Available online at http://ijcpe.uobaghdad.edu.iq and www.iasj.net
Iraqi Journal of Chemical and Petroleum Engineering
Vol.21 No.1 (March 2020) 67 – 74
EISSN: 2618-0707, PISSN: 1997-4884

Simulation of Two Phase Flow Mixing Co – Current in T Junction Using Comsol

Wallaa A. Noori, Dhiyaa A. Hussein AlTimimi and Bashar Jawad Kadhim

University of Technology

Abstract

The analysis, behavior of two-phase flow incompressible fluid in T-juction is done by using "A Computational Fluid Dynamic (CFD) model" that application division of different in industries. The level set method was based in "Finite Element method". In our search the behavior of two phase flow (oil and water) was studed. The two-phase flow is taken to simulate by using comsol software 4.3. The multivariable was studying such as velocity distribution, share rate, pressure and the fraction of volume at various times. The velocity was employed at the inlet (0.2633, 0.1316, 0.0547 and 0.0283 m/s) for water and (0.1316 m/s) for oil, over and above the pressure set at outlet as a boundary condition. It was observed through the program that the shear rate increased in the mixing area and begins to decrease after the mixing area, for the pressure suddenly decreases in the mixing area and after this area begins to decrease linearly with the length of the tube.

Keywords: Two-phase Flow, comsol CFD, level Method, Simulation

Received on 31/05/2018, Accepted on 23/03/2019, published on 30/03/2020

https://doi.org/10.31699/IJCPE.2020.1.10

1- Introduction

In nature and industrial application, the two-phase flow appears broadly, the term mixture called on the two-phase flow [1].

The parameter design can be simply modeled in a single-phase but the difficulty appears in two-phase of fluid flow, the complexity still exists to understand due to difficult behavior [2].

Advances in technology the mathematical models were interested to define the two-phase flow this attention becomes a challenge to the examiner [3], [4] , [5].

Usually, In various forms, two-phase can appear such as gas flow against the liquid flow, two liquids, liquid with existing solid.

In petroleum manufacturing such as the production of crude oil and carrying one of the important phenomena in two-phase flow between oil and water. Different phases influence every other hydrodynamically the combination for two-phase such as gas phase with liquid phase, liquid-liquid phase with solid, and gas-phase with solid small certain kind of two-phase flow, frequently saw.

In broad variety the two or multiphase occurrence in nature such as drops of rain in the air, volcanoes, sandstorms and snowstorm to additional practical uses in industries such as power manufacture, chemical processing, medicine and pharmaceuticals.

In interior combustion engines, thinly atomized oil spray is injected into compressed air for efficient combustion and hence, fewer atmospheric contamination.

The mixture that consists of oil and gas in the pipe in the system such as refrigeration, air-condition and water-oil mixture such as flow in the pipe, the transport of this system shows the two-phase flow. Not needed to say, a well understanding of the physical phenomenon occurrence in multiphase flows will advantage to make present applications additional efficient and environmentally friendly[6].

Several sorts of glide forms were expected by specific section speeds of the waft of binary immiscible liquids over the parallel pipelines by means of the volume of fluid approach.

The projectile goes with the stream looked at small speeds, but the flow in the annular, the pressure behavior of mix inverse relationship with the oil velocity. Also, it was observed that the quantity portion of oil is in two position center pipe and higher side of the pipe in the upper if the applied flow is maximum [2].

The relationship between the pressure gradient and the velocity in case stratified for two-phase flow oil-water is positive [7], [8].

Today, Computational Fluid Dynamic (CFD) has progressed toward becoming a very important tool in simulation studies.

Several multidimensional statistical models were moving ahead to suggest stratified flows. These approaches for example Lagrangian, the volume of fluid (VOF), and level set. These approaches are in rule capable to minutely capture most of the physics of the applied flows.

