



BOBO CLAUDE KOUADJO

INVESTIGATION OF MULTIPHASE FLOW INSTABILITIES AND
DEVELOPMENT OF TURBULENCE MODELS FOR TRANSIENT
HYDRODYNAMIC SLUG FLOW IN HORIZONTAL PIPELINES

SCHOOL OF AEROSPACE, TRANSPORT AND
MANUFACTURING
(COMPUTATIONAL ENGINEERING SCIENCES)

DOCTOR OF PHILOSOPHY (PhD)
Academic Year: 2020 - 2023

Supervisor: Dr László Könözy
Associate Supervisor: Professor Karl W. Jenkins
May 2023



SCHOOL OF AEROSPACE, TRANSPORT AND
MANUFACTURING
(COMPUTATIONAL ENGINEERING SCIENCES)

DOCTOR OF PHILOSOPHY (PhD)

Academic Year 2020 – 2023

BOBO CLAUDE KOUADJO

INVESTIGATION OF MULTIPHASE FLOW INSTABILITIES AND
DEVELOPMENT OF TURBULENCE MODELS FOR TRANSIENT
HYDRODYNAMIC SLUG FLOW IN HORIZONTAL PIPELINES

Supervisor: Dr László Könözy

Associate Supervisor: Professor Karl W. Jenkins

May 2023

This thesis is submitted in partial fulfilment of the requirements for
the degree of Doctor of Philosophy

© Cranfield University 2023. All rights reserved. No part of this
publication may be reproduced without the written permission of the
copyright owner.

ABSTRACT

The knowledge of the slug flow is essential to the design and safety of hydrocarbon transportation in the pipelines systems of the petroleum industry. This thesis presents a theoretical and computational investigation of hydrodynamic slug flow in horizontal pipelines. As slug flow is part of multiphase flows, a literature review of multiphase models and their instabilities including the turbulent models involved. A focus on the anisotropic aspect of the prediction of turbulence has been considered by setting up a new turbulence model that will help us to solve this multiphase problem (hydrodynamic slug flow).

A User-Defined Function based C code has been implemented to model the anisotropic turbulence for the new multiphase turbulence model created as a contribution to the knowledge.

The Ansys Fluent computational fluid dynamics (CFD) software package has been used to set up the simulations with the aim of investigating the characteristics of slug flows such liquid holdup, pressure drops in different pipe sizes (diameter 16mm, 26mm and 50mm). The challenges including the three-dimensional simulations were carried out with both isotropic and anisotropic turbulent models to investigate the differences, similarities, and improvements.

The isotropic simulations were validated with a journal paper (Deendarlianto *et al.*, 2018). As a result, different flow patterns such as stratified Fig. (4-14.) and Fig. (4-15), pseudo slug Fig. (4-23.), slug Fig. (4-5) and Fig. (4-8) have been obtained. The analysis shows that at some time steps, numerical and experimental data values are very closed, showing a good validation.

A comparative study has been performed to compare and assess the performance of isotropic and anisotropic simulations results for an industrially relevant engineering application. The validation shows a better result Fig. (4-43) compared to isotropic simulation.

Keywords: Multiphase Anisotropic Turbulence, Computational Fluid Dynamics, Slug Flows

ACKNOWLEDGEMENTS

I would like to send my sincere appreciation to Professor Karl W. Jenkins, Head of Centre, and Program Director, who gave me the opportunity to pursue this PhD degree.

I would also like to express my sincere gratitude to Dr László Könözy, my primary supervisor who continuously guided my progress, despite the difficult time caused by the recent pandemic.

Thanks to all Cranfield members of staff, students and others who generously supported me during this laborious period.

Finally, my deeply thanks to my family for their unceasing support and understanding throughout my study at Cranfield University.

TABLE OF CONTENTS

ABSTRACT	i
ACKNOWLEDGEMENTS.....	ii
TABLE OF CONTENTS	iii
LIST OF FIGURES.....	v
LIST OF TABLES	viii
LIST OF ABBREVIATIONS	ix
Chapter 1: INTRODUCTION	11
1.1 Background.....	11
1.2 Project Objectives	12
1.3 Contribution to the knowledge.....	14
1.4 Summary of subsequent Chapters.....	14
Chapter 2: LITERATURE REVIEW	16
2.1 Summary	16
2.2 Two-phase Flow Regimes.....	16
2.2.1 Description of Flow Patterns.....	18
2.3 Introduction to slug Flows	22
2.3.1 Terrain-Induced Slugs	22
2.3.2 Hydrodynamic Slugs.....	24
2.3.3 Operationally Induced Slugs	24
2.4 Turbulence Modelling.....	25
2.4.1 Reynolds Averaging	27
2.4.2 Reynolds Stress tensor	29
2.4.3 Isotropic Turbulence Models	30
2.4.4 Anisotropic Turbulence Models	32
2.5 Mathematical Modelling	42
2.5.1 Quasi-Multiphase models.....	44
2.5.2 Multi-Fluid Models	46
2.5.3 Moving Grid Methods	49
2.5.4 Interface Tracking Methods.....	51
2.5.5 Particle based Models/Methods	54
2.5.6 Hybrid Models	60
2.6 Numerical Methods	61
2.6.1 Discretisation Methods	62
2.6.1.1 Finite Difference Method.....	62
2.6.1.2 Finite Volume method	63
2.6.1.3 Finite Element Method	64
2.6.1.4 Spectral Methods	65
2.7 Hydrodynamic Slug Flow Modeling.....	65
2.7.1 Slug Flow Mechanisms	67
2.7.2 Slug Flow Modelling Approaches	73

2.7.3 Slug Flow Parameters	77
2.7.4 Slug Frequency	84
Chapter 3: METHODOLOGY AND NEW APPROCH FOR MODELLING MULTIPHASE ANISOTROPIC TURBULENCE.....	86
1.1 Summary.....	86
1.2 The New Governing Equations and the Contribution to the Knowledge.....	86
Chapter 4: VALIDATION OF TURBULENCE APPROACHES FOR THE HYDRODYNAMIC SLUGS FLOWS IN COMPARISON WITH EXPERIMENTAL DATA.....	90
1.1 Summary.....	90
1.2 Numerical Solutions.....	90
1.3 Validation.....	113
1.4 Conclusions.....	119
Chapter 5: Conclusion and recommendations for future work.....	123
5.1 Conclusions.....	123
5.2 Recommendations and Future work.....	125
Bibliography.....	126
Appendix A: Two-phase Flow Terminology.....	135
Appendix B: User Defined Functions.....	140
Appendix C: UDF C Code.....	153
Appendix D: Experimentation.....	184

LIST OF FIGURES

Figure 2-1 Flow pattern map of Baker (1954), modified by Scott (1964)	17
Figure 2-2 Flow pattern maps of Mandhane (1974) and Taitel & Dukler (1976).....	18
Figure 2-3 Stratified flow pattern	19
Figure 2-4 Slug evolution inside the pipe	20
Figure 2-5 Annular flow pattern	21
Figure 2-6 Description of severe slug flow phenomenon.....	24
Figure 2-7 Transition from slug flow to stratified flow.....	68
Figure 2-8 Definition sketch of system with wave stationary (after Gardner (1979)	72
Figure 4-1 Pipeline design using workbench.....	91
Figure 4-2 Hexahedral mesh using workbench.....	91
Figure 4-3 Pressure drops with coarse mesh (300000hexahedral cells) with max 500.....	94
Figure 4-4 Pressure drops with fine mesh (600000 hexahedral cells) with max 250 [Pa].....	94
Figure 4-5 Water volume fraction contours (sluggish) with coarse mesh (300000 hexahedral cells).....	95
Figure 4-6 Water volume fraction contours (sluggish) with fine mesh (600000 hexahedral cells).....	95
Figure 4-7 Velocity Profile at Z = 4.5 m with max velocity 0.27 m/s with coarse mesh (300000 hexahedral cells).....	96
Figure 4-8 Velocity Profile at Z = 4.5 m with max velocity 0.26 m/s with fine mesh (600000 hexahedral cells).....	96
Figure 4-9 Liquid Holdup at pipeline with coarse mesh (300000 hexahedral).....	97
Figure 4-10 Liquid Holdup at pipeline with coarse mesh (600000 hexahedral).....	97

Figure 4-11 Pressure drops with coarse mesh (300000 hexahedral cells) with max 130 [Pa].....	98
Figure 4-12 Pressure drops with coarse mesh (600000hexahedral cells) with max 130 [Pa].....	98
Figure 4-13 Water volume fraction contours with coarse mesh (300000 hexahedral cells).....	99
Figure 4-14 Water volume fraction contours with fine mesh (600000 hexahedral cells).....	99
Figure 4-15 Velocity Profile at Z = 4.5 m with max velocity 0.75 m/s with coarse mesh (300000 hexahedral cells).....	100
Figure 4-16 Velocity Profile at Z = 4.5 m with max velocity 0.75 m/s with fine mesh (600000 hexahedral cells).....	100
Figure 4-17 Pressure drops with fine mesh (500000 hexahedral cells) with max 3900 [Pa].....	101
Figure 4-18 Liquid Holdup at pipeline with fine mesh (500000 hexahedral cells).....	101
Figure 4-19 Water volume fraction contours (sluggish) with fine mesh (500000) hexahedral cells.....	102
Figure 4-20 Velocity Profile at Z = 5 m with max velocity 1.10 m/s with coarse mesh (500000 hexahedral cells).....	102
Figure 4-21 Pressure drops with fine mesh (500000 hexahedral cells) with max 1500 [Pa].....	103
Figure 4-22 Liquid Holdup at pipeline with fine mesh (500000 hexahedral cells)	103
Figure 4-23 Water volume fraction contours (sluggish) with fine mesh (500000 hexahedral cells)	104
Figure 4-24 Velocity Profile at Z = 5 m with max velocity 1.4 m/s with coarse mesh (500000 hexahedral cells)	104
Figure 4-25 Pressure drops with fine mesh (500000 hexahedral cells) with max 490 [Pa].....	105

Figure 4-26 Liquid Holdup at pipeline with fine mesh (500000 hexahedral)	105
Figure 4-27 Water volume fraction contours (sluggish) with fine mesh (500000 hexahedral cells).....	106
Figure 4-28 Velocity Profile at Z = 5 m with max velocity 1.85 m/s with coarse mesh (500000 hexahedral cells)	106
Figure 4-29 Pressure drops with fine mesh (600000 hexahedral cells) with max 3200 [Pa].....	107
Figure 4-30 Liquid Holdup at pipeline with fine mesh (600000 hexahedral)	107
Figure 4-31 Water volume fraction contours (sluggish) with fine mesh (600000 hexahedral cells).....	108
Figure 4-32 Velocity Profile at Z = 9 m with max velocity 1.05 m/s with coarse mesh (600000 hexahedral cells).....	108
Figure 4-33 Pressure drops with fine mesh (600000 hexahedral cells) with max 2500 [Pa].....	109
Figure 4-34 Liquid Holdup at pipeline with fine mesh (600000 hexahedral)	109
Figure 4-35 Water volume fraction contours (sluggish) with fine mesh (600000 hexahedral cells).....	110
Figure 4-36 Velocity Profile at Z = 9 m with max velocity 1.1 m/s with coarse mesh (600000 hexahedral cells).....	110
Figure 4-37 Pressure drops with fine mesh (500000 hexahedral cells) with max 790 [Pa].....	111
Figure 4-38 Liquid Holdup at pipeline with fine mesh (600000 hexahedral).....	111
Figure 4-39 Water volume fraction contours (sluggish) with fine mesh (600000 hexahedral cells).....	112
Figure 4-40 Velocity Profile at Z = 9m with max velocity 1.1 m/s with coarse mesh (600000 hexahedral cells).....	112
Figure 4-41 Comparison graph bet Numerical and Experimental data case 1.....	113
Figure 4-42 Graph of experimental data for case 1.....	114

Figure 4-43 Comparison graph between Anisotropic and Experimental data case 1...117

LIST OF TABLES

Table 2-1 Mean slug lengths in horizontal pipes.....83

Table 2-2 Summary of fully developed slug frequency models.....85

Table 4-1 Data recorded between t = 15.7s and t = 15.78s.....114

Table 4-2 Data recorded between t = 43.13s and t = 43.24s.....115

Table 4-3 Data recorded between t = 47.1s and 47.21s.....116

Table 4-4 Data recorded between t = 43.13s and t = 43.25s.....118

LIST OF ABBREVIATIONS

IT	Information Technology
RANS	Reynolds Averaged Navier-Stokes models
LES	Large Eddy Simulations
DNS	Direct Numerical Simulations
SPH	Smooth Particle Hydrodynamics
LBM	Lattice Boltzmann Method
DEM	Discrete Element Method
DDPM	Dense Discrete Phase Model
DPD	Dissipative Particle Dynamics
ASM	Algebraic-Slip Mixture model
EE	Eulerian-Eulerian Model
IAC	Interface Area Concentration Model
VOF	Volume of Fluid Model
LS	Level Set Method
FT	Front Tracking Method
DPM	Discrete Phase Model
DDPM	Dense Discrete Phase Model
DEM	Discrete Element Method
SPH	Smooth Particle Hydrodynamics
LBM	Lattice Boltzmann Method
DPD	Dissipative Particle Dynamics
(CLSVOF)	Coupled Level Set and Volume of Fluid Model

NOMENCLATURE

Symbols:

a_{int} :	interfacial area concentration
$B_{i,B}$:	bubble birth rate in size group i due to breakup of larger bubbles
$B_{i,C}$:	bubble birth rate in size group i due to coalescence of smaller bubbles
$C\alpha$:	interface compression coefficient
CD :	drag coefficient c_s : speed of sound
$D_{i,B}$:	bubble death rate in size group i due to
D :	turbulence diffusion coefficient
D :	number of space dimensions
d :	diameter of bubble
dc :	damping constant
e_{12} :	unit normal vector between particle 1 and 2
f :	particle distribution function
F :	force term for LBM
F :	momentum source (force per unit volume)
G :	strength of the interaction
g :	gravity
h :	smooth length parameter
K_{ij} :	momentum transfer coefficient from phase i to j
kn :	normal stiffness
m :	mass (kg)
M :	total number of bubble size groups

N:	total number of discrete particles
n:	normal vector to interface
n:	total number of phases
Δn :	deformation of a particle in normal direction
p:	pressure
Q:	number of discrete velocities
r:	discrete particle position vector
r_{ij} :	relative position ($r_i - r_j$)
r_b :	position vector for a neighbor particle
s_i :	size fraction of bubble size
$i = a$	number of bubble in size group total number of bubbles i
$\dot{S}_i$:	continuity equation source term for size group i
SRC:	coalescence through random collision driven by turbulence eddies
SWE:	coalescence through collision due to acceleration by wake
STI:	breakup due to turbulent eddies
SRO:	shearing off around the base rim of the cap bubbles
SSI:	breakup of large cap bubbles due to surface instability
SRV:	collision due to the difference in the bubble rise velocity
SLS:	breakup due to the laminar shear in viscous fluid
Svg:	collision due to the velocity gradient
SMg:	momentum transfer to velocity group j due to coalescence and breakup of bubbles
Sct :	turbulent Schmidt number
t:	time

u :	fluid velocity field
U :	time averaged fluid velocity
u_{ck} :	compression velocity for phase k
V :	volume
v :	discrete particle velocity
w :	kernel function
x :	position
α :	volume fraction
γ :	damping constant
κ :	curvature
μ :	viscosity
μ_f :	friction coefficient
Π_{ij} :	artificial viscosity between particle i and j
ρ :	density
σ :	surface tension
ϕ :	interface function
ν :	kinematic viscosity
η :	SPH model constant ($\eta = 0.1 h$)
τ :	relaxation time
ψ :	density dependent interaction potential
ω :	weight function Subscripts
b :	bin number (for LBM)
B :	buoyancy
D :	drag

dr,k:	drift velocity of phase k
eq:	equilibrium
i:	particle ID
j:	pair particle of particle i
k:	arbitrary phase (liquid or gas)
q:	the other phase
G:	gravity
g:	gas
gj:	gas phase velocity group j
L:	lift
m:	mixture
M:	momentum transfer from different size groups by breakup or coalescence
p:	particle
P:	pressure gradient force
rel:	relevant
t:	turbulent
TD:	turbulence dispersion
tan:	tangential
ter:	terminal
V:	virtual mass
WL:	wall lubrication
0:	reference

Superscript

n:	new
o:	old
T:	transpose

Math

∇ :	gradient
∇_i :	gradient in particle 'i' velocity direction
$\nabla \cdot$:	divergence
$ $:	absolute value
∂_t :	partial derivative with respect to time
dt :	exact derivative with respect to time
D_t :	material derivative
$\times$:	cross product
Σ :	summation

Roman symbols:

$\tilde{A}$	dimensionless wetted area (-)
A_{LF}	cross-sectional area of the liquid film in front of the slug (m ²)
$\tilde{D}_L$	dimensionless hydraulic liquid diameter (-)
$\tilde{h}_L$	dimensionless height (-)
$\dot{m}$	mass velocity (kg/s)
$\tilde{S}$	dimensionless wetted perimeter (-)
$\dot{V}_G$	net gas volumetric flow rate per unit area (m/s)
$\dot{V}_L$	net liquid volumetric flow rate per unit area (m/s)
B_o	distribution parameter of Zuber and Findlay (1965)
C	wave propagation velocity(m/s)

C_R	real part of the wave velocity (m/s)
C_o	distribution parameter for bubble velocity (Nicklin et al., 1962)
f	friction factor
f_{wG}	friction factor between the gas and the pipe wall (-)
f_{wL}	friction factor between the liquid and the pipe wall (-)
f_i	friction factor between the liquid and the gas (-)
F	dimensionless liquid frequency (-)
Fr	Froude number (-)
Fr_c	critical Froude number - Bendiksen (1984)
Fr_G	gas Froude number in Taitel and Dukler (1977) model
Fr_s	Froude number evaluated using the liquid velocity within the liquid slug (-)
Fr_{T-D}	Taitel and Dukler parameter
f_s	friction factor in the slug body (-)
f_m	two-phase friction factor (-)
$\bar{h}_G$	mean height of the gas phase
h_{Lf}	liquid film height immediately in front of the slug (m)
k	wave number
$K_{s(T-D)}$	Taitel and Dukler parameter
K_{S-I}	stratified to intermittent transition coefficient
Ku	Kutateladze number
l	length (m)
l_f	film length (m)
l_s	slug length (m)

l_u	slug unit length (m)
$\dot{M}_L$	liquid mass flow rate (kg.m/s)
$\dot{M}_L$	liquid pick up mass flow rate (kg.m/s)
N_μ	ratio of viscous and gravitational force (Gokcal et al., 2009)
P	pressure (N/m ²)
Re	Reynolds number (-)
Re_s	Reynolds number in the slug body (-)
S	wetted perimeter (m)
s_{T-D}	sheltering coefficient
t	time (s)
T	temperature (K)
T_{T-D}	Taitel and Dukler parameter - Eqn. 2.48 (Taitel and Dukler, 1976)
U_{SL}	Superficial velocity (m/s)
u_b	velocity of dispersed bubbles in liquid slug (m/s)
u_d	drift velocity (m/s)
$\bar{u}_{GStr}$	average gas velocity in the stratified layer (m/s)
$\bar{u}_{LStr}$	average liquid velocity in the stratified layer (m/s)
$\bar{u}_\alpha$	average of the velocities of the two phases (m/s)
$\bar{u}_{Lbubble}$	average velocity of liquid droplets in elongated bubble (Bendiksen and Espedal, 1992)
u_{Lf}	liquid velocity in the film region immediately in front of the slug (m/s)
)	
u_{LS}	liquid velocity within the slug body (m/s)
u_{MO}	limiting velocity (m/s) (Andreussi and Bendiksen, 1989)

u_{Mf}	limiting velocity (m/s) (Andreussi and Bendiksen, 1989)
U_m	mixture velocity (m/s)
U_T	translational velocity of the slug (m/s)
V_T	net loss rate of small bubbles at the slug tail (-) (Andreussi & Bendiksen, 1984)
V_F	the net entrainment rate at the slug front (-) (Andreussi & Bendiksen, 1984)
V_{PR}	bubble production rate (-) (Andreussi & Bendiksen, 1984)
V_{PB}	small bubble shedding rate to elongated bubble (-) (Andreussi & Bendiksen, 1984)
V_W	wave crest velocity
We	Weber number
X_{L-M}	Lockhart and Martinelli parameter (-)

Greek symbols:

ε	holdup (-)
ε_G	volume fraction of gas (-)
ε_{Ghog}	volume fraction of gas in homogeneous void fraction model (-)
ε_{Gsep}	volume fraction of gas in separated void fraction model (-)
η	wave profile (-)
$\varepsilon_{Lbubble}$	volumetric fraction of liquid droplets in elongated bubble (Bendiksen and Espedal, 1992)
ε_{LEmod}	modified stratified equilibrium liquid holdup (-) (Hill and Wood, 1994)
ε_{LS}	liquid holdup in the slug body (-)

ε_{Gs}	gas holdup in the slug body (-)
ε_{Lfilm}	average liquid holdup in an aerated film (Bendiksen et al., 1996)
α	input liquid fraction
λ_B	Fluid property correction factor – Eqn. 2.3 (Baker, 1954)
μ	dynamic viscosity (Ns/ m ²)
μ_m	mixture viscosity
μ_{LB}	reference dynamic viscosity (Baker, 1954)
ρ_m	mixture density
ρ_{LB}	reference liquid density (Baker, 1954)
σ	surface tension (N/m)
σ_B	reference surface tension (Baker, 1954)
τ	shear stress (N/m ²)
τ_s	fluid-wall shear stress in the slug body
ϑ_G	relative gas velocity (m/s)
ϑ_L	relative liquid velocity (m/s)
ψ_B	fluid property correction factor – Eqn. 2.4 (Baker, 1954)

Subscripts:

SL	superficial liquid
G	gas
G_m	gas mixture
Gf	gas properties within the film region
Gs	gas properties within the slug body
Gu	gas properties within the slug unit

L	liquid
L_m	liquid mixture
L_f	liquid properties within the film region
L_s	liquid properties within the slug body
s	Properties evaluated for the slug body
u	Properties evaluated for the entire slug unit.

Chapter 1: INTRODUCTION

1.1 Background

Multiphase flows are present in different engineering processes including, e.g., conventional, and nuclear power plants, combustion, propulsion, chemical, biological and metallurgical, oil and gas production industrial sectors. Multiphase flows are also part of our natural environment, i.e., they can be found in rainy or snowy winds, air and water pollution, and volcanic activities. Multiphase systems must be able to behave predictably for this industrial application. This explains why there are currently so many more research articles in this area.

The subject of our study consists of interacting gas and liquid phases that are dispersed in a very complicated way spatially and temporally. Momentum, energy, and mass are all exchanged from one phase to another at each gas-liquid interface. These transfers should be a significant factor in the prediction of two-phase configurations in practice. The transport mechanisms and interface regions over which they occur are challenging to measure experimentally, thus our understanding of two-phase flow is far from complete. The main challenge in creating trustworthy two-phase flow systems has been and will continue to be the existence of interfaces and the challenging work of characterizing them. The importance of two-phase flow in many fields has sparked interest in it.

The two phases (gas and liquid) can be categorized in a wide range of flow regimes or patterns. The flow pattern may consist of one continuous phase and another that is present as liquid drops or gas bubbles (bubble flow). The two phases (gas and liquid) can be geometrically separated in a stratified multiphase flow, e.g., in a long horizontal pipe or in a vertical channel. They might be switching between different fluid flow patterns, resulting in a pattern called a "slug flow" or "churn flow." The overall and local time behavior of all two-phase flow patterns is intermittent, and the changes across time are much larger than for a single phase. Building computer codes that deal with the detailed two-phase multiphase fluid flow structures in various flow regimes is a real challenge.

In two-phase system, the flow can be known as co-current where the phases flow in the same direction or countercurrent when they flow in opposite directions.

The derivation of mathematical models describing multiphase flows should become a real challenge or even impossible if we must take into account all the preceding requirements. Therefore, approximations and averaging schemes are required to be used in most mathematical models and computer codes.

Computational engineering models that describe multiphase flows with their different phase distribution patterns can be classified in several groups as follows:

Quasi-Multiphase models, multi-Fluid models, Interface Tracking models, Moving Grids models and Particles Based models.

Quasi-Multiphase models or mixture models treat fluid properties such density, viscosity, and velocity as mixture. Multi-Fluid models or separated flow models focus on identifiable phases when they interact with each other. An inclined drainpipe can be considered as an example where the liquid phase flows down while the gas phase flows in the upper part of the pipe.

Particles based models are used to model one phase as discrete particles, e.g., solid and gas phases, distributed in the continuous primary phase. In this case, there is a huge interaction between phases.

1.2 Project Objectives

The objective of this project is to develop an anisotropic multiphase engineering turbulence model for accurately predicting slug flows, because these flows are relevant in industrial applications. This complex type of flow is a highly intermittent flow regime in which large bubbles flow alternate with liquid slugs at randomly fluctuating frequency. The liquid slugs may be either of pure liquid or aerated.

Slug flow can be created in horizontal pipes by the natural development of hydrodynamic instabilities, where small random perturbations with short wavelengths may emerge and build into larger and longer waves on the surface of the liquids. This growth is caused by the standard Kelvin-Helmholtz (KH) instability mechanism. Slugs are created when such waves increase until they

cross the cross section of the pipe by absorbing liquid flowing in front of them. If the slug fronts move faster than the tails, the slugs may develop; otherwise, they would disintegrate. The slug flow becomes stable when the speed of the front and tail are equal. All of these things typically happen at various periods, causing some slugs to grow and others to shrink. Additionally, certain slugs may move at different rates, which might cause some slugs to merge with others.

This type of flow exhibits unsteady behaviour with high fluctuating vorticity and instabilities due to the complex interactions that can be rotational, fully three dimensional and time dependent. Therefore, the flow is highly turbulent and can display anisotropic behaviour in terms of the distribution of the Reynolds stresses.

Most of the current turbulent models available in commercial computer codes such as Fluent, are assuming isotropic turbulence.

In isotropic simplification of mathematical and physical description of turbulent flows proposed by Taylor, all normal components of the Reynolds stress tensor are equal and non-diagonal shear stress component in the Reynolds stress tensor are assumed to be equal to zero (Könözsy, 2019). This approach usually considers special physical circumstances such as turbulence far from any solid boundary. However, in real turbulent flows such slug flows, the shear stress in the Reynolds stress tensor could easily break near to any solid boundary when the effect these shear stresses become dominant.

Therefore, for those turbulent flows when effect of any solid boundary is present, an anisotropic mathematical description of the Reynolds stress tensor is required. It means that all the normal stresses components of the Reynolds stress tensor are not equal to each other and all the non-diagonal shear stress components in the Reynolds stress tensor are assumed to be not equal to zero (Könözsy, 2019).

Our aim should be to find a plausible mathematical description of the anisotropic Reynolds stress tensor for multiphase slug flows. It means that to propose a physically correct description of the fluctuating velocity components that can support the contribution to the knowledge in this thesis.

1.3 Contribution to the knowledge

Within the RANS models, the Reynolds stress plays a huge role as it can make models become either isotropic or anisotropic. In the case of isotropic turbulence models, the Reynolds stresses are proportional to the mean velocity gradients, in others words the deviatoric Reynolds stress is directly proportional to the local mean rate of strain, where eddy viscosity is viewed as scalar. In the case of anisotropic turbulence models, where One example of problem may be the case of 3D turbulent flows multiphase slugs in horizontal pipelines, a new mathematical description of the Reynolds stress tensor has been proposed within the context of URANS/RANS turbulence modelling approach. A new tensor has been included in addition to the mean rate-of-strain tensor to capture the anisotropic behaviour of the turbulent flow. This is a multiphase anisotropic RANS modelling approach which is a further extension of a single-phase anisotropic turbulence model to multiphase fluid flow problems.

