京都大学防災研究所年報 第 60 号 B 平成 29 年
DPRI Annuals, No. 60 B, 2017

Hydrodynamic Modeling of Air-Water 2-Phase Flow in Urban Sewer System : Turbulence Modelling of Stratified Flows

Chong Soo HAN, Kenji KAWAIKE and Hajime NAKAGAWA

Synopsis

Stratified 2-phase flow is one of the flow regimes that is of importance in multiphase flow transport through channels and pipelines. The phenomenon of turbulence complicates the stratified flow behavior. Conducting accurate simulation in a turbulent channel or pipe flow can lead to better designs of 2-phase flow systems.

In this paper, the common RANS turbulence models produce too much turbulence at the water-gas interface. Therefore, these model need to be modified at the interface. To ensure this, ω which is function of the surface roughness factor(k_s) need to represent the effect of interface waves.

The main objective is to find and test a model for k_s and to apply that to Standard k-ω turbulence model and compare with former experimental data and Fluent simulation model. MATLAB calculates the flow rates better than Fluent comparing with experimental data, because MATLAB can impose an interface condition straightforward.

Keywords: 2-phase flow, 2-phase Interface, Turbulence Models

1. Introduction

Urban inundation disasters due to increasing of torrential local rainfall and decreasing of impermeable area cause serious problem in many countries. To mitigate the damage of urban inundation, various structural strategies have been carried out, one of which is an underground drainage pipe system. Deeply underground, especially development area in mega city, drainage pipe system with huge-diameter has been laid in recent years. In order to estimate the effect of pipe system, diversion flow rate must be estimated precisely.

However, when the water flows into the pipes, the air will be taken into the flow as well which makes it difficult to estimate the flow discharge. This kind of flow containing air-mass or bubbles is referred as 2-phase flow. In a drainage pipe shows the different hydraulic characteristics than usual channel and pipes flows.

Stratified 2-phase flow is one of the flow regimes that is of importance in 2-phase flow transport through channels and pipelines. The phenomenon of turbulence complicates the stratified flow behavior. Conducting accurate simulation in a turbulent channel or pipe flow can lead to better designs of 2-phase flow systems.

2. Literature Review

2.1 2-phase Flow

Multiphase flow can be defined as the presence of multiple immiscible phases which are flowing together in a system. For example, water and gas or water and oil in two-phase flow or water, oil and gas in a three-phase flow. Similarly, the flow can be broadly classified topologically as separated, dispersed or mixed and classified in various flow regimes : stratified flow, slug flow, annular flow and bubbly flow. The hatched areas that are shown in the example flow pattern map in Fig.1 give an approximation to where the flow patterns changes, and the solid lines are theoretical predictions. In this research focused on stratified flow in horizontal channels and pipes. As shown in Fig. 1, stratified flow is found for low gas and liquid flow rate.

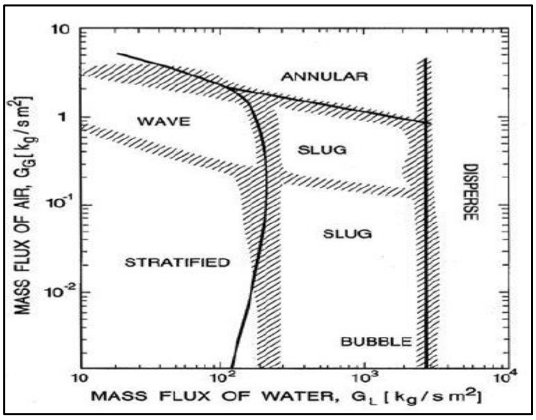


Fig. 1 Typical flow pattern map of horizontal flow (Weisman, 1983)

Stratified flow is important in many industrial applications. The stratified flow is characterized by a sharp interface between the phases. For the proper design of 2-phase flows, the accurate simulations should be done in the system.

