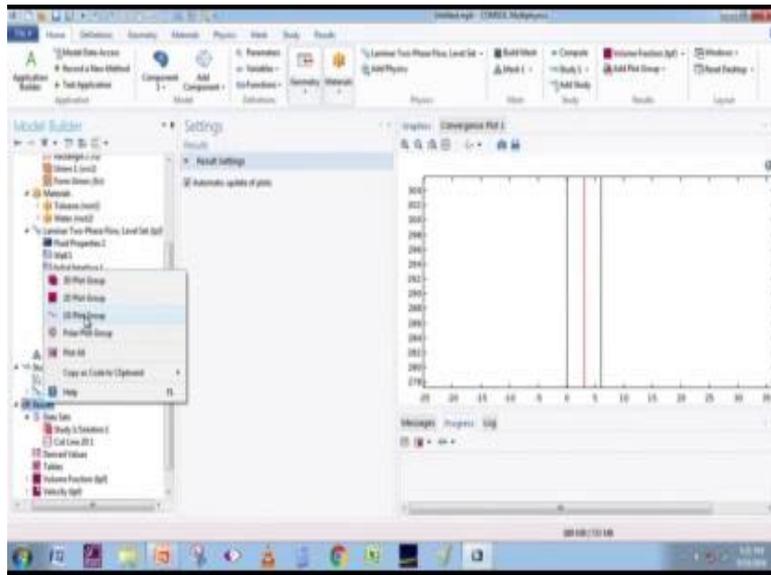
(Refer Slide Time: 46:23)



So, in this way we saw with given our input condition, we can extract or see foresee our flow pattern, what flow pattern can accruing the particular set of conditions, now in our daily requirement many to extract some graphs from our experimentation or some input which we provide, we need to extract some graphs results such are pressure graphs velocity profile plot and likewise. So, for this I am going to show all of you that, how to ex extract simple pressure plot across the length of the conduit.

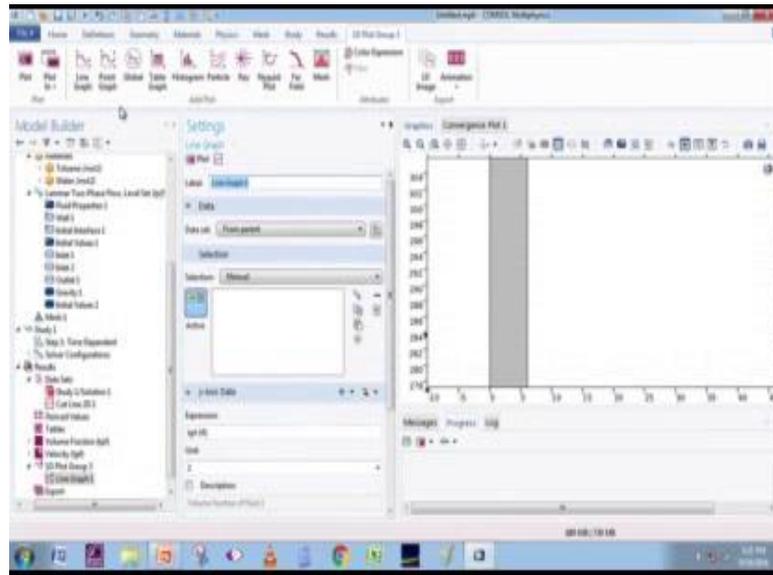
Now, I click on results and the data sets for extracting any clean graph or graphical plot first you need to define our cut line. So, I click on data sets and select cut line 2D, its 2D geometry and I specify two points that is x, x my select it as 3 and y as 0, rather I it let it remain 0, then after that three, then this as 1506.

(Refer Slide Time: 47:38)



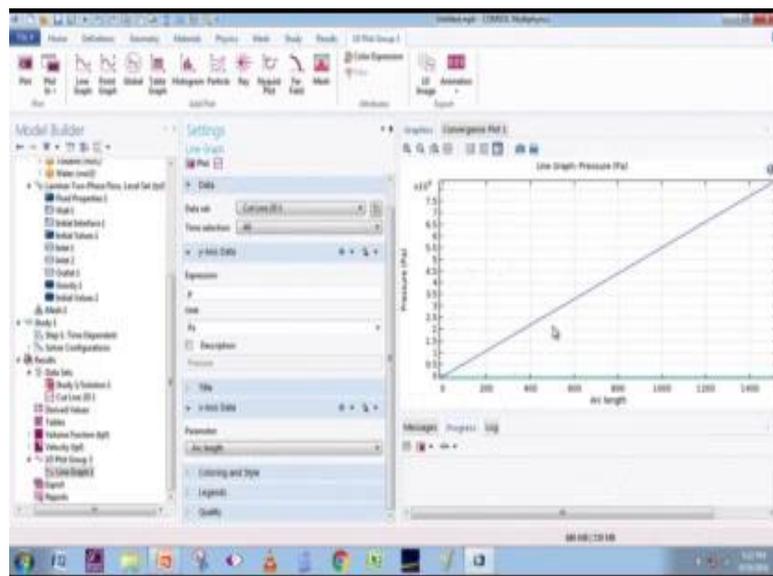
Then click on plot, this is my cutline, which is running through the entire geometry across the length of the conduit I click on zoom extends again check, click on zoom box this cut line runs to the entire geometry and the non I am going to specify the cutline for extracting results and the plot will be I will takes the values from this cut line, now again go for in the results section and I right click there are different options, 3D plot group 2D plot group 1D plot group, as I select one d plot group for a line graph here and here is my selection 1D plot group.

(Refer Slide Time: 48:18)



Again click on the selection and go for line graph, in this line graph this line graph window, I am now the server is asking from where it should be take the values, I give it as cut line 2D1 then y axis data in this icon what variable which we want to put in a y axis, I click on this x icon and see click on this laminar two phase node and select double click on pressure, I want to keep my pressure as y axis and the default x axis arc length or which is the length of the conduit here.

(Refer Slide Time: 48:57)



So, after this I click on this plot, but a plot has been generated here across the conduit as we have seen, this explains to how we can extract a plot basically from our simulation. So, after this we need to extract some reports from our simulation, there is other the reports which Comsol Multiphysics in maroon provide we click on this report option then click on right click, this report brief report intermediate report, complete report different option group options are there based on our requirement we extract our report. One thing is important is that it showing that across the length of our conduit it, that is this one my conduit was initially this section this 1500 mm section, across the conduit it is plotting the graph and showing the how the pressure is changing with respect to the length.

So, we see there is (Refer Time: 49:48) linear evolution a pressure across the length of the conduit. So, in this manner we are going to simulate a 2D flow problem. So, we in this video we saw that how we can that how we can with the with given inlet velocities and outlet pressure, discharge pressure, we can we can simulate what flow pattern can be going, can be prevailing of those condition and to extract some simple graph from our simulation, that is all hope that we you will all of you benefit from this video.

Thank you.