1.4 Summary of subsequent Chapters

Chapter 2 will exclusively be about literature review. In this chapter, we set the background on two phase flow regimes or patterns. Depending on the flow characteristics, flow regimes can be classified as stratified flow, slug flow, annular flow and dispersed bubble flow. A particular focus is made on slug flow by introducing its different types that can be terrain induced slug or severe slugging, hydrodynamic slug and pigging slugs. Another attention is made on hydrodynamic slug modelling as it has been the subject of our work. After describing the importance of slug flows in the petroleum industry, we investigated the slug flow mechanisms by highlighting the importance of the Kelvin-Helmholtz instability criterion and discuss the slug modelling approaches. We finally end our discussion on slug flow parameters.

Secondly, we discuss about turbulence modelling that can be investigated by using different approaches including RANS models, LES models and DNS models. A particular attention is considered for URANS/RANS models as they are the object of our investigations. Within the RANS turbulence models we will discuss about Eddy viscosity models (isotropic) such as $k - \varepsilon$ model and

Reynolds stress models (anisotropic). Further development of the anisotropic turbulence models will help to create a new anisotropic multiphase mixture flows model that will be a contribution to the knowledge.

Finally, a review multiphase flow models, which can be divided into six different categories as follows: a) quasi-multiphase models, b) multi-fluid models, c) moving mesh techniques, d) interface tracking approaches, e) particle-based methods and f) hybrid models which are described in this chapter. The Volume of Fluid model (VOF) serves as the foundation for our work, understanding other multiphase flow models would be extremely beneficial for researchers.

Chapter 3 will exclusively focus on the methodology and new approach for modelling multiphase anisotropic mathematical (VOF) model. The new model will be solved by using UDF (see Appendix B and C) that will be added to the ANSYS Fluent software.

Chapter 4 will focus on the validation of turbulence approaches for hydrodynamic slugs flows in comparison with experimental data. Experimental data have been selected and numerical solutions to the experimental data have been conducted. The results between experimental and numerical have been compared for isotropic validation and anisotropic validation.

Chapter 5 summarizes the contribution of each chapter in reaching the overall objectives. The main conclusions of each section are stated and areas for further work are recommended.

Chapter 2: LITERATURE REVIEW

2.1 Summary

This chapter starts with description of two-phase regimes or patterns by highlighting that flow regimes can be depicted as stratified flow, slug flow, dispersed bubble flow and annular flow in section 2.2. An introduction of slug is done in section 2.3 showing that there are three types of slug flows such as Hydrodynamic slug flow, severe or terrain induced slug flows and pigging. Section 2.4 deals with turbulence modelling and identifies that turbulence models can be Direct Numerical Simulations (DNS), Reynolds Averaged Navier-Stokes models (RANS) and Large Eddy viscosity models (LES). A review of multiphase flow models has followed in section 2.5 dealing with quasi multiphase models, Multi-Fluid Models in section, Moving Grid Methods in section, Interface tracking Methods, Particle based Models/Methods in section and Hybrid models. Another review is conducted in section 2.6 on numerical methods where discretization methods can be classified as finite difference methods, finite volume methods, finite elements methods and spectral methods. The final review is conducted on Slug flow modelling in section 2.7 where slug flow mechanisms are conducted in section 2.7.1, Slug flow modelling approaches in section 2.7.2, and Slug flow parameters in section 2.7.3.

2.2 Two-phase Flow Regimes

For oil and gas flows in pipelines, numerous different configurations are known as flow patterns or multiphase fluid flow regimes. In horizontal and vertical pipelines, flow patterns depend on component volume fluxes, volume fraction and on fluid properties such as density, viscosity, and surface tension. The results can be found or displayed in the form of flow regime map shown in Figure 1-1 that identifies the flow patterns occurring in various parts of a parameter space defined by the component flow rates, that can vary from being volume fluxes, mass fluxes and momentum fluxes. One of the most widely used multiphase flow pattern map for horizontal gas/liquid flows was created or built by Baker (1954).

Authors including Wallis (1969) and Weisman (1983) participated in the study of flow patterns.

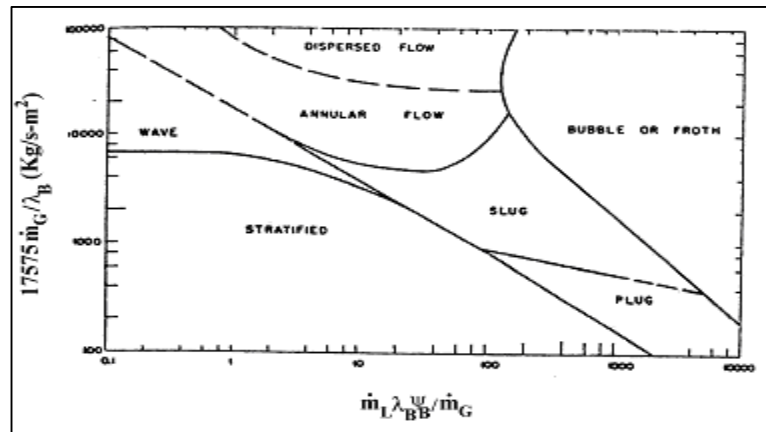


Figure 2-1 Flow pattern map of Baker (1954), modified by Scott (1964)

The flow pattern maps boundaries between the different flow patterns are formed as a result of a regime becoming unstable as it approaches a boundary and growing instability preparing a transition to another flow pattern. These multiphase flow transitions, like the change from laminar to turbulent flow in single phase flow, can be surprising because they may be influenced by minute characteristics of the flow, including the wall roughness or the entrance circumstances. Therefore, we can conclude that flow pattern boundaries are poorly designed transition zones rather than distinct lines. These maps are frequently dimensional, which means that they apply to the particular pipe sizes and fluids that researchers utilise, which is a challenging problem in fluid mechanics. Researchers like Schicht (1969), Baker (1954), Weisman and Kang (1981) worked on the identification of generalised coordinates that would enable maps to include a range of fluids and pipelines. These generalisations are only of limited use, because most multiphase flow pattern maps show multiple transitions, and the corresponding instabilities are determined by various sets of fluid properties. The crucial Weber number may mark one of the transitions, however, a specific Reynolds number may define another limit. Note that a

general non-dimensional flow pattern map does not exist which can consider the complete parametric dependence of fluid properties on the boundaries.

Researchers such as Levy (1999), Taitel & Dukler, (1976) and Mandhane et al. (1974) (see Figure 1-2) describe knowledge on flow patterns in various inclinations to highlight complications and differences in this research field. We can focus on a few qualitative features of the boundaries between multiphase flow patterns and the underlying instabilities leading to transitional flows.

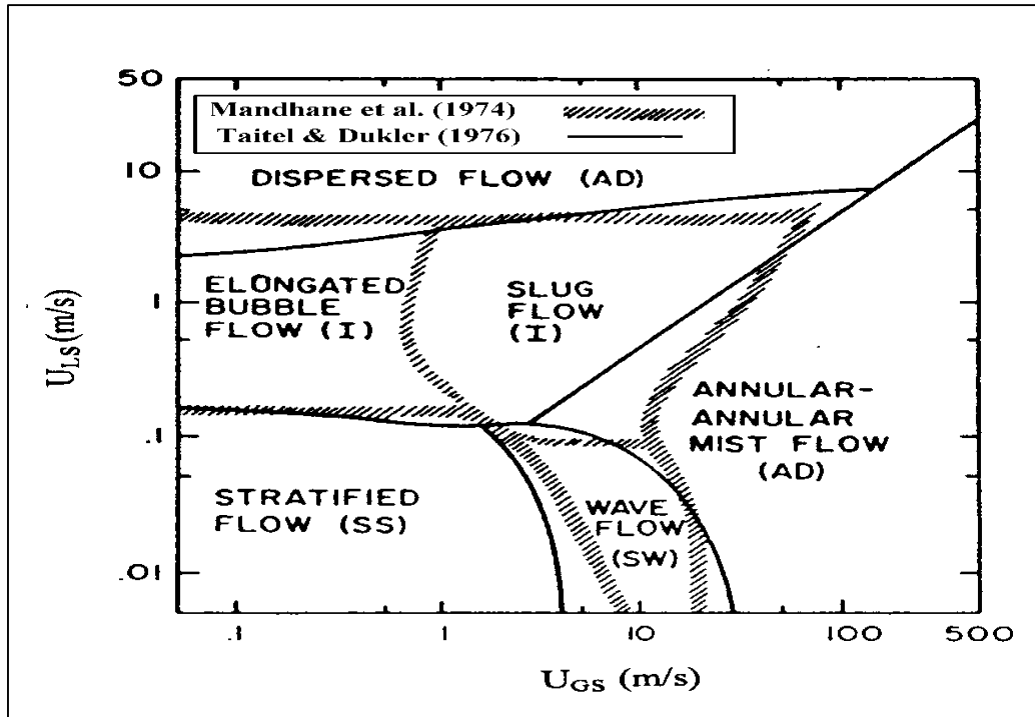


Figure 2-2 Flow pattern maps of Mandhane (1974) and Taitel & Dukler (1976)

2.2.1 Description of Flow Patterns

As above mentioned, researchers like Schicht (1969), Baker (1954), Weisman and Kang (1981) worked on the identification of generalised coordinates that would enable maps to include a range of fluids and pipelines.

2.2.1.1 Stratified flow

According to Figure 1-3, a stratified multiphase flow pattern is when the liquid and gas flows are separated, i.e., the liquid flows downstream along the bottom of the horizontal pipe and the gas flows simultaneously above it. Low gas velocities make the gas-liquid contact look unaltered. As the gas velocity rises, waves start

to form at the gas-liquid interface, giving the flow pattern its other name: stratified wavy. As gas velocity increases further, large-amplitude, long-wavelength waves are produced, and the interface becomes incredibly irregular. When the liquid volume fraction is high, this may result in the production of slug.

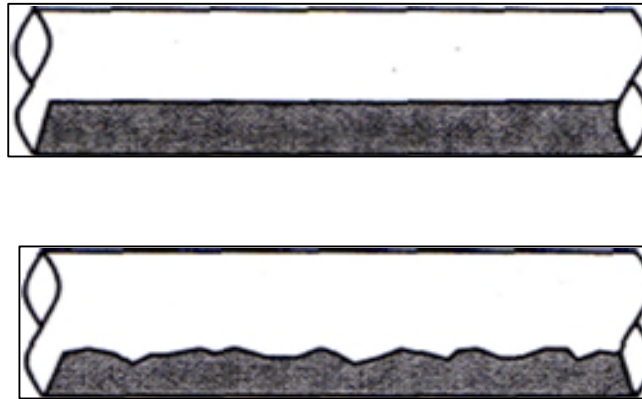


Figure 2-3 Stratified flow pattern

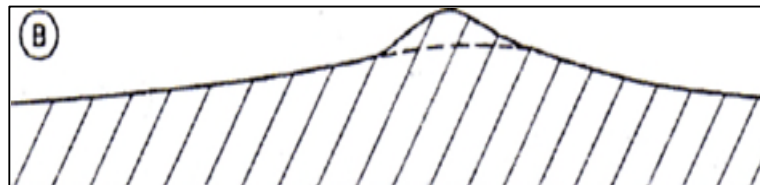
2.2.1.2 Slug flow

A series of liquid slugs separated by huge gas bubbles with diameters that are close to the tube diameter define slug flow. In Figures 1-4, the unsteady development process of slug generation shows liquid and gas in segment *A* arranged in a layered manner. As the liquid layer travels down the pipe, its speed slows down and its level rises. The surface of the liquid also exhibits waves. In segments, labelled as *B* and *C*, waves can develop to a certain size where the liquid level is increasing and the wave height is sufficient to span the pipe at the right gas and liquid velocities. For bridging, the liquid phase in the bridge is sped up to gas velocity capturing the slowly moving liquid in the film ahead.

Large time variations can be present in the mass flow rate, pressure and velocity at any cross section that is normal to the multiphase fluid flow pattern, which is a characteristic of this type of flow pattern. Furthermore, the slug flow pattern is affected by the material properties of the fluid, the geometry of the channel and its vertical or horizontal direction.



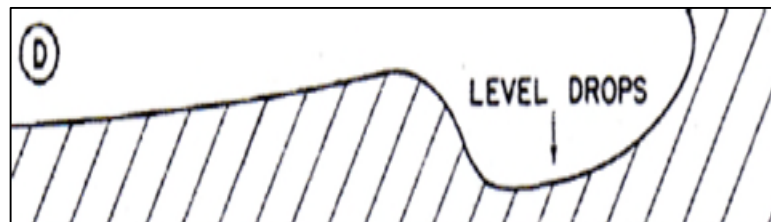
PASSES OUT OF VIEW TO THE RIGHT. LEVEL DROPS



LEVEL REBUILDS AND WAVE NEARLY BRIDGES PIPE



BRIDGING OF PIPE BY LIQUID: SLUG FORMATION.



Slug sweeps up liquid: level drops

Figure 2-4 Slug evolution inside the pipe

2.2.1.3 Dispersed bubble flow

The gas is spread in the liquid as bubbles of varying size and shape. The liquid is the continuous phase. The chemical process industry places a great deal of emphasis on bubble flow since it is important for bubbles to rise through liquids both individually and in swarms. A great deal of research has been done on single gas bubbles rising through liquids, and it has been discovered that surface tension forces cause the bubbles to become spherical when they are very small, and that configuration tends to be maintained as long as their velocity rising, i.e., the Reynolds number stays modest. The interface of gas and liquid can change

over time and distance for the bubbles to take on diverse forms, and in the majority of practical situations, the single gas bubbles are not spherical.

2.2.1.4 Annular Flow

Figure 1-5 depicts an annular flow, which is distinguished by a continuous liquid film on the channel wall encircling a centre core of gas that is loaded with changing numbers of entrained liquid droplets. Many real-world applications, including liquid film evaporators and the majority of high-pressure steam-water systems, use annular flow. Because so many distinct and comparable dynamic forces must be taken into account, it is particularly challenging to analyse.

Three different annular flow patterns exist. The first type, which is observed close to the transition from annular to slug flow, is a liquid film with waves that have a disproportionately large and irregular amplitude. They are referred to as unsteady or flooding waves in this context and are a result of slug flow developing into annular flow as gas flow rate is increasing and their amplitude is decreasing. The second one can be observed when the liquid flow rate is modest, a minimum critical gas velocity can be taken into account when the interface waviness is small, and the core is carried with the least amount of liquid droplets possible. The "ideal" annular flow pattern can be considered when the gas and liquid phases are presumed to be separated with a smooth contact surface which is similar to the flow pattern presented here.

Roll waves develop on the surface of the liquid film as the liquid flow is increasing and the upper gas velocity is attained. They move on top of the liquid layer at a speed several times faster than the liquid surface and are periodic in nature. The roll wave front portion rises abruptly from the liquid film, while their back portion progressively tapers off. As drops are sheared off the top of the waves by the gas flow, the roll waves appearance is accompanied by an increase in entrainment.



Figure 2-5 Annular flow pattern

2.3 Introduction to slug Flows

Due to the excessive demands that big variations in oil and gas flow rates make on processing facilities, multiphase surges are a major flow assurance concern in many oil and gas complexes including multiphase flow lines. Hydrodynamic slugs, terrain-induced slugs, and operationally induced slugs are the three basic types of these surges. They differ greatly in their origin, development, and behaviour; as a result, different methodologies are needed to analyse them.

2.3.1 Terrain-Induced Slugs

Most offshore pipelines deliver both gas and oil at the same time. The liquid phase can build up in the low spots of the pipeline, forming long liquid bridges that can be blown out from one pipeline section to the next by gas pressure, depending on factors like the terrain topography and local flow conditions. In pipelines built on hilly terrain, a phenomenon known as terrain-induced slugging causes significant fluctuations in the liquid outflow rate. If processing facilities are too small, this might lead to equipment tripping and unanticipated shutdowns. A good estimation of the volume of the liquid entering the separator is necessary in order to size the gas-liquid separator at the pipeline exit; yet, such an estimate is challenging to acquire under conditions of terrain-induced slugging.

The study of severe slugging in pipeline-riser systems with a constant separator pressure is the bulk of the research on terrain-induced slugging. Researchers like [Schmidt et al., 1985] and [Fabre et al., 1990] have carried out experimental studies of severe slugging in pipeline riser systems.

Models can be seen in Figure 1-6 to predict slugging in pipeline-riser systems which were developed by (Schmidt et al., 1985; Taitel et al., 1990a; Fabre et al., 1990; Sarica & Shoham, 1991). In a pipeline-riser system severe slugging exhibits a cyclic behaviour according to the literature which is similar to the one observed for terrain-induced slugging in undulating pipelines. An attempt was made by (Taitel et al., 1990b) to consider the upstream and downstream influence, extended a slugging model to handle fluid flow instability in a pipeline with multiple uphill and downhill sections. This model is valid for low volume flow

rates of gas and liquid, because all dynamics terms and friction losses in the momentum balance were neglected in the model. Predictions are also obtained by (Taitel et al., 1990b) for a hypothetical undulating pipeline where a cyclic behaviour can be observed in the pressure which is typical for terrain-induced slugging. However, note that it is not known if the predicted cycle times are realistic and it was mentioned by (Taitel et al., 1990b) that this model is restricted to terrain-induced slugging and cannot be applied to transient multiphase two-phase flow problems in general when pipelines are investigated.

Several transient two-fluid models were created recently to represent transient pipeline phenomena. For instance, (Taitel et al., 1989) suggested a simplified transient model that relies on a local momentum balance and a quasi-steady gas flow. They assumed that the gas flow rate was constant at any cross-section of the pipeline which is a drawback of this model with relation to terrain-induced slugging. A transient two-fluid model known as OLGA was created by (Bendiksen et al. 1991) in which case the interface interactions are taken into consideration through correlation that is dependent on the flow regime. For diverse stationary and transient flows in pipelines, this model worked well. The virtual mass force which may lead to non-physical oscillations (De Henau & Raithby, 1995), was not considered by OLGA in the slug flow regime.

The discussing each of the major types of flow patterns, it is necessary not only to specify such global properties as the pressure drop and the volumetric gas fraction, but also the conditions that determine the transition from one flow to another. Furthermore, interface areas and the transfer processes at the interfaces need to be covered. Also, because the geometry at the interface is quite complex, it is often necessary to simplify it considerably to obtain an analytical representation.

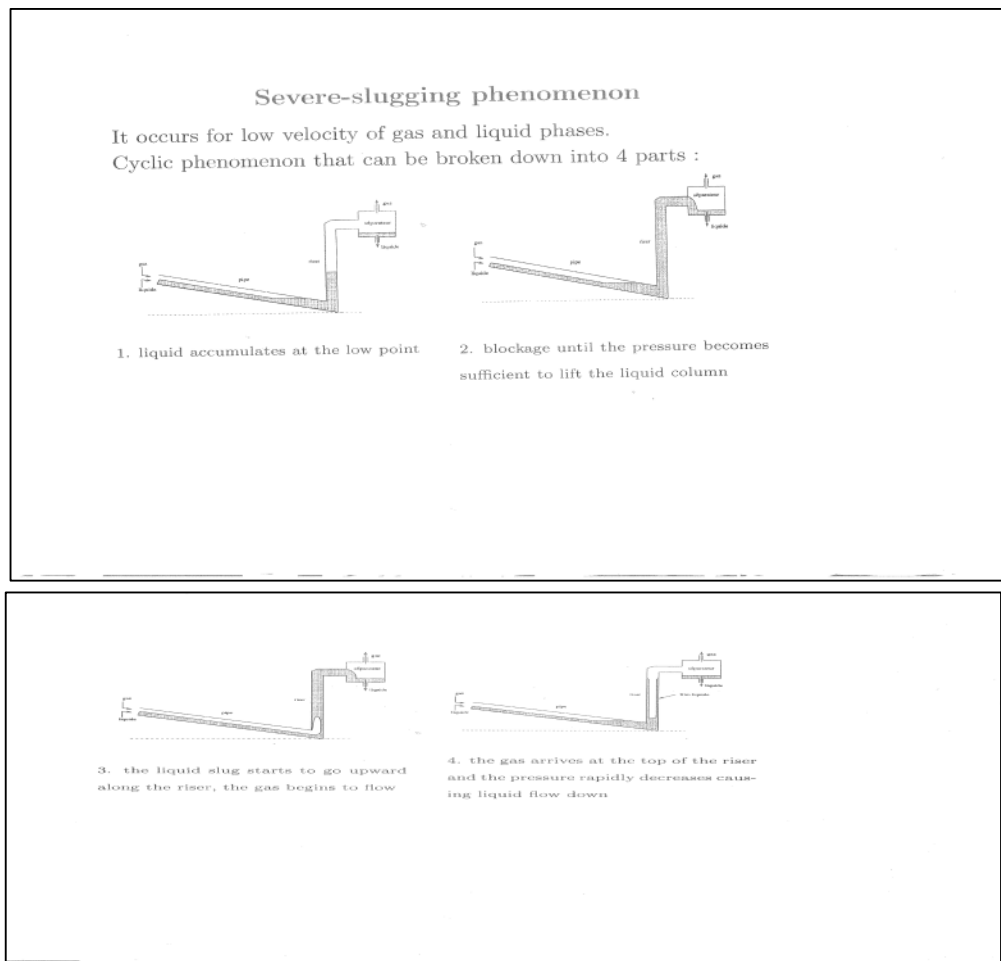


Figure 2-6 Description of severe slug flow phenomenon

2.3.2 Hydrodynamic Slugs

It is believed that hydrodynamic slugs are caused by the instability of waves on the gas-liquid interface in stratified flow under specific circumstances. The pipeline is operating in a slug flow regime when there are numerous hydrodynamic slugs. Complex gas-liquid fluxes that frequently exhibit chaotic behaviour are what define this type of flow. Correlations are used to describe the intricate three-dimensional multiphase phenomena within a slug using commercially available one-dimensional transient flow codes (OLGA). Our study will focus on hydrodynamic slugs.

2.3.3 Operationally Induced Slugs

By accelerating the system from one stable state to another, these slugs are produced. This commonly occurs when pigging operations are underway.

Pigging procedures involve launching water-inflated rubber, neoprene, or polyethylene spheres during lab activities. The entering two-phase mixture pushes those. During this operation, several flowing zones are represented.

- An undisturbed flow zone is located far from the pig. The effects of this pig are not yet felt.
- In front of the moving pig, the liquid slug zone is created and grows more and more.
- The last zone is a redeveloping two-phase flow zone: a very low liquid hold-up zone forms since the pig removes most of the liquid phase.

By this way, pipelines subject to pigging are, always, operated under transient conditions.

Pigging activities are carried out to maintain the pipes dry, which lowers the overall pressure drop and improves the flow efficiency. It can be utilised as a synthetic lift system for wax management or oil production.

2.4 Turbulence Modelling

A large range of length and time scales, no repeatability (i.e., sensitivity to initial conditions), enhanced diffusion (mixing), enhanced dissipation (both of which are mediated by viscosity at molecular scales), three-dimensionality, time dependency, rotationality, and intermittency in both time and space are all characteristics of turbulent flows that can be expected to be present.

The foundation of turbulent flow modelling is the idea that a fictitious field of continuously defined mathematical functions may be used to replace the actual flow field. Typically, these functions represent measurable physical quantities in the flow field. The Navier-Stokes and Reynolds Stress equations are approximated in many turbulence modelling techniques. The degree to which certain turbulence models are complex may be highly influenced by the specifics that one intends to observe and analyze through any numerical simulations. Multiphase turbulence can be thought of as stochastic process and fluid flow instability at high Reynolds numbers (Re). These instabilities result from interactions between the viscous and non-linear convective terms which appear in the momentum balance equation.

Turbulence modeling can be investigated by using different approaches.

- **Direct Numerical Simulation (DNS)** represents, conceptually speaking, the most straightforward approach. To resolve motions at all scales, Navier-Stokes equations are solved with a fine mesh and tiny time increments. To manage and reduce drag on solid surfaces, the effect of compressibility on turbulence, the mechanism of turbulence production, energy transfer and dissipation can be considered to calibrate experimental procedures for near wall flows. This approach is mostly employed in research. The number of grid points is proportional to $Re^{3/4}$ and the size of the grid is determined by the Kolmogorov scale. The finest scale of turbulence, where Re is based on the size of the velocity fluctuations and integral length scale. This strategy requires a lot of effort and computational time.
- **Large Eddy Simulation (LES)** represents a trade-off between the accuracy and computational demands of RANS and DNS. It is a workable research technique for precise simulation of big Re flows; it is more expensive than RANS for the same flow but less expensive than DNS. The best carriers of conserved quantities like mass, momentum, and energy are large eddies. are therefore more active than minor eddies. This methodology, which is time-dependent and three-dimensional by nature and is the preferred method for getting accurate time histories for high Reynolds numbers and complex geometry flows, resolves the biggest scales of flow motion while modelling the smallest scales of motion.
- **Reynolds-Averaged Navier-Stokes (RANS) Models** is the focus of our investigation and is based on a model that ensures closure and time-averages new components that occur in the governing equations of the Navier-Stokes equations. The modelling lessens the need for extremely precise grids. This method is the mainstay of industrial CFD for design analysis, and the underlying turbulence models determine how accurate the simulations will be. The analysis of RANS will be more thorough Reynolds-Averaged Navier-Stokes (RANS) Models

2.4.1 Reynolds Averaging

Engineering systems can be designed, and their performance can be predicted if experimental data and conceptual models are available that can accurately represent physical processes. Different properties of such conceptual models and processes should be precisely formulated on rational foundations in conjunction with experimental evidence. Therefore, it should be expected that conceptual models describing the stationary and dynamic properties of multiphase or multi-component flow will also be expressed in terms of suitable field and constitutive equations. However, it is significantly more difficult to derive these equations for multiphase flows. It means that it is important to keep in mind that in continuum mechanics, field theories are built on integral balances of mass, momentum, and energy to understand the challenges in deriving balance equations for inhomogeneous flows with interfacial discontinuities. Leibnitz's rule can be used to obtain an Eulerian-type differential balance equation if variables in the integration region are continuously, at least twice differentiable. More specifically, the Reynolds transport theorem enables the interchange of differential and integral operations.

The existence of interfacial surfaces of the flow greatly complicates the mathematical and physical formulation of the problem in multiphase or multicomponent flows. A multiphase flow can be conceptualised mathematically as a field divided into single-phase areas with shifting borders dividing the component phases. For each sub-region, the differential balance is maintained. However, it cannot be applied to the collection of these sub-regions in the conventional sense without going against the continuity requirements. The presence of the interface and the fact that both the steady state and dynamic characteristics depend upon the interfacial structure of the system make it difficult to derive the field and constitutive equations appropriate to multiphase flow systems from a physics perspective.

The structure and dynamics of the interface have a significant impact on the rates of mass, heat, and momentum transfer as well as the stability of the system for separated flow, such as stratified flow.

The explanation above leads to the conclusion that describing the local properties of the flow is important to obtain the field and constitutive equations suitable for structured multiphase flow. By using an appropriate averaging approach, the macroscopic properties should be deduced from that flow.