Corresponding Authors: Name: Wallaa A. Noori , Email: 80020@uotechnology.edu.iq

Bubbles and droplets are examples for small morphological formations these formations they cannot capture if the mesh is not sensible minor enough. One of the primaries tries to simulate mix inflows as joined the VOF method with a two-fluid model in order to transfer simultaneously the advantages of both analytical formulations. Another try a multifluid simulation of slugging phenomena in horizontal channels that existing a mechanistic move toward the guess of hydrodynamic slug beginning, growth and resulting improvement into nonstop slug stream in lines [9].

In our work, the model of two-phase flow is worked out by means of the simulation program based on the level set approach. The numerical technique was used to find an approximation solution using the finite element method. It similarly offers a lot of roles that make the difference of differential equations between the node [10]. The oil-water streams are exceptionally normal in the oil ventures. For the most part, the oil stage is moved in a multiphase stream state as water and oil are ordinarily delivered collectively. The closeness of water has a noteworthy impact amid the carriage of the oil [11]-**Error! Reference source not found.**.

In our investigation, the two-phase flow of mix oil-water is assumed based on the finite element method (FEM) wetted wall boundary condition. The investigation includes study behavior of velocity, pressure, a fraction of volume through a pipe with a certain condition.

2- Selection Of The Discretization Technique

The equations of the statistical approaches for multiphase flows are typically numerical method by means of the finite volume model (FVM) or FEM Method in an area over with limited finite-difference mode (FDM) for time discretization. An essential variance amidst these two approaches is that the FEM could be simply enhanced in giving high request precision with additional computational monetary value, while it ends up mind-boggling and hard to comprehend in high request exactness in alternate strategies **Error! Reference source not found.**.

2.1. The Level Set Method

The set technique is a statistical method that became added in osher & sethian's paper [16]. The set strategy has been one extremely advantageous structure from both hypothetical and numerical perspectives. It permits to define dramatic alteration in areas (as well as topological alteration) in a very strong way [17]. Particular data on the set approach could be determined by way of osher and fedkiw [18].

The concept of the set technique is to describe the interface among the multi fluids with the aid of the usage of tacit function, the extent set feature ∅(x), as shown in Fig. **1**. later decide on the kind of the tacit function, the area value that provide ∅(x) = 0 create the interface among the multi incompressible liquids.

Another benefit of the set approach is that geometries that trade topology including division, and evolving holes, be able to be followed effortlessly and any other one is that the grid of the area does no longer want to be changed. The set method from its algebraic thought to its statistical calculations is obtainable within the next segments **Error! Reference source not found.**.

2.2. Mathematical Model Formulation

In this segment, an algebraic form could be advanced to talk about the two-phase flow. the fluid Flow simulation can be built by the computational domain. An appropriate grid is taken into concern as the simulation. The condition situations are for the multi-phase (oil-water) in a level set. In this research, two-phase consist of two fluid incompressible and flow laminar,

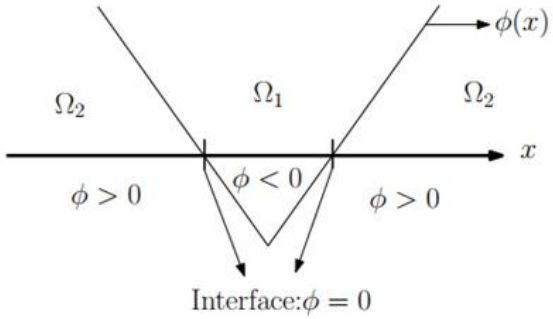


Fig. 1. Viewing the interface by an employ set function∅

The governing equations of this situation become as follows at the Initial condition (IC) and Boundary condition (BC).

In the computational area Ω, where u present the mixture velocity, ρ present density, and μ present viscosity, g present the gravity and the surface tension force is present by F_{st}.

$$\nabla . u = 0 \quad (1)$$

$$\rho\left(\frac{\partial u}{\partial t} + u\,.\nabla u\right) = \nabla p + \nabla.\left(\mu(\nabla u + \nabla u^T)\right) + \rho g + F_{st}, \quad (2)$$

$$\frac{\partial \emptyset}{\partial t} + u.\nabla\emptyset = \gamma\nabla.(\varepsilon\nabla\emptyset - \emptyset(1-\emptyset)\tfrac{\nabla\emptyset}{|\nabla\emptyset|}, \emptyset = phils \quad (3)$$

Where: ϕ is the phase arena dimensionless changeable, γ movement, ε is a regulatory interface factor, the mixture properties such as viscosity and density are the function of the water volume fraction V_w . The water and oil volume fraction are Vw =(1 + ϕ)/2 and V0 =(1- ϕ)/2 respectively.