2.2 2-phase flow analysis techniques

The ability to simulate multi-phase flows is an important addition to the Computational Fluid Dynamics(CFD) techniques and increases its range of applicability to flows of engineering interest considerably. The problem in the CFD analysis of 2-phase flow systems is how to represent the multi-phase nature of the flow. A multi-phase flow consist of many single-phase flow regimes including interface between each phases or physical boundaries enclosing flows. However, the interfaces between phases are small, often highly distorted with the local flow conditions, this would lead to incorrect prediction, so this approach requires detailed calculations within each phase (David P.H, 1998).

In the past, 'Direct Simulation' approach has not been adopted because it poses significant numerical problems. Recent advanced approaches is applied to the study of free surface flows and the motion of large bubbles. For the turbulent flows, it become possible to calculate simple turbulent flows with a transitional Reynolds number (HK Versteeg and W Malalasekera, 1995)

For the single-phase turbulent flows, the process of averaging the microscopic equations are well established (HK Versteeg and W Malalasekera, 1995), but in the 2-phase case, the problem is more complicated and the exact form chosen depends on the physical nature of the flow situation.

According the Lorencez et al. (1997) the gas flow over the liquid imposes a shear stress which consequently results in the formation of the interfacial waves. The research also highlighted that turbulent eddies are generated at the wall as well as at the interface. This implies that the interface behaves as a rough wall for the gas phase and a moving wall for the liquid phase.

Holmås et al. (2005) claimed that the regular turbulence models do not accurately predict the interfacial effects. In their simulations an upward shift of the gas phase velocity was noticed, which was due to an overestimation of the turbulence (e.g. reflected by the turbulent viscosity) at the interface. The lack of turbulence damping at the interface gives an asymmetric gas velocity profile because turbulence is damped along the top wall not along the interface. They concluded that the present turbulence models cannot conduct accurately for the 2-phase flows, it needs to modify the turbulence models. Wilcox (1998, 2006) has defined a boundary condition for rough fixed walls in his low Reynolds k- ω model for liquid-gas interface.

2.3 Channel flow experiment

The experimental cases that will be used for the validation of the simulations in this thesis are Fabre et al. (1987) and Akai et al. (1980).

Fabre et al. (1987) performed experiments in a channel that is 12.6 m long, 0.2 m wide and 0.1 m high and it has an inclination of 0.1%. The length of the channel ensures that the flow is fully developed. The flow rate of air could be changed, while that of water was kept constant. The corresponding pressure drops and interface heights were measured. The three test cases or runs are shown in Table 1, where the interface is smooth for Run 250 and it is slightly wavy for Run 400 and rough/wavy for Run 600.

The second set of experiments was obtained by Akai et al. (1980), who run experiments for a stratified mercury-air case. The channel was 3.6 m in length, 0.048 m in width and 0.018 m in height. The channel was kept perfectly horizontal implying that there was no streamwise effect of gravity. The strength of the shear stress at the walls was a third of that of the shear stress at the interface. The three test cases or runs are shown in Table 1, where the interface is slightly wavy for Run 1 and it is wavy for Run 2 and wavy for Run 600. In the table Q_g means gas discharge, Q_l means liquid discharge, dp/dx means pressure gradient and h means water height, respectively.

The gas flow rate was varied while the liquid flow rate remained constant. The pressure drop and liquid level were measured and presented in terms of the Lockhart-Martinelli correlation (1949).

Table 1 Experimental results

	Q_g (m^3/s)	Q_l (m^3/s)	dp/dx (Pa/m)	h (m)
Fabre et al. (1987)	Run 250 (Smooth)			
	0.0454	0.003	2.1	0.0380
	Run 400 (Slightly Wavy)			
	0.0754	0.003	6.7	0.0315
	Run 600 (Wavy/Rough)			
	0.1187	0.003	14.8	0.0215
Akai et al. (1980)	Run 1 (Slightly Wavy)			
	0.005	4.2×10^{-5}	84.52	0.63
	Run 2 (Wavy)			
	0.007	4.2×10^{-5}	154.3	0.54
	Run 3 (Wavy)			
	0.01	4.2×10^{-5}	283.652	0.48

3. Turbulence modeling

The relevant theory for this project involves the understanding of stratified two phase flow and of turbulence modelling.

