27/28th International Conference
ENGINEERING MECHANICS 2022
Milovy, Czech Republic, May 9 –12, 2022
Paper **#55,** pp. 241–244, doi: 10.21495/51-2-241

COMPUTATIONAL SIMULATION OF TWO-PHASE IMMISCIBLE FLOW IN HORIZONTAL PIPELINE

Lunda F.[*], Fialová S.[**], Pírková M.[***]

Abstract: *A computational simulation of two phased immiscible flow in the horizontal pipeline is presented in this paper. This flow type is commonly found in several industrial applications, for example pipelines in a subsea oil production system, mixing microfluids and so on. The first part contains basic information about this phenomenon, where the origin of slug flow is described more specifically. The second part of this paper deals with numerical simulation of two-phase immiscible flow in the horizontal pipeline where wavy stratified flow was simulated. Boundary conditions were taken from experimental observations. This regime of flow is simulated in Ansys Fluent using the Volume of Fluid model (VOF) The article describes the procedure needed to successfully simulate the flow regime for two models (2D and 3D) from creating geometry and mesh to setting the correct boundary conditions in the computational software. As a results, two simulations were created, where both solutions were compared between each other and with the experimentally observed flow.*

Keywords: Two-phased immiscible flow, Horizontal pipeline, Ansys Fluent, Volume of fluid method.

1. Introduction

Two-phase immiscible flow in the pipeline is demanding problem. Unfortunately for now, we cannot fully understand this problem that we encounter in several industrial applications from microfluidic mixers to subsea pipelines (Russel 1959). Therefore, several studies were conducted in the past (Russel 1959; Angeli 1997), where these studies have focused on the observation and understanding of the two-phase immiscible flow and the final creation of a flow regime map, in which several flow regimes can be observed in the horizontal pipeline. The flow type depends on the kind of liquids, their velocity and type of pipeline. Up to 9 types of regimes were observed in the past (Angeli 1997).

One flow regime is the so-called slug flow, which was initially the primary goal of our research. Two experimental observations were carried out on a specially designed track, where the pipeline was horizontal. For liquids water and corn germ oil were used. In the first observation (Malá 2020), the oil was fed from below and in the second measurement (Lunda 2021), the oil was flowed to pipe from above. For very low velocities, immiscible liquids formed into a straight stratified flow in both cases. With increasing velocity, waves started to appear on the interface, this regime is called wavy stratified flow. After that, bubbles began to separate from the interface and for the highest velocities the flow became dispersed.

One major difference was the occurrence of the slug flow in observation with oil inlet form below. If the oil enters from above, due to its lower density it does not pass through the water layer and thus no oil slugs form. When oil enters from below, it must pass through the layer of water, and the slug flow can be observed for appropriate velocities.

In the present study, wavy stratified flow was simulated based on measurements on the experimental track (Lunda 2021), from where we got the boundary conditions and properties of used liquids. Simulations were done both for 2D and 3D. The results could be compared and evaluated between each other.

[*] Ing. Filip Lunda.: V. Kaplan Department of Fluid Engineering, Brno University of Technology, Technická 2896/2; 616 69, Brno; CZ, 191563@vutbr.cz

[**] Doc. Simona Fialová, PhD.: V. Kaplan Department of Fluid Engineering,Brno University of Technology, Technická 2896/2; 616 69, Brno; CZ, fialova@fme.vutbr.cz

[***] Ing. Marcela Pírková: V. Kaplan Department of Fluid Engineering, Brno University of Technology, Technická 2896/2; 616 69, Brno; CZ, 182476@vutbr.cz

2. Method

Simulation of wavy stratified flow was performed in order to get as close as possible to the experimental observation, where all boundary conditions and properties of liquids were obtained from this experiment (Lunda 2021; Malá 2020).