In the literature, various averaging techniques have been put forth. The Eulerian averaging, the Lagrangian averaging, and the Boltzmann statistical averaging are the three basic categories into which these numerous methodologies can be divided. Since Eulerian averaging has been widely used in continuum mechanics and is closely related to human observations and the majority of instruments, we will concentrate on it for this study. The time and spatial description of physical phenomena is the fundamental idea supporting this methodology. The time and space coordinates are treated as independent variables in a so-called Eulerian description, and several dependent variables reflect their changes in relation to these coordinates.

- Time-averaging (Ishii, 1975 and Delhay, 1981) where F is an arbitrary function, and the time scale is Δt as follows:

$$\bar{F}^t = \frac{1}{\Delta t} \int_{\Delta t}^t F(\vec{X}, t) dt' \quad (2-1)$$

- Volume-averaging example (Nigmatulin, 1979, and Soo, 1990, and Whitaker, 1999) over a control volume V as follows:

$$\bar{F}^V = \frac{1}{\Delta V} \int_{\Delta V} F(\vec{X}, t) dV \quad (2-2)$$

- Flow area averaging (Yadigarolu & Lahey, 1976, and Boure & Delhay, 1982) over a cross-section S as follows:

$$\bar{F}^S = \frac{1}{\Delta S} \int_{\Delta S} F(\vec{X}, t) dS \quad (2-3)$$

The concept of Reynolds-averaging is to express the flow variable ϕ in time and space variation as, its mean value and its fluctuating component. It is known as Reynolds decomposition.

ϕ can be velocity, pressure, ...etc.

$$\phi_T = \phi + \phi' \quad \text{where } \phi \text{ is the mean value and } \phi' \text{ the fluctuating value.} \quad (2-4)$$

Using this (2-4) in the Navier-Stokes equation will give the following Reynolds (RANS) equation.

$$\frac{\partial D u_i}{\partial t} = \rho \frac{\partial u_i}{\partial t} + \rho u_i \frac{\partial u_j}{\partial x_i} = \rho g_i - \frac{\partial P}{\partial x_i} + \mu \frac{\partial^2 u_j}{\partial x_i \partial x_i} + \frac{\partial \tau_{ij}^R}{\partial x_i} \quad (2-5)$$

where the Reynolds stress tensor can be expressed

$$\tau_{ij}^R = -\rho \overline{u_i u_j} \quad (2-6)$$

2.4.2 Reynolds Stress tensor

A time correlation of velocity component fluctuation is a component of Reynolds' stress. The primary effect of turbulence velocity fluctuations is to increase shear stresses, which in turn facilitates the movement of momentum within the flow. The Reynolds stress tensor is a symmetric tensor with nine velocity varying correlations as its components.

The symmetrical Reynolds stress tensor from Equation (2-6) can alternatively be stated in matrix form for incompressible turbulent flows as:

$$\tau_{ij}^R = -\rho \begin{bmatrix} \overline{u_1 u_1} & \overline{u_1 u_2} & \overline{u_1 u_3} \\ \overline{u_2 u_1} & \overline{u_2 u_2} & \overline{u_2 u_3} \\ \overline{u_3 u_1} & \overline{u_3 u_2} & \overline{u_3 u_3} \end{bmatrix} \quad (2-7)$$

Taylor (1930) proposed a homogeneous simplification of its mathematical and physical description known as **isotropic turbulence** [Könözy, 2019]. In that case, all diagonal components of the Reynolds stress tensor (2-7) are equal, thus

$$-\rho \overline{u_1 u_1} = -\rho \overline{u_2 u_2} = -\rho \overline{u_3 u_3} \quad (2-8)$$

All non-diagonal components in the Reynolds stress tensor are taken to be equal to zero for isotropic turbulence as

$$-\rho \overline{u_1 u_2} = -\rho \overline{u_2 u_1} = 0 \quad (2-9)$$

$$-\rho \overline{u_1 u_3} = -\rho \overline{u_3 u_1} = 0 \quad (2-10)$$

$$-\rho \overline{u_3 u_2} = -\rho \overline{u_2 u_3} = 0 \quad (2-11)$$

According to Taylor isotropic assumption of the Reynolds stress tensor, the equations above can be roughly met by simply considering unique physical conditions, including turbulence far from solid boundaries.

Since all isotropic turbulent flow conditions according to the equations could

easily break down close to any solid boundary when the influence of shear stresses becomes dominant, the Taylor isotropic turbulence assumption is a significant simplification compared to actual turbulent flows.

It implies that a new mathematical and physical description of the Reynolds stress tensor known as anisotropic approach is needed for those turbulent flows that have the influence of any solid boundary. For anisotropic turbulence, the normal components of the Reynolds stress tensor are not equal, it means:

$$-\overline{\rho u_1 u_1} \neq -\overline{\rho u_2 u_2} \neq -\overline{\rho u_3 u_3} \neq 0 \quad (2-12)$$

And because of the **anisotropic** mathematical and physical description of the turbulent flows, all the non-diagonal shear stress components in the Reynolds stress tensor are assumed to be not equal to zeros, giving the following.

$$-\overline{\rho u_1 u_2} = -\overline{\rho u_2 u_1} \neq 0 \quad (2-13)$$

$$-\overline{\rho u_1 u_3} = -\overline{\rho u_3 u_1} \neq 0 \quad (2-14)$$

$$-\overline{\rho u_3 u_2} = -\overline{\rho u_2 u_3} \neq 0 \quad (2-15)$$

The closure problem of anisotropic turbulence is to find a mathematical and physical description of the fluctuating components of the Reynolds stresses.

The anisotropic conditions describing Reynolds stresses could provide a more realistic physical description of turbulent flows compared to isotropic turbulence assumptions (Könözsy, 2019).

2.4.3 Isotropic Turbulence Models

The eddy viscosity theory, advanced by J. Boussinesq, (1877), postulated that Reynolds stresses are proportional to mean velocity gradients, or, put another way, that the deviatoric Reynolds stress is exactly proportional to the local mean rate of strain. Thus, it follows:

$$\tau_{ij}^R = -\overline{\rho u_i u_j} = \mu_T S_{ij} = \frac{2}{3} \rho K \delta_{ij} \quad (2-16)$$

where:

$$\mu_T = \rho \nu_t \quad (2-17)$$

is the dynamic viscosity, which is made up of the scalar kinematic eddy viscosity ν_t and the fluid density ρ .

$$S_{ij} = \frac{1}{2} \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \quad (2-18)$$

is the second-rank mean strain (deformation) rate tensor, and

$$K = \frac{1}{2} (\overline{u'^2} + \overline{v'^2} + \overline{w'^2}) \quad (2-19)$$

Is the turbulent kinetic energy.

Researchers such as Taylor, G. I. (1915), Prandtl, L. (1925), Von Karman, Th. (1930) and others used the eddy viscosity hypothesis to conduct their work and created some turbulence models known as eddy viscosity-based RANS models. Among these, our study will focus on the Mixing length model, the Spalart-Allmaras model, and the Standard K- ϵ model.

2.4.3.1 K – ϵ Turbulence Model

This method uses two model equations: one for the rate of dissipation of turbulent kinetic energy per unit mass ϵ , and one for the turbulent kinetic energy K . Researchers Launder & Spalding (1974) made the suggestion.

Using K and ϵ , velocity, length scales are defined as:

$$\vartheta = \sqrt{K} \quad (2-20)$$

$$l = \frac{K^{3/2}}{\epsilon} \quad (2-21)$$

$$V_T = C_\mu \vartheta l \quad (2-22)$$

Transport equations for K and ϵ are given as:

$$\frac{\partial(\rho K)}{\partial t} + \frac{\partial(\rho K \bar{v}_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[\frac{\mu_T}{\sigma_K} \frac{\partial K}{\partial x_j} \right] + 2\mu_T S_{ij} S_{ij} - \rho \epsilon \quad (2-23)$$

$$\frac{\partial(\rho \epsilon)}{\partial t} + \frac{\partial(\rho \epsilon \bar{v}_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[\frac{\mu_T}{\sigma_\epsilon} \frac{\partial \epsilon}{\partial x_j} \right] + C_{1\epsilon} \frac{\epsilon}{K} 2\mu_T S_{ij} S_{ij} - 2C_{2\epsilon} \rho \frac{\epsilon^2}{K} \quad (2-24)$$

With widely used values of the fine adjustable constants are defined as:

$$C_\mu=0.09, \quad \sigma_K=1.0, \quad \sigma_\epsilon=1.3, \quad C_{1\epsilon}=1.44, \quad C_{2\epsilon}=1.92$$

This approach has great advantages such as simplicity of implementation, stability and convergence of calculations, good performance for various flows that are relevant to industry and has received the highest validation. For whirling and rotating flows, flows with strong separation, and axisymmetric jets, this technique, however, can demonstrate poor prediction.

Improved two equation models such K- ϵ RNG model, realizable K- ϵ model have been created to enhance to model.

2.4.4 Anisotropic Turbulence Models

The scalar eddy viscosity assumption contributes to several serious errors in eddy viscosity models. According to measurements and simulations such as multiphase flow in horizontal pipelines, eddy viscosity is transformed into a tensor quantity in 3D turbulent flows. As a result, employing a scalar eddy viscosity to calculate Reynolds stresses is no longer appropriate. The use of their own dynamic transport equations to explicitly determine Reynolds stresses is preferred. The Reynolds Stress Model (RSM) is based on this. Thus, it follows:

$$\tau_{ij}^{RA} = -\rho \overline{u_i u_j} = 2\mu_T S_{ij} - \frac{2}{3} \rho K \delta_{ij} + \mu_\Theta \Theta G_{ij} \quad (2-25)$$

Where $\mu_T S_{ij} - \frac{2}{3} \rho K \delta_{ij}$ (2-16) represents the Boussinesq hypothesis.

$$\text{and } \mu_\Theta \Theta G_{ij} \quad (2-26)$$

represents the Anisotropic modification.

Based on experiments for various flow issues, (2-26), $\mu_\Theta (0 < |\mu_\Theta| \leq 1)$ accounts for the physical impact of the anisotropic Reynolds stress distribution $\mu_\Theta \Theta G$.

$$\Theta = \rho K^2 L^2 |\Omega|^2 = \mu_t |\Omega| = \rho a_1 K \quad (2-27)$$

Is the principal (dominant) turbulent shear stress,

L is the turbulent length scale,

a_1 is the Bradshaw constant ($0.3 \leq a_1 \leq 0.31$),

K represents turbulent fluctuations kinetic energy per mass unit,

$|\Omega|$ is the magnitude of the mean vorticity vector with

$$|\Omega| = \sqrt{|\Omega|_1^2 + |\Omega|_2^2 + |\Omega|_3^2} = \quad (2-28)$$

$$\sqrt{\left(\frac{\partial u_3}{\partial x_2} - \frac{\partial u_2}{\partial x_3}\right)^2 + \left(\frac{\partial u_1}{\partial x_3} - \frac{\partial u_3}{\partial x_1}\right)^2 + \left(\frac{\partial u_2}{\partial x_1} - \frac{\partial u_1}{\partial x_2}\right)^2}$$

In (2-27), the turbulent kinetic energy K is defined as:

$$K = \frac{\overline{u' \cdot u'}}{2} = \frac{1}{2} (\overline{u'_1 u'_1} + \overline{u'_2 u'_2} + \overline{u'_3 u'_3}) = \frac{1}{2} \sum_{i=1}^3 \overline{u'_i u'_i} = \frac{1}{2} \overline{u'_i u'_i} \quad (2-29)$$

The tensor $\underline{\underline{G}}$ reflects the orthogonal coordinate transformation of the modified anisotropic deviatoric similarity tensor H^* between the physical x_1, x_2 , and x_3 coordinate systems and the fluctuating x'_1, x'_2, x'_3 coordinate systems.

$$\underline{\underline{G}} = E \cdot H^* \cdot E^T \quad \text{and} \quad E \cdot E^T = E^T \cdot E = I \quad (2-30)$$

E is the matrix of transformation and its transpose, and H is the modified deviatoric tensor of the similarity tensor (Könözsy, 2019, 2021). The unit base vectors of the fluctuating x'_1, x'_2, x'_3 are combined to form E^T and the physical x_1, x_2, x_3 coordinate systems as

$$E = \begin{bmatrix} e_1 \cdot e'_1 & e_1 \cdot e'_2 & e_1 \cdot e'_3 \\ e_2 \cdot e'_1 & e_2 \cdot e'_2 & e_2 \cdot e'_3 \\ e_3 \cdot e'_1 & e_3 \cdot e'_2 & e_3 \cdot e'_3 \end{bmatrix}, \quad E^T = \begin{bmatrix} e'_1 \cdot e_1 & e'_1 \cdot e_2 & e'_1 \cdot e_3 \\ e'_2 \cdot e_1 & e'_2 \cdot e_2 & e'_2 \cdot e_3 \\ e'_3 \cdot e_1 & e'_3 \cdot e_2 & e'_3 \cdot e_3 \end{bmatrix},$$

$$H = \begin{bmatrix} \alpha_{CZ} & 1 & \mu_{CZ} \\ 1 & \beta_{CZ} & \vartheta_{CZ} \\ \mu_{CZ} & \vartheta_{CZ} & \gamma_{CZ} \end{bmatrix} \quad (2-31)$$

Note that $\underline{\underline{G}} = \underline{\underline{G}}^T$ (2-32)

$\underline{\underline{G}}$ can be written in a matrix form as;

$$\begin{bmatrix} G_{11} & G_{12} & G_{13} \\ G_{21} & G_{22} & G_{23} \\ G_{31} & G_{32} & G_{33} \end{bmatrix} = \begin{bmatrix} E_{11} & E_{12} & E_{13} \\ E_{21} & E_{22} & E_{23} \\ E_{31} & E_{32} & E_{33} \end{bmatrix} \cdot \begin{bmatrix} \lambda_{11}^* \alpha_{CZ}^* & 1 & \mu_{CZ} \\ 1 & \lambda_{22}^* \beta_{CZ}^* & \vartheta_{CZ} \\ \mu_{CZ} & \vartheta_{CZ} & \lambda_{33}^* \gamma_{CZ}^* \end{bmatrix} \cdot \begin{bmatrix} E_{11} & E_{21} & E_{31} \\ E_{12} & E_{22} & E_{32} \\ E_{13} & E_{23} & E_{33} \end{bmatrix} \quad (2-33)$$

where λ_{11}^* , λ_{22}^* , λ_{33}^* are anisotropic scale factors and α_{CZ}^* , β_{CZ}^* , γ_{CZ}^* are the main diagonal elements of the deviatoric part of the anisotropic similarity tensor (Könözsy, 2019, 2021). The transformed modified deviatoric similarity tensor (Könözsy, 2019, 2021) scalar major diagonal members are denoted by:

$$G_{11} = \lambda_{11}^* \alpha_{CZ}^* E_{11}^2 + \lambda_{22}^* \beta_{CZ}^* E_{12}^2 + \lambda_{33}^* \gamma_{CZ}^* E_{13}^2 + 2(E_{11}E_{12} + \mu_{CZ}E_{11}E_{13} + \vartheta_{CZ}E_{12}E_{13}) \quad (2-34)$$

$$G_{22} = \lambda_{11}^* \alpha_{CZ}^* E_{21}^2 + \lambda_{22}^* \beta_{CZ}^* E_{22}^2 + \lambda_{33}^* \gamma_{CZ}^* E_{23}^2 + 2(E_{21}E_{22} + \mu_{CZ}E_{21}E_{23} + \vartheta_{CZ}E_{22}E_{23}) \quad (2-35)$$

$$G_{33} = \lambda_{11}^* \alpha_{CZ}^* E_{31}^2 + \lambda_{22}^* \beta_{CZ}^* E_{32}^2 + \lambda_{33}^* \gamma_{CZ}^* E_{33}^2 + 2(E_{31}E_{32} + \mu_{CZ}E_{31}E_{33} + \vartheta_{CZ}E_{32}E_{33}) \quad (2-36)$$

Due to the symmetric property of (2-32) of the tensor $\underline{\underline{G}}$ (2-33), the scalar non diagonal elements G_{12} , G_{13} , G_{23} can be written as:

$$G_{12} = G_{21} = \lambda_{11}^* \alpha_{CZ}^* E_{11}E_{21} + \lambda_{22}^* \beta_{CZ}^* E_{12}E_{22} + \lambda_{33}^* \gamma_{CZ}^* E_{13}E_{23} + E_{11}E_{22} + E_{12}E_{21} + \mu_{CZ}(E_{11}E_{23} + E_{13}E_{21}) + \vartheta_{CZ}(E_{12}E_{23} + E_{13}E_{22}) \quad (2-37)$$

$$G_{13} = G_{31} = \lambda_{11}^* \alpha_{CZ}^* E_{11}E_{31} + \lambda_{22}^* \beta_{CZ}^* E_{12}E_{32} + \lambda_{33}^* \gamma_{CZ}^* E_{13}E_{33} + E_{11}E_{32} + E_{12}E_{31} + \mu_{CZ}(E_{11}E_{33} + E_{13}E_{31}) + \vartheta_{CZ}(E_{12}E_{33} + E_{13}E_{32}) \quad (2-38)$$

$$G_{23} = G_{32} = \lambda_{11}^* \alpha_{CZ}^* E_{21}E_{31} + \lambda_{22}^* \beta_{CZ}^* E_{22}E_{32} + \lambda_{33}^* \gamma_{CZ}^* E_{23}E_{33} + E_{21}E_{32} + E_{22}E_{31} + \mu_{CZ}(E_{21}E_{33} + E_{23}E_{31}) + \vartheta_{CZ}(E_{22}E_{33} + E_{23}E_{32}) \quad (2-39)$$

Where α_{CZ}^* , β_{CZ}^* , γ_{CZ}^* , μ_{CZ} and ϑ_{CZ} the modified anisotropic deviatoric similarity tensor scalar components and H^* is defined as:

$$H^*(x'_1, x'_2, x'_3, t) = \begin{bmatrix} \lambda_{11}^* \alpha_{CZ}^* & 1 & \mu_{CZ} \\ 1 & \lambda_{22}^* \beta_{CZ}^* & \vartheta_{CZ} \\ \mu_{CZ} & \vartheta_{CZ} & \lambda_{33}^* \gamma_{CZ}^* \end{bmatrix} = \begin{bmatrix} \lambda_{11}^* \alpha_{CZ}^* & 1 & \mu_{CZ} \\ 1 & \lambda_{22}^* \beta_{CZ}^* & \vartheta_{CZ} \\ \mu_{CZ} & \vartheta_{CZ} & \lambda_{33}^* \gamma_{CZ}^* \end{bmatrix} - \frac{1}{3} \text{tr}[H(x'_1, x'_2, x'_3, t)] \cdot \begin{bmatrix} \lambda_{11}^* & 0 & 0 \\ 0 & \lambda_{22}^* & 0 \\ 0 & 0 & \lambda_{33}^* \end{bmatrix} \quad (2-40)$$

where $\mathbf{H}$ stands for the similarity tensor with anisotropy (Könözsy, 2019, 2021). It follows that.

$$H^*(X', t) = \begin{bmatrix} \lambda_{11}^* \frac{(2\alpha_{CZ} - \beta_{CZ} - \gamma_{CZ})}{3} & 1 & \mu_{CZ} \\ 1 & \lambda_{22}^* \frac{(2\beta_{CZ} - \alpha_{CZ} - \gamma_{CZ})}{3} & \vartheta_{CZ} \\ \mu_{CZ} & \vartheta_{CZ} & \lambda_{33}^* \frac{(2\gamma_{CZ} - \alpha_{CZ} - \beta_{CZ})}{3} \end{bmatrix} \quad (2-41)$$

Where α_{CZ} , β_{CZ} , γ_{CZ} , μ_{CZ} and ϑ_{CZ} are constants [Könözsy, 2019, 2021]. The scalar components of the matrix corresponding to the transformation $\mathbf{E}$ (direction Cosines) and its transpose $\mathbf{E}^T$ are

$$E_{11} = \frac{1}{\sqrt{1-\lambda^2}} \left[\frac{\Delta U_1^{(t)}}{|\Delta U^{(t)}|} - \lambda \frac{\Omega_1^{(t)}}{|\Omega^{(t)}|} \right] \quad (2-42)$$

$$E_{12} = \frac{1}{\sqrt{1-\lambda^2}} \frac{\Delta U_2^{(t)} \cdot \Omega_3^{(t)} - \Delta U_3^{(t)} \cdot \Omega_2^{(t)}}{|\Delta U^{(t)}| \cdot |\Omega^{(t)}|} \quad (2-43)$$

$$E_{13} = -\frac{\Omega_1^{(t)}}{|\Omega^{(t)}|} \quad (2-44)$$

$$E_{21} = \frac{1}{\sqrt{1-\lambda^2}} \left[\frac{\Delta U_2^{(t)}}{|\Delta U^{(t)}|} - \lambda \frac{\Omega_2^{(t)}}{|\Omega^{(t)}|} \right] \quad (2-45)$$

$$E_{22} = \frac{1}{\sqrt{1-\lambda^2}} \frac{\Delta U_3^{(t)} \cdot \Omega_1^{(t)} - \Delta U_1^{(t)} \cdot \Omega_3^{(t)}}{|\Delta U^{(t)}| \cdot |\Omega^{(t)}|} \quad (2-46)$$

$$E_{23} = -\frac{\Omega_2^{(t)}}{|\Omega^{(t)}|} \quad (2-47)$$

$$E_{31} = \frac{1}{\sqrt{1-\lambda^2}} \left[\frac{\Delta U_3^{(t)}}{|\Delta U^{(t)}|} - \lambda \frac{\Omega_3^{(t)}}{|\Omega^{(t)}|} \right] \quad (2-48)$$

$$E_{33} = -\frac{\Omega_3^{(t)}}{|\Omega^{(t)}|} \quad (2-49)$$

$$E_{32} = \frac{1}{\sqrt{1-\lambda^2}} \frac{\Delta U_1^{(t)} \cdot \Omega_2^{(t)} - \Delta U_2^{(t)} \cdot \Omega_1^{(t)}}{|\Delta U^{(t)}| \cdot |\Omega^{(t)}|} \quad (2-50)$$

where the auxiliary scalar is λ the Cosine of the angle between the vorticity vector defined by $\Omega^{(t)}$ and the velocity difference vector $\Delta U^{(t)}$.

$$\lambda = \cos \varphi = \frac{\Delta U^{(t)} \cdot \Omega^{(t)}}{|\Delta U^{(t)}| \cdot |\Omega^{(t)}|} \quad (2-51)$$

$$|\Delta U^{(t)}| = \sqrt{[\Delta U_1^{(t)}]^2 + [\Delta U_2^{(t)}]^2 + [\Delta U_3^{(t)}]^2} \quad (2-52)$$

$$|\Omega^{(t)}| = \sqrt{[\Omega_1^{(t)}]^2 + [\Omega_2^{(t)}]^2 + [\Omega_3^{(t)}]^2} \quad (2-53)$$

In this case, the scalar elements of the invariant mean vorticity vector at time label t are:

$$\Omega_1^{(t)} = \frac{\partial U_3^{(t)}}{\partial x_2} - \frac{\partial U_2^{(t)}}{\partial x_3}, \quad (2-54)$$

$$\Omega_2^{(t)} = \frac{\partial U_1^{(t)}}{\partial x_3} - \frac{\partial U_3^{(t)}}{\partial x_1}, \quad (2-55)$$

$$\Omega_3^{(t)} = \frac{\partial U_2^{(t)}}{\partial x_1} - \frac{\partial U_1^{(t)}}{\partial x_2} \quad (2-56)$$

2.4.4.1 The Matrix Form of the new Hypothesis on the Anisotropic Reynolds Tensor

The new anisotropic Reynolds stress tensor for incompressible turbulent flows can be expressed as follows using Eq. (2-26):

$$\tau^{RA} = -\overline{\rho \mathbf{U}' \otimes \mathbf{U}'} = 2\mu_t \mathbf{S} + \mu_\Theta \Theta \underline{\underline{\mathbf{G}}} - \frac{2}{3} \rho K \mathbf{I} \quad (2-57)$$

$$= \rho v_t \left[(\nabla \otimes \mathbf{U}) + (\nabla \otimes \mathbf{U})^T \right] + \mu_\Theta \Theta \underline{\underline{\mathbf{G}}} - \frac{2}{3} \rho K \mathbf{I} \equiv \quad (2-58)$$

$$\equiv \rho v_t \left(\frac{\partial U_j}{\partial x_i} + \frac{\partial U_i}{\partial x_j} \right) + \mu_\Theta \Theta G_{ij} - \frac{2}{3} \rho K \delta_{ij} \quad (2-59)$$

Which can be also referred in matrix form as:

$$\tau^{RA} = -\overline{\rho \mathbf{U}' \otimes \mathbf{U}'} = -\rho \begin{bmatrix} \overline{u'_1 u'_1} & \overline{u'_1 u'_2} & \overline{u'_1 u'_3} \\ \overline{u'_2 u'_1} & \overline{u'_2 u'_2} & \overline{u'_2 u'_3} \\ \overline{u'_3 u'_1} & \overline{u'_3 u'_2} & \overline{u'_3 u'_3} \end{bmatrix} = \begin{bmatrix} T_{11}^{RA} & T_{12}^{RA} & T_{13}^{RA} \\ T_{21}^{RA} & T_{22}^{RA} & T_{23}^{RA} \\ T_{31}^{RA} & T_{32}^{RA} & T_{33}^{RA} \end{bmatrix} \quad (2-60)$$

where $\otimes$ the dyad tensor product.

$$T_{11}^{RA} = 2\rho v_t \frac{\partial U_1}{\partial x_1} + \mu_\Theta \Theta G_{11} - \frac{2}{3} \rho K = 2\rho v_t \frac{\partial U_1}{\partial x_1} + \mu_\Theta \rho a_1 K G_{11} - \frac{2}{3} \rho K \quad (2-61)$$

$$T_{22}^{RA} = 2\rho v_t \frac{\partial U_2}{\partial x_2} + \mu_\Theta \Theta G_{22} - \frac{2}{3} \rho K = 2\rho v_t \frac{\partial U_2}{\partial x_2} + \mu_\Theta \rho a_1 K G_{22} - \frac{2}{3} \rho K \quad (2-62)$$

$$T_{33}^{RA} = 2\rho v_t \frac{\partial U_3}{\partial x_3} + \mu_\Theta \Theta G_{33} - \frac{2}{3} \rho K = 2\rho v_t \frac{\partial U_3}{\partial x_3} + \mu_\Theta \rho a_1 K G_{33} - \frac{2}{3} \rho K \quad (2-63)$$

The non-diagonal components of the Reynolds shear stresses, or anisotropic Reynolds stress tensor can be expressed (Könözsy, 2019, 2021) as follows:

$$T_{12}^{RA} = T_{21}^{RA} = 2\rho v_t \left(\frac{\partial U_2}{\partial x_1} + \frac{\partial U_1}{\partial x_2} \right) + \mu_\Theta \Theta G_{12} = 2\rho v_t \left(\frac{\partial U_2}{\partial x_1} + \frac{\partial U_1}{\partial x_2} \right) + \mu_\Theta \rho a_1 K G_{12} \quad (2-64)$$

$$T_{13}^{RA} = T_{31}^{RA} = 2\rho v_t \left(\frac{\partial U_3}{\partial x_1} + \frac{\partial U_1}{\partial x_3} \right) + \mu_\Theta \Theta G_{13} = 2\rho v_t \left(\frac{\partial U_3}{\partial x_1} + \frac{\partial U_1}{\partial x_3} \right) + \mu_\Theta \rho a_1 K G_{13} \quad (2-65)$$

$$T_{32}^{RA} = T_{23}^{RA} = 2\rho v_t \left(\frac{\partial U_2}{\partial x_3} + \frac{\partial U_3}{\partial x_2} \right) + \mu_\Theta \Theta G_{23} = 2\rho v_t \left(\frac{\partial U_2}{\partial x_3} + \frac{\partial U_3}{\partial x_2} \right) + \mu_\Theta \rho a_1 K G_{23} \quad (2-66)$$

The dynamic eddy viscosity $\mu_t = \rho v_t$ by using the dynamic eddy viscosity coefficient μ_t of the $K - \varepsilon$ model of Falding (1974) as a closure equation, the scalar elements of the anisotropic Reynolds stress tensor may be calculated.

