Flow Dynamics of Two Phase (air and water) Stratified Flow by a Simulation Package COMSOL MULTIPHYSICS

Muhammad Hashim[1*], Ali Nawaz Mengal[2]

[1]Department of Petroleum and Gas Engineering, [2]Department of Mining Engineering, Balochistan University of Information Technology, Engineering and Management Sciences, Quetta

Abstract

In this paper simulation of the two phase stratified flow is carried out by a Commercial Simulation Package Comsol Multiphysics version 3.4 using level set approach for tracking the movements of the interface. The problem involves the Simulation (based on Transient analysis) of two phase stratified flow between two planes in (2D) two Dimension. Some boundary problems arose in defining specific geometry, that is why a little different geometry approach is presented to overcome this problem. Further, gravity that usually stabilizes the flow by providing specific flow pattern (specially separation of the fluids in gravity field having different densities) when flow is occurring in gravity field was observed that gravity constant destabilize the flow (carrying simulation in Comsol Multiphysics) and convergence problem arises. That is why the flow has checked by keeping the value of gravity zero at first then, giving a much less value than the gravity really has, and increased gradually to a value beyond which again the flow did not converge. The effects of Surface tension Coefficient on the flow, are also observed and presented.

Key words: Stratified flow, Level Set Approach, Interface, Gravity, Surface tension

Corresponding Authors email: m.hashim@buitms.edu.pk

INTRODUCTION

Two phase (gas, liquid) stratified flow occurs in many engineering design problem, such as long distance transportation of oil and gas, oil platforms, process industry, may be in horizontal section of horizontal Oil/Gas wells, reactors etc. To simulate or to calculate the flow pattern, shear forces(Wongwises and kalinithenko, 2002) involved and different aspect of flow are of the utmost importance in designing of such wide range of engineering problems (Yap et al., 2005; Ghorai and Nigam, 2000).

Multiphase/ two phase flow phenomenon has been a challenging task through all times and to understand this sort of flow, a wide range of research have been carried out for past several years.

In petroleum industry specially in surface gathering systems, in horizontal section of horizontal wells and in oil/gas transportation industry, in many cases gas and oil flows simultaneously either it is wanted to transport both at the same time or if condensation occurs in pipes. To avoid problems in such cases, the behaviours like forces, flow pattern and instabilities etc involved in the flow has been studied widely. Different approaches have been investigated to understand the stratified flow phenomenon.

A variety of CFD based Commercial simulation packages are available in industry to simulate the problem and to apply further in modelling of such designs

(Wongwises and kalinithenko, 2002).

The Simulation package Comsol Multiphysics with its built-in levelset approach is used here to simulate the two phase (air-water) flow between two plane transiently. The results may be applied to different two fluid flow problems, especially Oil-Gas (Moguedet et al., 2005).

2d Models Prepared in Comsol Multyphysics

The simulation is carried out in a commercial CFD (Computational Fluid Dynamics) based simulation package Comsol Multiphysics version 3.4.
Since, two phase (air-water) stratified flow is understudy, there for the level set approach is used in particular, tracking the moving interface.

An Overview of the Level Set Approach

A large number of fluid flow problems involve moving interfaces, like air-water, breaking surface waves, combustion and reacting flows etc. The interplay between the interface dynamics and the surrounding fluid motions is of great consideration in such many applications. Factors such as density ratios and temperature jumps across the interface, surface tension effects, topological connectivity and boundary conditions play significant roles in the dynamics (Berthelsen and Ytrehus, 2005).

To tackle some of the most complex problems in fluid interface, a class of numerical techniques have been built over the past several years. Relying on an implicit representation of the interface whose equation of motion is numerically approximated using schemes built from those for hyperbolic conservation laws, Osher and Sethian Level Set methods which are the computational techniques for tracking moving interfaces. The resulting techniques are able to handle problems in which the speed of the evolving interface may sensitively depend on local properties such as curvature and normal direction, as well as complex physics of the front and internal jump and boundary conditions determined by the interface location. In particular, Level Set methods are designed for problems in multiple space dimension in which the topology of the evolving interface changes during the course of events and problems in which sharp corners ad cusps are present (Rune, 2007).

