Modelling Gas and High-Viscous-Oil Slug Flow Regime to Estimate the Dispersed Phase Distribution Coefficient

Victor Pugliese [a,b], *Amin Ettehadtavakkol, PhD.* [a]
[a] *Bob L. Herd Department of Petroleum Engineering, Texas Tech University, TX 79409, Lubbock, Texas, United States.*
[b] *Department of Mechanical Engineering, Universidad del Norte, Km. 5 Vía a Puerto Colombia, Barranquilla, Colombia.*

Bob L. Herd Department of Petroleum Engineering, Texas Tech University, Lubbock, TX 79409-3111, Box 43111
Degree: Master's Degree
Phone Number: +1 806 500 0798
E-mail address: victor.pugliese@ttu.edu

Abstract

In multiphase flow in pipes models there are more variables involved than conservation equations available. Thus, constitutive equations based on experiments are required to relate fluids properties, pipe geometry and inclination with flow variables such as shear stresses and velocities. In this research, an experiment to study the hydrodynamic of the slug flow regime was modelled using OpenFOAM. The turbulent scale-resolving strategy used in the numerical simulation was Large-Eddy Simulation (LES). Additionally, the Volume of Fluid (VOF) Method was used as the Interface Tracking Method (ITM).

A gas and high-viscous liquid mixed flow experiment was implemented to predict the Dispersed Phase Distribution Coefficient. The velocity at the gas and liquid inlets were selected to promote the formation of slug flow regime. A total of 56 simulation cases were run to study the effect of pipe inclination (from horizontal direction to vertical upward direction), Reynolds Number and Froud Number on the Dispersed Phase Distribution Coefficient. The numerical model was validate using experimental results carried out in horizontal and vertical pipes. The results indicate that the correlation to predict the Distribution Coefficient correlation has less than 10% absolute average relative error for the viscosity range of 0.14 to 1.120 $Pa \cdot s$, and pipe inclinations from 0° to 90°.

Keywords: Multiphase Flow, High-Viscous Oil, Distribution Coefficient, Large-Eddy Simulation, Volume of Fluid Method.

1. Introduction

The main constitutive equation required by the drift flux model is the estimation of the mean gas velocity, $\langle\langle v_g \rangle\rangle\,[m/s]$:

$$\langle\langle v_G \rangle\rangle = C_0 \langle j \rangle + \langle\langle V_{Gj} \rangle\rangle \quad (1)$$

The parameter C_0 is a dispersed phase distribution coefficient related to the velocity and concentration profiles in dispersed systems, $\langle j \rangle [m/s]$ is the average volumetric flux, and $\langle\langle V_{Gj} \rangle\rangle\,[m/s]$ is the local gas drift velocity (Ishii and Hibiki, 2011).

In traditional multiphase flow models, it is accepted that for laminar flow in the slug body the value of the distribution coefficient C_0 is close to 2. For the turbulent flow, the value of C_0 is close to 1.2. These numerical values correspond to the theoretical ratio between the maximum velocity

and the average velocity in pipelines for each flow regime. However, this simplification does not consider the effect of pipe inclination, and high viscous liquids.
The complexity of the slug flow requires the combined use of numerical approaches to overcome the optical problems reported in experimental methods. The general expression for the Average Volumetric Concentration (Zuber and Findaly, 1965) can be used with 3D-CFD results.

$$C_0 = \frac{\langle \alpha j \rangle}{\langle \alpha \rangle \langle j \rangle} \tag{2}$$

The gas volume fraction, α, and volumetric flux field, j, is information that is available in 3D-CFD slug flow simulation for each cell.