Over the interface viscosity and density are computed for the two-phase model according to:

$$\rho = \rho_0 + (\rho_w + \rho_0)\, V_w \quad (4)$$

$$\mu = \mu_0 + (\mu_w - \mu_0) V_w \quad (5)$$

Where: w and o that subscript to indicate the water and oil respectively **Error! Reference source not found.**

The velocity of water (0.2633, 0.1316, 0.0547 and 0.0283 m/s) and oil (0.1316 m/s). The velocity u = u0, is the base at the inlet and the condition at the wall wetted Fig. **2** present 2d domain for the circular tube with dimension (5 m Width x 12.7[mm] Height) and Fig. **3** present mesh design.

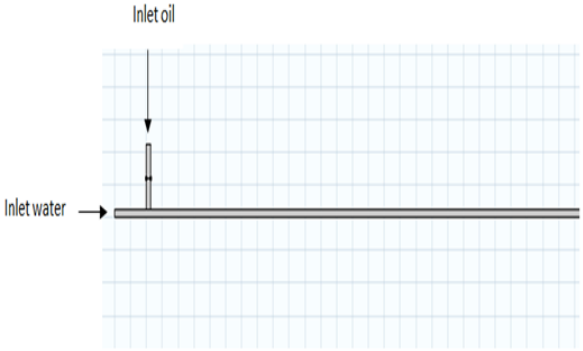


Fig. 2. The field geometry

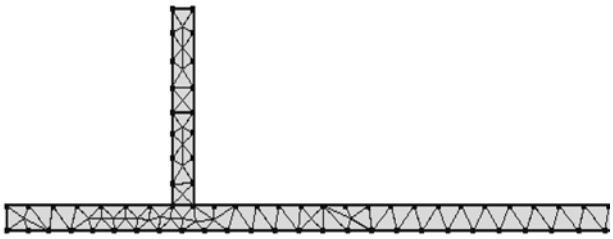


Fig. 3. The field meshes

The physical properties of two fluid are given in Table **1** and the parameter value is given in Table **2**.

Table 1. Fluid phases of physical properties

Property	sign	Phase of H_2O	Phase of oil
Dynamic viscosity(Pa·s)	μ	0.001003 Pa·s	0.0055
Density (kg/m^3)	ρ	998.2 kg/m3	828
Oil-water Interfacial Tension([mN/m])	τ	39.6	

Table 2. Parameters values used in simulation

sign	Parameters	Values
ε	Parameter controlling interface thickness	tpf.hmax/2 m
g	Gravity	9.8 m/s^2
ϒ	Reinitialization	0.9 m/s

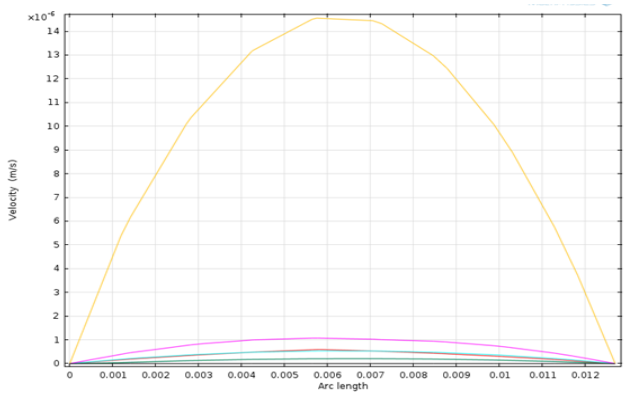
3- Results and Discussion

In our research, the version of comsol that employed was 4.3 to simulate two-phase flow (oil-water). Fig (4) present the velocity distribution for different positions (distance range from edge (0,1,5)).