3.1 Multiphase flow

The following quantities are commonly used to describe or model two phase flow in channels or pipes:

1. Holdup fraction (α_k)

2. Superficial velocity (U_{sk})

The subscript s denotes "superficial" and k denotes the phase. The holdup fraction denotes the part of the volume that is occupied by each phase.

The superficial velocity of a phase is defined by:

$$U_{sk} = \frac{\dot{V_k}}{A} \quad (1)$$

Where, $V\dot{k}$ is the volumetric flow rate of the kth phase and A is cross sectional area of the channel or pipe.

Modelling of the stratified flow can be done by combining the two force balances for each phase, i.e. of the gas (G) and of the liquid (L).

The force balance equations for the gas and liquid phase in a channel are:

$$-\propto_G \frac{dp}{dx} = \frac{\tau_{wG}}{H} + \frac{\tau_i}{H} + \propto_G \rho_G g sin\theta \quad (2)$$

$$-\propto_L \frac{dp}{dx} = \frac{\tau_{wL}}{H} + \frac{\tau_i}{H} + \propto_L \rho_L g sin\theta \quad (3)$$

In a 2D flow, $\alpha_G=(H-h)/H$ and $\alpha_L=h/H$

The interfacial shear can be calculated by subtracting equations (2) and (3)

$$\frac{\tau_{wG}}{\alpha_G H} - \frac{\tau_{wL}}{\alpha_L H} + \frac{\tau_i}{\alpha_G \alpha_L H} - (\rho_L - \rho_G) g sin = 0 \quad (4)$$

The wall shear stresses can be expressed as

$$\tau_{wk} = \mu_k \frac{\partial \mu_k}{\partial y} \quad (5)$$

3.2 Turbulence

The theory behind the phenomenon of turbulence will be explained below.

$$\frac{\partial \bar{u}_i}{\partial t} + u_j \frac{\partial \bar{u}_i}{\partial x_j} = \frac{1}{\rho}\frac{\partial \bar{P}}{\partial x_j} + \frac{\partial}{\partial x_j}\left(\frac{v \partial \bar{u}_i}{\partial x_j} - \overline{u_i' u_j'}\right) + g_i \quad (6)$$

Where the extra fluctuating term ($\overline{u'_i u'_j}$) is the Reynolds Stress tensor. Equation (8) is known as the Reynolds Averaged Navier Stokes equation (RANS). To solve the RANS equation closure relations are needed for the Reynolds stresses.

Standard k- ω model

The kinetic energy and specific dissipation rate transport equations for the SKW are as follows.

$$\frac{\partial}{\partial t}(\rho k) + \nabla(\rho v k) = \nabla\left(\mu_{eff,k} \nabla k\right) + P_k - \beta^* \rho k \omega \qquad (7)$$

$$\frac{\partial}{\partial t}(\rho \omega) + \nabla(\rho v \omega) = \nabla\left(\mu_{eff,\omega} \nabla \omega\right) + C_{\alpha 1} \frac{\omega}{k} P_k - C_{\beta 1} \rho k \omega^2 \qquad (8)$$

k : kinematic energy, w: the rate of dissipation of turbulence kinetic energy per unit mass due to viscous stresses, $C_{\alpha 1} = \frac{5}{9}$, $C_{\beta 1} = 0.075$, $\beta^* = 0.09$.

The Baseline (BSL) k- ω model

The BSL model is the basic combination of the Standard k - ω and Standard k - ϵ models.