2. CFD Simulation Procedure

2.1. Large-Eddy Simulation (LES)

The CFD opensource software OpenFOAM v.1806 was used to simulate the two-phase flow in a pipe. The turbulent scale-resolving strategy used in the numerical simulation was Large-Eddy Simulation (LES). LES is based on unifying the phase averaging concept and the turbulent-scale filtering operation into one single process (Lakehal, 2018). Using this approach, the dispersed mixed flow regions are solved using a phase-averaged formulation, while separate flow regions (stratified and elongated bubble flow) are simulated using an Interface Tracking Method (ITM) since the interface is large enough to be resolved according the CFD grid resolution.
To simulate the flow of a two-phase fluid, the Volume of Fluid (VOF) Method (Hirt and Nichols, 1981) was used as the Interface Tracking Method (ITM). VOF formulation assumes that the two phases do not mutually dissolve. In each control volume of the computational domain, the volume fraction of both phases adds up to unity. The flow field for all variables and properties are shared by both phases and represent volume-averaged values, provided that the volume fraction of each of the phases is known at each location.
Two-phase flow was modeled by the three-dimensional continuity equation and Navier-Stokes equations for two incompressible isothermal immiscible fluids. The eddy or turbulent viscosity was estimated using the Smagorinsky (1963) turbulence model.

2.2. Numerical Simulations

The computational domain consists of a pipe with two inlet sections cut by his vertical symmetry plane. In the axial direction, mesh is uniformly distributed. In the radial direction, mesh is finer near the wall. Figure 1 displays the mesh used.

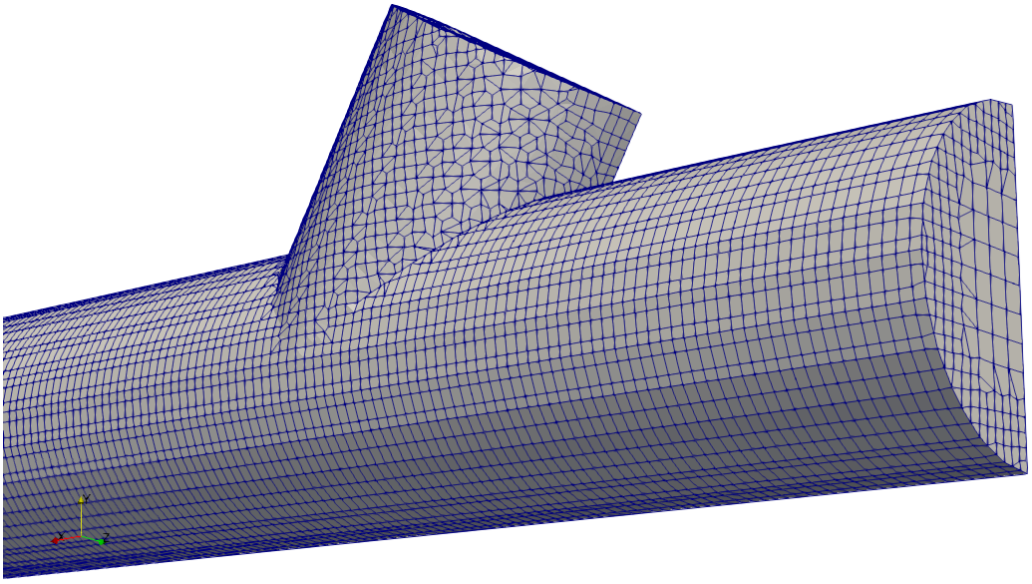

Figure 1 Computational domain. It is composed by hexahedral mesh uniformly distributed along the pipe axis. The mesh is finer near the wall. In this figure, the gas inlet (30° bifurcation) and the liquid inlet are presented.

The aim of the study is to investigate the effect of the pipe inclination and liquid viscosity on the Dispersed Phase Distribution Coefficient C_0. The Dispersed Phase Distribution Coefficient was obtained by simulating the continuous mixture of gas and high-viscous oil in a 0.0508 m-ID and 7 meters pipe during 60 seconds. The inclination angle of the pipe range from 0° (horizontal direction) to 90° (vertical upward direction). The physical properties of each fluid used can be found in Table 1, and the Table 2 presents the factors and levels for the numerical experiments. A total of 56 numerical simulations were carried out.

Fluid	**Density** $[\boldsymbol{kg/m^3}]$	$\boldsymbol{Viscosity}$ $[\boldsymbol{Pa \cdot s}]$	**Surface tension with air** $[\boldsymbol{N/m}]$
Air	1.225	1.7894e-05	--
Oil	889	0.14 – 1.12	0.025

Table 1 Fluid Properties.