Governing Equations

To simulate numerically the model a level set approach is coupled to the classical Navier-Stokes equations. Actually this method is very well studied to describe the motion of interface.

$$\rho\frac{\partial u}{\partial t}+\rho u.\nabla u=\nabla.\left[-pId+\eta\left(\nabla u+(\nabla u)^T\right)\right]+\rho g+\sigma\kappa\delta n \quad 5.1$$

$$\nabla.u=0 \quad 5.2$$

$$\frac{\partial\Phi}{\partial t}+u.\nabla\Phi=\gamma\nabla.\left[\varepsilon\nabla\Phi-\Phi(1-\Phi)\frac{\nabla\Phi}{|\nabla\Phi|}\right] \quad 5.3$$

Where in equation 5.1,
ρ is the density
η is the dynamic viscosity
g is the gravity constant
σ is the surface tension coefficient
κ = - .n the curvature of the fluid interface
δ is the delta function concentrated at the interface between the fluids
u is the velocity field
p is the pressure

The term σκδn defines the surface tension forces.
In equation 5.2, Φ represents the level set function and in this approach Φ at a point is the distance from the interface, such that an interface corresponds to a surface where Φ = 0. γ and ε are the level set parameters where γ is the reinitialization parameter and ε is the parameter controlling the interface thickness.

Moving Air and Water Concurrently between Two Interface

Air and water both are flowing co currently between two planes from left to the right. The interface and exit positions of the two fluids are not the same. The configuration consists of two sections (subdomains) water and air. The air (upper) section is a bit ahead of the

water (lower) section. The pressure at the two points at outlets is defined zero (see Comsol Model Reports). this is done to establish an interface between two fluids. While at the exit, the water exits first. This is done to avoid the problem that arises in defining the boundary condition.
The total length of the model is 4.5 with a common interface of 3.5. The length of each section is 4. The total height of the sections is 0.8, and each section with the height of 0.4.

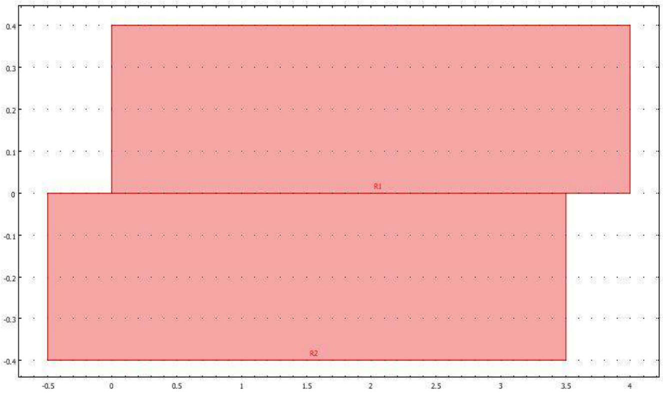


Figure.1. Basic geometry for the model of moving air and water co currently.

Subdomains
The model consists of two subdomains 1 and 2. Subdomain 1 is provided by water while subdomain 2 by air.

The following table describes the numerical values of functional parameters used in the simulation.

Parameter	Num. Value (air)	Num. Value (water)	Unit
Density: ρ	1	1000	kg/m^3
Dynamic viscosity: η	1e-5	1e-3	Pa.s
Gravity constant: gy (in y direction)	0 and -2		m/s^2
Surface tension coefficient: σ	0, 0.073		N/m
Mean inlet velocity	0.1	0.01	m/s
Parameter controlling interface thickness: ε	0.05		M
Reinitialization parameter: γ	0.2		m/s

The mesh statistics of the subdomains is given in the following table.