2.4.4.2 Mathematical Formulation and Derivation

An examination of the mathematical formulation of the anisotropic hybrid K- ϵ /STM closure model in comparison to the K- ϵ model of Spalding is necessary because computer code is needed to execute our model.

The mass conservation equation for incompressible turbulent flows is written as:

$$\nabla \cdot \mathbf{U} = 0 \quad (2-67)$$

The Reynolds momentum equation of the anisotropic hybrid K- ϵ STM turbulent model, which is based on the new theory of the anisotropic Reynolds stress tensor can be formulated as follows:

$$\rho \frac{\partial \mathbf{U}}{\partial t} + \rho \mathbf{U} \cdot (\nabla \otimes \mathbf{U}) = \rho \mathbf{g} - \nabla P + \mu \nabla^2 \mathbf{U} + \nabla \cdot \left\{ \rho \nu_t [(\nabla \otimes \mathbf{U}) + (\nabla \otimes \mathbf{U})^T] + \mu_\Theta \Theta \underline{\underline{\mathbf{G}}} \right\} - \frac{2}{3} \rho \nabla K \quad (2-68)$$

where $\mu_t = \rho \nu_t$ is the scalar dynamic eddy viscosity coefficient.

The role of μ_Θ ($0 < |\mu_\Theta| < 1$) the weight parameter is to consider the scope of the anisotropic tensorial source term $\mu_\Theta \Theta \underline{\underline{\mathbf{G}}}$ physical effects in relation to experiments for various flow issues such as hydrodynamic slug flows in horizontal pipelines.

The magnitude $\mathbf{S}$ of the mean rate-of-strain $\mathbf{S}$ tensor-based formulation of the eddy viscosity coefficient is substituted for the vorticity magnitude Ω , and the resulting scalar dynamic eddy viscosity μ_t is used to calculate the principal turbulent shear stress Θ (Könözsy, 2019, 2021).

The scalar elements of the transformed modified anisotropic deviatoric similarity tensor $\underline{\underline{\mathbf{G}}}$ are computed based on Equations (2-34) – (2-39).

By resolving the turbulent kinetic energy transport equation, the turbulent kinetic energy (2-69) K in the Reynolds momentum equation can be calculated.

$$\rho \frac{\partial K}{\partial t} + \rho (\mathbf{U} \cdot \nabla) K = P_k^A - \rho \epsilon_k + D_k \quad (2-69)$$

where, P_k^A is the production term, ε_k is the dissipation term and D_k is the diffusion term. They need to be modelled in the context of URANS/RANS turbulent modelling approaches. The kinetic energy dissipation equation ε can be expressed as

$$\rho \frac{\partial \varepsilon}{\partial t} + \rho (\mathbf{U} \cdot \nabla) \varepsilon = \frac{\varepsilon}{K} C_{1\varepsilon} P_k - \frac{\varepsilon^2}{K} \rho C_{2\varepsilon} + \nabla \cdot \left(\frac{\mu_t}{\sigma_\varepsilon} \nabla \varepsilon \right) \quad (2-70)$$

where $C_{1\varepsilon}=1.44$, $C_{2\varepsilon}=1.92$, $\sigma_\varepsilon=1.3$

2.4.4.2.1 Tensor Divergence of the New Anisotropic Reynolds Stress Tensor for Incompressible Turbulent Flows

The Reynolds momentum equation of the anisotropic hybrid K- ε /STM is different compared to K- ε model of Spalding because the anisotropic Reynolds stress tensor (2-57) includes an extra tensorial term called $\mu_\theta \underline{\underline{\Theta G}}$ as a result of the anisotropic adjustment to the Boussinesq hypothesis. This indicates that the divergence of the anisotropic Reynolds stress tensor is distinct from the tensor divergence of the generalised Boussinesq hypothesis for the Reynolds stress tensor. As a result, in contrast to the Reynolds equation of the K- ε model of Spalding, the Reynolds momentum equation (2-5) has an additional tensor divergence term $\nabla \cdot (\mu_\theta \underline{\underline{\Theta G}})$. Consequently, its tensor divergence of the anisotropic hybrid K- ε /STM closure model can be written as:

$$\begin{aligned} \text{Div} T^{\text{RA}} &= \nabla \cdot T^{\text{RA}} = \nabla \cdot \left\{ \rho v_t [(\nabla \otimes \mathbf{U}) + (\nabla \otimes \mathbf{U})^T] + \mu_\theta \underline{\underline{\Theta G}} - \frac{2}{3} \rho K \mathbf{I} \right\} \\ &= \nabla \cdot \left\{ \rho v_t [(\nabla \otimes \mathbf{U}) + (\nabla \otimes \mathbf{U})^T] + \mu_\theta \underline{\underline{\Theta G}} \right\} - \frac{2}{3} \rho \nabla K \end{aligned} \quad (2-71)$$

$$\begin{aligned} &= \sum_{j=1}^3 \left\{ \sum_{i=1}^3 \frac{\partial}{\partial x_j} \left[\rho v_t \left(\frac{\partial U_j}{\partial x_i} + \frac{\partial U_i}{\partial x_j} \right) + \mu_\theta \Theta G_{ij} \right] \cdot \mathbf{e}_j \right\} - \frac{2}{3} \rho \sum_{i=1}^3 \left(\frac{\partial K}{\partial x_i} \cdot \mathbf{e}_i \right) \equiv \\ &\equiv \frac{\partial}{\partial x_i} \left[\rho v_t \left(\frac{\partial U_j}{\partial x_i} + \frac{\partial U_i}{\partial x_j} \right) + \mu_\theta \Theta G_{ij} \right] - \frac{2}{3} \rho \frac{\partial K}{\partial x_i} \end{aligned} \quad (2-72)$$

This equation can be written with its scalar components as:

$$\begin{aligned}
\nabla \cdot \mathbf{T}^{\text{RA}} = & \nabla \cdot \left\{ \rho v_t [(\nabla \otimes \mathbf{U}) + (\nabla \otimes \mathbf{U})^T] + \mu_\theta \underline{\underline{\Theta}} \underline{\underline{G}} \right\} - \frac{2}{3} \rho \nabla K \\
= & \left\{ \frac{\partial}{\partial x_1} \left(2\rho v_t \frac{\partial U_1}{\partial x_1} \right) + \frac{\partial}{\partial x_2} \left[\rho v_t \left(\frac{\partial U_1}{\partial x_2} + \frac{\partial U_2}{\partial x_1} \right) \right] + \frac{\partial}{\partial x_3} \left[\rho v_t \left(\frac{\partial U_1}{\partial x_3} + \frac{\partial U_3}{\partial x_1} \right) \right] \right\} \mathbf{e}_1 + \\
& + \left\{ \frac{\partial}{\partial x_2} \left(2\rho v_t \frac{\partial U_2}{\partial x_2} \right) + \frac{\partial}{\partial x_1} \left[\rho v_t \left(\frac{\partial U_1}{\partial x_2} + \frac{\partial U_2}{\partial x_1} \right) \right] + \frac{\partial}{\partial x_3} \left[\rho v_t \left(\frac{\partial U_2}{\partial x_3} + \frac{\partial U_3}{\partial x_2} \right) \right] \right\} \mathbf{e}_2 + \\
& + \left\{ \frac{\partial}{\partial x_3} \left(2\rho v_t \frac{\partial U_3}{\partial x_3} \right) + \frac{\partial}{\partial x_2} \left[\rho v_t \left(\frac{\partial U_2}{\partial x_3} + \frac{\partial U_3}{\partial x_2} \right) \right] + \frac{\partial}{\partial x_1} \left[\rho v_t \left(\frac{\partial U_1}{\partial x_3} + \frac{\partial U_3}{\partial x_1} \right) \right] \right\} \mathbf{e}_3 \\
& + \left\{ \frac{\partial(\mu_\theta \Theta G_{11})}{\partial x_1} + \frac{\partial(\mu_\theta \Theta G_{12})}{\partial x_2} + \frac{\partial(\mu_\theta \Theta G_{13})}{\partial x_3} - \frac{2}{3} \rho \frac{\partial K}{\partial x_1} \right\} \mathbf{e}_1 + \\
& + \left\{ \frac{\partial(\mu_\theta \Theta G_{21})}{\partial x_1} + \frac{\partial(\mu_\theta \Theta G_{22})}{\partial x_2} + \frac{\partial(\mu_\theta \Theta G_{23})}{\partial x_3} - \frac{2}{3} \rho \frac{\partial K}{\partial x_2} \right\} \mathbf{e}_2 + \\
& + \left\{ \frac{\partial(\mu_\theta \Theta G_{31})}{\partial x_1} + \frac{\partial(\mu_\theta \Theta G_{32})}{\partial x_2} + \frac{\partial(\mu_\theta \Theta G_{33})}{\partial x_3} - \frac{2}{3} \rho \frac{\partial K}{\partial x_3} \right\} \mathbf{e}_3 \quad (2-73)
\end{aligned}$$

We can see that the spatial derivatives of the mathematical terms $\mu_\theta \Theta G_{ij}$, must be implemented in a User-Defined Function based C computer code in the Ansys Fluent software package in our case.

2.4.4.2.2 The New Turbulent Kinetic Energy Production Term

The new anisotropic Reynolds stress tensor (2-57) and the mean velocity gradient tensor can be used to calculate the new anisotropic production term P_k^A of the turbulent kinetic energy equation (2-69) as follows:

$$\begin{aligned}
P_k^A = & \mathbf{T}^{\text{RA}} : (\nabla \otimes \mathbf{U}) = \\
= & \left\{ \rho v_t [(\nabla \otimes \mathbf{U}) + (\nabla \otimes \mathbf{U})^T] + \mu_\theta \underline{\underline{\Theta}} \underline{\underline{G}} - \frac{2}{3} \rho K \mathbf{I} \right\} : (\nabla \otimes \mathbf{U}) \\
= & \rho v_t \{ [(\nabla \otimes \mathbf{U}) + (\nabla \otimes \mathbf{U})^T] : (\nabla \otimes \mathbf{U}) \} + \mu_\theta \Theta \left[\underline{\underline{G}} : (\nabla \otimes \mathbf{U}) \right] - \frac{2}{3} \rho K [\mathbf{I} : (\nabla \otimes \mathbf{U})] = \\
= & \rho v_t \sum_{i=1}^3 \left[\sum_{j=1}^3 \left(\frac{\partial U_j}{\partial x_i} + \frac{\partial U_i}{\partial x_j} \right) \cdot \frac{\partial U_i}{\partial x_j} \right] + \mu_\theta \Theta \sum_{i=1}^3 \left(\sum_{j=1}^3 G_{ij} \cdot \frac{\partial U_i}{\partial x_j} \right) \quad (2-74)
\end{aligned}$$

Due to the mass conservation equation of incompressible flows (Könözsy, 2019, 2021), we can write:

$$\mathbf{I} \cdot (\nabla \otimes \mathbf{U}) = \nabla \cdot \mathbf{U} = 0 \quad (2.75)$$

Where $(\cdot)$ denotes the double inner dot (scalar) product of two second – rank tensors, which often is denoted by the symbol “:”.

It follows that:

$$\begin{aligned} P_k^A &= \rho v_t \{ [(\nabla \otimes \mathbf{U}) + (\nabla \otimes \mathbf{U})^T] \cdot (\nabla \otimes \mathbf{U}) \} + \mu_\Theta \Theta [\underline{\underline{G}} \cdot (\nabla \otimes \mathbf{U})] \equiv \\ &\equiv \rho v_t \left(\frac{\partial U_j}{\partial x_i} + \frac{\partial U_i}{\partial x_j} \right) \cdot \frac{\partial U_i}{\partial x_j} + \mu_\Theta \Theta G_{ij} \frac{\partial U_i}{\partial x_j} \end{aligned} \quad (2-76)$$

With scalar components, it can be written as

$$\begin{aligned} P_k^A &= \rho v_t \{ [(\nabla \otimes \mathbf{U}) + (\nabla \otimes \mathbf{U})^T] \cdot (\nabla \otimes \mathbf{U}) \} + \mu_\Theta \Theta [\underline{\underline{G}} \cdot (\nabla \otimes \mathbf{U})] = \\ &= 2\rho v_t \left[\left(\frac{\partial U_1}{\partial x_1} \right)^2 + \left(\frac{\partial U_2}{\partial x_2} \right)^2 + \left(\frac{\partial U_3}{\partial x_3} \right)^2 + \frac{\partial U_2}{\partial x_1} \frac{\partial U_1}{\partial x_2} + \frac{\partial U_3}{\partial x_1} \frac{\partial U_1}{\partial x_3} + \frac{\partial U_3}{\partial x_2} \frac{\partial U_2}{\partial x_3} \right] + \\ &+ \rho v_t \left[\left(\frac{\partial U_1}{\partial x_2} \right)^2 + \left(\frac{\partial U_2}{\partial x_1} \right)^2 + \left(\frac{\partial U_3}{\partial x_1} \right)^2 + \left(\frac{\partial U_1}{\partial x_3} \right)^2 + \left(\frac{\partial U_2}{\partial x_3} \right)^2 + \left(\frac{\partial U_3}{\partial x_2} \right)^2 \right] + \\ &+ \mu_\Theta \Theta \left[G_{11} \frac{\partial U_1}{\partial x_1} + G_{22} \frac{\partial U_2}{\partial x_2} + G_{33} \frac{\partial U_3}{\partial x_3} + G_{12} \left(\frac{\partial U_1}{\partial x_2} + \frac{\partial U_2}{\partial x_1} \right) + G_{13} \left(\frac{\partial U_1}{\partial x_3} + \frac{\partial U_3}{\partial x_1} \right) + G_{23} \left(\frac{\partial U_3}{\partial x_2} + \frac{\partial U_2}{\partial x_3} \right) \right] \end{aligned} \quad (2-77)$$

All of the components containing the G_{ij} tensor must be incorporated in a computer program as additional source terms for the turbulent kinetic energy equation of Spalding (1974) $K - \varepsilon$ model.

2.4.4.3 Governing Equations of the Anisotropic Hybrid $K - \varepsilon$ /STM Closure Model

To summarize, using the previous equations, the Reynolds averaged governing equations of the anisotropic hybrid $K - \varepsilon$ /STM closure model for incompressible flows can be written as:

1. Continuity equation

$$\nabla \cdot \mathbf{U} = 0 \quad (2-78)$$

also, can written with the cartesian index notation

$$\frac{\partial U_i}{\partial x_i} = 0 \quad (2-79)$$

2. The Reynolds momentum equation

$$\rho \frac{\partial \mathbf{U}}{\partial t} + \rho \mathbf{U} \cdot (\nabla \otimes \mathbf{U}) = \rho \mathbf{g} - \nabla P + \mu \nabla^2 \mathbf{U} + \nabla \cdot \left\{ \rho \mathbf{v}_t [(\nabla \otimes \mathbf{U}) + (\nabla \otimes \mathbf{U})^T] + \mu_\Theta \Theta \underline{\underline{\mathbf{G}}} \right\} - \frac{2}{3} \rho \nabla K \quad (2-80)$$

In the cartesian index notation

$$\rho \frac{\partial U_i}{\partial t} + \rho U_i \frac{\partial U_j}{\partial x_i} = \rho g_i + \frac{\partial P}{\partial x_i} + \mu \frac{\partial^2 U_j}{\partial x_i \partial x_i} + \frac{\partial}{\partial x_i} \left[\rho v_t \left(\frac{\partial U_j}{\partial x_i} + \frac{\partial U_i}{\partial x_j} \right) + \mu_\Theta \Theta G_{ij} \right] - \frac{2}{3} \rho \frac{\partial K}{\partial x_i} \quad (2-81)$$

3. The kinetic energy transport equation can be written as

$$\rho \frac{\partial K}{\partial t} + \rho (\mathbf{U} \cdot \nabla) K = \rho v_t \{ [(\nabla \otimes \mathbf{U}) + (\nabla \otimes \mathbf{U})^T] \cdot (\nabla \otimes \mathbf{U}) \} + \mu_\Theta \Theta [\underline{\underline{\mathbf{G}}} \cdot (\nabla \otimes \mathbf{U})] \quad (2.82)$$

2.5 Mathematical Modelling

Multiphase flows have a completely different appearance with complex physics that are challenging to manipulate, predict, and solve. In contrast to single phase flows, where all fluid properties such as density and viscosity are constant or can be calculated using different commercial computational fluid dynamics (CFD) tools has a long history and is now a standard in the developments for different engineering problems. Essential parameters like density, viscosity, and thermal conductivity, if they depend on the temperature, cannot be computed directly for these flows due to the complicated physics involved, making the application of CFD very useful.

It is also possible to consider multiphase gas-liquid flows as interacting phases when boundaries between the phases present one of the main difficulties in the

mathematical modelling of the behaviour of multiphase flows.

According to Wang (2015) and Zhang (2008), another type of multiphase flow behaviour is when solid particles are present in the flow. Cleary, P. W. (2007) and Prakash, M. (2007) considered other interactions between fluid and powder mould slag layers with molten steel at the top of the mould when the interface shape is changing. These are just a few examples of free surfaces between the flowing molten steel and the surrounding gas that give rise to various multiphase flow situations and call for multiphase flow analysis according to V. Singh (2006).

The extremely difficult settings where these multiphase flow phenomena occur make experimentation tough, hence numerical simulation represents a useful tool to deal with and research them. Water model experiments show and exhibit accuracy concerns in multiphase flows in general, because of different material properties such as relative density, viscosity, and surface tension, contact angle and domain length. According to Chaudhary, (2010), experimental work carried out with water can be considered as accurate for single phase fluid flows.

The governing equations that represent multiphase flow systems should offer a tool for analyzing and forecasting the characteristics of mean flow while being aware of the constraints and their uncertainties. The creation of the physical model is one of the most crucial steps in the modelling process, because numerical models created from inadequate conceptual models may provide non-physical outcomes and solutions that are very different from the physical systems they typically represent. Numerical errors in flow and transport estimates have the potential to be exceedingly expensive in the petroleum business, and notably in the pipeline engineering application.

Various turbulent models have been used, depending on the flow situation, to handle turbulent flows that are relevant to processes like slug flows in pipelines or continuous casting. They consist of RANS, LES, and DNS [Moin, P., 1998]. DNS uses a precise computational grid to resolve the fluid flow specifics. In that case, an empirical turbulent model is not necessary. However, DNS has an extremely high computational cost that rises with the Reynolds number. DNS is

essentially impossible to employ, e.g., in continuous casting due to the high velocities. LES is a different approach to simulate turbulent flow that has good accuracy since it accounts for turbulent eddies that are smaller than the grid size by utilising a sub-grid scale model proposed by John Smagorinsky (1963). RANS models use coarse grids to mimic time-averaged fields by resolving additional scalar transport equations for turbulence. They are widely used in research and industry and have inexpensive computing costs; however, they perform less accurately in transient simulations according to Rodi, W. (1993).

Like single phase flows, multiphase flows are modelled by solving a set of Navier-Stokes equations for momentum balance and the continuity equation for the mass conservation. To depict the phase interactions like averaged interfacial pressure and pressure correction term, constitutive relations for multiphase flows are required which are approximations within the field of continuum mechanics.

There are many different instances of multiphase flow, and the abundance of options makes it challenging for users to select an appropriate model for their application and goals. First, by grouping them into categories, we shall summarize the various multiphase models in this chapter that are currently accessible in the literature. Next, go over each model and their benefits and drawbacks while emphasizing how it can be used. According to their shared characteristics, multiphase flow models can currently be divided into six classes, each of which includes several related models and methodologies. The governing equations for these models are outlined here including examples and their engineering applications for multiphase flows.

2.5.1 Quasi-Multiphase models

This modeling approach has two models: The **Algebraic-Slip Mixture model (ASM) Ishii, M. (1976)** and the **Modified Mixture Model**. Its main purpose is to approximate multiphase flow as an inhomogeneous mixture of gas and liquid.

For the gas-liquid mixture, the traditional ASM solves one continuity equation (2-83) and one momentum equation (2-84) and tracks the gas phase which is the

secondary phase in this case with an additional third transport equation for solving the volume fraction Eq. (2-85) as follows:

$$\frac{\partial \rho_m}{\partial t} + \nabla \cdot (\rho_m \mathbf{u}_m) = 0 \quad (2-83)$$

$$\frac{\partial}{\partial t} (\rho_m \mathbf{u}_m) + \nabla \cdot (\rho_m \mathbf{u}_m \mathbf{u}_m) = -\nabla p + \nabla \cdot (\mu_m (\nabla \mathbf{u}_m + \nabla \mathbf{u}_m^T)) + \rho_m \mathbf{g} + \nabla \cdot (\alpha_g \rho_g \mathbf{u}_{dr,g} \mathbf{u}_{dr,g} + \alpha_l \rho_l \mathbf{u}_{dr,l} \mathbf{u}_{dr,l}) \quad (2-84)$$

$$\frac{\partial}{\partial t} (\alpha_g \rho_g) + \nabla \cdot (\alpha_g \rho_g \mathbf{u}_m) = -\nabla \cdot (\alpha_g \rho_g \mathbf{u}_{dr,g}) \quad (2-165)$$

Where:

$$\mathbf{u}_m = \frac{\alpha_g \rho_g \mathbf{u}_g + \alpha_l \rho_l \mathbf{u}_l}{\rho_m} \quad (2-86)$$

$$\mu_m = \alpha_g \mu_g + \alpha_l \mu_l \quad (2-87)$$

$$\rho_m = \alpha_g \rho_g + \alpha_l \rho_l \quad (2-88)$$

The fluids properties such as According to the regional liquid and gas phase fractions, the density (ρ) (2-88), viscosity (μ) (2-87) and velocity ($\mathbf{U}$) (2-86) of the mixture are predicted based on mass- or volume-weighted averages. From an algebraic equation (2-89) and a slip equation (2-90), the drift velocity ($\mathbf{u}_{dr,g}$) and the relative velocity between the two phases, i.e., gas and liquid ($\mathbf{u}_{gl}$) are determined.

$$\mathbf{u}_{dr,g} = \mathbf{u}_g - \mathbf{u}_m = \left(1 - \frac{\alpha_g \rho_g}{\rho_m}\right) \mathbf{u}_{gl} \quad (2-89)$$

$$\mathbf{u}_{gl} = \mathbf{u}_g - \mathbf{u}_l = \frac{\rho_g d_g^2}{18 \mu_g C_D} \frac{(\rho_g - \rho_m)}{\rho_g} \left(\mathbf{g} - (\mathbf{u}_m \cdot \nabla) \mathbf{u}_m - \frac{\partial \mathbf{u}_m}{\partial t} \right) \quad (2-90)$$

This model is most effective for handling two phase flows as it only needs to consider a single additional transport equation for the volume fraction. It concentrates on finding a flow field within a certain accuracy acceptable for engineering purposes with a minimal computational expense to include the gas phase effect. The ASM model cannot be used to study bubble characteristics,

because it needs an input initial value, such as bubble drag (size and shape). The impacts of the Electro-Magnetic Level Accelerator (EMLA) on the flow for argon bubbles and free surface estimation with a pre-fixed bubble size were studied by Kubo et al. (2007) using the ASM technique.

A Modified-Mixture model devised by Thomas, B.G. (1994) is another quasi-multiphase flow model created to streamline the ASM technique. This method adds turbulent diffusion term to the gas-phase transport equation by replacing equation (2-85) with equation (2-92) where the diffusivity of gas bubbles is predicted by replacing the slip equation with a relative velocity fixed to the bubble terminal velocity according to Geng et al. (2010) as follows:

$$D = \frac{\mu_t}{\rho S c_t} \quad (2-91)$$

$$\frac{\partial \alpha_g}{\partial t} + \nabla \cdot (\alpha_g (u_l + u_{ter})) = \nabla \cdot (D \nabla \alpha_g) \quad (2-92)$$

This method works well for flows with bubbles where the gas quickly reaches its terminal velocity as pointed out by Thomas et al. (1994) and Bessho et al. (1991).

Quasi-multiphase models have the significant benefit of being computationally inexpensive. The effects of the gas phase can be incorporated using simply straightforward equations. The fundamental disadvantage of these models will, however, be their simplicity. Only bubbly flows, i.e., the gas volume fraction is about 10% and the bubble size remains roughly constant from its initial value that can accurately define gas behaviour through the relative velocity.

2.5.2 Multi-Fluid Models

Multi-fluid approach includes four models such as Eulerian-Eulerian model (EE) Harlow et al. (1975), Interface Area Concentration model (IAC) Ishii et al. (2006), Homogeneous MUSIG model Lo, S. (1996), and Inhomogeneous MUSIG model Krepper et al. (2008).

2.3.2.1 Eulerian-Eulerian Model (EE)

The EE multiphase two-fluid model solves the continuity equation (2-93) and the momentum equations (2-94) for each phase in addition to the volume fraction

equation and shared pressure field. This model uses distinct sets of conservation equations for each phase, it is referred to as a split flow model.

The Eulerian model equations are as follows:

$$\frac{\partial(\alpha_l \rho_l)}{\partial t} + \nabla \cdot (\alpha_l \rho_l \mathbf{u}_l) = 0 \quad (2-93)$$

$$\frac{\partial(\alpha_l \rho_l \mathbf{u}_l)}{\partial t} + \nabla \cdot (\alpha_l \rho_l \mathbf{u}_l \mathbf{u}_l) = -\alpha_l \nabla p + \nabla \cdot (\mu_l \alpha_l (\nabla \mathbf{u}_l + \nabla \mathbf{u}_l^T)) + \alpha_l \rho_l \mathbf{g} + \mathbf{F}_{\text{drag}} + \mathbf{F}_{\text{lift}} + \mathbf{F}_{\text{others}} \quad (2-94)$$

$$\mathbf{F}_{\text{drag}} = \frac{3}{4} \frac{C_D}{d_g} \alpha_g \rho_l |\mathbf{u}_g - \mathbf{u}_l| (\mathbf{u}_g - \mathbf{u}_l) \quad (2-95)$$

$$\mathbf{F}_{\text{lift}} = -C_L \alpha_g \rho_l (\mathbf{u}_g - \mathbf{u}_l) \times (\nabla \times \mathbf{u}_l) \quad (2-96)$$

For the gas phase, a related set of Eulerian equations is also solved. $\mathbf{F}_{\text{lift}}$, $\mathbf{F}_{\text{drag}}$ and $\mathbf{F}_{\text{others}}$ stand for forces acting between phases and represent lift, drag, and other forces. The bubble shape, size, and flow regime all affect the drag coefficient C_D and lift coefficient C_L .

There are different physical processes such as velocity gradients, bubble shapes and wall effects can cause the lift force ($\mathbf{F}_{\text{lift}}$) to occur. Wall lubrication and turbulent dispersion might be seen as additional momentum interactions ($\mathbf{F}_{\text{others}}$). To represent the hydrodynamic force caused by an asymmetric drainage of surrounding liquid at a wall, wall lubrication has been introduced. When RANS models are utilized, it is thought that turbulent dispersion will compensate for the turbulent fluctuation effects Ishii et al. (2006).