Factors	**Levels**
Liquid Viscosity $[Pa \cdot s]$	0.14 – 0.28 – 0.42 – 0.56 – 0.70 – 0.84 – 0.98 – 1.12
Pipe Inclination [°]	0 – 15 – 30 – 45 – 60 – 75 - 90

Table 2 Factors and corresponding Levels for the Numerical Simulations.

2.3. Initial and Boundary Conditions

The following initial and boundary conditions were used:

	Region	**Gas Volume Fraction**	**Velocity Field**	**Turbulent Kinetic Energy**	**Turbulent Viscosity**	**Pressure**
Initial conditions	For $0 < x < 7m$	$\alpha = 0$	$\vec{V} = 0$	$k = 0$	$\mu_t = 0$	$P = P_{0_{@outlet}}$

Boundary conditions	At Liquid Inlet	$\alpha = 0$	$V_x = \langle j_L \rangle$	k calculated based in Eq.(13a)	μ_t calculated based in and (12a)	$\frac{\partial P}{\partial x} = 0$
	At Gas Inlet	$\alpha = 1$	$V_x = \langle j_G \rangle$	k calculated based in Eq.(13a)	μ_t calculated based in and (12a).	$\frac{\partial P}{\partial x} = 0$
	At Outlet	$\frac{\partial \alpha}{\partial x} = 0$	$\frac{\partial \vec{V}}{\partial x} = 0$	$\frac{\partial k}{\partial x} = 0$	$\frac{\partial \mu_t}{\partial x} = 0$	$P = P_0$
	At $z = 0$	Symmetry plane				
	At Pipe Wall	Stationary wall and No-Slip condition. Wall adhesion for liquid with contact angle 20° measured for the liquid phase.				

Table 3 Initial and Boundary Conditions used in Numerical Simulations.

2.4. Solver and discretization scheme settings

System of partial differential equations formed by the conservation equation is treated in the segregated way, meaning that they are solved one at a time, with the inter-equation coupling treated in the explicit manner. Appropriate discretization schemes and linear solvers selection is required to guarantee rapid convergence and accuracy of the solution fields. The terms requiring a discretization scheme include derivatives and interpolations. Table 4 shows all the schemes used for each term (time derivative, gradient, divergence and Laplacian terms) in the continuity and momentum equation. In Finite Volume Method (FVM), the divergence theorem is used to convert the volume integral in each grid block to surface integrals. Thus, the fields are considered as flux at the surfaces. Variables have to be approximated at the center of the faces using interpolation schemes. *snGradSchemes* is used for the gradient component normal to a cell face.

Term	Subdictionary	Type
ddtSchemes	default	Euler
gradSchemes	default	Gauss linear
divSchemes	div(rhoPhi, U)	Gauss linearUpwind grad(U)
	div(phi, alpha)	Gauss vanLeer
	div(phirb, alpha)	Gauss linear
	div(phi,k)	Gas linearUpwind grad(k)
	div(phi,epsilon)	Gas linearUpwind grad(epsilon)
	div(((rho*nuEff)*dev2(T(grad(U)))))	Gauss linear
laplacianSchemes	default	Gauss linear corrected
interpolationSchemes	default	linear
snGradSchemes	default	corrected

Table 4 Discretization schemes applied in the interFoam application

In case we want to apply a specific discretization scheme to a variable or flux, we could use the Subdictionary name associated. Gauss schemes is the standard finite volume discretization of Gaussian integration and linear refers to linear interpolation or central differencing. When *default* is used, all terms use the scheme type associated.

In order to solve the discretize continuity and momentum equations, the PISO (Pressure-Implicit-Split-Operators) algorithm (Issa, 1986) is selected. The PISO algorithm is a non-iterative transient calculation procedure. It relies on the temporal accuracy gained by the discretization practice. One predictor step and three corrector steps were used in the present simulation.