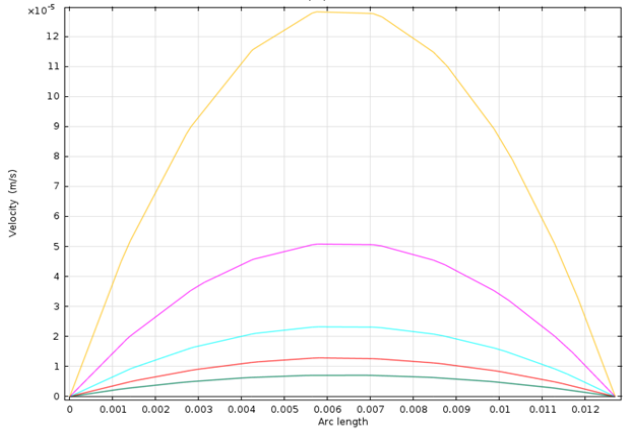
Figs. show it shows the pressure suddenly drops at the mixing zone and after this zone its decreases linearly with pipe length, this means the velocity in pipe increase until reaching the end tube.

The volume fraction of fluid one (water) and fluid two (oil) was studied at a different velocity, the **Fig. 7** & Fig. **8** present volume fraction of water phase and oil phase line graph and two dimensions respectively, the figs. display the different behavior according to the velocity of the water middle pipe.

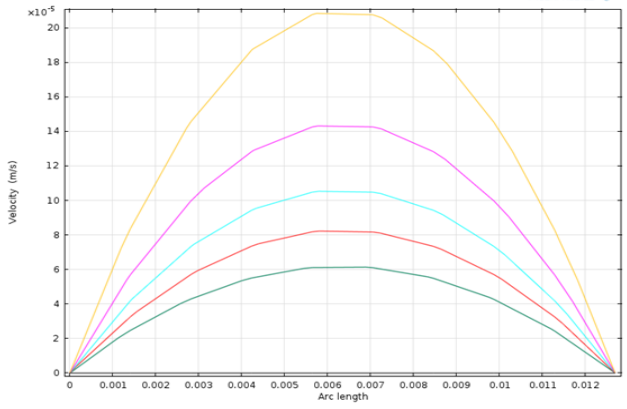
Fig. **5** clearly shows it shows the distribution of shear rate with increases in the mixed zone and after this zone it begins to decrease with length with some phases, but in general, the velocity is low at the boundary of pipe and then increases at the fluctuation difference between shapes because of different velocity and volume fraction. The pressure drop along the pipe at different velocity was studied and present in Fig. **6**.



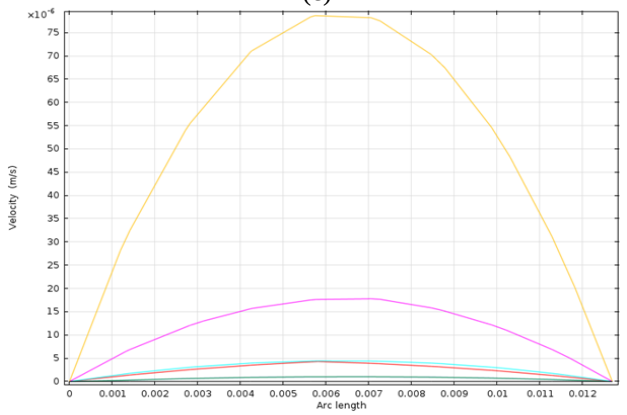
(a)



(b)



(c)



(d)

Fig. 4. Velocity dist. At diff. distance range (0, 1, 5) (a) v1 (b) v2 (c) v3 (d) v4

