
Enass Massoud

- PhD Student: Department of Naval Architecture, Ocean and Marine Engineering (NAOME), University of Strathclyde, Glasgow, United Kingdom.
- Teaching Assistant: Mechanical Engineering Department, Arab Academy for Science, Technology & Maritime Transport (AASTMT), Alexandria, Egypt.

enass-zakaria-shafik-massoud@strath.ac.uk

August 2016
Presentation Overview

1. Introduction.
2. Aim of the study.
3. Model development;
   a) Governing equations.
   b) Model geometry and boundary conditions.
   c) Solution strategy and convergence criterion.
4. Results and discussions;
   a) Taylor bubble shape.
   b) Taylor bubble rise velocity.
   c) Liquid film.
   d) Wake flow structure.
   e) Wall shear stress distribution.
5. Conclusions.
6. Recommendations for future work.
1. Introduction

a. What is Multiphase Flow?

- A multiphase flow is defined as any fluid system in which there is interaction between two or more phases (gas, solid and liquid) where the interface between the phases is influenced by their motion.
1. Introduction

a. What is Multiphase Flow?

- The multiphase flow is involved in natural, technologically processes and industrial applications including nuclear, chemical and petroleum industries.
1. Introduction

b. Classifications of Multiphase Flow?

- Multiphase flow can be classified according to:
  - Classification of Multiphase flow
    - Combination of the phases
      - Two phase flow
      - Three phase flow
      - Four phase flow
    - Structure of the interface (flow regime or flow pattern)
      - Dispersed bubbly flow
      - Intermittent flow
      - Annular flow
1. Introduction

\textit{b. Classifications of Multiphase Flow?}

1) \textbf{Two Phase Flow}

\textit{Gas-liquid flows:}

- It is found widely in a whole range of industrial applications;
  - \textit{Pipeline systems for the transport of oil-gas mixtures},
  - Evaporators,
  - Boilers,
  - Condensers,

\textit{Liquid-liquid flows:}

- \textit{Flow of oil-water mixtures in pipelines.}

\textit{Liquid-solid flows:}

- Liquid-solid suspensions in; crystallization systems, food-mineral transportation, oil processing and nuclear waste management.
1. Introduction

b. Classifications of Multiphase Flow?

1) Two Phase Flow

**Gas-solid flows:**
- Flows of solids suspended in gases are important in;
  - Pulverized fuel combustion,
  - Fluidized beds.

2) Three Phase Flow

**Gas-liquid-liquid flows:**
- Gas-oil-water flows in oil recovery systems.

**Gas-liquid-solid flows:**
- Two Phase Fluidized bed.

**Solid-liquid-liquid flows:**
- Presence of sand or solid particles in oil-water mixtures in pipelines.
1. Introduction

b. Classifications of Multiphase Flow?

3) **Four Phase Flow**

- The three phase flow is often encountered as a mixture of gas, oil and water.
- The presence of sand or other particle may result into four phase flow (G-L-L-S).
1. Introduction

b. Classifications of Multiphase Flow?

Gas-Liquid flow regimes in vertical pipes

<table>
<thead>
<tr>
<th>Sketches of flow regimes for two-phase flow in a vertical pipe</th>
</tr>
</thead>
</table>

Weisman (1983)
A slug unit cell (fundamental unit) consists of an elongated bullet shaped bubbles that fills almost the entire pipeline cross section, known as Taylor Bubble, a liquid film flowing downwards between the bubble interface and pipe wall, known as liquid slug.

The body of the slug is liquid. But in cases of high gas flow rates, some gases diffuse into the liquid slug and is referred to as Aerated Slug.
1. Introduction

c. Slug Flow

- Severe slugging may appear for low gas and liquid flow rates when a section with downward inclination angle (pipeline) is followed by another section with an upward inclination (riser). This configuration is common in offshore petroleum production systems.