The BSL k – ω model is derived by multiplying the k – ω with a blending function F_1 and the k – ω formulation of the k- ϵ model equations by ($1-F_1$), yielding the following equations for k and ω.

$$\frac{\partial}{\partial t}(\rho k) + \nabla(\rho v k) = \nabla\left(\mu_{eff,k} \nabla k\right) + P_k - \beta^* \rho k \omega \qquad (9)$$

$$\frac{\partial}{\partial t}(\rho \omega) + \nabla(\rho v \omega) = \nabla\left(\mu_{eff,\omega} \nabla \omega\right) + C_\alpha \frac{\omega}{k} P_k - C_\beta \rho k \omega^2 + 2(1-F_1)\sigma_{\omega 2} \frac{\rho}{\omega} \nabla k \nabla \omega \qquad (10)$$

These equations are formally very similar to those of the standard k – ω model, however all their coefficients depend on the blending function F_1.

The blending function F_1 depends on the solution variables and on the distance z from the nearest wall.

$$F_1 = \tanh(\gamma_1{}^4) \qquad (11)$$

$$\gamma_1 = \min(Max\left(\frac{\sqrt{k}}{\beta^* z}, \frac{500v}{z^2 \omega}\right), \frac{4\rho\sigma_{\omega 2} k}{CD_{kw} z^2}) \qquad (12)$$

$$CD_{k\omega} = Max(2\rho\sigma_{\omega 2} \frac{1}{\omega} \nabla k \nabla \omega, 10^{-10}) \qquad (13)$$

$C_{\alpha 1} = 0.5976$, $C_{\beta 1} = 0.075$, $\beta^* = 0.09, \sigma_{k1} = 2, \sigma_{\omega 1} = 2, Pr_t = 0.9$

The BSL model has a similar performance as the k- ω model for boundary layer flows and is nearly identical to the k – ω model for free shear flows. Its robustness is close to that of the k – ω model.

The shear Stress Transport (SST) k- ω model

Menter et al. (1993, 2003) have revised the BSL model. This has changed the formulation of the turbulent viscosity, another blending function F_2 was introduced, and the original closure coefficients are replaced by low Reynolds number corrections.

The expression for the shear stress as resulting from Bradshaw's assumption is shown below. In this assumption the shear stress is taken proportionally to the turbulent kinetic energy:

$$\tau_{xy} = \rho \alpha_1 k \qquad (14)$$

On the other hand, the principal shear stress for conventional 2-equation turbulence models can be computed as

$$\tau_{xy} = \rho \sqrt{\frac{Production\ of\ k}{Dissipation\ of\ k}} \alpha_1 k \qquad (15)$$

$F2$ is the second blending function denoted by

$$F_2 = \tanh(\gamma_2{}^2) \qquad (16)$$

$$\gamma_2 = Max(2\frac{\sqrt{k}}{\beta^* wz}, \frac{500v}{z^2 \omega}) \qquad (17)$$

Moreover, to maintain the original formulation of the eddy-viscosity for free shear layers, the same blending unction approach as for the baseline model is also adopted in the SST k – ω model. The modification is related to the production of turbulence kinetic energy P_k in the k equation, which is replaced by

$$P_k = \min(P_k, 10\beta^* \omega k) \qquad (18)$$

3.3 Wilcox Low Re k – ω model

Wilcox (1998) made the $k-\omega$ model robust by adding low Reynolds number correlations. Also, the advantage that the $k-\omega$ equation has over the $k-\epsilon$ model is that the value of ω can be arbitrarily specified at the surface. Hence, incorporating surface effects like roughness becomes rather easy.

In his low Reynolds number modification Wilcox proposed that

$$\omega = u^2{}_{\tau_i} \frac{S_R}{v_k} \ \ at\ y = 0 \qquad (19)$$

Where $u_{\tau i}$ is the wall friction velocity and ν_k is the viscosity of the considered phase. ω is the specific dissipation rate and its value at the wall depends on the non-dimensionalized surface roughness height k_s+ which is a function of the sand grain roughness or surface roughness k_s:

$$k^+{}_s = \frac{u^2{}_{\tau_i} k_S}{v_k} \tag{20}$$

If $k_s^+ \leq 5$ the surface is considered to be almost smooth or slightly rough.