Because the volume fraction at the current time step is directly calculated based on known quantities at the previous time step, the explicit formulation does not require an iterative solution, resulting in a reduced simulation time. However, a stability criterion is required in order to guarantee the convergence of the solution. The global Courant number (Courant, Friedrichs and Lewy, 1967) is set to 0.95, and time step is adjustable. In multiphase flow problems the recommended Courant Number is 0.35, due to the interface, to guarantee stability. This issue was addressed solving the transport equation for α three times within each time step; thus, the effective courant number is 0.95/3 in the α transport equation. In OpenFOAM, this was carried out using the function *nAlphaSubCycles* in the *Control* dictionary. The pressure and velocity field obtained at the end of the PISO process with a suitably small time step are considered to be accurate enough to proceed to the next time step.

Finally, the linear-solver for the discretize equations are selected. Generalized geometric-algebraic multi-grid (GAMG) was used to solve the pressure field. GAMG generates a quick solution on a mesh with small number of cells; mapping this solution onto a finer mesh; using it as an initial guess to obtain an accurate solution on the fine mesh. Using this solver increases the calculation speed considerably. Others used solver are listed in Table 5.

Field	Linear-Solver	Smoother	Tolerance
$\boldsymbol{p-\rho gh}$	GAMG	GaussSeidel	$1e^{-07}$
$\boldsymbol{u_i}$	smoothSolver	symGaussSeidel	$1e^{-07}$
$\boldsymbol{\alpha}$	smoothSolver	symGaussSeidel	$1e^{-07}$
$\boldsymbol{k}$	smoothSolver	symGaussSeidel	$1e^{-07}$

Table 5 Linear solvers applied to discretized equation.

3. Validation of the Models against Experimental Models

The model proposed by Bendiksen, Langsholt and Liu (2018) uses the Froud Number based on the average volumetric flux Fr_j as the independent variable to predict the dispersed phase distribution coefficient C_0.

$$C_0(R_t) = C_0(0)(1 - R_t^2) \tag{3a}$$

where $C_0(0)$ is the maximum value of C_0 at the center of the pipe. R_t is the dimensionless bubble tip radial position measured from the center of the pipe.

$$R_t = \begin{cases} k_1 Fr_j, & Fr_j < Fr_{ML} \\ k_2 + k_3 \ln(Fr_j), & Fr_j \geq Fr_{ML} \end{cases} \tag{3b}$$

$$Fr_j = \langle j \rangle / \sqrt{gD(1 - \rho_G/\rho_L)} \tag{3c}$$

The parameters k and Fr_{ML} depend on the liquid viscosity and Reynold Number Re_j, and are presented in Table 6.

Flow condition		**Model parameters**				
Re_j	$\mu_L[Pa \cdot s]$	Fr_{ML}	k_1	k_2	k_3	$C_0(0)$
≤ 900	< 0.7	1	0.35	0.35	0.1	2.15
	≥ 0.7	0.8	0.35	0.28	0.1	
> 900	> 0.1	1	0.35	0.35	0.18	

Table 6 Distribution Coefficient Model Parameters.

The distribution coefficient C_0 was calculated using equation (2) for 56 Simulation Cases. However, only 8 of them correspond to horizontal pipe. Bendiksen, Langsholt and Liu (2018) model is based on 241 experiments in a horizontal pipe with high-viscous liquid, thus a comparison between the distribution coefficient predicted by Bendiksen, Langsholt and Liu (2018) model and the CFD simulation is a good indication of how well the CFD model is capable to reproduce an actual two-phase flow. Figure 2 shows there is an agreement of more than 90% between the distribution coefficient estimated from the CFD results and the experiments performed by Bendiksen, Langsholt and Liu (2018).