As the gas phase behaves differently than the liquid phase and this modelling approach is more accurate than other quasi-multiphase models. For (EE) models, solving a second full set of partial differential equations comes at a large additional expense. Like quasi-multiphase, experimental work on the bubble size distribution or data taken from other models must be used to input average bubble features like size and shape into (EE) models. These circumstances are among (EE) models' constraints.

The study of particle transport has been expanded using (EE) models by the addition of a transport model for the solid-phase inclusion, see Liu et al. (2014).

To simulate the behaviour and size distribution of inclusions in gas-stirred ladles, Lou et al. (2013) employed their model with a Population Balance Theory.

2.3.2.2 Interface Area Concentration Model (IAC)

Ishii et al. (2006) developed the interfacial area concentration model where a novel bubble parameter was considered to get over the (EE) and quasi-multiphase model constraints that the local bubble size cannot be determined from the volume fraction field on their own. By resolving an additional transport Eq. (2-97) that includes the interfacial area concentration, a_{int} , along with the volume fraction and velocity fields taken from other multiphase models including an initial bubble size distribution. The goal of the IAC model developed by Ishii et al. (2006) was to evaluate the evolution of the bubble size distribution.

$$\frac{\partial a_{int}}{\partial t} + \nabla \cdot (a_{int} u_g) = S_{RC} + S_{WE} + S_{TI} + S_{RO} + S_{SI} + S_{RV} + S_{LS} + S_{vg} \quad (2-97)$$

The average bubble size can be predicted through the computation of the gas volume fraction (α_g) using a mean diameter as follows:

$$D_{sm} = \frac{6\alpha_g}{a_{int}} \quad (2-98)$$

The impacts of various coalescence and breakdown procedures are taken into consideration by the source terms at the RHS of (Eq. 2-97). The interfacial area will expand during breakdown and shrink during coalescence. This model has a minimal computational cost since the bubble size distribution may be calculated by solving a single extra transport equation. One of the issues related to this model is that its accuracy is linked to the breakup and coalescence modeling, that need empirically based calibrations.

2.3.2.3 Homogeneous MUSIG Model

According to Lo, S. (1996), the prediction of the bubble size distribution evolution can be further improved with the use of the Homogeneous Multi-Size Group (MUSIG) model which was developed as an extension of the (EE) model. Using the Population Balance Theory, a modified continuity equation was constructed:

$$\frac{\partial (\rho_g \alpha_g S_i)}{\partial t} + \nabla \cdot (u_g \rho_g \alpha_g S_i) = S_i(x,t) = B_{i,B} - D_{i,B} + B_{i,C} - D_{i,C} \quad (2-99)$$

$$\alpha_l + \alpha_g \sum_{i=1}^M S_i = 1, \quad \sum_{i=1}^M S_i = 1 \quad (2-100)$$

For each predetermined bubble size group in the gas phase, and for the liquid phase individually, this model solves a continuity equation. The gas velocity field is the same for all bubble sizes, and the momentum interaction between the gas and liquid phases is calculated using the average bubble size. The initial volume fractions of each size group must be predefined as an initial input, which is another restriction of this approach. This initial input must be found from somewhere and can affect the results. Researchers including, Shi et al. (2001) and Liu, *et al.* (2016) have used MUSIG model to investigate multiphase flows.

2.3.2.4 Inhomogeneous MUSIG Model

To improve one of the limitations of the Homogeneous MUSIG model when all the bubble shares the same velocity field regardless of size, a more elaborate model known as Inhomogeneous MUSIG model was developed by allowing different bubble size groups to have different velocity fields. Therefore, more than one set of momentum equations for the gas phase in addition to the modified continuity, Eqs. (2-99) and (2-100) are solved as follows:

$$\frac{\partial(\rho_g \alpha_{gj} u_{gj})}{\partial t} + \nabla \cdot (\rho_g \alpha_{gj} u_{gj} u_{gj}) = -\alpha_{gj} \nabla p + \nabla \cdot (\mu_g \alpha_{gj} (\nabla u_{gj} + \nabla u_{gj}^T)) + \alpha_{gj} \rho_g g + F_D + F_L + F_M \quad (2-101)$$

One big advantage of this model is to allow different bubble sizes behaving differently. On the other hand, the requirement of the computational cost becomes huge, by solving multiple set of Navier-Stoke equations, Liu et al. (2015) reiterates the used of only two velocity groups in practice. Like the other models such as (EE) and homogeneous model, this model also needs the initial bubble size distribution to be input and the application of turbulent models such as LES to become challenging.

2.5.3 Moving Grid Methods

The Arbitrary Lagrangian Eulerian (ALE) approach is one of the moving mesh techniques which is widely used. Moving the grid cells or nodes that make up the mesh of the computational domain is possible with the ALE approach according

to Hirt et al. (1974). The domain lies between the Eulerian approach where the fluid moves through a mesh that remains fixed in space and time about other points and the Lagrangian approach where cells move with the flow.

To capture a certain element of the fluid, the grid points shift in relation to one another, for example, placing the gas-liquid interface at the cell boundaries in multiphase flow. Since the cell surfaces are attached to the interface between phases and the grid deforms over time, the cell/grid velocities on that boundary are set to the interface velocity.

Eqs. (2-102), (2-103) and (Eqs. (2-93), (2-94) with $\alpha_l = 1$ or $\alpha_g = 1$ and all $\mathbf{F} = 0$), which represent a continuity equation and a set of momentum equations within each phase above and below the interface through an iteration technique which are used to impose exact boundary conditions at the interface to predict the location of the interface, i.e., where the interface is located.

$$\text{Kinematic boundary conditions:} \quad (\mathbf{U}_l - \mathbf{U}_g) \cdot \mathbf{n} = 0 \quad (2-102)$$

Dynamic boundary conditions:

$$n(P_l - P_g) = -n \cdot (\mathbf{T}_l - \mathbf{T}_g) + n(K_1 - K_2)\mathbf{d} \quad (2-103)$$

After determining the interface position, the mesh coordinates are changed to distribute the interface cells' deformation over a larger number of cells.

The (ALE) approach has a significant advantage: it allows for the precise determination of the boundary conditions that are favoured by the alignment of the interface with the cell borders, allowing for sharp and accurate recording of the exact interface form, including surface tension effects. However, there are some drawbacks to this moving grid method, such as convergence issues, high computational costs, and the restriction to simple interfaces since it cannot handle interfaces with significant deformation, break apart, or intersect.

Many researchers, including Liu, R. (2014), Takatani *et al.* (2001) have used this method to develop their work. Theodorakakos et al. (1998) and Panaras et al. (1998) applied a two-dimensional stationary version of the moving grid method to capture the interface wave between oil and water.

The moving grid method has the following general procedure:

1. Establish an initial pressure field and interface form as educated guesses.
2. Using the present interface shape and the presumptive pressure field, solve momentum balance equations with the dynamic specification of boundary conditions (Eqs. (2-103)).
3. Determine the mass fluxes passing through the interface by solving the continuity equation.
4. Adjust the interface to the magnitude and direction of the mass fluxes.
5. Reconstruct the remaining portions of the mesh using the rezoning procedure.
6. Repeat steps 1 through 5 until equation (2-102) is met.

2.5.4 Interface Tracking Methods

The Front Tracking (FT) method, Volume of Fluid model (VOF), and Level Set (LS) method are the three basic models that make up this methodology. This group of multiphase models uses a non-moving mesh to solve the continuity equation, a single set of momentum equations to determine the volume of the 3D domain, and equations (2-84) and (2-85) in conjunction with an additional technique to determine where the 2D interface between phases is located.

2.5.4.1 Volume of Fluid Model (VOF)

The VOF, developed by Hirt et al. (1981), is a quasi-multiphase model that solves equations (2-83) to (2-88). It determines the gas fraction field by solving a gas volume fraction transport equation like equation (2.85), with the exception that since there is only one field, the terms with drift velocity in equations (2.84) and (2.85) are dropped. As a result, from the cells with volume percentages between 0 and 1, typically 0 for pure liquid and 1 for pure gas, it is possible to pinpoint the location of the gas-liquid interface. For the surface tension forces used in Eq.

(2.84) by Brackbill et al. (1992), a technique is needed to quantify the curvature of the estimated interface shape.

$$S_{surf} = \int \sigma \kappa n \vec{z} \cdot (\vec{x} - \vec{x}') ds \quad (2-104)$$

The VOF approach, as opposed to moving grids, has the singular ability to monitor complex interface shapes from a simple volume fraction transport equation. However, resolving the interface becomes challenging unless a very fine mesh is used, particularly for small gas volumes like bubbles that require at least 8 cells across their diameter. Saw Kumar et al. (2016). The numerical dispersion of the gas volume fraction can occasionally make it difficult to calculate the surface tension because the contact can become murky. According to Aulisa et al. (2007), the interface must be sharpened using high-order solution methods and strategies for redistributing the gas volume, including geometric reconstruction. Spurious interface velocity can be avoided by applying the sharp surface force (SSF) approach to bubble motion.

The treatment of the advection terms in the volume fraction transport equation to avoid mass loss can restrict the time step following the Courant number limit Pletcher et al. (2012),

$\frac{u\Delta t}{\Delta x} + \frac{v\Delta t}{\Delta y} + \frac{w\Delta t}{\Delta z} < 1$, requiring the time step to decrease with the cell size. This can affect the computation, that can become costly.

Many researchers such as Ilgbusi and Szekely et al. (1994), Wang et al. (2010) have used the VOF approach to implement their work.

As the main purpose of the VOF method is to track the interface between two or more phases, it is appropriate for flows with sharp interfaces, including slug, stratified or free-surface flows.

The equations solved in this approach are written below.

$$\frac{\partial \rho_m}{\partial t} + \nabla \cdot (\rho_m \vec{U}_m) = 0 \quad (2-105)$$

$$\frac{\partial \rho_m \vec{U}_m}{\partial t} + \nabla \cdot (\rho_m \vec{U}_m \vec{U}_m) = -\nabla P + \nabla \cdot (\mu_m (\nabla \vec{U}_m + \nabla \vec{U}_m^T)) + \rho_m \vec{g} + \sigma \quad (2-1-06)$$

$$\frac{\partial}{\partial t} (\alpha_g \rho_g) + \nabla \cdot (\alpha_g \rho_g \vec{U}_m) = 0 \quad (2-107)$$

The mixture density can be computed as

$$\rho_m = \alpha_L \rho_L + \alpha_g \rho_g \quad (2-108)$$

where α_L , α_g are the volume fraction of the liquid and gas phases, ρ_L , ρ_g are the densities of the liquid and gas phases and σ is the surface tension.

The mixture velocity can be given as

$$u_m = \frac{\alpha_L \rho_L u_L + \alpha_g \rho_g u_g}{\rho_m} \quad (2-109)$$

The mixture viscosity can be computed as

$$\mu_m = \alpha_g \mu_g + \alpha_L \mu_L \quad (2-110)$$

where α_L , α_g are the volume fraction of the liquid and gas phases, and μ_L , μ_g are the viscosities of the liquid and gas phases.

2.5.4.2 Level Set Method (LS)

The Level Set (LS) approach looks like VOF but has another way of depicting the interface. In fact, the interface is estimated by a transport equation of scalar function ϕ , representing the distance from the interface.

$$\frac{\partial \phi}{\partial t} + u \cdot \nabla \phi = 0 \quad (2-111)$$

The smooth and continuous curve describes the interface shape. In this approach, fluid parameters are determined using ϕ -weighted averages, like how gas fraction in VOF is calculated. To provide more realistic material property and its gradients across the interface, the Level Set method includes an extra reinitialization procedures such as interface smoothing and field sharpening in comparison with the VOF approach.

The Level Set (LS) method is free of the Courant number limit for the time step size and accurately estimates curvature, which affects surface tension estimation. However, because the interface function is not conservative, problems with mass conservation result. To prevent this numerical smearing of

the interface, a fine mesh with high order numerical schemes is also required. Additionally, the inability to resolve small bubbles and bubble interactions is another weakness shared by LS and VOF.

2.5.4.3 Front Tracking Method (FT)

The FT method tracks the moving interface between phases by combining a standard fixed (Eulerian) mesh of the flow domain with a moving (Lagrangian) mesh. In 1992, Tryggvason et al. By creating a Lagrangian surface mesh that connects massless markers on the interface and records each time step in the Eulerian mesh, Tryggvason et al. (2001) put this strategy into practise. Compared to other interface tracking approaches, the FT method offers a crisp interface. The interplay between fluid properties in the Eulerian domain and the Lagrangian interface, however, needs to be handled carefully. Therefore, additional modelling is needed to handle interactions like coalescence and breakup. Similar to the LS technique, the advection of the moving interface and its rebuilding might result in mass loss. High resolution Eulerian mesh is required for precision, while the Courant number limit limits the time step size.

2.5.5 Particle based Models/Methods

This approach views multiphase flow situations as a carrier phase of dispersed particles, where particles can be solid, liquid, gas or droplets and the carrier phase considered as continuous is either liquid or gas. Particles flow field can be modelled using the Lagrangian approach where individual particles are tracked, or particles to be treated as cloud with continuum like equations known as Eulerian approach, or particles distribution functions can be used to describe the particle flow properties known as PDF approach.

The classification of the nature of these kinds of multiphase flows depends on the interaction between the particle motion and its environment. In one-way coupling, the continuous fluid affects the motion of the particles (for example, a vortex can rotate a particle), and in two-way coupling, the continuous fluid motion can also be affected by the particles' motion (for example, particle wakes can increase dissipation). Three-way coupling considers the two-way coupling conditions in

addition to the localized effects of fluid disturbances on other particles' motion, such as trailing particle drafting. Four-way coupling considers both particle collisions and the three-way coupling criteria.

Smooth Particle Hydrodynamics (SPH), Lattice Boltzmann Method (LBM), Discrete Element Method (DEM), Dense Discrete Phase Model (DDPM), and Dissipative Particle Dynamics (DPD) are examples of particle-based models.

2.3.5.1 Discrete Phase Model (DPM)

The velocity field dominated by the carrier phase, which is in the Eulerian approach, requires a different model in the DPM approach, because it handles the secondary phase as a discrete particle in a Lagrangian manner Maxey, M.R. (1987). By integrating two ordinary differential equations using Newton's momentum equation for the velocity field Eq. (2-112) and the particle trajectory equation (2-113) for the particle position, each particle is tracked separately as a point mass travelling across the flow domain Yeoh et al. (2009):

$$\rho_P \frac{dv_i}{dt} = \sum F = F_D + F_L + F_B + F_V + F_P \quad (2-112)$$

$$\frac{dr_i}{dt} = v_i \quad (2-113)$$

F_D , F_L , F_B , F_V , and F_P are respectively, drag, lift, buoyancy, virtual mass, and pressure gradient forces. According to Grace et al. (2005), virtual mass (F_V) and pressure gradient (F_P) forces are still significant for transient simulations when the particles have highly curved paths.

To represent these forces and consider the geometry of the particles and the boundary layer around each one, separate models are required.

With the use of a Gaussian-distributed random velocity component, the time averaged liquid velocity component U_l can be estimated using RANS turbulence models, the discrete random walk (DRW) approach used by Gosman et al. (1983), are a well-known methodology to account for turbulent fluctuations.

$$u_i = U_l + \epsilon \sqrt{2k/3} \quad \text{with } -1 < \epsilon < 1 \quad (2-114)$$

DPM approach is often used to simulate particles in the liquid steel in continuous casting systems and in the studies of flow pattern in the mold, Pfeiler et al. (2005). Many researchers such as Yu et al. (2008), Kubo et al. (2004), Lopez et al. (2014) have used this approach to implement their work. One of the advantageous features of the DPM approach is its availability to be combined with several different models such as EE, VOF, and other models.

2.3.5.2 Dense Discrete Phase Model (DDPM)

An alternative method called as the Dense Discrete Phase Model (DDPM) was created by Patankar et al. (2001) to enhance the DPM with high particle fractions. Like the DPM, this novel method uses the governing equations of the Eulerian model, Eqs. (2-93) and (2-94) for the continuum phase. DDPM, on the other hand, uses particles to estimate a liquid volume fraction field by adding up the contribution of each particle volume based on where it is located. A volume fraction limit organised as packing structure is also imposed by DDPM to stop particles from overlapping in any computational cell.

2.3.5.3 Discrete Element Method (DEM)

This method (DEM) was initially developed to mimic fluidized beds or strongly packed solid particles, such as granular materials (Collection of Solid Particles, 1979). DEM should be viewed as a continuum phase velocity field-free Lagrangian-based particle tracking method like DPM. However, a velocity field of a continuum phase could be added and used to increase the forces on the particles for thick fluids or low-density particles.

By resolving the DPM Eqs. (2-112) and (2-13) as well as collisions between all the particles, the DEM technique records each particle at each time step, adding additional calculations $O(n^2)$. Along with other forces, Eq. (2-112) also includes a collision force F_{coll} for the collision effect.

$$F_{coll} = F_n + F_t \quad (2-115)$$

The normal collision force, F_n , is dependent on the mechanical characteristics of the particle. While soft particles produce damped, inelastic collisions, collisions between hard particles produce elastic collisions with conserved kinetic energy.

Hard particle linear spring model:

$$F_n = (k_n \Delta_n) e_{12} \quad (2-116)$$

Soft particle linear spring-dashpot model:

$$F_n = (k_n \Delta_n + \gamma (\vartheta_{rel} \cdot e_{12})) e_{12} \quad (2-117)$$

Friction is what causes the tangential force, F_t (Coulomb's law):

$$F_t = \mu_f |F_n| \quad (2-118)$$

2.3.5.4 Smooth Particle Hydrodynamics (SPH)

An example of a hybrid model that combines a continuum method with a discrete one is the Smooth Particle Hydrodynamics model, see Chen et al. (2015). The Navier-Stokes momentum equations (2-120) and the continuity equation (2-119), the equation of state (2-121) are considered to predict the pressure field from the interaction forces between the system of particles, and the governing equations solved for each particle as follows:

$$\frac{d\rho_i}{dt} = -\rho_i \sum_{j=1}^N V_j (v_i - v_j) \cdot \nabla_i w(r_i - r_j, h) \quad (2-119)$$

$$\frac{dv_i}{dt} = -\frac{1}{\rho_i} \sum_{j=1}^N V_j (P_j + \rho_j \Pi_{ij}) \cdot \nabla_i w(r_i - r_j, h) + \sum_{j=1}^N \frac{2V_j v_j r_{ij} \cdot \nabla_i w(r_i - r_j, h)}{(|r_{ij}|^2 + n^2)} v_{ij} + g_i \quad (2-120)$$

$$P_i = C_s^2 (\rho_i - \rho_0) + P_0 \quad (2-121)$$

The pressure is found by using an artificial viscosity function, Eq. (2-120). The density is interpolated with a kernel function, Eq. (2-121).

$$\Pi_{ij} = \frac{-\alpha C_s h v_{ij} \cdot r_{ij}}{|r_{ij}|^2 + n^2} \quad (v_{ij} r_{ij} < 0)$$

$$\Pi_{ij} = 0 \quad (v_{ij} r_{ij} > 0) \quad (2-122)$$

$$w(r, h) = \frac{1}{3\pi h^2} (q^2 - 6q + 6), \quad 0 \leq q < 1 \quad (2-123)$$

$$q = \frac{|r - r_b|}{h} \quad (2-124)$$

The following are the steps in this model to calculate density from the particle

distribution, use Eq. (2-119) and the kernel function, Eq. (2-123). To calculate particle pressures, use equation (2-121) and the density equation of state. To solve for particle velocities, use equation (2.120) and the new pressures, use Eq. (2-113) to determine new locations of each particle, and repeat these steps.

The (SPH) model is capable of handling significant gas/liquid interface deformation in intricate free surface flows, like splashing inlet streams. There is no requirement for a computational mesh because the fluid is a collection of particles tracked in a Lagrangian framework. The interface does not require modelling because it is a result of particle distribution. State functions that incorporate the contributions of all the particles in the local area are used to capture pressure and density. Additionally, compared to other particle-based methods, the SPH approach uses a relatively small number of particles. However, this model becomes costly as the number of particles increases, and details of free surface such as bubble size cannot be resolved adequately. Natsui et al. (2017) investigated transient behavior of interfaces with in molten slag by using SPH approach. Prakash et al. (2007) applied SPH models to free surface flows in die casting and wheel casting of aluminum ingots Prakash et al. (2007).

2.3.5.5 Lattice Boltzmann Method (LBM)

This particle-based approach is a linearized version of the solution of the Boltzmann equation that describes the flow of liquid and gas through two sets of particles on a predefined lattice described by Shan et al. (1993). It means that LBM solves for the particle distribution function f with seven dimensions as

($\mathbf{X}$ (space), $\mathbf{C}$ (velocity), $\mathbf{t}$ (time)).

$$f_b(\mathbf{x} + \mathbf{v}_b \Delta t, \mathbf{t} + \Delta t) - f_b(\mathbf{x}, \mathbf{t}) = -\frac{f_b - f_{b,eq}}{\tau} + \mathbf{F}_b \quad (2-125)$$

When dealing with multiphase flow problems, Swift et al. (1995) and Shan et al. (1993) found that a unique body force modelling was required to accurately recover interface curvature and surface tension even at high density ratios.

$$\mathbf{F}_b(\mathbf{x}) = -Gc_s^2 \varphi(\mathbf{x}) \sum_b \omega(|\mathbf{v}_b|^2) \varphi(\mathbf{x} + \mathbf{v}_b) \mathbf{v}_b \quad (2-126)$$

LBM can parallelize and mimic mesoscale events, such as flow in a boundary layer. However, in multiphase flow systems with high density ratios, this approach can show numerical instability and anisotropy because of the lattice it uses. The same issue with relation to bubble interactions is also brought up by LBM, rejoining VOF and LS techniques Deen et al. (2005).

LBM models were used by researchers like Zhang et al. (2006) to represent transient turbulent asymmetric single-phase behaviour in continuous casting moulds. To replicate oscillations and binary collisions of liquid droplets as well as rising bubbles in a square duct that were consistent with theoretical expectations, Inamuro et al. (2004) also used the LBM technique.

2.3.5.6 Dissipative Particle Dynamics (DPD)

According to Groot et al. (1997), the DPD technique is a coarse-grained variant of Molecular Dynamics (MD) that can model mesoscale events. Each particle in this method is composed of a sizable group of molecules, in contrast to MD, where each particle is still made up of a single atom or molecule. The DPD technique solves the particle trajectory equation (Eq. 2-113) and the Newton's equation interaction forces between neighbouring groups of particles (Eq. 2-127), just like other particle-based models like DEM, SPH, and MD.

$$\frac{dv_i}{dt} = \sum_{i \neq j} F_{ij}^D + F_{ij}^C + F_{ij}^R \quad (2-127)$$

In this equation, 1) F^D denotes the dissipative force, which works to lower relative velocities between particles and create a viscous effect, 2) F^C denotes the conservative force from an inter-particle potential, like Lennard Jones in MD, and 3) F^R denotes the random force, which adds additional degrees of freedom to account for particle internal behaviour. (2006) Tiwari et al.

In flows with multiple phases and significant deformations, the DPD technique can capture meso-scale/molecular-based phenomena without the intrinsic anisotropic motion issues that plague LBM. This method, however, costs a lot to compute because it needs a lot of particles to capture.

2.5.6 Hybrid Models

As the above previously presented methods have advantages and disadvantages, combination of models known as hybrid models should be good to improve the existing ones. Researchers can create new models.

To estimate the curvature more accurately, hybrid models like the Coupled Level Set and Volume of Fluid model (CLSVOF) Sussman et al. (2000) have been developed. EEDPM is a different hybrid model that combines EE and DPM.

2.3.6.1 Coupled Level Set and Volume of Fluid Model (CLSVOF)

The goal of this method is to accurately estimate curvature by combining the greater mass conservation capabilities of VOF with the better interface shape tracking capabilities of Level Set approaches. This will make it much easier to assess the interfacial tension for applications like behaviour modelling of specific bubbles. Sussmann's team used CLSVOF techniques to implement their work on free surfaces Sussman et al. The handling of complex free surface shapes with substantial deformations, such as bubble coalescence and breakdown, presents several challenges for this method.

2.3.6.2 NP2 Phase Model

The NP2 phase method combines the VOF and inhomogeneous MUSIG models, according to Tomiyama et al. (2003). In this method, the vast gas regions and free surfaces are represented by the VOF equations, while the inhomogeneous MUSIG equations can simulate the sub-grid-scale bubbles. To exchange gas between the VOF and MUSIG phases, source terms for mass and momentum exchange are required. This offers a significant benefit.

Converging all the mass and momentum transfers while solving a large number of sets of continuity and momentum equations can be extremely computationally expensive.

To reduce the high computing cost, researchers like Tomiyama et al. (2003) implemented their work using this strategy by supposing that all discrete bubble phases are inviscid.

2.3.6.3 Multifluid VOF Model

In this method, Weller et al. (2013) attempt to depict a hybrid of the EE model and the VOF model. This model is a duplicate of the EE model with the exception

that extra terms, such as the divergence of an artificial compression velocity, have been added to the continuity equations of both phases., $\nabla \cdot (u_{ck} \alpha_k (1 - \alpha_k))$.

This artificial velocity for phase k , $u_{ck} = C_\alpha |u_k| \frac{\nabla \alpha_k}{|\nabla \alpha_k|}$ is only activated ($C_\alpha = 1$) when the cell sharpens the spread in the volume fraction field in the normal direction to the interface because it is situated on the interface between phases. This interface tracking happens when the mesh, which is simulated using the IAC model or population balance theory, can resolve the bubble size. The artificial compression is disabled for very small bubbles ($C_\alpha = 0$), hence the EE model is used to simulate the phase k velocity.

The fact that this method is faster in terms of performance does not mean that it loses accuracy for the sub-grid bubbles. This method still needs a small courant number to track the interface and artificial drag to allow the fluid velocities to equalise at the interface.

2.3.6.4 DMP + VOF Model

To handle some multiphase flow issues, this method combines the VOF model with the DPM model. This method has been utilised by researchers, such as Pirker et al. (2015), to imitate the steel flow with argon bubbles in the nozzle of a continuous caster, allowing bubbles to build up and create VOF gas regions, resulting in enormous gas pockets and small DPM bubbles flowing together.

One of the drawbacks of this strategy is that it enables single cells to change into VOF gas regions which could be improperly resolved according to Kumar et al. (2016). The failure to consider the fragmentation of VOF gas regions into DMP bubbles is another difficult constraint common to VOF models.

2.6 Numerical Methods

Computational Fluid Dynamic (CFD) is an alternative to measurements for solving large-scale fluid dynamical systems that has become a key element for computer aided designs in industries across the world. CFD has evolved as a design tool for various industries namely aerospace, mechanical, automobile, chemical, metallurgical, electronic, and even food processing industries by using numerical techniques to solve these problems. CFD approach has huge

advantages with the development of high-speed computers manufactured and cheaper in terms of cost.

Numerical techniques or methods for computing turbulent flows are derived from methods set up for laminar, viscous flows. Several features must be considered for the classification of numerical methods for computing turbulent flows. They include firstly, the discretization method used to approximate the differential or the integral conservation equations by a system of algebraic equations that can be solved on a computer; secondly, the time integration method; and thirdly, the method of pressure-velocity-density coupling.

The most used discretization methods are known as finite-difference (FD), spectral (S), finite-element (FE), and finite-volume (FV) methods. However, some mixed type methods such as spectral-element methods (Henderson and Karmiadakis, 1995) or control-volume finite-element methods (Baliga, 1997) can also be used.