Geometry

For the geometry, a t-piece with a subsequent horizontal pipeline was chosen. All dimensions were the same as in the experimental (Lunda 2021), so the pipe diameter was 19 mm. The t-piece was specially designed to have no notches in it, which made it very easy for us to create the geometry. T-piece was adjusted so that the oil flowed into the pipe from above. The domain was further simplified by shortening the area where each fluid flowed alone for 26 mm. The length of the horizontal line was 2 meters. Two geometries were created. One is 2D and the second one is 3D.

For meshing, only quadrilateral and hexagonal elements were used. These elements are computationally more accurate and converge faster. In the 2D simulation, a mesh with a size of one element was 0,5 mm led to 160 thousand elements. Meshing in 3D was a little more complicated and led to 4,1 million elements.

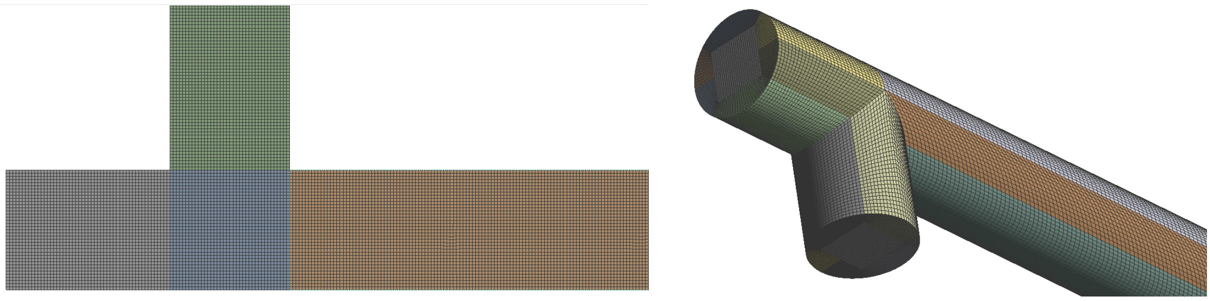

Fig. 1: 2D (left) and 3D (right) meshed geometry

Simulation

The multiphase flow was simulated in the Ansys Fluent 20 using an explicit Volume of Fluid (VOF) model with sharp interface modeling. The VOF model is based on the idea of fraction function. In every cell volume of each fluid is tracked. The cells containing pure liquid are identified by the volume fraction of 0 or 1. The surface tension coefficient was 0.04164 (Malá 2020).

Velocity inlet and pressure outlet were chosen for boundary conditions, where both oil and water velocities were taken from experimental measurement (Lunda 2021). The water velocity was 0,126 m/s, and the oil velocity was 0,024 m/s. Atmospheric pressure was chosen for the pressure boundary condition.

Gravity was considered for all solutions. The simulation was performed at very low velocities, indicating a small Reynolds number. Thanks to that, we used the laminar flow model in the PISO computational method. The QUICK method was used for the moment calculation and the Geo-Reconstruct method for the volume quantity. The time step was limited by the Curant number, which indicates the displacement of elements to the network. If the Curant number is greater, then the calculation does not have time to calculate the values of each element (Trallero 1997). Therefore, we tried to keep the Curant number below 0,5.

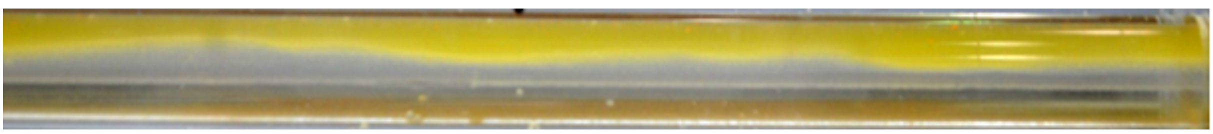

Fig. 2: Experimentally observed flow

Results

Two types of simulation were calculated. Simulation in 2D and 3D, where simulation in 2D was performed transiently for the whole duration of the calculation. First, only water was allowed to flow through the pipe with a time step of 0,01s. After the water flowed along the entire pipeline, the oil supply has also opened, after which we had to reduce the time step to 0,001s. From the solution, we can observe that the stratified flow formed with a smooth interface. The interface starts to curl after a few centimeters. The wavy stratified flow is observable for the rest of the pipeline.