Number of degrees of freedom	16636
Number of mesh points	1321
Number of elements	2464
Triangular	2464
Quadrilateral	0
Number of boundary elements	228
Number of vertex elements	8
Minimum element quality	0.785
Element area ratio	0.218

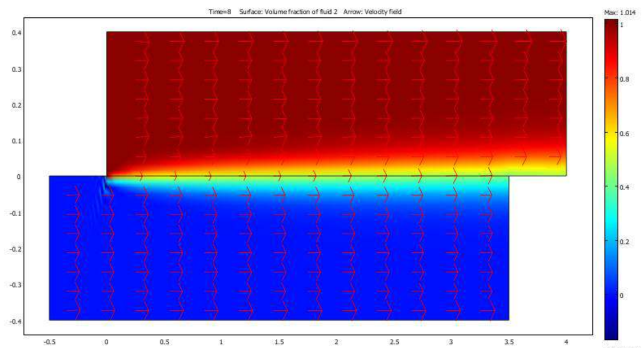
Boundaries, boundary conditions and settings

Boundary	Boundary type	Boundary condition	Settings
1	Inlet	Velocity	$u = u_0$, $\varphi = \varphi_0$
2	Wall	No-slip	$u = 0$
3	Wall	No-slip	$u = 0$
4	Inlet	Velocity	$u = u_0$, $\varphi = \varphi_0$
5	Interior boundary	Initial fluid interface	
6	Wall	No-slip	$u = 0$
7	Outlet	Pressure	$p_0 = 0$
8	Wall	No-slip	$u = 0$
9	Outlet	Pressure	$p_0 = 0$

At boundary 1, the fraction of fluid 2(subdomain 2(air) is defined as fluid 2) is zero here. The velocity in x-direction is expressed as 6*V_mean*s*(1-s)+t. Multiplying V_mean by 6 and adding with t (time step) is done to increase (maintain) the velocity during the flow. At boundary 4, the fraction of fluid 2 is 1.

RESULTS AND DISCUSSION.

Result without effects of Gravity and Surface tension



2. Result of simulation of moving air and water. The arrows are velocity field

The result showed above is obtained keeping the surface tension and gravity zero. Density plays an important role in fluid flow problems, flowing in gravity field. In our case giving to gravity a value of zero looks quite unrealistic. Also the water surface tension that differentiates between the water and air surface is kept zero to stabilize the flow at this stage.
Assigning a value of 9.81 m/s^2 to the gravity, the flow problem does not converge. Thus at this stage, it is tried here if no gravity acts on the flow.

Since the flow is transient, and is solved for the time of 8 seconds, the analysis of the flow pattern in different times show that initially when flowing air interacts with the flowing water, instability occurs and, a first water wave rises highly and approaches approximately the air wall contact which looks like forming a slug for a short interval of time. Then the first wave falls a little and a second water wave rises, but not so high as the first wave. At this stage the the interface rises quite high and much water flows out than air. No other wave rises, the second wave falls down slowly, and after some time a complete stratified smooth flow develops.

With each time step the velocity is defined to rise. With the rise in velocity of air and water both, the instability was considered to happen after some time but it is observed that flow stabilizes with a complete stratified smooth flow.

The pressure distribution (given in Appendix A) in two sections is almost uniform in the range of 1000 Pa approximately, except at the entrance of the water section, where the pressure is higher, in the range of 2000 Pa approximately.

The velocity distribution (given in Appendix A) shows that at the solid walls the velocity is zero. In air section, just below the wall the velocity of the air is quite high due to much lower pressure. Below this, the air velocity attains uniform value. But, just above the interface, it looks like water vapors rise in the air section and a pressure reduction in result. Due to this pressure reduction again velocity increases. At the Air-Water interface, the velocity decreases due to the shear forces acting at the interface. Below the interface the shear forces reduces and also the pressure due to the mixing air in water, and the velocity rises again. The velocity below this region is again decreased (less than the velocity in the air section). The velocity then gains a higher value just above the water-wall contact.

When surface tension was introduced, no difference was observed in flow patterns. The pressure and velocity distribution are also in the same pattern.