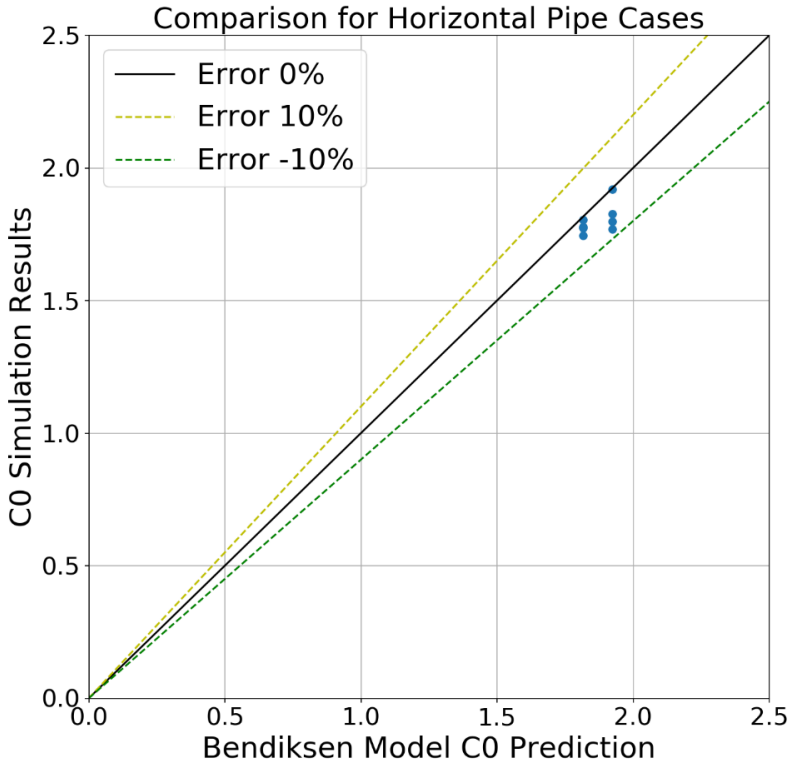

Figure 2 Horizontal two-phase flow simulation cases.

Pinto, et al. (2005) presented a correlation for C_0 when Taylor bubbles move in vertical co-current liquid flow, in which C_0 does not depend solely on Re_j:

$$C_0 = \begin{cases} 2.0 \pm 0.1, & Re_j We_{V_{Gj}}^{0.21} \left(\dfrac{\langle j \rangle}{\langle\langle V_{Gj} \rangle\rangle} \right)^{0.28} < 1{,}000 \\ 2.08 - 1.38 \times 10^{-4} Re_j We_{V_{Gj}}^{0.21} \left(\dfrac{\langle j \rangle}{\langle\langle V_{Gj} \rangle\rangle} \right)^{0.28}, & 1{,}000 < Re_j We_{V_{Gj}}^{0.21} \left(\dfrac{\langle j \rangle}{\langle\langle V_{Gj} \rangle\rangle} \right)^{0.28} < 6{,}000 \\ 1.2 \pm 0.1, & Re_j We_{V_{Gj}}^{0.21} \left(\dfrac{\langle j \rangle}{\langle\langle V_{Gj} \rangle\rangle} \right)^{0.28} > 6{,}000 \end{cases} \quad (4a)$$

where Re_j is the Reynolds number in the slug body; and $We_{V_{Gj}}$ is the Weber number for the bubble in stagnant fluid. These dimensionless numbers are given by

$$Re_j = \rho_L \langle j \rangle D / \mu_L \quad (4b)$$

$$We_{V_{Gj}} = \rho_L \langle\langle V_{Gj} \rangle\rangle^2 D / \sigma \quad (4c)$$

The 8 vertical upward two-phase flow simulation cases were compared against Pinto, et al. (2005) model, which is based on low Reynold number vertical two-phase flow experiments. Figure 3 presents the comparison between the distribution coefficients predicted by Pinto, et al. (2005) and the distribution coefficient calculated based on the CFD results. Again, there is an agreement greater than 90% for all 8 cases.