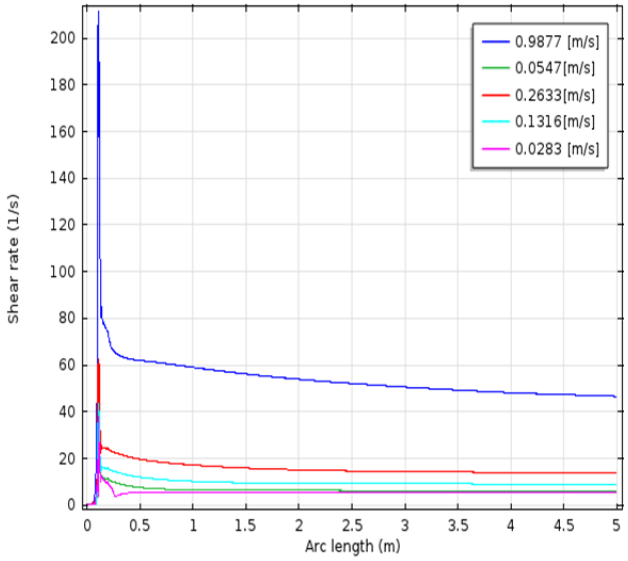


Fig. 5. Share rate against length domain

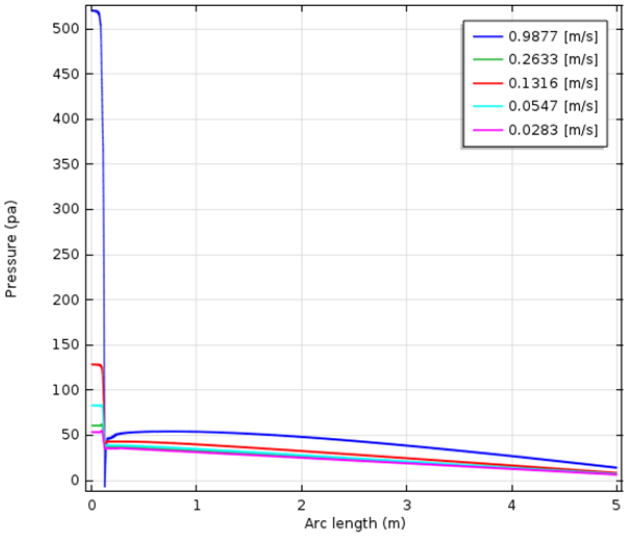


Fig. 6. Pressure drop against length domain