Result with a Small Gravity Value and Surface Tension Coeefficient

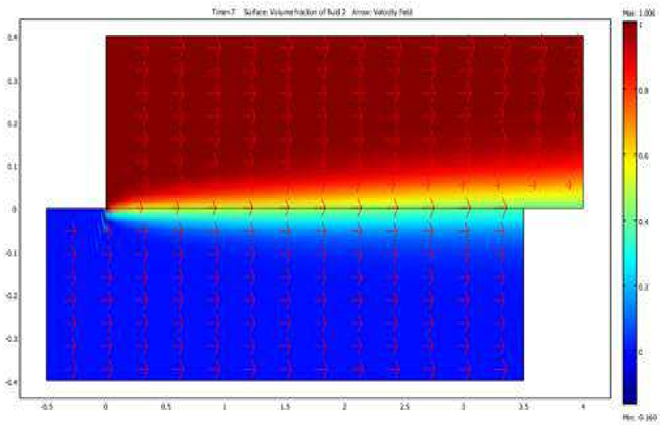


Figure 3. Air moving with an inlet mean velocity of 0.1 m/s^2 and water is flowing with an inlet mean velocity of 0.01 m/s^2. The arrows show the velocity field.

As the flow problem does not converge when introducing gravity, it is checked by introducing a little value of gravity that is 2 m/s^2 to observe the effect of gravity. This value obtained by starting from 1m/s^2 and the final convergence obtained at 2 m/s^2. The value of gravity more than this destabilizes the flow and the flow does not converge. But still we can check the effect of gravity if it has a small value though.

By introducing a little gravity keeping the value of surface tension coefficient as it is, it is observed that when the first water wave rises, at the same time water level falls down at the exit of the water section. When second water wave rises consecutive to the first one, the water level at the exit of water rises again. As time passes the water level at the entrance falls down and water level at the exit rises up slowly. After some time again a stratified smooth flow develops but with an interface a little higher at the exit. This means that at the end of the interface the area of the flowing air reduces slightly. By introducing Gravity, the pressure distribution changes slightly (See appendix A). Pressure in the lower(water) section is a little higher than in the upper (air) section. The over all velocity of the model reduces by introducing gravity (See appendix A). At the solid walls, the velocities are zero. Just below the solid wall in air section, since a low pressure exists, the velocity gets a high value. Again at the interface, due to shear forces acting here, the velocity reduces. Checking the flow just with the gravity (2m/s2 here) and vanishing the effect of surface tension coefficient, it follows the pattern as just the surface tension was introduced.

Flowing Air over Water Which Is Initially at Rest but Provided an Outlet

The configuration consists of two sections; air (upper) and water (lower). In this configuration the air flows (left to right) over water which is at rest (no inlet for the water) but provided an outlet. The basic idea in modelling such configuration is to observe the drag applied by air to move the water.
The geometry is designed such that, initially the air flows as single phase to establish a stabilized flow, and then flows over water.
The length of the air section is 10 while the length of the water section is 8. The heights

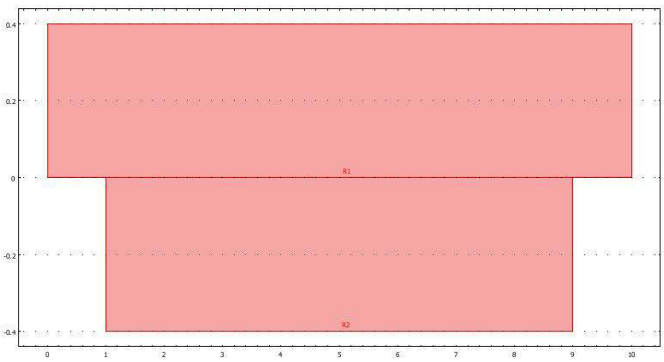


Figure 4. Basic geometry of the flow problem of flowing air over water which is initially at rest but provided an outlet. Mean inlet air velocity is 0.1m/s^2.

SUBDOMAINS

The configuration consists of two sub domains. Water is provided in subdomain 1 while air in 2. The following table describes the numerical values of functional parameters used in the simulation.

Parameter	Num. Value (air)	Num. Value (water)	Unit
Density: ρ	1	1000	kg/m^3
Dynamic viscosity: η	1e-5	1e-3	Pa.s
Gravity constant: gy (in y direction)	0 and -3		m/s^2
Surface tension coefficient: σ		0, 0.073	N/m
Mean inlet velocity	0.1	0.0	m/s
Parameter controlling interface thickness: ε	0.1		m
Reinitialization parameter: γ	0.1		m/s

The mesh statistics of the subdomains is given in the following table.