The boundary condition for slightly rough surfaces proposed by Wilcox (2006).

$$\omega = \frac{40000 v_k}{k_s} \ \ at\ y = 0 \tag{21}$$

Equation (16) was indeed incorporated in the previous MATLAB code of Chinello (2015). Here it was also used as a condition at the liquid-gas interface, where the "surface wall roughness" now in fact represents the "interface waviness". However, the limitation of this approach is that the value of ks needs to be specified (estimated) for each case. Moreover, ks has an SI unit of metre (i.e. it is not dimensionless). To improve its predictability, the code must be able to calculate the value of the surface roughness (interface waviness) using existing models(Charnock, Cohen & Hanratty, Fernandez-Flores and Oliemans).

4. Results

The surface roughness factor is imposed at the interface. The value of ω_i is calculated from the SR expression. What follows is the comparison and testing of different models from the literature. We uses different method to calculate the value of the surface roughness.

4.1 Surface roughness

We use the models given by Charnock (1955), Cohen & Hanratty (1968), Fernandez-Flores (1984) and Oliemans (1987) obtained their expression through experiments and empirical calculations for 2-phase flows.

Tables 2 show the formulae for each of these models for the different cases of Fabre et al. (1987) and shows numerical simulation results using MATLAB code. The result of the model of Charnock (1955) give the closest comparison to the experimental values for the pressure drop and liquid level, followed by the Oliemans (1987) model.

The predicted values by Charnock are closest to the experiment, but the model applies a value for β were chosen by user-specified among 0.39 ~ 0.97 according to interface condition such as smooth, a slightly wavy and a rough (Berthelsen & Ytrehus, 2005). It means the calculation cannot be automated and it depends on user's knowhow whether the interface would be wavy or smooth, therefore it is not reasonable to conduct simulation. So, other there models should be compared with experimental results in terms of the accuracy.

Fig 2. Shows comparison of the surface

Table 2 Simulation and Experimental results

CASE		Experiment	Charnock	Cohen & Hanratty	Fernandez-Flores	Oliemans
	Equations		$k_s = \frac{\beta u_{\tau_2}}{g}$	$k_s = 3\sqrt{2}\,\delta$	$k_s = \delta\sqrt{2[1-(\frac{56 v_G}{u_{\tau_i} \times D_{hG}})^2}$	$2\delta < h_l,\ k_s = 2\sqrt{2.2}\,\delta$ $2\delta \geq h_l,\ k_s = 3\sqrt{2}\,h_l$
Run250 (V_{sl}=0.15m/s, V_{sg}=2.27m/s)	Pressure gradient (Pa/m)	2.1	1.54	1.42	1.4	1.43
	Liquid level (m)	0.04	0.04	0.04	0.04	0.04
	Surface Roughness (m)	5x10^{-4}	3.5x10^{-3}	3.38x10^{-4}	7.8x10^{-5}	4.7x10^{-4}
Run400 (V_{sl}=0.15m/s, V_{sg}=3.27m/s)	Pressure gradient (Pa/m)	6.7	5.2	3.4	3.13	3.53
	Liquid level (m)	0.032	0.030	0.033	0.034	0.0333
	Surface Roughness (m)	0.0154	0.0152	3.1x10^{-3}	8.0x10^{-4}	4.2x10^{-3}
Run600 (V_{sl}=0.15m/s, V_{sg}=3.77m/s)	Pressure gradient (Pa/m)	14.8	13.08	10.7	7	11.6
	Liquid level (m)	0.022	0.021	0.023	0.028	0.022
	Surface Roughness	0.018	0.016	0.013	4.0x10^{-3}	0.0154

roughness with gas inflow rates for each models. The values shown in circles represent the experimental values for each case of 250, 400 and 600 by Fabre et al. It shows big difference values of surface roughness for all models for the runs 250 & 400. The results of runs 600 shows more better agreement than other runs when using Oliemans method. This is why the surface roughness models are known to give better predictions when the gas flow rate increases (Espedal, 1998).