Malekzadeh, R. (2012)
1. Introduction

c. Slug Flow

According to Xing et al. (2014), the liquid slugs generated in the Oil and Gas multiphase flowlines can be classified based on their *initiation mechanism* into:

- At early and late stages of production
  - Terrain-induced slugs

- At middle stages of production
  - Hydrodynamic slugs

- At start up and regular pigging operation
  - Operation induced slugs
## 1. Introduction

### c. Slug Flow

### Causes

<table>
<thead>
<tr>
<th>Hydrodynamic Slugging</th>
<th>Terrain-Induced Slugging</th>
<th>Riser-Based Slugging</th>
<th>Operation-Induced Slugging</th>
</tr>
</thead>
</table>

### Effects

| Higher pressure drop for flow | Could cause platform trips and plant shutdown | Reduces capacity of separation and compression units | Enhance corrosion, erosion and fatigue in pipelines |

---

5. Slug Flow in Subsea Oil & Gas Production System

v. Slug flow problems-Why model slug flow?

- Flooding of downstream processing facilities.
- Reservoir flow oscillations and poor management.
- Severe pipe corrosion and structural instability of pipeline.
- The slug induces unsteady loading on pipeline and also on the receiving devices, such as separators. This causes design problems and hence lower the system efficiency and size.
- Instabilities in liquid control system of separators due to high sudden flow rates which may lead to complete shut down.
- Production loss due to the slug induced high average back pressure.
Presentation Overview

1. Introduction.
2. **Aim of the study.**
3. Model development;
   a) Governing equations.
   b) Model geometry and boundary conditions.
   c) Solution strategy and convergence criterion.
4. Results and discussions;
   a) Taylor bubble shape.
   b) Taylor bubble rise velocity.
   c) Liquid film.
   d) Wake flow structure.
   e) Wall shear stress distribution.
5. Conclusions.
6. Recommendations for future work.
2. Aim of the study

- The hydrodynamic analysis for two-phase gas-liquid vertical slug flow can be governed by applying the Buckingham theorem, namely, viscous, inertial, and interfacial forces.

- Eötvös number: the ratio between gravitational forces, and surface tension forces.

\[ E_0 = \frac{\rho_L \cdot g \cdot D^2}{\sigma} \]

- Froude number: the ratio between the inertia and gravitational forces

\[ Fr_{TB} = \frac{U_{TB}}{\sqrt{g \cdot D}} \]

- Inverse viscosity number: the ratio between Eötvös number and Morton number

\[ M = \frac{g \cdot \mu_L^4 (\rho_L - \rho_G)}{\rho_L^2 \cdot \sigma^3} \]

\[ N_f = \frac{\rho_L \cdot (g \cdot D^3)^{0.5}}{\mu_L} \]
2. Aim of the study

- The main aim of the present investigation is to study the hydrodynamics characteristics of single Taylor bubble rising in a stagnant Newtonian liquid using the volume-of-fluid (VOF) methodology implemented in the computational dynamic software package, ANSYS Fluent (Release 15.0).

- The results accounts for; Taylor bubble shape, Taylor bubble rise velocity \( U_{TB} \), liquid film \( U_{LF}, \delta_{LF} \), wake flow structure, and wall shear stress distribution \( \tau_W \).

- The simulation has been performed for 2D, unsteady, laminar flow with constant fluid properties.

- The two phases were assumed as incompressible, viscous, immiscible, and not penetrating each other.
Presentation Overview

1. Introduction.
2. Aim of the study
3. **Model development;**
   a) Governing equations.
   b) Model geometry and boundary conditions.
   c) Solution strategy and convergence criterion.
4. Results and discussions;
   a) Taylor bubble shape.
   b) Taylor bubble rise velocity.
   c) Liquid film.
   d) Wake flow structure.
   e) Wall shear stress distribution.
5. Conclusions.
6. Recommendations for future work.
3. Model Development

a. Governing Equations

- The simulation domain is a vertical pipe of diameter, $D$, and length, $L$.
- The flow consists of single Taylor bubble rising in stagnant fluid.
- The volume-of-fluid (VOF) method is used to track/capture the sharp interfaces between two immiscible fluids (free surface flow).
- This model solves a single set of Navier Stokes equations (NSE) that is shared by the two fluids, and the volume fraction of each of the fluids in each computational cell throughout the domain.
3. Model Development

a. Governing Equations

1. Reynolds Average Continuity Equation

\[ \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho U) = \sum_{q=1}^{n} S_q \]