Number of degrees of freedom	14320
Number of mesh points	1141
Number of elements	2112
Triangular	2112
Quadrilateral	0
Number of boundary elements	228
Number of vertex elements	8
Minimum element quality	0.713
Element area ratio	0.438

Boundaries, Boundary Conditions and Settings

Boundary	Boundary type	Boundary condition	Settings
1	Inlet	Velocity	$u = u_0$, $\varphi = \varphi_0$
2	Wall	No-slip	$u = 0$
3	Wall	No-slip	$u = 0$
4	Inlet	Velocity	$u = u_0$, $\varphi = \varphi_0$
5	Interior boundary	Initial fluid interface	
6	Wall	No-slip	$u = 0$
7	Outlet	Pressure	$p_0 = 0$
8	Wall	No-slip	$u = 0$
9	Outlet	Pressure	$p_0 = 0$

The figure showing different boundaries is given in Appendix A.

Results without Surface Tension and Gravity

The figure below is the result of flow of air over water at rest but provided an outlet. Basically in this flow problem the water is shear driven, shear forces act at the interface of flowing air and still water.

With no gravity and surface tension coefficient, it is observed that when air flows over water, the water level falls at start of the water section. The fall in water section goes approximately to the water wall contact, forming slug, but the water level at the exit of the water section remains constant. It was expected since the cause of this pattern is the absence of gravity, as in the absence of gravity no external force acts on the flow and the water level does not falls at the exit. At the entrance when air just contacts the water surface, the fall of water is due to drag, as there is no inflow of water and, water flows to the exit by interfacial shear force.

The pressure distribution is quite different here (Appendix A). The pressure decreases with the flow from left to the right in three stages, in the beginning, in the middle and in the end of flow. All pressure reductions takes place suddenly. It means, no hydrostatic pressure acts here.

Initially in the water section, water is at rest with no inflow. When the flowing air with velocity interacts with water, the direction of the velocity changes to the down side due to the water displacement and creation of wave. Air moves the water with a little velocity. The air gets maximum velocity at the top of the wave due to pressure reduction by wave generation.

When, only Surface tension coefficient was introduced as in the first model, no effects were observed. It just behaves as it does without Surface tension coefficient.

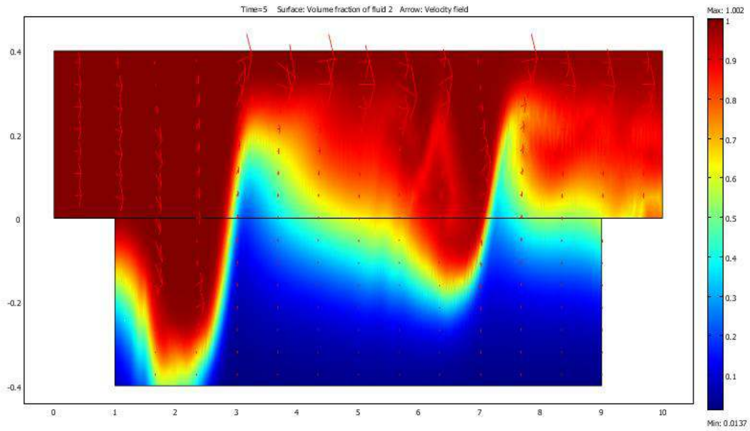


Figure 5. Result of the flow when Gravity and Surface tension are not considered.

Results with a Little Gravity and Surface Tension

Here again it was tried to check the effect of gravity(which must be taken into account) by introducing a little value of Gravity 2m/s2 and also by introducing Surface tension coefficient. Here the result is a bit satisfactory as the water level at the exit falls down mainly due to gravity and somewhat due to shear. As the water is shear driven, and is under effect of gravity, the water level falls abruptly at the exit and slowly at the start. If this goes on, the water will be drained out of the section. By introducing the gravity, the pressure distribution is quite natural here, higher at the bottom due to hydrostatic pressure of water and lower at the top wall (air-wall contact). (See appendix A) As the water falls down due to gravity, a decrease in pressure takes place above in the air zone and back flow of the air occurs. The result just by introducing a value of small gravity but keeping the surface tension zero, the flow pattern is as when both quantities were introduced.