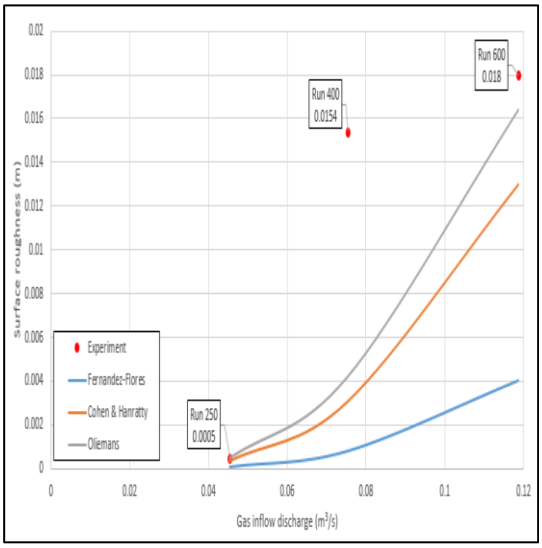


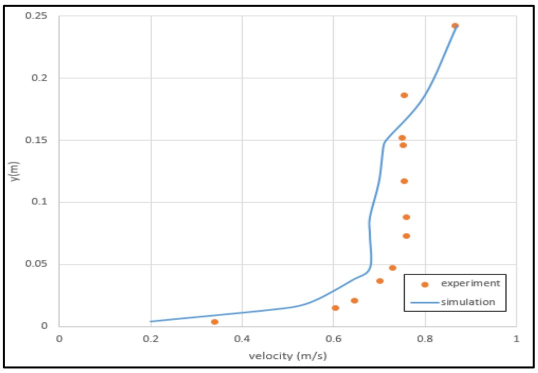
Fig. 2 Simulation surface roughness comparison with the experimental cases by Fabre et al.

All models underpredict the surface roughness. This gives a too low turbulent viscosity and a too high value of specific dissipation rate. This gives a too low pressure drop in the simulations. The inaccurate prediction of ω_i shows why the pressure drop is not the same with experiment.

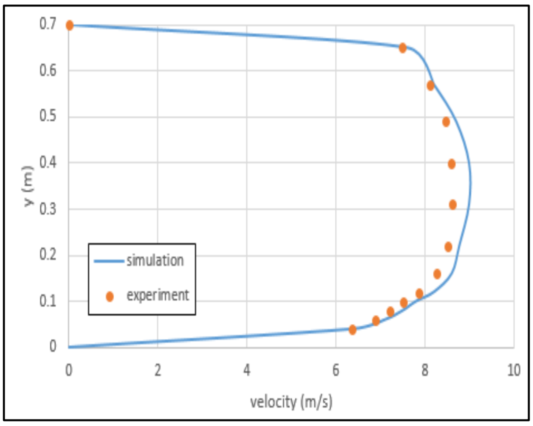
In this simulations, Wilcox (2006) value is being imposed at the interface.

4.2 Velocity using k – ω model

The predicted velocity profiles, using Oliemans method for the surface roughness are shown in Fig. 3. While the predicted liquid velocities are in good agreement with the experiment, the gas velocities shows considerable difference. For the wavy cases (i.e. Run 600) the velocity profiles have a rather gradual shape, while the experiments show a clear peak. This difference in shape is due to the over-prediction of the turbulence at the interface. The gas velocity at the interface side is reduced along the upper wall and it leads to the asymmetric profile.



(a) Velocity profiles in water



(b) Velocity profiles in air

Fig. 3 Predicted velocity profiles compared with the experiments by Fabre et al.