- \( \rho \): volume-fraction-averaged density
- \( U \): time average velocity which is computed by solving the momentum equation for the mixture (not individual components).
- \( n \): number of the phases (for the present two phase flow, \( n=2 \)).
- \( S_q \): mass source (set to zero in the present case)

2. Reynolds Average Momentum Equation

\[ \frac{\partial (\rho U)}{\partial t} + \nabla \cdot (\rho U U) = -\nabla P + \nabla \cdot [\mu (\nabla U + \nabla U^T)] + (\rho g) + F \]

- \( P \): pressure in the domain,
- \( \mu \): volume-fraction-averaged viscosity of the flowing fluid,
- \( g \): acceleration due to gravity,
- \( F \): body force.
3. Model Development

A. Governing Equations

The NSE formulation is dependent on the fact that fractions of all phases (phases) are not interpenetrating. For each additional phase that you add to your model, a variable is introduced which is the volume fraction of the phase. In a control volume, the volume fractions add up to unity.

\[
\rho = \sum_{q=1}^{n} \rho_q \alpha_q = \alpha_L \rho_L + \alpha_G \rho_G = \alpha_L \rho_L + (1 - \alpha_L) \rho_G
\]

\[
\mu = \sum_{q=1}^{n} \mu_q \alpha_q = \alpha_L \mu_L + \alpha_G \mu_G = \alpha_L \mu_L + (1 - \alpha_L) \mu_G
\]
3. Model Development

a. Governing Equations

3. Volume fraction equation

- Tracking the interface between the two phases is achieved by the treatment of the volume fraction of the $q^{th}$ fluid, $\alpha_q$ through solving a separate continuity equation, given by:

$$\frac{1}{\rho_q} \left[ \frac{\partial}{\partial t} (\alpha_q \rho_q) + \nabla \cdot (\alpha_q \rho_q U_q) \right] = S_{aq} + \sum_{p=1}^{n} (m_{pq}^\circ - m_{qp}^\circ)$$

- $S_{aq}$: mass source term,
- $m_{pq}^\circ$: mass transfer from phase $p$ to phase $q$,
- $m_{qp}^\circ$: mass transfer from phase $q$ to phase $p$. 
3. Model Development

b. Model Geometry and Boundary Conditions

- The initial guess of Taylor bubble velocity, $U_{TB}$, is estimated according to Wallis (1969)

$$Fr = \frac{U_{TB}}{\sqrt{g \times D}} = 0.345 \left(1 - e^{-0.01N_f^{0.35}}\right) \left(1 - e^{\frac{3.37 - E_0}{m}}\right)$$

where:

$$m = \begin{cases} 
25, & N_f < 18 \\
69N_f^{0.35}, & 18 < N_f < 250 \\
10, & N_f > 250 
\end{cases}$$

- The simulation is performed by

- The initial guess of the liquid film thickness, $\delta_{LF}$, is estimated using Brown (1965) equation:

$$\delta_{LF} = \left[\frac{3 \times \nu}{2 \times g \times (R - \delta_{LF}) \times U_{TB} \times (R - \delta_{LF})^2}\right]^{1/3}$$
FRF and MRF

- Consider single TB rising up with velocity, $U_N$, relative to the stationary walls. The transformation coordinate from FRF to MRF are:
  
  \[ x = \dot{x} - U_N t \]
  \[ y = \dot{y} \]

- The TB becomes stationary if a new velocity variable, $u$, is assigned as follow:
  
  \[ u = \dot{u} + U_N \]
**Fixed Reference Frame (FRF)**

- Outflow:
  - $U_{out} = w_y$
  - $V_{out} = 0$

**Axis of Symmetry**
- $\frac{\partial U}{\partial y} = 0$
- $V_{out} = 0$

**Stationary Wall**
- $U_{wall} = V_{wall} = 0$

**Moving Reference Frame (MRF)**

- Inlet Flow:
  - $U_{in} = U_{TB} \cdot w_y$
  - $V_{in} = 0$

**Axis of Symmetry**
- $\frac{\partial U}{\partial y} = 0$
- $V_{out} = 0$

**Moving Wall**
- $U_{wall} = U_{TB}$
- $V_{wall} = 0$
**G-L Interface BC**

- The pressure variation in the gas phase is assumed to be constant.