Fig. 3: 2D simulation – Inlet part

Fig. 4: 2D simulation - Irregular waves

Fig. 5: 2D simulation – Bigger waves

When calculating the flow in the 3D pipeline, due to the high complexity, we omitted the procedure in which only water is first to let into the channel, and only then the oil supply is opened. Therefore, we let the calculation run stationary in the first phase of the simulation. After its convergence, the calculation was switched to transient with a time step of 0,001s.

From the results, we see no mixing of water and oil in the front part of the pipeline and immediately, a stratified straight flow is created. It has a smooth interface at first, but a ripple begins to appear after a few centimeters, which is unusually regular. With increasing time, this regularity disappears and small irregular waves with the occasional occurrence of a large wave appear. A significant benefit of a 3D simulation is observing the cross-section in individual parts of the pipeline. However, there are no substantial changes in this regime.

Fig. 6: 3D simulation – Inlet part

Fig. 7: 3D simulation – Irregular waves

Fig. 8: 3D simulation – Bigger waves

Fig. 9: 3D simulation – Cross-section

3. Conclusions

This study successfully simulated the two-phased immiscible flow in the horizontal pipeline in the Ansys Fluent using the VOF method. For the observed flow regime, the stratified wavy flow was chosen. The flow was simulated in 2D and 3D, wherein it was possible to calculate the corresponding flow regime in both cases. In both cases, the magnitude and frequency of the waves matched our experimental observation.

Thanks to this, we can note that it is sufficient to simulate only in 2D pipes for basic observation of immiscible liquids. This significantly simplifies the computation complexity of the simulation.

One way to refine the quality of the simulation is to experimentally measure the contact angle between fluids. However, 3D simulation should be used for more complex observation, as the results are more accurate, and cross-sections can be observed. The main difference between 2D and 3D is what type of pipeline software computes. The main reason for the different results between 2D and 3D simulation is that the 3D pipeline is cylindrical and in 2D the pipeline is prismatic.

Further simulations will focus on other types of flow regimes. These regimes are more complex and should be simulated in a 3D pipeline.

Acknowledgment

This paper was supported by the project "Computer Simulations for Effective Low-Emission Energy Engineering" funded as project No. CZ.02.1.01/0.0/0.0/16_026/0008392 by Operational Program Research, Development and Education and BD FSI-S-20-6235 "Application of the principles of fluid mechanics as a tool for a sustainable resource".

References

Angeli, P. (1997) *Liquid-liquid dispersed flows in horizontal pipes*. Imperial College, University of London, London.

Lunda, F. (2021) *Study of the flow at the interface of immiscible liquids*. Master thesis University of Technology, Faculty of mechanical engineering, Energy institute. Supervisor: Simona Fialová Master thesis, Brno University of technology, Brno https://www.vutbr.cz/studenti/zav-prace/detail/132500, (in Czech).

Malá, K. (2020) *Study of the flow of the immiscible liquids*. Master thesis, . University of Technology, Faculty of mechanical engineering, Energy institute. Supervisor: Simona Fialová Master thesis, Brno University of technology, Brno, https://www.vutbr.cz/studenti/zav-prace/detail/124883, (in Czech).

Russel, T. Hodgson, and Govier, G. W. (1959) Horizontal pipeline flow of mixtures of oil and water. *The Canadian Journal of Chemical engineering*, 37(1), 9-17.

Trallero, J. L. Sarica, and C. Brill, J. P. (1997) A study of oil-water flow patterns in horizontal pipes. *Pipes. SPE Production & Facilities* [online]. 12(03), 165-172 ISSN 1064-668X.