In comparison to the previous model of Chinello (2015), the new simulation model is now more accurate in terms of calculating the value of the surface roughness. But there still differences between the simulations and experiments.

4.3 Comparison with Fluent model

For a meaningful comparison between the Fluent and MATLAB results, the same interface condition ω should be applied in MATLAB. Comparison conducted with Syed results (2016).

For Run 400, the value imposed at the interface according to the output of the Modified MATLAB was $220s^{-1}$. Fig. 4 shows for the velocity profile that due to the UDF the turbulence at the interface in Fluent is indeed being damped (blue curve) which was not the case in the original approach (red curve)

(Syed, 2016). It shows considerable improvement.

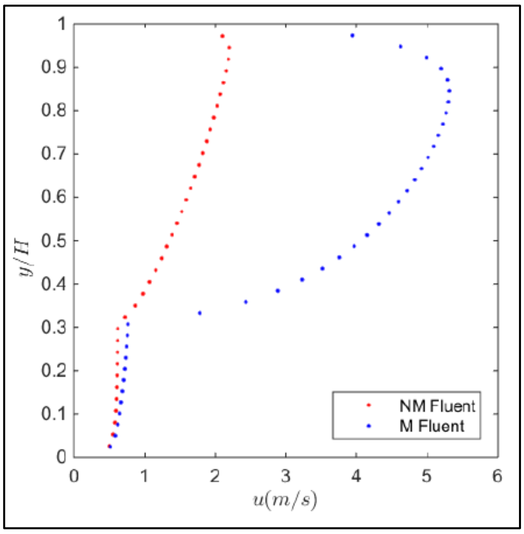


Fig. 4 Fluent results using SKW model with and without interface condition (Syed, 2016)

When comparing with the Modified SKW model, as Fig. 5, there is a clear mismatch between profiles, even though the imposed value for both is same.

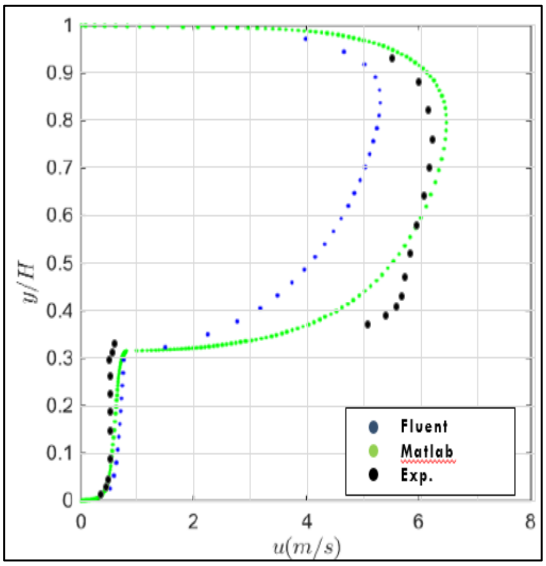


Fig. 5 Comparison with MATLAB, Fluent with interface condition and experiment result.

The predictions with MATLAB and Fluent are not the same. Difference can be explained that Fluent causing the turbulence to be damped.

5. Conclusion

2-Phase flow has various industrial applications. Simulating such flows with a CFD approach is not a trivial task. RANS models have difficulties in handling the turbulence at the interfaces.

The main aim is to improve these models and demonstrate performance to calculate 2-phase flows using MATLAB.

In addition to the interface modelling, also the type of turbulence model must be considered. It can improve the predictions of the pressure gradient, liquid level and velocities by changing turbulence models. The Modified k – ω that is implemented in MATLAB) considerably increased the accuracy results for Run 600.

MATLAB results are compared with predictions by the commercial CFD software package ANSYS Fluent. Fluent accurately predicts single phase flows with or without turbulence in complex geometries with relative ease. However, when a two phase stratified flow is considered the results can be very inaccurate.