- The boundary conditions at the gas-liquid interface are given by:

  1. The kinematic condition assuming full slip at the interface is applied:

     \[ \rho_{iL} + \sigma K = \text{constant} \]

     Where; \( \tau, \hat{n}, \hat{s}, \sigma, \rho_{iL} \) and \( K \) are the shear stress, unit normal vector at the interface, unit tangential vector at the interface, surface tension, liquid phase side pressure, and curvature of the interface, respectively.

  2. The dynamic boundary condition, which is also known by stress jump condition can be divided into two separate boundary conditions:

     a) the tangential stress balance assuming zero interfacial shear stress along the interface,

     \[ K = \frac{1}{r_1} + \frac{1}{r_2} \]

     According to Mao and Dukler (1990), the curvature of the interface is expressed in terms of radii of the curvature of the bubble surface, as follows;
3. Model Development

   c. Solution Strategy and Convergence Criterion

1. Transient solver and a time step size of $0.0001\text{s}$ was employed.
2. The simulation was carried out using the explicit VOF model.
3. A simple scheme for the pressure-velocity coupling is considered.
4. A spatial discretization scheme used are as follows;
   - Green Gauss Cell Based for Gradient.
   - PRESTO for Pressure.
   - Compressive for Volume fraction.
   - Quick scheme for Momentum.
   - First order implicit for transient formulation
5. The scaled absolute values of the residual of the calculated values of mass, velocity in x and y directions were monitored and convergence criterion was set to $10^{-3}$ for each time step.
6. Results were obtained using the Engineering and Physical Sciences Research Council (EPSRC) funded ARCHIE-WeSt high performance computer (www.archie-west.ac.uk).
1. Introduction.
2. Aim of the study.
3. Model development;
   a) Governing equations.
   b) Model geometry and boundary conditions.
   c) Solution strategy and convergence criterion.
4. Results and discussions;
   a) Taylor bubble shape.
   b) Taylor bubble rise velocity.
   c) Liquid film.
   d) Wake flow structure.
   e) Wall shear stress distribution.
5. Conclusions.
6. Recommendations for future work.
4. Results and Discussions

Four cases were simulated according to the experimental work of Campos and Carvalho (1988) using MRF. Velocity fields and streamlines of gas-liquid slug flow in vertical pipe of 19mm diameter, and 209mm length with \( R_e = 84 \), \( N_M = 0.289 \), and \( M = 66.3 \) are shown in the figure.
4. Results and Discussions

a. Taylor Bubble Shape

- Effect of inverse viscosity number $N_f$ on Taylor bubble shape.

- The simulated results are in good agreement with:
  
  1. the experimental observations of Goldsmith and Mason (1962); in highly viscous flow (viscosity dominated flow) the Taylor bubble has spheroid shape where the top end of bubble is prolate, and the bottom end is oblate. While, in low viscosity flow the flattening or concaving shape of Taylor bubble bottom end is observed.

  2. the numerical work of Araújo et al. (2012), Taha and Cui (2006), and Zheng et al. (2007).

(1) $N_f = 84$, (2) $N_f = 176$, (3) $N_f = 205$, and (4) $N_f = 325$
4. Results and Discussions

a. Taylor Bubble Shape

Validation of simulation results for Taylor bubble shape profile for cases 1, and 2 with the work of Taha and Cui (2006) - $x$ is axial distance from bubble nose.

Effect of $N_f$ on Taylor bubble shape profile for cases 2, 3 and 4 - $x$ is axial distance from bubble nose.
4. Results and Discussions

\textit{b. Taylor Bubble Rise Velocity,} \( U_{TB} \)