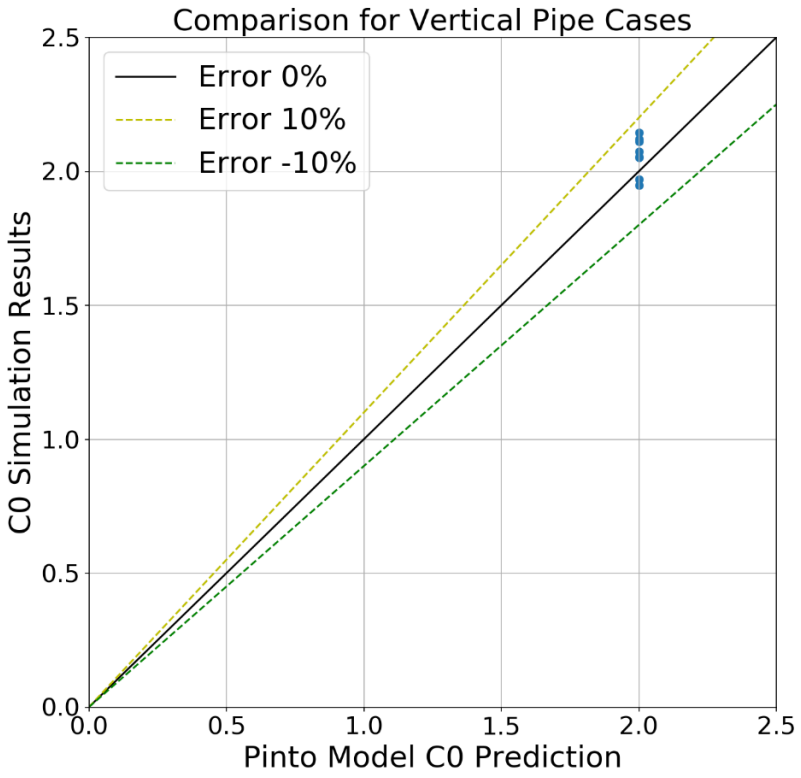

Figure 3 Vertical Upward two-phase flow simulation cases.

4. Outcome

The inclined simulation cases only differ from the vertical and horizontal cases in the gravity acceleration components. Thus, we expect that the CFD model is capable to reproduce actual slug flow regime in pipes. The dataset composed by the 56 numerical experiments can be used to propose a new correlation to estimate the dispersed phase distribution coefficient.

References

Bendiksen, Kjell H., Morten Langsholt, and Lan Liu. 2018. "An experimental investigation of the motion of long bubbles in high viscosity slug flow in horizontal pipes." *International Journal of Multiphase Flow. Vol. 104* 60-73 DOI:10.1016/j.ijmultiphaseflow.2018.03.010.

Courant, R., K. Friedrichs, and H. Lewy. 1967. "On the Partial Difference Equations of Mathematical Physics." *IBM Journal of Research and Development* 215-234. DOI: 10.1147/rd.112.0215.

Hirt, C. W., and B. D. Nichols. 1981. "Volume of fluid (VOF) method for the dynamics of free boundaries." *Journal of Computational Physics* 201-225 DOI:10.1016/0021-9991(81)90145-5.

Ishii, Mamoru, and Takashi Hibiki. 2011. *Thermo-Fluid Dynamics of Two-Phase Flow. 2nd Edition.* West Lafayette, IN, USA: Springer. DOI:10.1007/978-1-4419-7985-8.

Issa, Raad. 1986. "Solution of the implicitly discretised fluid flow equations by operator-splitting." *Journal of Computational Physics. Volume 62. Issue 1* 40-65.

Lakehal, Djamel. 2018. "Status and future developments of Large-Eddy Simulation of turbulent multi-fluid flow (LEIS and LESS)." *International Journal of Multiphase Flow. Vol. 104* 322-337 DOI:10.1016/j.ijmultiphaseflow.2018.02.018.

Pinto, Alexandra, Maria N. Coelho, S. Nogueira, and Joao B. Lares. 2005. "Experimental Study on the Transition in the Velocity of Individual Taylor Bubbles in Vertical Upward Co-Current Liquid Flow." *Chemical Engineering Research and Design. Vol. 83* 1103-1110. DOI:10.1205/cherd.03238.

Smagorinsky, Joseph. 1963. "General circulation experiments with the primitive equations: I. The basic experiment." *Monthly Weather Review. Vol. 91* 99-163. DOI:10.1175/1520-0493(1963)091<0099:GCEWTP>2.3.CO;2.

Zuber, Z., and J A. Findaly. 1965. "Average Volumetric Concentration in Two-Phase Flow Systems." *Journal of Heat Transfer. Vol. 87* 453-468 DOI:10.1115/1.3689137.