This results in a shift of the location of the maximum gas velocity in channel flow towards the top wall. Fluent also has issues when simulating two phase flows, with regards to convergence and oscillating residuals. For the Modified model, the output value of ω_i increases for a certain input value and decreases for some other input values.

The surface roughness calculations with MATLAB are less accurate for lower flow rates. It is recommended to further investigate the encountered turbulence damping phenomenon at the 2-phase interface.

The simulation results are not in agreement with the Run 250 and 400. It should be verified by carrying out with other method or variables or carrying out a similar experiment.

This research has conducted various aspects of RANS models applied to 2-phase flows. Also the possibilities of using CFD tools.

Acknowledgements

This research was supported by a grant (16CTAP-C095650-02) from Infrastructure and Transportation Technology Promotion Program funded by Ministry of Land, Infrastructure and Transport of Korean government.

References

Akai, M., Inoue, A., Aoki, S. & Endo, K. (1980): A Co-Current Stratified Air-Mercury Flow with Wavy Interface. International Journal of Multiphase Flow, Volume 6, pp. 173-190.

Berthelsen, P. A. & Ytrehus, T. (2005): Calculation of stratified wavy two-phase flows in pipes. International Journal of Multiphase Flow, Volume 31, pp. 571-592.

Charnock, H. (1955): Wind Stress on a Water Surface. Quarterly Journal of the Royal Meteorological Society, 81(350), pp. 639-640.

Chinello, G. (2015): CFD Modelling of Two Phase Stratified Flow in Oil and Gas Transportation, Pisa.

Cohen, L. S. & Hanratty, T. J. (1968): Effect of Waves at a Gas-Liquid Interface on a Turbulent Air Flow. Journal of Fluid Mechanics, 31(3), pp. 467-479.

David P.H (1998): The computer simulation of dispersed two-phase flows, Imperial college of science, technology and medicine.

Espedal, M. (1998): An Experimental Investigation of Stratified Two-Phase Pipe Flow at Small Inclinations, Norwegian University of Science and Technology.

Fabre, J., Masbernat, L. & Suzanne, C. (1987): Stratified Flow, Part I: Local Structure. Multiphase Science and Technology, Volume 3, pp. 285-301.

Fernandez-Flores, R. (1984): Etude des Interactions Dynamiques en Ecoulement Diphasique Stratifie, s.l.: Institut National Polytechnique de Toulouse.

HK Versteeg and W Malalasekera (1995): An introduction to computational fluid dynamics: The finite volume method. Longman Scientific and Technical.

Holmas, K. et al. (2005): Simulation of Wavy Stratified Two-Phase Flow using Computational Fluid Dynamics (CFD), BHR Group.

Lorencez, C., Nasr-Esfahany, M., Kawaji, M. & Ojha, M. (1997): Liquid Turbulence Structure at a Sheared and Wavy Gas-Liquid Interface. International Journal of Multiphase Flow, 2(23), pp. 205-226.

Menter, F. R. (1993): Zonal Two Equation k-w Turbulence Models for Aerodynamic Flows. s.l., American Institute of Aeronautics and Astronautics.

Menter, F. R., Kuntz, M. & Langtry, R. (2003): Ten Years of Industrial Experience with the SST Turbulence Model. Turbulence, Heat and Mass Transfer, pp. 625-632.

Oliemans, R. V. A. (1987): Modelling of Gas-Condensate Flow in Horizontal and Inclined Pipes. s.l., ASME Pipeline Engineering Symposium.

Syed Aaquib Hazari (2016): Turbulence modeling of two phase stratified channel and pipe flows, Delft University of Technology.

Weisman, J. (1983): Two Phase Flow Patterns. Chapter 15 of Handbook of Fluids in Motion, pp. 409-425.

Wilcox, D. C. (1998): Turbulence Modeling for CFD. 2nd ed. s.l.:DCW Industries.

Wilcox, D. C. (2006). Turbulence Modeling for CFD. 3rd ed. s.l.:DCW Industries.

(Received June 13, 2017)