The time-integration methods are either explicit or implicit, and the choice of each method remains essential for both the accuracy and computational efficiency reasons.

The method of pressure-velocity-density coupling remains very important for both compressible and incompressible flows.

2.6.1 Discretisation Methods

In this work, we will only go through an overview of the different discretization methods.

2.6.1.1 Finite Difference Method

This approach is defined as the easiest method to deal with simple geometries. It is regarded as the oldest method for numerical solution of PDEs introduced by Euler in the 18th century.

In this approach, we start with the conservation equation in differential form. The solution domain is covered by a mesh or grid and at each grid point, the

differential equation is approximated by replacing the partial derivatives by approximations in terms of the nodal values of the functions. This will result in one algebraic equation per grid node, where the variable at the node and a certain number of neighbour nodes appears as unknowns.

Although the Finite Difference approach can be applied to any grid type, most of authors have used the structured grids for their applications with grid lines serving as local coordinate lines. The use of structured grids makes this approach very simple and effective, and particularly easy to obtain higher-order schemes.

Approximations to first and second derivatives of the variables with respect to the coordinates are obtained by Taylor series expansion or polynomial fitting.

This approach is not suitable to solve problems with increasing degree of physical complexity such as flows with higher Reynolds numbers and flows around arbitrary shaped bodies.

2.6.1.2. Finite Volume method

The Finite Volume approach uses the integral form of the conservation equations for starting point. Then, the solution domain is subdivided into a finite number of contiguous control volume (CVs), and the conservation equations are applied to each CV. The variable values are calculated at a computational node lying at the centroid of each CV. Interpolation is used to name variable values at the CV surface in terms of the nodal (CV-center) value.

Suitable quadrature formulae are used to approximate surface and volume integrals. As a result, we get an algebraic equation for each CV, containing several neighbour nodal values.

The Finite Volume approach is suitable for complex geometries, so it can be used for any grid type. We must mention for precision that the grid is concerned by the control volume boundaries and not the coordinate system.

This approach is the simplest to program and understand. It remains popular for engineering purpose because all the terms that need to be approximated have physical meaning.

The drawback with Finite Volume methods compared to the FD schemes is that methods of order higher than second are more difficult to develop in 3D, because the FV approach needs three levels of approximation: interpolation, differentiation, and integration.

2.6.1.3 Finite Element Method

The Finite Element approach has many similarities with the FV methods. The domain is divided into a set of discrete volumes or finite elements that are unstructured in general. These elements are either triangles or quadrilaterals in 2D, and tetrahedra or hexahedra in 3D.

The distinguishing feature of the FE methods is that the equations are multiplied by a weight function by being integrated over the entire domain. In the simplest FE methods, the solution is approximated by a linear shape function within each element in a manner guaranteeing continuity of the solution across element boundaries. Such a function can be constructed from its values at the corners of the elements.

This approximation is then substituted into the weighted integral of the conservation law and the equations to be solved by requiring the derivative of the integral with respect to each nodal value to be zero; this means selecting the best solution within the set of allowed functions (the one with the best residual). The result is a set of non-linear algebraic equations.

One of the advantages of this approach is that the grids can be easily refined or subdivided. FE methods are relatively easy for mathematical analysis and have optimal properties for certain types of equations. The main drawback of FE methods, which is common to any method using unstructured grids, is that the matrices of the linearized equations are not well structured compared to those of regular grids. This can make it more challenging to find efficient solution methods.

Researchers, including Oden (1972), Chung (1978), Baker (1983), and Fletcher (1991) have done a lot of work on FE methods.

2.6.1.4 Spectral Methods

Spectral methods approach is used for specialized applications, and particularly for Large Eddies (LE) and Direct Numerical Simulations (DNS) flows in simple geometries. In this approach, Fourier series or generalizations of them are used to evaluate spatial derivatives. The grids remained uniformly spaced, and the method is especially accommodated for problems allowing periodic boundary conditions to be applied at the opposite boundaries.

In this approach, the discretization error decreases exponentially with the number of grid points N , making them very accurate if N is sufficiently large Canuto et al. (1987).

These methods have been used so far to solve the Navier-Stokes equations, and mathematically are more complicated compared to the other methods presented so far.

2.7 Hydrodynamic Slug Flow Modeling

The phrase "transient pressure in a pipeline" refers to the wave phenomenon that happens when the velocity of fluid suddenly changes. The word "surge" pressure is often used by authors to refer to a brief pressure that has no negative effects, whereas the term "water-hammer" may be used to refer to a brief pressure that, if not appropriately addressed and reduced, could result in significant repercussions like rupture.

The majority of transient stresses are predictable and recurrent. Examples include turning on and off a pump and opening and closing an in-line valve. These transients are easily tolerated in the pipeline architecture because they are predictable.

The quick loss of power in a system of pumped pipelines, which results in an abrupt halt to flow in the pipeline and a significant transient pressure, is frequently the most severe transient. In this case, the pipeline experiences cavitations, and

real water-hammer events—which are extremely brief but extremely damaging—might be produced.

Some transient pressures are the result of an abnormal or unforeseen circumstance, and the pipeline design does not always readily account for them. It might also be more challenging to find and measure these transients. Examples from case studies include installing valve actuators incorrectly during construction, installing combination air valves incorrectly, and installing unstable pressure relief valves.

Some transients are cyclic in nature and can be brought on by reciprocating pumps, unstable pressure reduction valves, and inappropriate pressure relief valves. These transients should be minimized or reduced to pressure changes that won't cause fatigue damage.

Nowadays petroleum sector depends heavily on transients of all kinds, particularly in the creation of multiphase flows in oil and gas pipelines. Offshore pipelines are made up of infield and export pipelines. From a platform or FPSO (Floating, Production, Storage, and Offloading) to the beach for further processing, oil and gas are transported via export pipes. Typically, oil or oil with a small amount of water flows through these types of pipelines. Infield pipelines deliver well stream to the platform or FPSO from manifolds or wells, respectively. For oil fields, this sort of pipelines typically have multiphase flows of gas, oil, and water, or gas, condensate, and water for gas fields.

Under certain flow conditions, the simultaneous flow of gas and oil in a pipeline can result in significant transient issues. Slug formation is one of these issues.

When a small break loss of coolant accident (SB-LOCA) occurs in a pressurised water reactor, for example, hydrodynamic slug flow, a multiphase flow regime, can occur. It can also occur in other settings, such as chemical plants or oil transportation pipelines, as was previously mentioned. The transition of fast liquid slugs, which carry a sizeable amount of liquid with high kinetic energy, and an acceleration of the gaseous phase often characterise the slug flow regime, a very complex and unstable regime. At any cross-section normal to the pipeline

axis, mass flowrates, phase velocities, and pressure show significant time-dependent variations.

As was already indicated, slug flow poses a risk to the pipeline structural system integrity because of the intense oscillating pressure levels that are created behind the liquid slugs and their mechanical velocity. Due to the erratic and intermittent nature of the slug flow, it is also important to demonstrate understanding of the general fluid dynamic mechanism causing this type of flow and to identify the critical flow parameters (such as slug length, frequency, and propagation velocity; pressure drop) affecting it.

2.7.1 Slug Flow Mechanisms

The following stages can be involved in the production of stable slugs, according to authors like Taitel and Duckler (1977), Manolis (1995), Woods & Hanratty (1999), Hale (2000), and Ujang et al. (2006), who all made this conclusion based on various experimental observations as follows: 1. Interface wave formation, 2. Waves growth and pipe bridging event, 3. Slug growth, 4. Gas entrainment, 5. Stratified layer formation (see Figure 2-7).

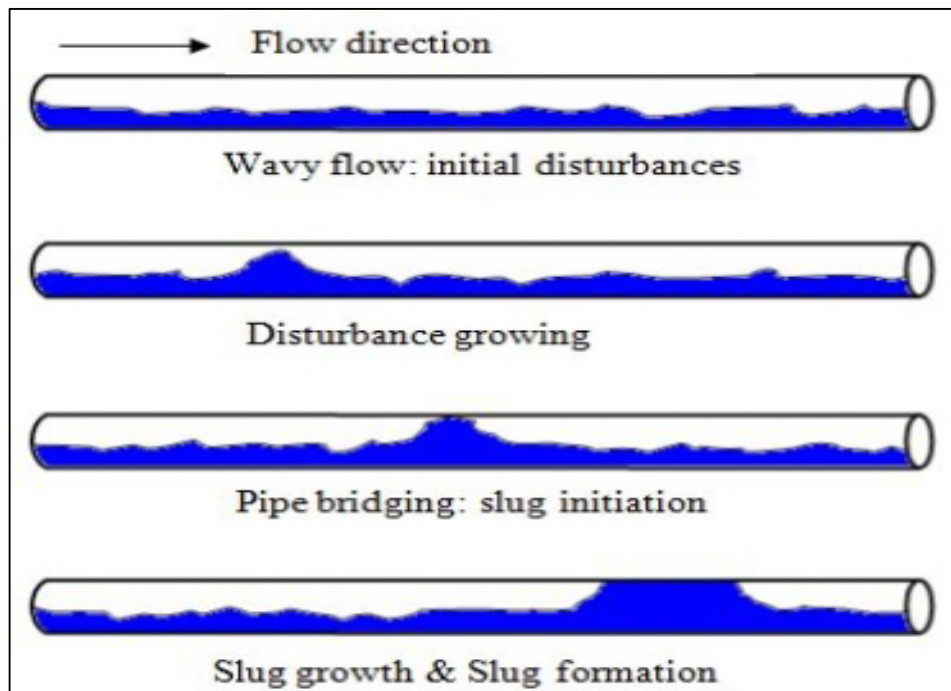


Figure 2-7 Transition of slug flow from stratified flow

2.7.1.1 Kelvin-Helmholtz Instability Criterion

Waves form at the point where gas and liquid are flowing simultaneously in a pipeline. The gas velocity varies throughout the wave, reaching a maximum at the peak and a minimum at the trough. The pressure in the gas, which fluctuates along the wave profile with a maximum at the trough and a minimum at the crest, is another factor that contributes to the destabilising effect of the wave by creating an aerodynamic lift. The waves will rapidly increase in amplitude as the stabilising influences of surface tension and gravity are overpowered by the destabilising effects of pressure changes in the gas phase and liquid inertia. Kelvin-Helmholtz (K-H) instability is the name given to this occurrence.

Consider the flow of a gas over a liquid in a horizontal channel where the densities of the two phases densities ρ_L and ρ_G , and the respective flow rates are velocity u_L and u_G . Milne-Thomson (1960) and Ishii (1982) discussed the criteria based on the Kelvin-Helmholtz (K-H) instability for the prediction of wave production depending on the stability of the two-phase fluid interface as follows. Following are the definitions of the surface waves C propagation velocity:

$$C = \frac{\rho_L u_L + \rho_G u_G}{\rho_L + \rho_G} \pm \left[C_\infty^2 - \rho_L \rho_G \left(\frac{u_L - u_G}{\rho_L + \rho_G} \right)^2 \right]^{1/2} \quad (2-128)$$

Where:

$$C_\infty^2 = \frac{g}{k} \left(\frac{\rho_L - \rho_G}{\rho_L + \rho_G} \right) + \frac{\sigma k}{\rho_L + \rho_G} \quad (2-129)$$

where k is the wave number and σ the surface tension. The instability criterion is then defined as

$$\frac{g}{k} \left(\frac{\rho_L - \rho_G}{\rho_L + \rho_G} \right) + \frac{\sigma k}{\rho_L + \rho_G} < \rho_L \rho_G \left(\frac{u_L - u_G}{\rho_L + \rho_G} \right)^2 \quad (2-130)$$

The three forces that control wave stability at interfaces are gravity, surface tension, and relative motion. While the relative motion between the two phases always serves to destabilise the system, the surface tension force always acts to stabilise it. The influence of pressure on the relative motion of the waves is accounted for by Bernoulli's principal. If $\rho_L > \rho_G$, then the gravity factor tends to stabilise the system. Researchers, e.g., McCready, (1989) found that for fluids with very large difference of viscosity between them, this analysis tends to fail.

2.7.1.2 Studies of Slug Flow Initialisation and Mechanisms

Generally, the slug flow is initiated either by the classical Kelvin-Helmholtz (K-H) long wave instability, or from other wave propagation theories. These initiation mechanisms can be classified into four groups or categories as suggested by researchers including Lin & Hanratty (1986), Wallis & Dobson (1973) and Kordyban & Ranov (1970).

- The analysis of K-H instability as the underlying principle for the wave formation using waves motion theories.
- The use of Bernoulli effect with the K-H instability to analyze the slug initiation mechanisms by ignoring the wave motion aspect.
- The analysis of the transition to slug flow by using the kinetic energy transfer mechanisms.
- The viscous K-H instability viewing slug flow initiation through linear stability analysis where inertia and friction effects are considered.

Category I

In this area, wave motion theories were used to analyse the K-H instability as the fundamental idea guiding wave generation. The changeover to slug flow was modelled as a K-H instability of stratified flow, where the channel is filled with slugs as waves and ripples gradually form and enlarge. The pioneers of this methodology included researchers like Kordyban & Ranov (1970), and the simplified instability criterion is provided by:

$$(\bar{u}_G - \bar{u}_L) > K_{S-I} \left(\frac{g \rho_L \bar{h}_G}{\rho_G} \right)^{1/2} \quad (2-131)$$

K_{S-I} is defined as the stratified to intermittent transition coefficient:

$$K_{S-I} \equiv K_{K-R} = \left[\frac{1}{k \bar{h}_G [\coth(k \bar{h}_G - 0.9) + 0.45 \coth^2(k \bar{h}_G - 0.9)]} \right]^{1/2} \quad (2-132)$$

where $\bar{u}_G$ and $\bar{u}_L$ are respectively gas and liquid velocities in the stratified layer, $\bar{h}_G$ represents the mean height of the gas phase, and with the assumptions that the surface tension effects are negligible, $\rho_L \geq \rho_G$, allowing the liquid contribution and using the experimental relationship $\eta k = 0.9$ where k is the wave number and η is the wave amplitude. The wave amplitude is determined by the proximity of the horizontal channel top wall and is dependent on the channel diameter. The amplitude of the waves increases as the gas flows over the liquid, which causes the waves to grow.

The findings of Kordyban and Ranov's work revealed a distinctive link between gas velocity and liquid level, and it was determined that their theory and the Baker flow regime map agreed in a reasonable manner for a purely horizontal layout. Even Kordyban (1977) suggested further that the wave instability occurrence was at approximatively:

$$\frac{1.35 V_c^2}{g h_c} \frac{\rho_G}{\rho_L - \rho_G} = 1 \quad (2-133)$$

where V_c is the velocity of the wave crest, and h_c represents the distance from the wave crest to the top of the channel. This experimentation was done to support the work of Wallis & Dobson (1973) and Taitel & Duckler (1976).

Category II

By neglecting the wave motion, this method connects the Bernoulli effect and the K-H instability to examine the slug initiation mechanisms.

Wallis & Dobson (1973) developed their experiments for the predictions of slugging in horizontal rectangular ducts with 25.4- and 305-mm heights of varying lengths and uniform water depths. Air and water were flowing concurrently at atmospheric pressure and suggested that $\dot{j} = 0.5\varepsilon_G^{3/2}$ could describe the transition to slug flow, where $\dot{j}$ represents the dimensionless gas velocity. They formulated that their results could be expressed as:

$$U_{SG} \left[\frac{\rho_G}{gH(\rho_L - \rho_G)} \right]^{1/2} = 0.5\varepsilon_G^{3/2} \quad (2-134)$$

Where ε_G is the void fraction and H represents the height of the channel.

These scientists hypothesised that slugging occurs when a big wave forms quickly, rolls over the liquid below, and eventually fills the channel to form a slug. By using Benjamin's (1968) bubble theory on transition to slug flow in reverse, their model was created as follows:

$$U_{SG} = 0.5 \left[\frac{gh_G(\rho_L - \rho_G)}{\rho_G} \right]^{1/2} \quad (2-135)$$

which is an illustration of the gas speed required to keep a stationary liquid wave on top of a liquid layer.

Some authors, including Taitel and Duckler (1976), disagreed with the modelling approach described above because they believed there was no theoretical support for using it to describe the transition to slug flow. According to Taitler & Duckler (1976), slug creation should be thought of as a "entrance phenomenon"

that takes place right at the channel entrance. For their research, they used visual records of slug cycle observations.

Category III

This method used kinetic energy transfer to view and understand the mechanisms that lead to slug flow. Gardner (1979) disagreed with Taitler and Duckler (1976) approach and thought that the initiation of slug flow should occur through energy transfer from the gas to the liquid phase and slugging would occur at the point of maximum energy transfer. He proposed that the simultaneous solution of the following two equations will yield the wave height at which the commencement of slugging can occur, thus:

$$(h_2 - h_1) = \frac{h_2(1-h_2)^2 \frac{h_2^3}{P^2}}{(3h_2-1)(1-h_2) + 3\left(\frac{h_2}{P}\right)^2} \quad (2-136)$$

$$\text{and } (1-h_2) = (1-h_2) + (h_2-h_1) \quad (2-137)$$

where the subscripts 1 and 2 represent station 1 and 2 in the figure below.

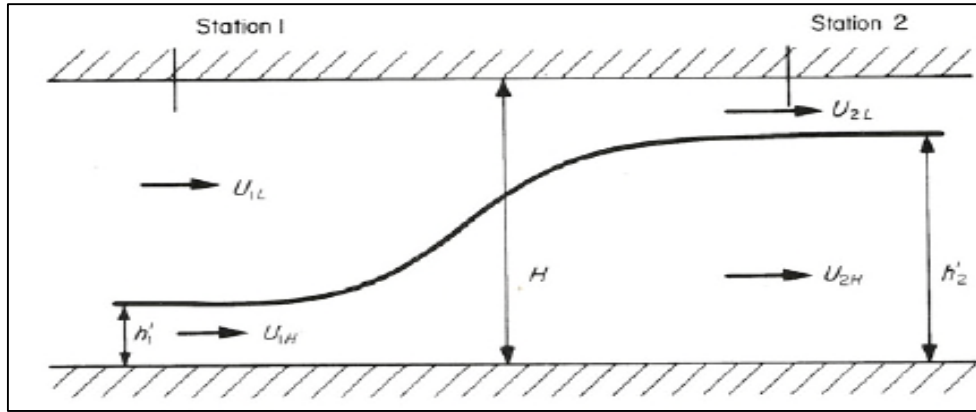


Figure 2-8 Definition sketch of system with wave stationary (after Gardner (1979))

By assuming that this liquid kinetic energy provided a lifting force to that of the gas phase accelerating over the wave crest, researchers like Minato et al. (1986) used the contribution of the liquid kinetic energy to the growth of waves to set up a model that predicts the transition to slug flow.

Category IV

This method, also known as viscous K-H instability, looked at the beginning of slug flow through an examination of linear stability that took friction and inertia into account.

Phase inertia and shear stresses were incorporated into the theory by researchers like Lin and Hanratty (1986), who also calculated continuity and momentum equations for stratified flows and then added a small amplitude long wavelength disturbance to obtain the equation for neutral stability as follows:

$$\frac{U_{SL}^2}{gH \sin \theta} \frac{1}{(1-\epsilon_G)^3} \left(\frac{C_R}{\bar{u}_a} - 1 \right) + \frac{U_{SG}^2}{gH \sin \theta} \frac{\rho_G}{\rho_L} \frac{1}{\epsilon_G^3} - \frac{(\rho_L - \rho_G)}{\rho_L} = 0 \quad (2-138)$$

Where C_R is the real component of the wave velocity, $\bar{u}_a$ denotes the average of the velocities of the two phases. In the equation above, the first term represents the liquid inertia destabilising effect whereas the second and third terms describe the destabilising impacts of gas phase pressure changes and the stabilising effects of gravity, respectively. The following need for neutral stability follows.

$$\left(\frac{U_{SG}}{U_{SL}} \right)^{1.75} \sqrt{\frac{\rho_G}{\rho_L - \rho_G}} = K \epsilon_G^{3/2} \quad (2-139)$$

$$\frac{1}{K^2} = 1 + \left(\frac{\epsilon_G}{1-\epsilon_G} \right)^{0.714} \left(\frac{C_R}{\bar{u}_a} - 1 \right)^2 \left[1 + (1+\emptyset) \left(\frac{1-\epsilon_G}{\epsilon_G} \right) \right]^{1.14} \left(\frac{\rho_G}{\rho_L} \right)^{0.143} \left(\frac{f_i}{f_s} \right)^{1.17} \left(\frac{\vartheta_G}{\vartheta_L} \right)^{0.286} \quad (2-140)$$

where $\emptyset$ is of the wall interfacial shear stresses, f_i is the interfacial friction factor and f_s is the interfacial friction factor assuming a hydraulically smooth interface. They found some good agreement with the revised K-H instability criterion devised by Taitel & Duckler (1976) regarding the inertia air-water flow, however they found that the comparison between the inviscid K-H instability theory and their viscous K-H instability was not appropriate, as the inviscid K-H instability over predicted the critical superficial liquid velocity relative to their own prediction.

2.7.2 Slug Flow Modelling Approaches

Slug flow irregularity is a key characteristic. In fact, using a typical time averaging approach to describe this kind of flow would often be constrictive. An analysis

that considers the intrinsic erraticness and makes a distinction among the liquid slug region, which may contain dispersed gas bubbles and the massive gas bubble region that follows is instead necessary.

The two basic approaches that effectively represent the steady states and transient models known in the literature and which reflect this incredibly complicated flow are briefly described in the paragraphs that follow.

2.7.2.1 Steady state Models

Traditional slug flow models typically use the "unit-cell" technique, which involves analysing a control volume that includes both the long gas bubble and the liquid slug. Wallis from 1969. After choosing a moving frame of reference moving at the speed of the cell, a steady state is obtained, allowing for a steady state analysis based on mass and momentum balances over the control volume. Numerous other models built on the "equivalent-unit-cell" concept, which is more widely used, and related methodologies were created; examples include Dukler & Hubbard (1975) and Taitel and Barnes (1990).

The statistical averaging of the physical characteristics of the separated and scattered flow zones can also be used to describe steady state slug flow. Based on conditional averaging of the conservation equations proposed by Ferschneider (1983), Fabre et al. (1983), and Line (1983), a "statistical cellular model" predicts the dependent variables and the intermittency features. Fabre et al. (1992) improved this method and applied it to transient flows. Most steady state models are suitable for use in a unified scheme that consists of individual models for various predominant flow regimes and a generic model for flow pattern prediction. For a review, see Gomez et al. (1999).

However, steady state models are not necessarily able to predict phenomena occurring in inclined pipelines, such as the dissipation of slugs in downhill flow, as demonstrated by [Taitel et al. 2000], nor are they suitable for the prediction of the transition from one flow pattern to another. Therefore, a study of transient models is required.

2.7.2.2 Transient models

Slug tracking, slug catching, and empirical slug specification are the three main categories of transient models for slug flow.

2.7.2.2.1 Empirical slug specification models

The development, growth, decay, and shape (tail and front) of the slugs are all described using the empirical slug specification models. They can be divided into "slug growth" models and "stratified-slug transition" models.

For inviscid flow, stratified-slug transition models apply the traditional Kelvin-Helmholtz stability theory. Watson (1989), Espedal & Bendiksen (1989), Mishima & Ishii (1986), Lin & Hanratty (1986), Taitel & Dukler (1976), Kordyban & Ranov (1970), and others. De Henau and Raithby (1995b) created a one-dimensional two-fluid model with a sub-model that offers the constitutive relations and parameters required for the closure relations incorporated into the two-fluid model.

Slug growth Models begin with the supposition that a slug flow was caused by some mechanism, and correlations are used to show the slug front and tail. In 1989, Ruder et al., Jepson, Lunde & Asheim, and Bendiksen & Espedal published their research. The transient two-fluid model by Bendiksen et al. (1996) included a cell-unit kind of slug flow description. The bubble body, nose, and tail as well as the hydraulic leap at the back of the bubble should be calculated based on mass and momentum conservation, according to Fagundes Netto et al. (1999).

2.7.2.2.2 Slug-tracking models

Slug-tracking methods offer an alternate method for simulating slug flow. They differ from the "unit-cell" models mentioned earlier in that they model the propagation of several discrete slugs along the pipeline while taking into account each slug individually. Tracking individual slugs has an impact on how they migrate, develop, and vanish. The slug length distribution resulting from a specific pipeline layout can be calculated using this method instead of the unit-cell method, which has a considerable advantage over it. Empirical correlations are used in some degree by slug-tracking models (OLGA) Bendiksen et al. (1987,

1991, 2000) to model certain slug characteristics. A slug tracking approach that forecasts the growth, production, and dispersal of each slug separately was proposed by Zheng et al. in 1994. Using an object-oriented methodology, Nydal and Barnerje (1996) introduced a Lagrangian slug tracking method for dynamic gas-liquid slug flow. Slugs and bubbles are considered as discrete computational objects and are arranged in linked lists. Another model, developed by Taitel and Barnea in 1998, replicates the fundamental processes of slug formation, development, and dissipation with a focus on the impacts of gas compressibility.

2.7.2.2.3 Slug capturing using the two-fluid models.

This approach predicts the slug regime as a mechanism and the result of the development of hydrodynamic instabilities. In 1998, Issa and Woodburn solved the same governing equations which was a transient, one-dimensional multiphase two-fluid model. Closure models were used to describe the stratified, slug and transitional fluid flow regimes, eliminating the need for empirical flow models. The numerical prediction of the two-fluid model equations results in a rise in the liquid volume percentage in the computer simulations, which can finally exceed unity and cause the start of slug. The solution of the mass and momentum transport equations for each phase determines how slugs grow, merge, and collapse. For closure relations for the liquid wall, gas wall, and interfacial shear forces, Vernier & Delhay (1968), Boure & Reocreux (1972), Ishii (1975), and Ishii & Mishima (1984) are the sole sources of empirical data needed. This is a serious deficiency.

Commercial pipeline transient codes like (TACITE) and (OLGA) Bendiksen et al. (1987, 1991), as well as (PLAC) Black et al. (1990), are all well-established on the two-fluid models. This method of describing slug flow seems to be a natural one because it includes mechanisms like the generation of interfacial waves, which are the precursor to hydrodynamic slugs. There haven't been many studies on this strategy up to this point Bonizzi et al. (2001).

Issa & Woodburn (1998) and Bonizzi et al. (2001) employed a fine computational mesh to obtain their results, which are several orders of magnitude below those often used in commercial simulators. This may be the cause of the absence of

two-fluid slug flow experiments. In the context of an industrial scale pipeline, the high number of nodes required looks impractical. But as computing power keeps rising, this issue might eventually be resolved.

2.7.3 Slug Flow Parameters

In developing slug flow models, and notably in deriving closure connections like virtual mass, slug flow characteristics like Translational velocity, slug mean length, slug frequency, and slug Body hold-up play a crucial role. We will provide a thorough overview of these factors in the sections that follow.

2.7.3.1 Translational Velocity

The gas/liquid slug can be characterized by large, elongated bubbles which are separated by liquid slugs and the fluid flow pattern can sometimes contain very small dispersed bubbles. A complete model for the numerical study of slug requires knowledge on long bubbles and their **translational velocity** (V_T); which is a key parameter investigated theoretically and experimentally. Therefore, considerable amount of slug motion data was obtained for various flow conditions, fluid properties and pipe diameters and inclinations.