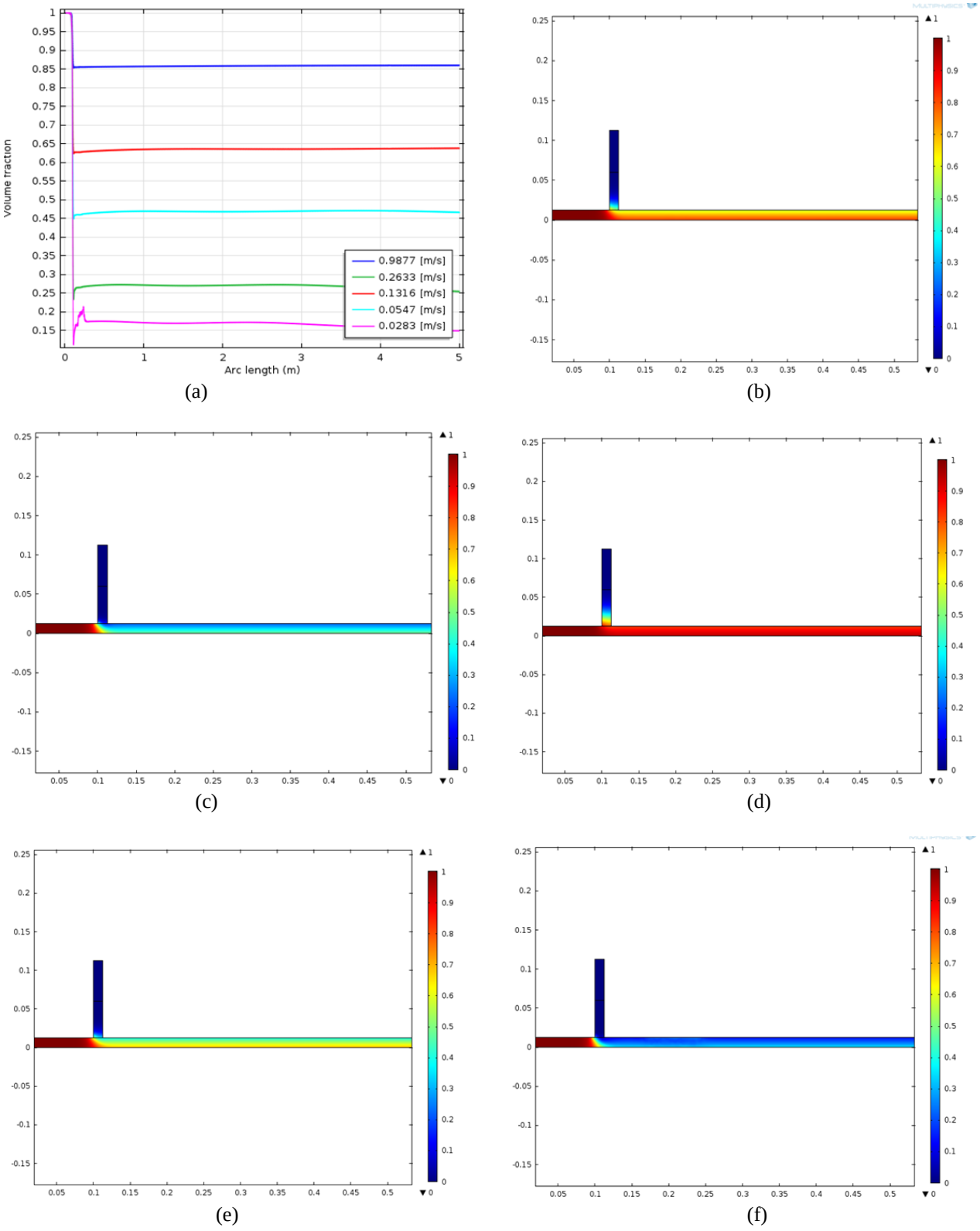


Fig. 7. present volume fraction of water against pipe length at different velocity (a) line graph ,(b) 2d v1, (c)2d v2, (d)2d v3, (e) 2d v4, (f) 2d v5.