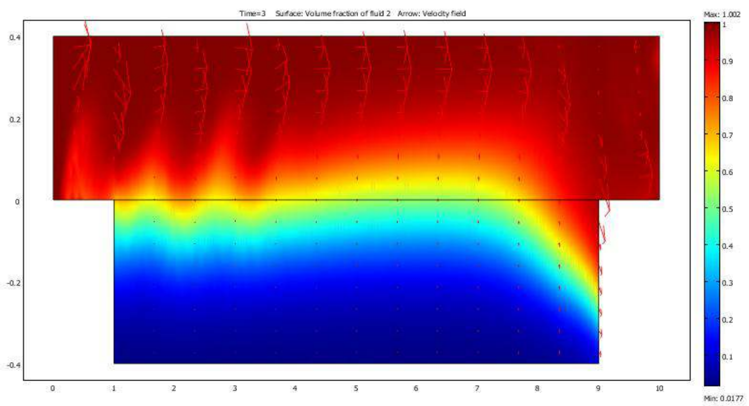


Figure 6. The result when Gravity and Surface tension are introduced.

CONCLUSION

Finally, the work shows the possibilities of Comsol Multiphysics to deal with two phase flow topics provided appropriate geometries, boundary conditions and parameters, since during the simulation, it is observed that Comsol Multiphysics is very sensitive to the above mentioned values and conditions. Minor changes to the boundary conditions, geometry and defined parameters cause the problem not to converge and thus no solution. The main conclusions we established on this work are, Without considering Gravity constant, the flow behaves in a different way, which does not show the natural way of fluid flow pattern. Thus Gravity plays vital role in establishing a natural and accepted flow pattern. Surface tension Coefficient, which is thought to have a role while stratified two phase (air-water) occurs, since its existence affects the surface of water in different way, but it is observed that it does not affect the flow. The level set method is a good approach when dealing with the moving interface, since the movement of fluid interface is a dynamic process and must be clearly observed to understand the flow.

The two fluids (air-water) used here may be displaced by Oil-Gas and different fluids having specific properties. The different flow problems which arise in the two phase flow industry/ Oil-Gas industry may be dealt in a good way.

Indeed, further works are needed to solve the two phase flow problem between two planes by Comsol. It looks quite possible to solve such problems by making further more and more good efforts. Further if the sensitivity issue of the Comsol Multiphysics is overcome, more possibilities will be attained to deal with such problems in a good time and wide range of parameters.

REFERENCES

- Wongwises S and Kalinithenko V. (2002). “Mean velocity distributions in a horizontal air-water flow”, *International Journal of Multiphase Flow.* 28:167-174.
- Yap YF, Chai JC, Toh KC, Wong TN, Lam YC. (2005). Numerical modelling of unidirectional stratified flow with and without phase change. *International journal of Heat and Mass Transfer*. 48:477-486.
- Ghorai S, Nigam KDP (2000). “CFD modelling of flow profiles and interfacial phenomena in two-phase flow in pipes” , Department of chemical Engineering, Indian Institute of Technology, Haus KHas, New Delhi 16 India.
- Rune W. Time “Two-Phase Flow in Pipelines (Course Compendium)”; Department of Petroleum Engineering, Faculty of Science and Technology, University of Stavanger, Norway, September 17th 2007.
- Berthelsen PA and Ytrehus T. (2005). Calculations of stratified wavy two-phase flow in pipes, The Fluids Engineering Group, Department of Energy and Process Engineering, Norwegian University of Science and Technology, Kolbjørn Hejes vei 2, N-7491 Trondheim, Norway
- Moguedet M, Namy P and Béreaux Y (2007). “ On the Use of Comsol Multiphysics® to Understand and Optimize the Filling Phase in Injection and Micro-Injection Molding Process. Excerpt from the Proceedings of the COMSOL Users Conference Grenoble.