Generally, translational velocity is view as the superposition of the bubble propagation velocity in a stagnant liquid, i.e., **the drift velocity** (V_D), and a contribution due to **mixture gas and liquid velocity** (V_M). The equation commonly used was proposed by [Nicklin et al., 1962] and is as follows:

$$V_T = C_0 V_M + V_D \quad (2-141)$$

The values of the coefficient C_0 that is the weighted flow distribution parameter and drift velocity V_D depend on the pipe or tube inclination. However, as we are interested in horizontal pipelines, our investigations will be focused on them.

2.7.3.2 Drift Velocity Coefficient

Davies & Taylor, (1950) obtained an expression for the drift velocity by performing some analysis around the nose of the elongated bubble as

$$V_D = C_1 \sqrt{gD} \quad (2-142)$$

where C_1 is the **drift velocity coefficient**, g the gravitational acceleration, and D the pipe diameter. It follows that the drift velocity depends on gravity. The viscous or surface tension can cause secondary effects and these effects were further investigated by different authors including the effects of the material properties of the fluids. Zukoski, (1996) investigated the pipe inclination to drift coefficient using the following general expression:

$$V_D = C_1 (N_f, E_0, \beta) \sqrt{\hat{g}D} \quad (2-143)$$

where $\hat{g} = g\Delta\rho/\rho_L$, $N_f = D^{3/2} \sqrt{\rho_L \Delta\rho g} / \mu_L$ is the non-dimensional inverse viscosity, β is the pipe inclination, and E_0 is the Eötvös number, which is the ratio of the gravitational to surface tension forces as follows:

$$E_0 = \frac{\Delta\rho g D^2}{\sigma} \quad (2-144)$$

Weber, (1981) showed that an increase of the surface tension implies a decrease of the drift velocity for small diameter pipes by using the following relationship.

$$C_1 = 0.54 - \frac{1.76}{E_0^{0.56}} \quad (2-145)$$

The hydrostatic pressure difference between the top and bottom of the bubble nose investigated by [Nicholson et al., 1978], [Ferre, 1979], [Webber, 1981], [Bendiksen, 1984], [Theron, 1989], [Manolis, 1995], and [Woods & Hanratty, 1996], which causes the drift velocity for horizontal pipe to exist. We examine several authors' works in the section that follows.

Ferre, (1979) demonstrated that the drift velocity dramatically changed values for two critical velocities, or Froude numbers, and proposed the correlation is

$$C_1 = \begin{cases} 0.44 & \text{Fr}_M \leq 2.26 \\ 0 & 2.26 \leq \text{Fr}_M \leq 8.28 \\ 3 & \text{Fr}_M \geq 8.28 \end{cases} \quad (2-146)$$

where the mixture Froude number Fr_M is given by

$$Fr_M = \frac{V_M}{\sqrt{gD}} = \frac{J_G + J_L}{\sqrt{gD}} \quad (2-147)$$

where J_G is the gas and J_L is the liquid superficial velocities. Theron, (1989) suggested a general relationship for the drift coefficient that considers all pipe inclination as

$$C_1 = \left(-0.5 + \frac{0.8}{\Gamma}\right) \cos \beta + 0.35 \sin \beta \quad (2-148)$$

where the parameter

$$\Gamma = 1 + \left(\frac{Fr_M}{Fr_C} \cos \beta\right)^{10} \quad \text{and} \quad Fr_C = 3.5 \quad (2-149)$$

Manolis, (1995) studied the translational velocity and proposed the following expression for the drift coefficient.

$$C_1 = \begin{cases} 0.477 & \& Fr_M < 2.86 \\ 0 & \& Fr_M \geq 2.86 \end{cases} \quad (2-150)$$

Wood & Hanratty, (1996) investigated slug initiation at high gas velocity and suggested the following expression for the drift coefficient as

$$C_1 = \begin{cases} 0.52 & \& Fr_M < 3.1 \\ 0 & \& Fr_M \geq 3.1 \end{cases} \quad (2-151)$$

2.7.3.3 Mean Bubble Motion Coefficient

The parameter C_0 plays an important role to describe the translational velocity which is a distribution coefficient in the dispersed system. Many authors used experimental investigations to deal with it. From various expressions found in the literature, a general expression has emerged:

$$C_0 = C_0(Fr_M, Re_S, \beta) \quad (2-152)$$

where β, Fr_M, Re_S are respectively, the pipe inclination, the Froude number and the slug Reynolds number are.

$$Fr_M = V_M / \sqrt{gD} \quad (2-153)$$

$$Re_S = \frac{\rho_L V_M D}{\mu_L} \quad (2-154)$$

Duckler et al., (1985) used a boundary layer analysis and power law model to report the following analytical expression that is:

$$C_0 = \frac{(2n+1)(n+1)}{2n^2} \rightarrow 1.225 \text{ for } n = 7 \quad (2-155)$$

where $n = 7$ corresponds to the 7th power law velocity profile for turbulent flow. The above expression agreed with the experimental value suggested by Nicklin et al., (1962) which is $C_0 = 1.2$ for fully developed turbulent flow.

Duckler & Hubbard, (1975) used a logarithmic velocity profile approach to propose the following correlation:

$$C_0 = 1.022 + 0.021 \ln \text{Re}_M D \quad (2-156)$$

$$\text{with } 3.10^4 < \text{Re}_M < 4.10^5 \text{ and } \text{Re}_M = \frac{\rho_M V_M D}{\mu_M} \quad (2-157)$$

ρ_M, μ_M are the density and viscosity of the mixture. Ferre, (1979) suggested two distinct Froude number values when the drift velocity changes. Using the coefficient:

$$C_0 = \begin{cases} 1.10 & \& \text{Fr}_M \leq 2.26 \\ 1.30 & \& 2.26 < \text{Fr}_M < 8.28 \\ 1.02 & \& \text{Fr}_M > 8.28 \end{cases} \quad (2-158)$$

Theron, (1989) proposed a continuous correlation that considers the pipe inclinations et the effect of the Froude number on the distribution coefficient.

$$C_0 = 1.3 - \frac{0.23}{\Gamma} + 0.13 \sin^2 \beta \quad (2-159)$$

where Γ is defined in the way as previously in Eq. (2-76). Manolis, (1995) suggested the following expression for the value of

$$C_0 = \begin{cases} 1.033 & \& \text{Fr}_M < 2.86 \\ 1.216 & \& \text{Fr}_M \geq 2.86 \end{cases} \quad (2-160)$$

Woods & Hanratty, (1996) presented an expression that was comparable to that of Manolis, (1995) and their connection was as follows:

$$C_0 = \begin{cases} 1.1 & \text{Fr}_M < 3.1 \\ 1.2 & \text{Fr}_M \geq 3.1 \end{cases} \quad (2-161)$$

2.7.3.4 Slug Body Hold-up

The volume percentage of liquid in the slug body or the slug body hold-up is a crucial parameter needed for the slug model closure relationship. Gas bubbles are entrained in the slug body through a highly intricate procedure. The prediction of the typical liquid hold-up R_{LS} within the slug is made using graphical representations and empirical relations based on small data sets. This section examines a few of them.

Gregory et al., (1978) proposed an empirical expression that depends only upon the mixture velocity.

$$R_{LS} = \frac{1}{1 + \left(\frac{V_M}{8.66}\right)^{1.39}} \quad (2-162)$$

Malne, (1982) suggested an alternative correlation by considering, the gravitational and surface tension effects.

$$R_{LS} = 1 - \frac{1}{1 + \left(\frac{83}{Fr_M Bo_L^{0.25}}\right)} \quad (2-163)$$

where Fr_M, Bo_L are the Froude number and the liquid Bond number with

$$Bo_L = \frac{g D^2 \rho_L}{\sigma} \quad (2-164)$$

Ferschneider, (1983) carried out some experiment at the Boussens flow loop in France and proposed the following correlation:

$$R_{LS} = \frac{1}{\left[1 + \left(\frac{V_M}{\sqrt{g D (\Delta \rho / \rho_L)}}\right)^2 / \left(\frac{A}{Bo^\beta}\right)^2\right]^2} \quad (2-165)$$

where the Bond number **Bo** is the ratio of gravitational to surface tension force:

$$Bo = \frac{g D^2 \Delta \rho}{\sigma} \quad (2-166)$$

Barnea & Brauner, (1985) suggested a method that considers the effect of turbulence, gravitational force and surface tension. The following correlation was derived by [Taitel & Barnea, 1990b]

$$R_{LS}=1-0.058 \left[2 \left(\frac{0.4\sigma}{\Delta\rho g} \right)^{0.5} \left(\frac{2f_s V_M^3}{D} \right)^{0.4} \left(\frac{\rho_L}{\sigma} \right)^{0.6} -0.725 \right]^2 \quad (2-167)$$

where f_s the Blasius friction factor is based on the liquid slug Reynolds number Re_S with

$$f_s=0.079 Re_S^{-0.25} \quad (2-168)$$

and $Re_S = \frac{\rho_L V_M D}{\mu_L} \quad (2-169)$

The above expression can be rewritten as

$$R_{LS}=1-0.058 \left[0.605 \left(\frac{Bo}{Re_S} \right)^{0.1} Fr_M^{1.2} -0.725 \right]^2 \quad (2-170)$$

where Bo, Re_S, Fr_M are the Bond, slug Reynolds and Froude numbers, and

$$0.48 < R_{LS} < 1$$

Marcano et al., (1998) suggested the following expression as

$$R_{LS} = \frac{1}{1.001+0.0179V_M+0.0011V_M^2} \quad (2-171)$$

This correlation worked well in term of accuracy compared to work of Gregory et al., (1978). Gomez et al., (2000) presented a correlation for upward inclined slug flow. The correlation was expressed as

$$R_{LS} = \exp \left(-(0.45\beta + 2.48 \cdot 10^{-6} Re_S D) \right) \quad (2-172)$$

with $0 \leq \beta \leq 1.57$ and $Re_S = \frac{\rho_L V_M D}{\mu_L} \quad (2-173)$

are the pipe inclination and the slug Reynolds number. Abdul-Majeed, (2000) proposed a correlation that takes into account, the influence of the dynamic viscosity of the fluids, the pipe diameter and the surface tension is as follows:

$$R_{LS} = (1.009 - CV_M)A \quad (2-174)$$

with $C = 0.006 + 1.3377 \frac{\mu_G}{\mu_L} \quad (2-175)$

$$A = \begin{cases} 1 & \beta \leq 0 \\ 1 - \sin(\beta) & \beta > 0 \end{cases} \quad (2-176)$$

Table 2-1 Mean slug lengths in horizontal pipes

Reference	D(mm)	Fluids	Mean Slug Length
Duckler & Hubbard (1975)	38	Air/Water	12-30D
Nicklson et al., (1978)	25, 51	Air/light oil	~30D
Gregory et al., (1978)	25, 51	Air/light oil	~30D (N.B. Some slugs reached 375D)
Barnea & Brauner (1985)	Theory	Theory	32D
Andreussi et al., (1988)	50	Air/Water	32D
Nydal et al., (1992)	53, 90	Air/Water	15-20D(53mm Pipe) 12-16D(90mm Pipe)
Manolis (1995)	78	Air/Water Air/ oil	10-25D

2.7.3.5 Slug length

The liquid slug length L_S is a crucial factor when sizing hydrocarbon pipelines, especially for slug catchers and other downstream separation equipment. The slug length appears to be preferred in most slug flow models, even though this parameter is related to the slug frequency. Through a review of the most prevalent correlations for slug length in the literature, we will quickly investigate.

Numerous researchers have studied the slug length parameter, including [Duckler & Hubbard, 1975], [Gregory et al., 1978], and [Manolis, 1995], and have discovered that the lengths are roughly 12 to 30 D, generally appearing to be constant values. The findings of certain observations are displayed in Table 2-1.

Despite the fact that these constant values offer reasonable estimates for pipelines with modest diameters (25-90D), they significantly underestimated typical values seen in those with big diameters (400-600D). Researchers have proposed a few logarithmic formulas in the absence of a theory to forecast the value of slug length. Brill et al., (1981) proposed the following correlation as

$$\ln(L_S) = -3.851 + 0.059 \ln\left(\frac{V_M}{0.3048}\right) + 5.445 \left[\ln\left(\frac{D}{0.0254}\right)\right]^{0.5} \quad (2-177)$$

Norris, (1982) modified the above expression to produce the following one.

$$\ln(L_S) = -3.287 + 4.589 \left[\ln\left(\frac{D}{0.0254}\right)\right]^{0.5} \quad (2-178)$$

Scott et al., (1986) suggested a more accurate version of this correlation by taking into account the two slug growth mechanisms. The following empirical equation was found for the mean slug length.

$$\ln(L_S) = -26.6 + 28.495 \left[\ln\left(\frac{D}{0.0254}\right)\right]^{0.1} \quad (2-179)$$

The above logarithmic expressions over-predict the slug length for small diameters. However, the chaotic nature of slug flow, made some researchers, to think that, slug length should be characterised statistically. They attempt to use probability density functions to predict the mean slug length, see King, (1998).

2.7.4 Slug Frequency

According to Gregory & Scott, (1969), the slug frequency is the typical number of slugs that pass a specific place in a system over time. Because slug flow patterns are statistical in nature, the slug frequency parameter accurately captures the intermittent nature of the flow. In the modelling of the closure relationships, this parameter is crucial. As a result, it needs to be correctly predicted.

The frequency of slugs varies according on the type of flow, whether it is developing or completely developed. As a result, there are two different kinds of models in the literature: phenomenological models and empirical models.

Table 2-2 Summary of fully developed slug frequency models

Model	D(mm)	Fluids	Slug Frequency (Hz)
Gregory & Scott (1969)	19	CO_2 -water	$\omega_S = 0.0226 \left[\lambda_L \left(\frac{2.02}{D} + Fr_M^2 \right) \right]^{1.2}$
Heywood & Richardson (1979)	42	Air/water	$\omega_S = 0.0226 \left[\lambda_L \left(\frac{2.02}{D} + Fr_M^2 \right) \right]^{1.02}$
Tronconi (1990)	----- -	-----	$\omega_S = 0.61 \left(\frac{\rho_G V_{GE}}{\rho_L h_{GE}} \right)$
Hill & Woods (1990)	50-590	Air/water kerosene	$\omega_S = 0.275 \frac{V_M}{3600D} 10^{(2.68R_{LS})}$
Hill & Woods (1990)	50-590	Air/water kerosene	$\omega_S = 2.74 \frac{(V_{GE} - V_{LE})}{3600D} \frac{R_{LE}}{(1 - R_{LE})}$
Stapelberg & Mewes (1994)	42, 101	Air/water	$\omega_S = \left[Fr (Fr_M^{0.1} - 1.1 Fr_L^{0.064})_{min} \right]^2 \sqrt{\frac{g}{D}}$
Manolis (1995)	78	Air/water Air/oil	$\omega_S = \frac{J_G}{D} \left(0.0363 \frac{F_L}{F_G} \frac{Z_L}{Eo^{0.2}} \right)^n$
Zabaras (1999)	25-203	Air/water Air/oil	$\omega_S = 0.0226 \left[\lambda_L \left(\frac{2.02}{D} + Fr_M^2 \right) \right]^{1.2} \varphi$ $\varphi = 0.836 + 2.75 \sin^{0.25}(\beta)$

Chapter 3: METHODOLOGY AND NEW APPROACH FOR MODELLING MULTIPHASE ANISOTROPIC TURBULENCE

3.1 Summary

Anisotropic multiphase turbulence modelling, a novel approach is the emphasis of discussion in this chapter. In section 3.2, which represents a contribution to knowledge, we explore THE NEW MULTIPHASE GOVERNING EQUATIONS FOR ANISOTROPIC MULTIPHASE MIXTURE FLOWS. The spatial derivatives of the mathematical terms of Eq. (3-4), must be implemented in a User-Defined Function based C computer code in the ANSYS Fluent software package to solve the problem by running a simulation. See Appendix: B and Appendix: C

3.2 The New Governing Equations and the Contribution to the Knowledge

A further development of (Könözy's, 2019; 2021) proposed anisotropic Reynolds stress tensor mathematical description to simulate multiphase turbulent mixed flows. A novel description of the anisotropic multiphase Reynolds stress tensor leads to the discovery of a new term in the momentum equation. As part of this PhD study, a fresh anisotropic multiphase $k-\varepsilon$ turbulence model has been created.

THE NEW MULTIPHASE GOVERNING EQUATIONS FOR ANISOTROPIC MULTIPHASE MIXTURE FLOWS

The set of governing equations for multiphase mixed flows that has been further extended to include a new anisotropic Reynolds stress tensor. The mixture continuity equation can be written as follows:

$$\frac{\partial \rho_m}{\partial t} + \nabla \cdot (\rho_m \mathbf{u}_m) = 0 \quad (3-1)$$

The momentum equation of the mixture can be given by

$$\frac{\partial(\rho_m u_m)}{\partial t} + \nabla \cdot (\rho_m u_m \otimes u_m) = -\nabla p + \sigma + \rho_m g + \nabla \cdot (\mu_m (\nabla u + \nabla u^T)) + \nabla \cdot \underline{\underline{\tau}}_m^{RA} \quad (3-2)$$

where σ represents the surface tension and τ_m^{RA} is the new anisotropic Reynolds stress tensor for multiphase flows:

$$\begin{aligned} \underline{\underline{\tau}}_m^{RA} &= -\rho_m \overline{u'_m \otimes u'_m} = \mu_{t,m} [(\nabla \otimes u_m) + (\nabla \otimes u_m)^T] + \mu_{\Theta_m} \Theta_m \cdot \underline{\underline{G}} - \frac{2}{3} [\rho_m k + \mu_{t,m} (\nabla \cdot u_m)] \cdot \underline{\underline{I}} = \\ &= \mu_{t,m} [(\nabla \otimes u_m) + (\nabla \otimes u_m)^T] + \mu_{\Theta_m} \rho_m C_\mu \frac{k^2}{\varepsilon} \cdot |\nabla \times u_m| \cdot \underline{\underline{G}} - \frac{2}{3} [\rho_m k + \mu_{t,m} (\nabla \cdot u_m)] \cdot \underline{\underline{I}} \end{aligned} \quad (3-3)$$

which is an anisotropic modification to the Boussinesq hypothesis for multiphase mixture flows where the new anisotropic multiphase Reynolds stress tensor term can be expressed by

$$\mu_{\Theta_m} \rho_m C_\mu \frac{k^2}{\varepsilon} \cdot |\nabla \times u_m| \cdot \underline{\underline{G}} \quad (3-4)$$

where μ_{Θ_m} is the anisotropic multiphase dimensionless scalar weight parameter. The new turbulence model $\underline{\underline{G}}$ is being calibrated using the Reynolds stress distribution. transformed deviatoric anisotropic similarity tensor which is defined in the book of (Könözy, 2019, 2021). The dominant (principal) turbulent shear stress of the mixture Θ_m can be computed as the product of the eddy viscosity coefficient of the mixture $\mu_{t,m}$ and the magnitude of the mixture vorticity as

$$\Theta_m = \mu_{t,m} |\Omega_m| = \rho_m C_\mu \frac{k^2}{\varepsilon} \cdot |\nabla \times u_m| \quad (3-5)$$

where the eddy viscosity coefficient of the mixture is

$$\mu_{t,m} = \rho_m C_\mu \frac{k^2}{\varepsilon} \quad (3-6)$$

and the magnitude of the mixture vorticity field can be computed as the absolute value of the rotation of the mixture velocity field as

$$|\Omega_m| = |\nabla \times u_m| \quad (3-7)$$

The turbulent kinetic energy equation k for anisotropic multiphase turbulent mixture flows:

$$\frac{\partial(\rho_m k)}{\partial t} + (\rho_m u_m \cdot \nabla) k = P_{k,m} - \rho_m \varepsilon + \nabla \cdot \left(\frac{\mu_{t,m}}{\sigma_k} \nabla k \right) \quad (3-8)$$

where the new anisotropic production term can be written as

$$\begin{aligned} P_{k,m} &= \mu_{t,m} [(\nabla \otimes u_m) + (\nabla \otimes u_m)^T] \cdot (\nabla \otimes u_m) + \left(\mu_{\Theta_m} \Theta_m \cdot \underline{\underline{G}} \right) \cdot (\nabla \otimes u_m) = \\ &= \mu_{t,m} [(\nabla \otimes u_m) + (\nabla \otimes u_m)^T] \cdot (\nabla \otimes u_m) + \mu_{\Theta_m} \rho_m C_\mu \frac{k^2}{\varepsilon} \cdot |\nabla \times u_m| \cdot \left[\underline{\underline{G}} \cdot (\nabla \otimes u_m) \right] \end{aligned} \quad (3-9)$$

which is a brand-new term for anisotropic production of turbulent energy derived from the Reynolds stress tensor of an anisotropic multiphase mixture (3-82).

The dissipation of turbulent kinetic energy transport equation

$$\text{is } \frac{\partial(\rho_m \varepsilon)}{\partial t} + (\rho_m u_m \cdot \nabla) \varepsilon = \frac{\varepsilon}{k} C_{1\varepsilon} P_{k,m} - \frac{\varepsilon}{k} C_{2\varepsilon} \rho_m \varepsilon + \nabla \cdot \left(\frac{\mu_{t,m}}{\sigma_\varepsilon} \nabla \varepsilon \right) \quad (3-10)$$

where C_μ , $C_{1\varepsilon}$, $C_{2\varepsilon}$, σ_k , σ_ε are model constants of the anisotropic multiphase mixture version of the k – ε turbulence model which must be further calibrated for anisotropic multiphase flows.

The mixture density can be computed as

$$\rho_m = \alpha_L \rho_L + \alpha_g \rho_g \quad (3-11)$$

where α_g , α_L are the volume fraction of the gas and liquid phases, and ρ_L , ρ_g are the densities of the liquid and gas phases.

The mixture velocity can be given as

$$u_m = \frac{\alpha_L \rho_L u_L + \alpha_g \rho_g u_g}{\rho_m} \quad (3-12)$$

The liquid-gas interface can be tracked with the Volume-Of-Fluid method, thus the continuity equation for the gas phase can be expressed by

$$\frac{\partial(\alpha_g \rho_g)}{\partial t} + \nabla \cdot (\alpha_g \rho_g u_m) = 0 \quad (3-13)$$

The system of equations from Eq. (3-1) to (3-13) consists of a closed set of equations where the model constants must be calibrated and validated for anisotropic multiphase turbulent flows.

We can see that the spatial derivatives of the mathematical terms of Eq. (3-4), must be implemented in a User-Defined Function based C computer code in the ANSYS Fluent software package to solve the problem by running a simulation. See Appendix: B and Appendix: C

Chapter 4: VALIDATION OF TURBULENCE APPROACHES FOR THE HYDRODYNAMIC SLUGS FLOWS IN COMPARISON WITH EXPERIMENTAL DATA

4.1 Summary

Chapter 4 describe the validation of turbulence approaches for hydrodynamic slugs flows in comparison with experimental data. The experimental apparatus, procedures and results are discussed in Appendix D. In section 4.2, the numerical solutions to the experimentation are run and the results displayed. The validation of the isotropic multiphase (VOF) model with the experimental data is done in section 4.3.1 and the validation of the new anisotropic multiphase model discussed in section 4.3.2. The conclusion is done in section 4.4.

4.2 Numerical Solutions

4.2.1 Introduction

For our issue, we used ANSYS FLUENT 2022/R2, one of the programmes available to solve computational fluid dynamics (CFD) difficulties. This computer programme offers unequalled analysis capabilities, all the tools required to develop and optimise new equipment and is a fully featured fluid dynamics solution for studying flow and other physical processes. The capabilities of FLUENT were developed by well-known specialists and supported by highly skilled engineers, enabling it to be used to create more inventive and high-quality products. High productivity is offered by the ANSYS Workbench platform, which directly integrates with CAD tools to mesh fluid volumes.

The solvers in the ANSYS FLUENT package can model the behaviour of a wide range of flows, including Newtonian and non-Newtonian, single-phase and multiphase, subsonic, and hypersonic flows. Each solution has been thoroughly tested, validated, and optimised for quick and accurate simulations. It is also very resilient and efficient.

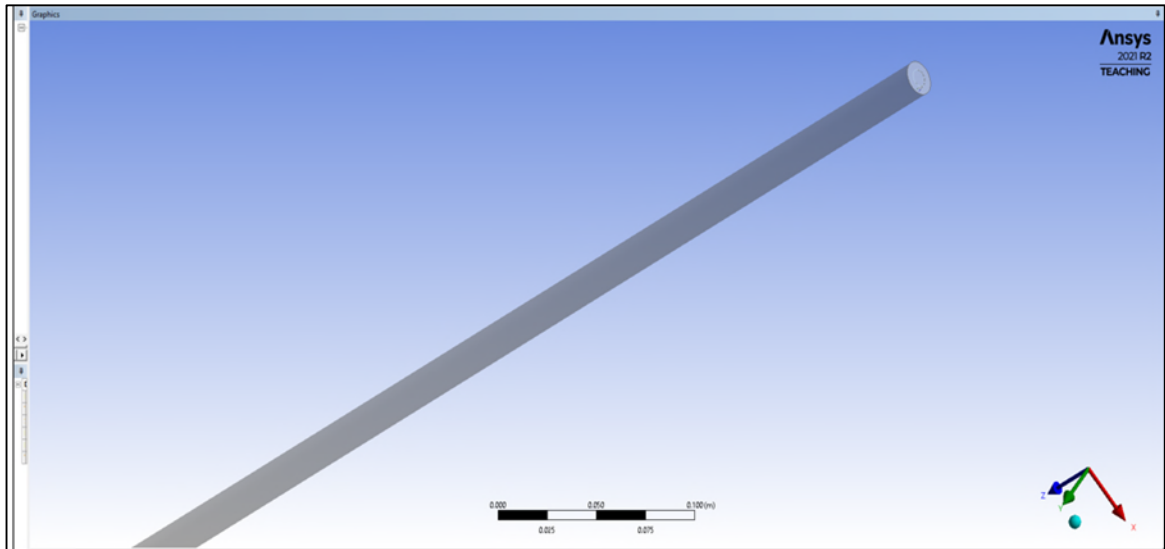


Figure 4-2 Pipeline design using workbench

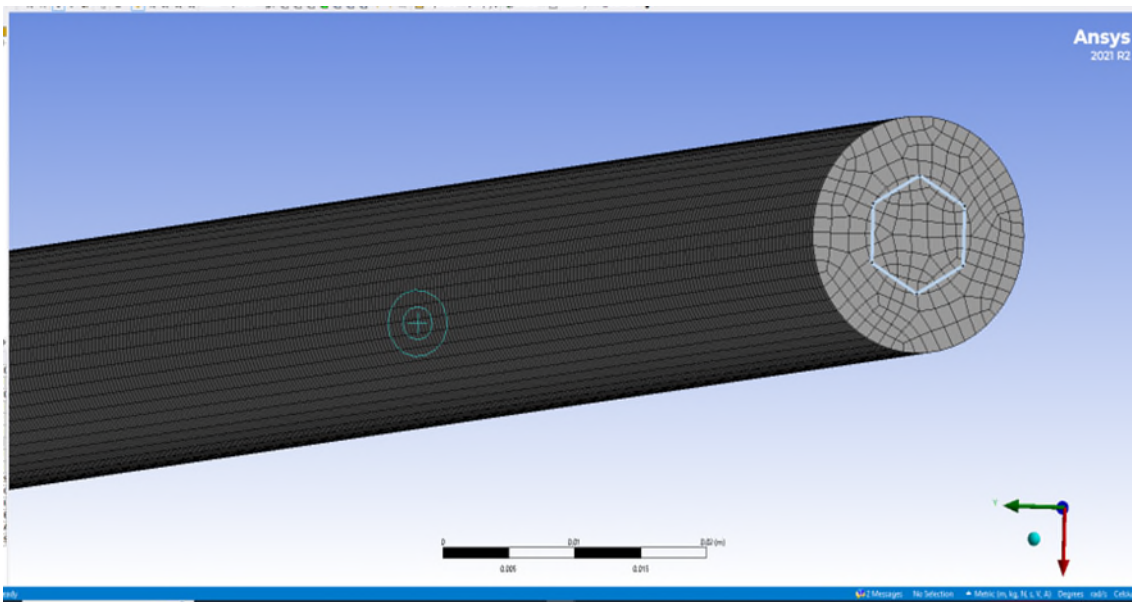


Figure 4-2 Hexahedral mesh using workbench

4.2.2 Transient Simulation

For our work, three cases have been considered.