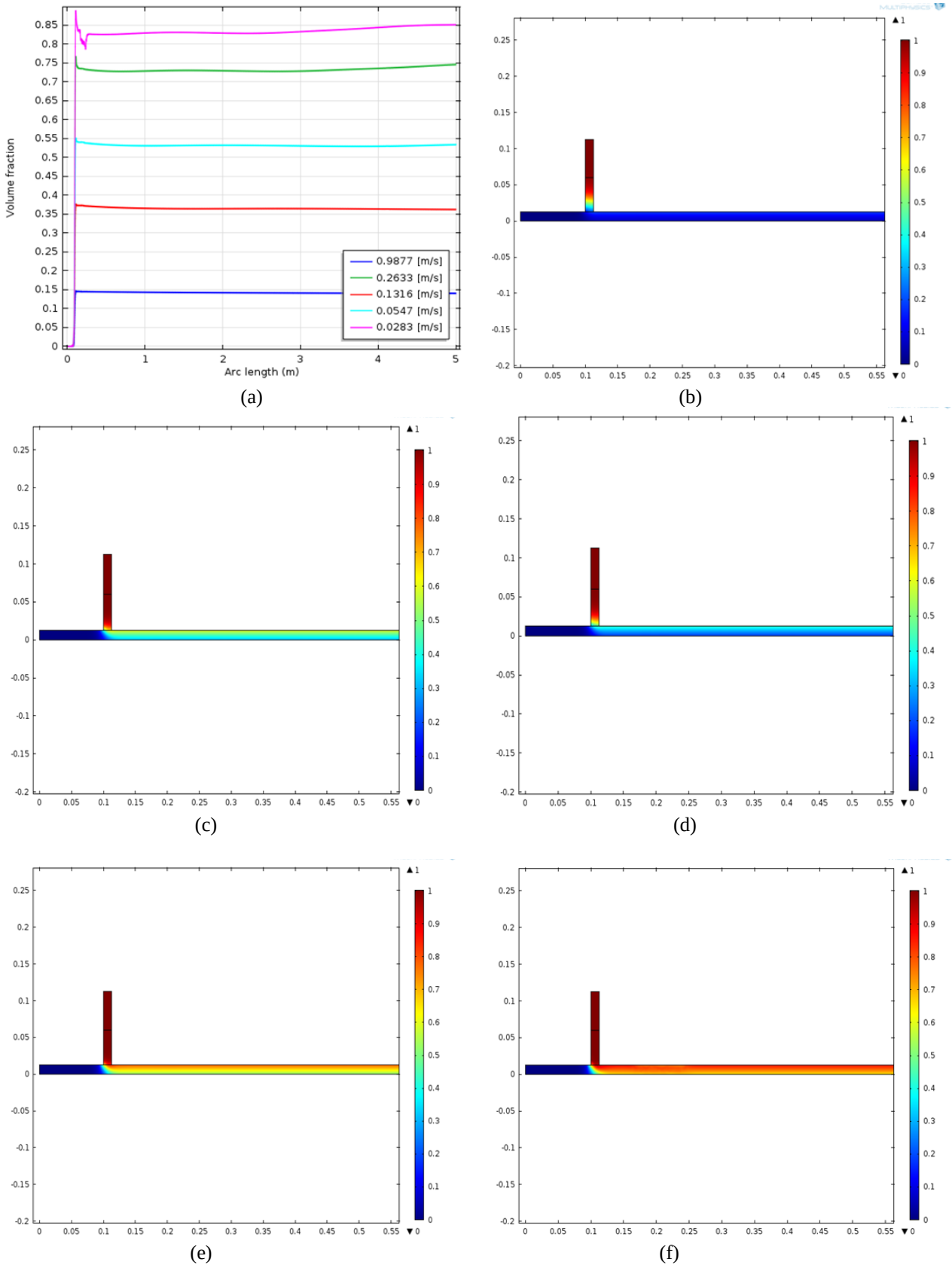


Fig. 8. present volume fraction of oil against the pipe length at different velocity (a) line graph ,(b)2d v1, (c)2d v2, (d)2d v3, (e) 2d v4, (f) 2d v5

4- Conclusion

Two-phase flow (water-oil) is simulated by comsol, the level set method was based on the CFD model. According to level set gained the difference report of the fluid flow observed the velocity magnitude is higher in pipe center and velocity at the boundary of the pipe is lowest. The shear rate increased in the mixed zone after this zone decreased with pipe length with small fluctuation. The volume fraction is variously with various velocities. The pressure drop observed in the pipe is a linear change expect mix zone.

References

[1] Drew, D.A., (1983), "Mathematical Modeling of Two-Phase Flow". Annual Reviews on Fluid Mechanics, 15, 261-291.

[2] Desamala, A.B., Dasamahapatra, A.K. and Mandal, T.K., (2014), "Oil-Water Two-phase Flow Characteristics in Horizontal Pipeline A Comprehensive CFD Study". International journal of Chemical, Molecular, Nuclear, Materials and Metallurgical Engineering, World Academy of Science, Engineering and Technology, 8, 360-364.

[3] Liao, Q., Liu, D., Ye, D., Zhu, X. and Lee, D., (2011), "Mathematical Modelling of Two-phase Flow and Transport in an Immobilized-Cell Photo Bioreactor". International Journal of Hydrogen Energy, 36,13939-13948.

[4] Morin, A., (2012), "Mathematical Modelling and Numerical Simulation of Two-phase Multi-component Flows of CO2 Mixtures in Pipes". PhD Thesis, Norwegian University of Science and Technology, Norway.

[5] Hudson, J. and Harris, D., (2006), "Numerical Modeling of Eulerian Two-Phase Gas-Solid Flow". Manchester Institute of Mathematical Sciences, Manchester, 1-41.