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Case 1</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>( U_{TB} )</td>
<td>0.1251</td>
<td>0.1381</td>
<td>0.1340</td>
</tr>
<tr>
<td>Error (%)</td>
<td>…</td>
<td>9.41</td>
<td>6.62</td>
</tr>
<tr>
<td><strong>Case 2</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>( U_{TB} )</td>
<td>0.1374</td>
<td>0.1467</td>
<td>0.1473</td>
</tr>
<tr>
<td>Error (%)</td>
<td>…</td>
<td>6.35</td>
<td>6.71</td>
</tr>
<tr>
<td><strong>Case 3</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>( U_{TB} )</td>
<td>0.1390</td>
<td>0.1467</td>
<td>0.1480</td>
</tr>
<tr>
<td>Error (%)</td>
<td>…</td>
<td>5.24</td>
<td>6.02</td>
</tr>
<tr>
<td><strong>Case 4</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>( U_{TB} )</td>
<td>0.1425</td>
<td>0.1424</td>
<td>0.1485</td>
</tr>
<tr>
<td>Error (%)</td>
<td>…</td>
<td>-0.07</td>
<td>4.0</td>
</tr>
</tbody>
</table>
4. Results and Discussions

c. Liquid Film

Effect of $N_f$ on liquid film thickness $\delta_{LF}$ - $x$ is axial distance from bubble nose.

Effect of $N_f$ on liquid film axial velocity $U_{LF}$ - $x$ is axial distance from bubble nose.
4. Results and Discussions

c. Wake Flow Structure

- Effect of inverse viscosity number $N_f$ on wake flow pattern.
- The simulated results are in good agreement with:
  1. the experimental observations of Campos and Carvalho (1988).
  2. the numerical work of Araújo et al. (2012), Taha and Cui (2006), and Zheng et al. (2007).
4. Results and Discussions

d. Wall Shear Stress Distribution

Validation of simulation results for the wall shear stress distribution $\tau_w$ around a slug unit with the work of Taha and Cui (2006) - $x$ is axial distance from bubble nose.

Effect of $N_f$ on the wall shear stress distribution $\tau_w$ around a slug unit - $x$ is axial distance from bubble nose.
Presentation Overview

1. Introduction.
2. Aim of the study.
3. Model development;
   a) Governing equations.
   b) Model geometry and boundary conditions.
   c) Solution strategy and convergence criterion.
4. Results and discussions;
   a) Taylor bubble shape.
   b) Taylor bubble rise velocity.
   c) Liquid film.
   d) Wake flow structure.
   e) Wall shear stress distribution.
5. Conclusions.
6. Recommendations for future work.
4. Conclusions

- The following conclusions can be pointed out;

1) The Taylor bubble bottom depends on the liquid viscosity where the increase in the inverse viscosity number, $N_f$, increases the concave shape of the Taylor bubble bottom surface.

2) The calculated Taylor bubble rise velocity, $U_{TB}$, is in an acceptable range when compared with experimental values and commonly used correlations in the literature.

3) The liquid film zone can be describes using the liquid film thickness, $\delta_{LF}$, and the liquid film axial velocity, $U_{LF}$ that are both directly affected by the inverse viscosity number, $N_f$.

4) The wake flow structure has a closed axisymmetric nature for all the simulation cases with the development of circulatory vortex in the bubble wake with the increase in $N_f$.

5) The wall shear stress, $\tau_W$, is mainly dependent on the liquid film thickness, $\delta_{LF}$, and has a peak positive values in the stabilized liquid film zone.
Presentation Overview

1. Introduction.
2. Aim of the study.
3. Model development;
   a) Governing equations.
   b) Model geometry and boundary conditions.
   c) Solution strategy and convergence criterion.
4. Results and discussions;
   a) Taylor bubble shape.
   b) Taylor bubble rise velocity.
   c) Liquid film.
   d) Wake flow structure.
   e) Wall shear stress distribution.
5. Conclusions.
6. **Recommendations for future work.**
4. Recommendations for Future Work

- This study set out to develop a basic simulation model for gas-liquid slug flow in vertical pipe under laminar flow regime.
- It is recommended that further research could be undertaken in the following areas;
  1. Investigating the hydrodynamic characteristics of slug flow for different fluid system including the effect of viscosity and density ratios,
  2. Investigating the hydrodynamic characteristics of slug flow under turbulent regime with $N_f > 500$,
  3. Exploring the hydrodynamic characteristics of slug flow including the flow of two consecutive Taylor bubbles in vertical pipe,
  4. Studying the wake flow pattern of single Taylor bubble or two consecutive Taylor bubbles under turbulent flow regime in terms of wake volume and length.
Any Questions?