[6] Rahul Garg (2009)."Modeling and Simulation of two-phase flows" lib.dr.iastate.edu/etd/10657

[7] Alias, A.Z.M., Koto, J. and Ahmed, Y.M., (2015), "CFD Simulation for Stratified Oil-Water Two-Phase Flow in a Horizontal Pipe". International Society of Ocean, Mechanical and Aerospace Scientists and Engineers, 2, 1-6.

[8] Hu, H. and Cheng, Y.F., (2016), "Modelling by Computational Fluid Dynamics Simulation of Pipeline Corrosion in CO2-Containing Oil-Water Two-phase Flow". Journal of Petroleum Science and Engineering, 146, 134-141.

[9] Thomas HÖHNE, (2009), "Experiments And Numerical Simulations Of Horizontal Two Phase Flow Regimes" Seventh International Conference on CFD in the Minerals and Process Industries CSIRO, Melbourne, Australia.

[10] Yu, W.H. and Wiwatanapattaphee, B., (2006), "Finite Element Method and Application". Mistercopy Publishing Company,Bangkok.

[11] Abed, E.M. and Auda, Z.A., (2014), "Simulation and Experiment of Oil-Water Flow with Effect of Heat Transfer in Horizontal Pipe". International Journal of Mechanical Engineering and Applications, 2,117-127.

[12] H. Neamah and A. A.Alrazzaq, "Torque and Drag Forces Problems in Highly Deviated Oil Well", *ijcpe*, vol. 19, no. 3, pp. 19-31, Sep. 2018..

[13] S. A. Lazim, S. M. Hamd-Allah, and A. Jawad, "Permeability Estimation for Carbonate Reservoir (Case Study/ South Iraqi Field)", *ijcpe*, vol. 19, no. 3, pp. 41-45, Sep. 2018.

[14] N. M. Farhan, "Effect of Nanoparticles and Surfactant on Phase Inversion of Two Phases", *ijcpe*, vol. 18, no. 1, pp. 121-128, Mar. 2017.

[15] Deniz Kennedy, (2014), "Level Set Methods for Two-Phase Flows with FEM" Examensarbete 30 hp Department of Information Technology Uppsala university.

[16] Giordano, M., Bonfiglioli, A. and Magi, V., (2006), "A Parallel Finite Element Method for Two-phase Flows". European Conference on Computational Fluid Dynamics, Glasgow, 1-20.

[17] M. Sussman, P. Smereka, and S. Osher, (1994), "A level set method for computing solutions to incompressible two-phase flow", J. Comput. Phys., 114, 146-159.

[18] S. Osher and R. Fedkiw, (2003), "Level set methods and dynamics implicit surfaces", Volume **153** of Applied Mathematical Sciences, Springer-Verlag, New York.

محاكاة عملية خلط نظام متعدد الاطوار في انبوب متعامد باستخدام برنامج الكومسل

ولاء عبد الهادي نوري, ضياء عبد الرسول حسين و بشار جواد كاظم

الجامعة التكنولوجية

الخلاصة

تحليل سلوكية الجريان المتعدد الاطوار للموائع الغير انضغاطية في في الانبوب المتعامد ينجز بواسطة المحاكاة باستخدام البرامج الحاسوبية والتي تستخدم لمختلف التطبيقات الصناعية. في بحثنا تم دراسة سلوك تدفق نظام متعدد (ماء مع نفط) باستخدام موديل المستويات باستخدام برنامج الكومسل اصدار 4.3. تم دراسة العديد من المتغيراة مثل توزيع السرعة و ومعدل القص والضغط و النسبة الحجمية للموائع. تم استخدام سرع مختلفة للماء (0.2633 ، 0.1316 ، 0.0547 و 0.0283 متر / ثانية) و (0.1316 متر/ثانية). لوحظ من خلال البرنامج تزايد معدل القص في منطقة الخلط ويبدأ بالتناقص بعد منطقة الخلط, بالنسبة للضغط ينخفض فجأة في منطقة الخلط وبعد هذه المنطقة يبدا بالانخفاض بصورة خطية مع طول الانبوب.

الكلمات الدالة: نظام متعدد الاطوار , كومسل , محاكاة, ديناميكية الموائع