In each case, the simulation is set up using the following steps.

1. General

The solver type is set to pressure based and the velocity to absolute. The simulation type is transient with 60s as time and 0.01s as time step size, and the gravity is set to $Y = -9.81 \text{ m/s}^2$.

2. Model

A multiphase Volume of Fluid (VOF) has been used to run this simulation with primary phase air (Phase1) and secondary phase water (Phase2). At the interface, surface tension has been added and the interface type is set to sharp. The volume fraction is set to explicit, as viscous model, $K - \varepsilon$ realisable with enhanced wall treatment is applied.

3. Boundary Conditions

The boundary conditions include the Inlet (Air-Inlet and Water-Inlet), the outlet and the wall conditions.

At the inlet, for the momentum, the velocity magnitude of air and water is normal to boundary and specified as mixture. The turbulence is specified by the intensity 5% and the viscosity ratio 10.

At the outlet, for the momentum, the backflow pressure is specified as total pressure and the backflow turbulence intensity 5% and viscosity ratio 10.

At the pipe wall, for the momentum, the wall remains stationary with no slip shear condition.

4. Initialisation

The initialisation is set hybrid with patching volume fraction as phase 2 (water) to 1.

5. Convergence of Solution

ANSYS FLUENT Different equations have solutions calculated by the solver. But residual values occur frequently. This happens because an equation is not completely satisfied. Of course, the residual should be 0 if the solution is exact. The outcomes in a solution, however, do not necessarily correspond to reality because equation answers can only approximate physics. The residuals for each

equation at the conclusion of each time-step can be plotted to provide an indication of how well a solution has converged. For a well converged solution, the expectation is that the maximum residual is around 10^{-3} . The RMS (Root Mean Square) residual will be an order of magnitude lower than this. i.e., 10^{-4} , 10^{-5}

Although a residual may increase after any time-step, this does not necessarily indicate that the solution is diverging. Residuals frequently grow from time to time, particularly at the start of a run.

Be aware that even in cases of good convergence, the residual can occasionally grow larger.

Additionally, it is possible for runs to simply deviate from the same value without ever converging.

6. Solution Methods

The pressure velocity coupling scheme is set to SIMPLE.

In the spatial discretisation:

- The gradient is set to least squares cell based.
- The pressure is set to Presto.
- The momentum is set to second order Upwind.
- The volume fraction is set to Geo-reconstruct.
- The turbulent kinetic energy is set to first order Upwind.
- The turbulent dissipation rate is set to first order Upwind.
- The transient formulation is set to first order implicit.

4.2.3 Results

4.2.3.1 Mesh Dependency (Case 1)

In **case 1**, two simulations have been run for coarse mesh (300000 hexahedral nodes) and fine mesh (600000 hexahedral nodes). The results are as follow.

At $t = 0.030s$

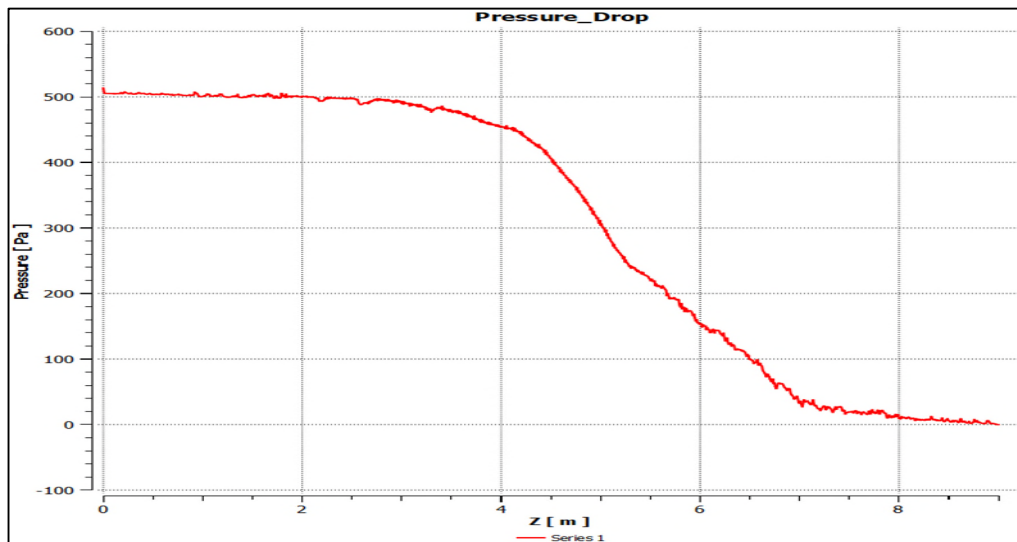


Figure 4-3 Pressure drops with coarse mesh (300000 hexahedral cells) with max 500 [Pa]

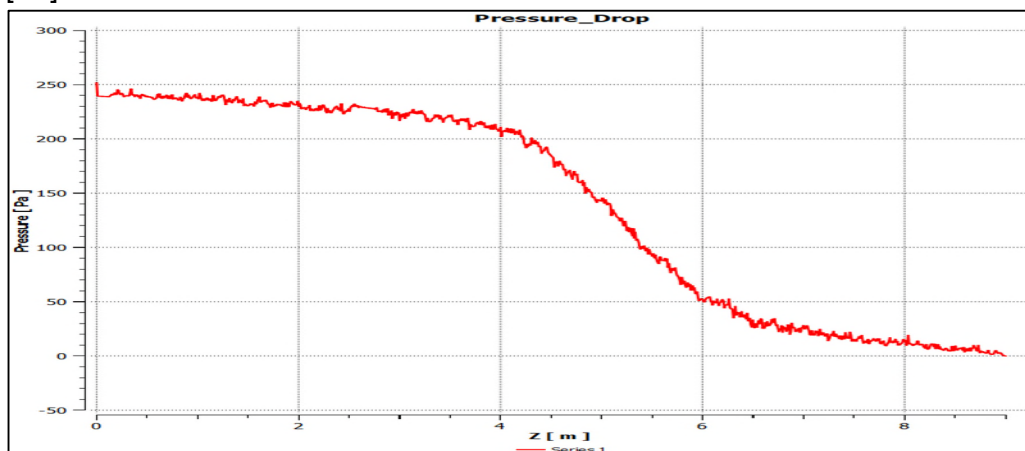


Figure 4-4 Pressure drops with fine mesh (600000 hexahedral cells) with max 250 [Pa]

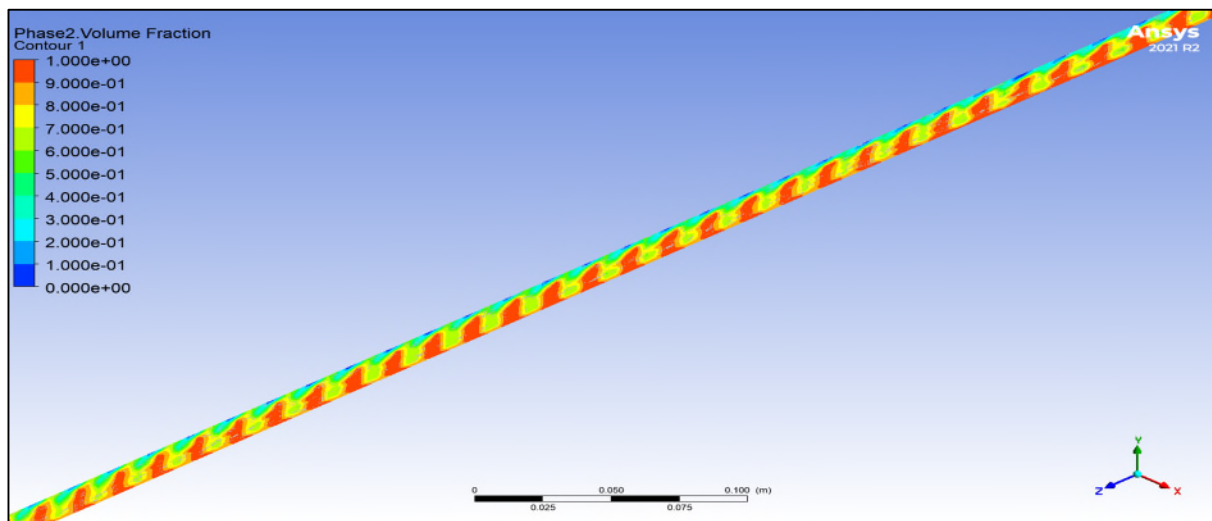


Figure 4-5 Water volume fraction contours (sluggish) with coarse mesh (300000 hexahedral cells)

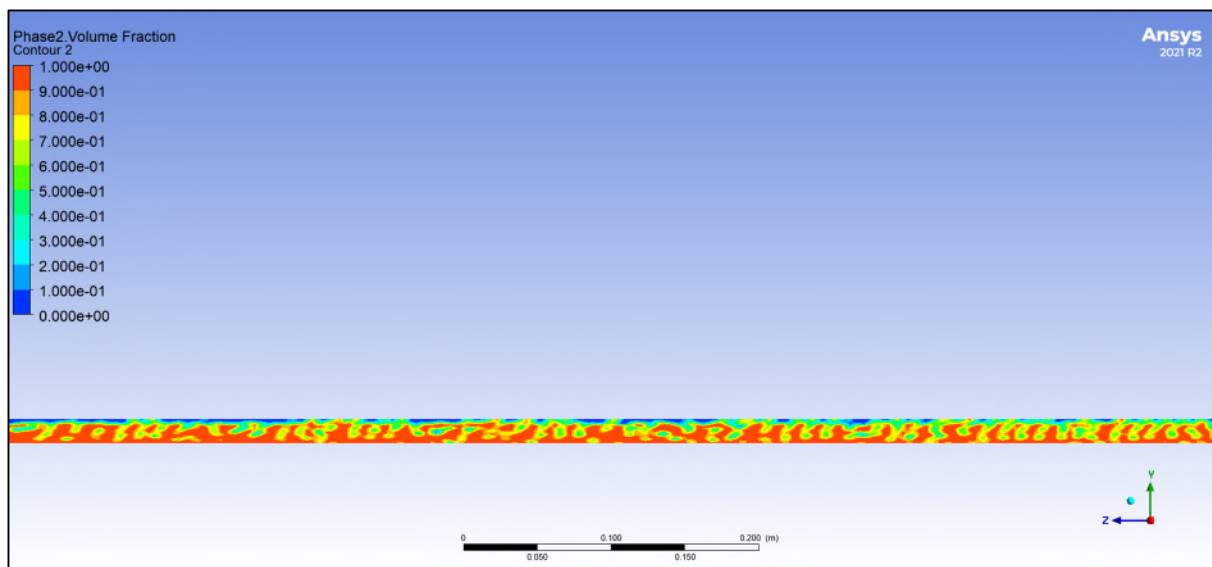


Figure 4-6 Water volume fraction contours (sluggish) with fine mesh (600000 hexahedral cells)

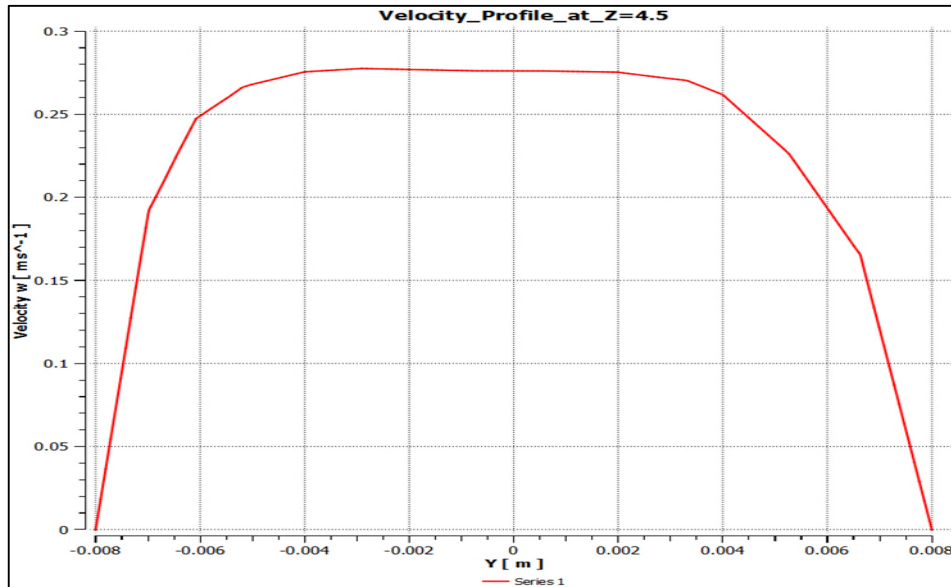


Figure 4-7 Velocity Profile at Z = 4.5 m with max velocity 0.27 m/s with coarse mesh (300000 hexahedral cells)

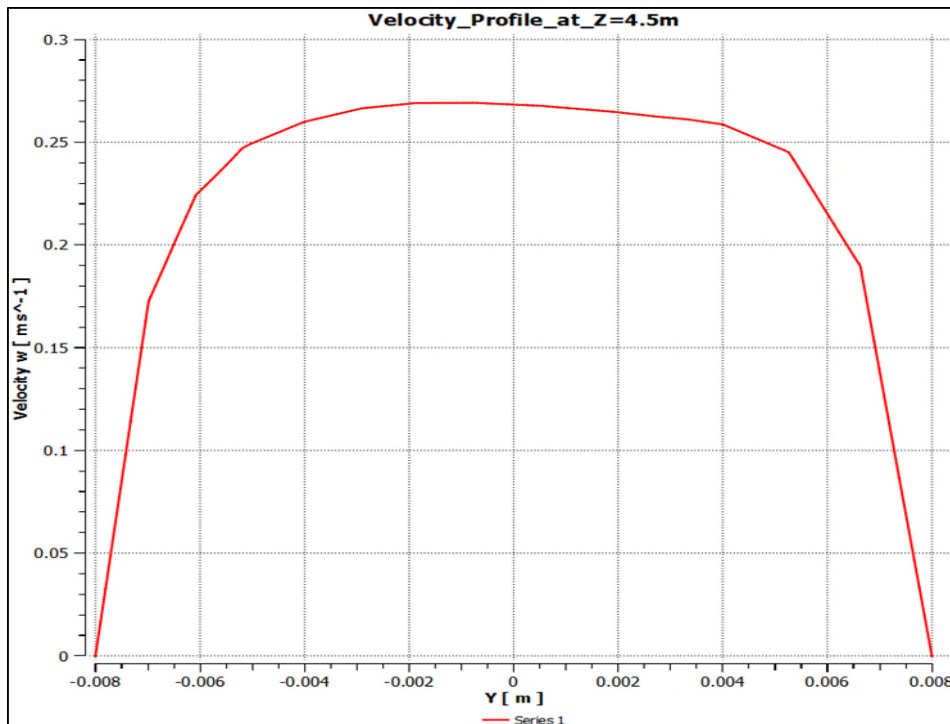


Figure 4-8 Velocity Profile at Z = 4.5 m with max velocity 0.26 m/s with fine mesh (600000 hexahedral cells)

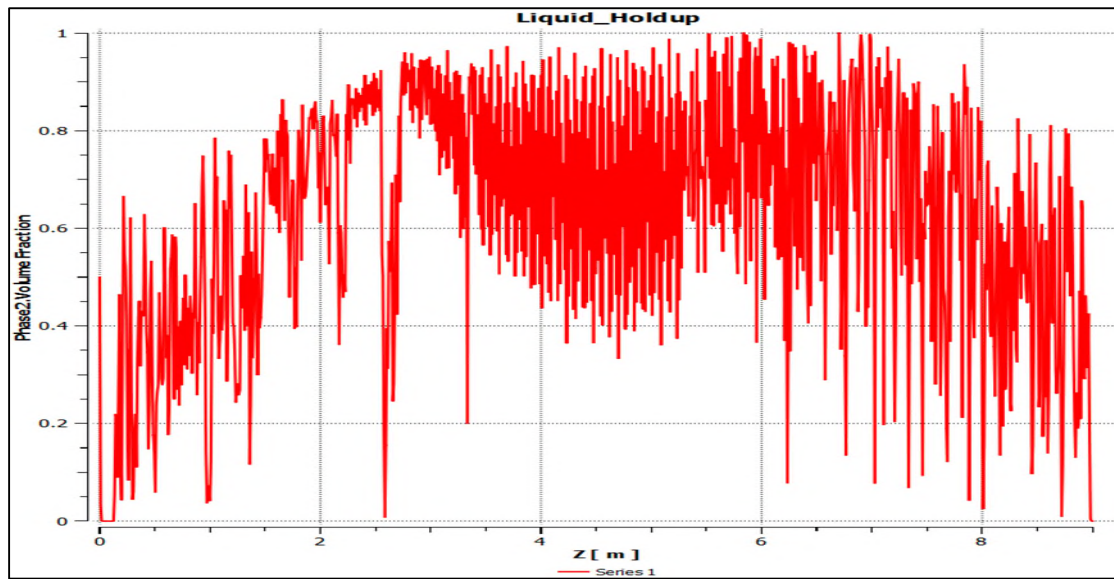


Figure 4-9 Liquid Holdup at pipeline with coarse mesh (300000 hexahedral)

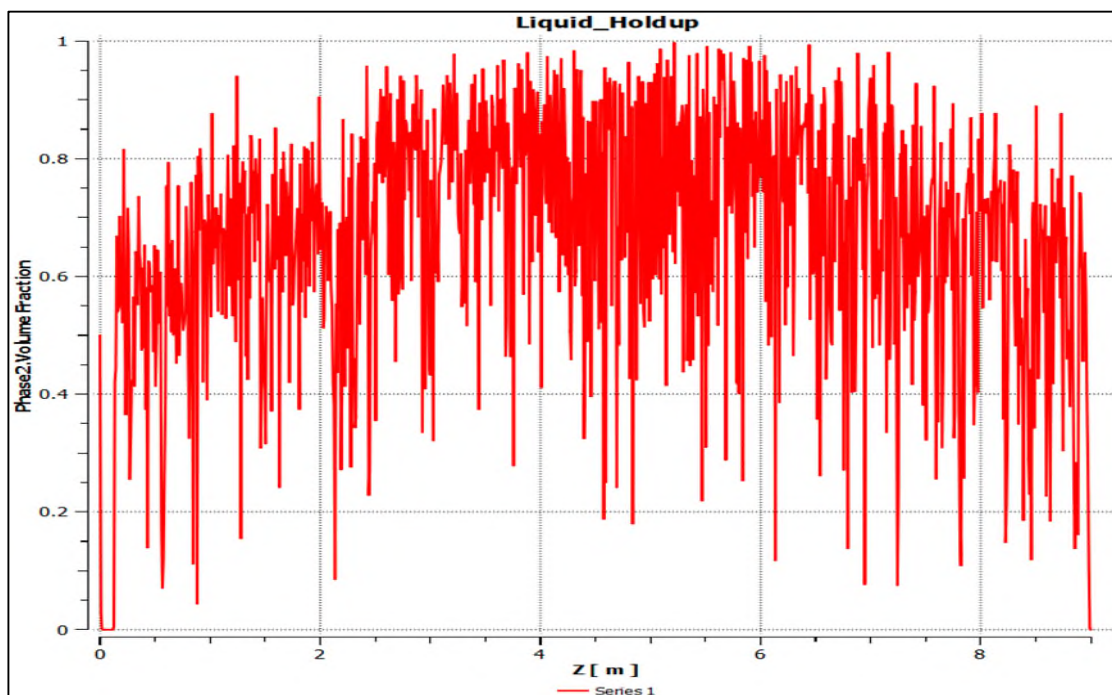


Figure 4-10 Liquid Holdup at pipeline with coarse mesh (600000 hexahedral)

At $t = 50s$

Using $U_{SL} = 0.075m/s$ and $U_{SG} = 0.7m/s$ the following results below:

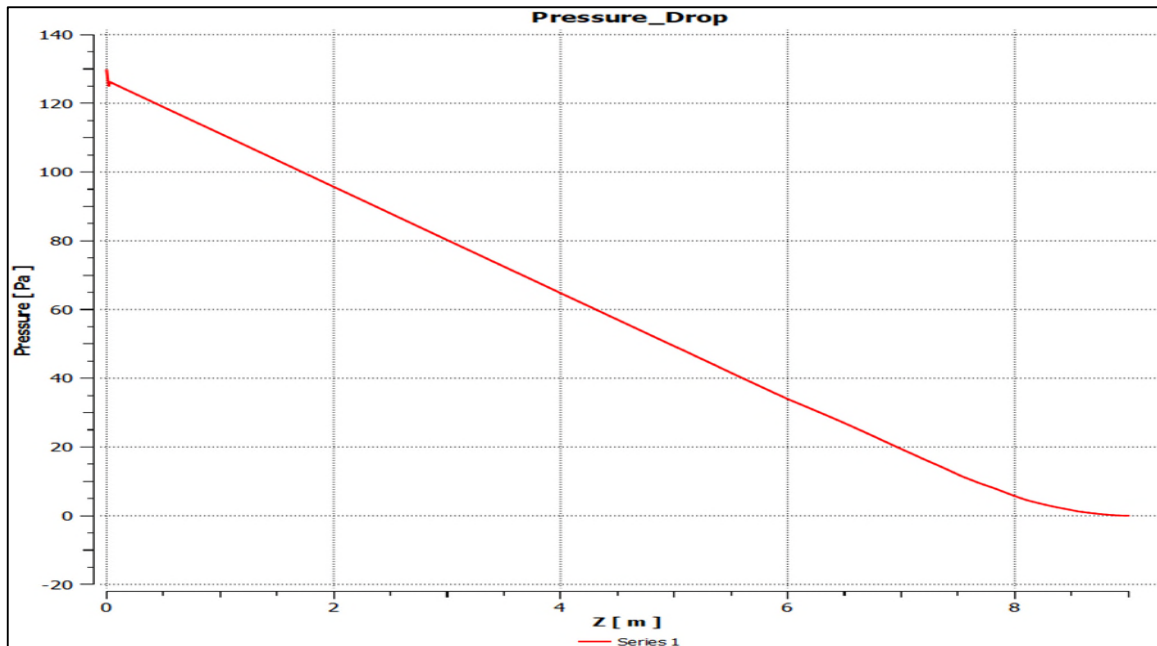


Figure 4-11 Pressure drops with coarse mesh (300000 hexahedral cells) with max 130 [Pa]

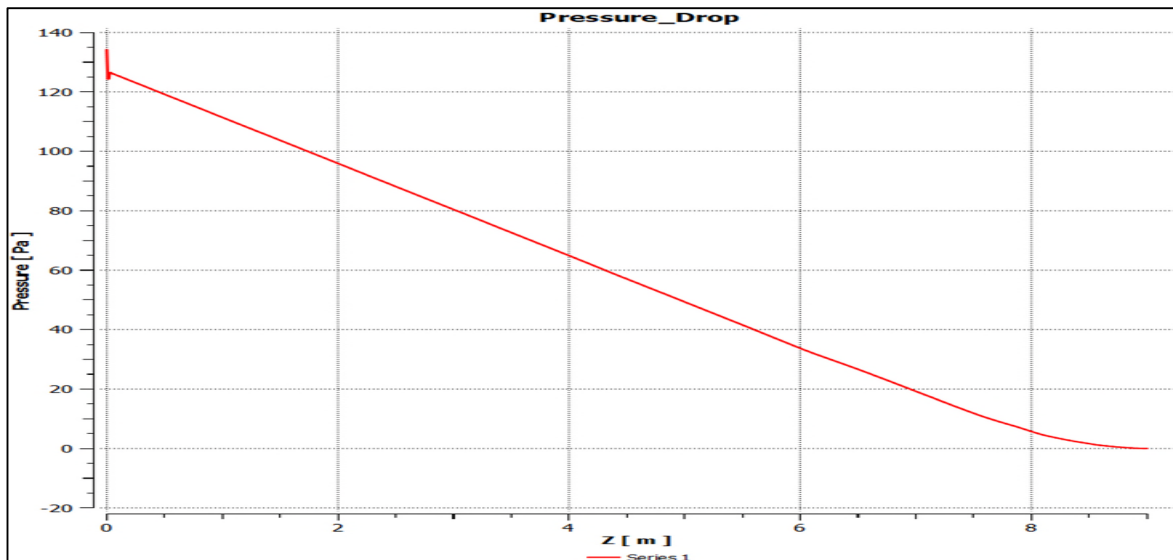


Figure 4-12 Pressure drops with coarse mesh (600000hexahedral cells) with max 130 [Pa]

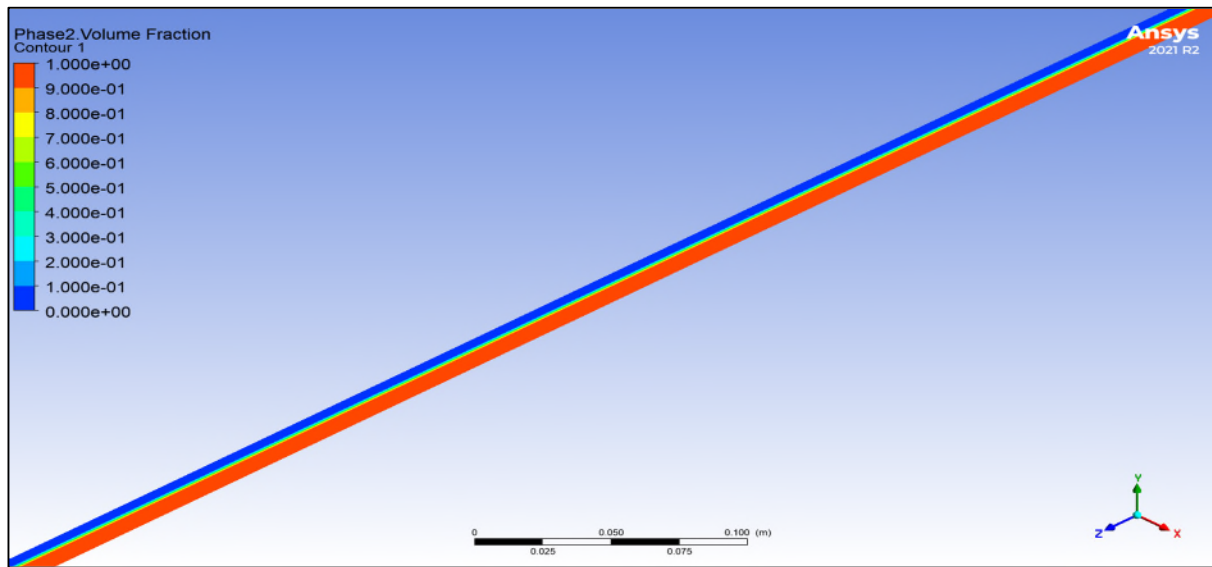


Figure 4-13 Water volume fraction contours with coarse mesh (300000 hexahedral cells)

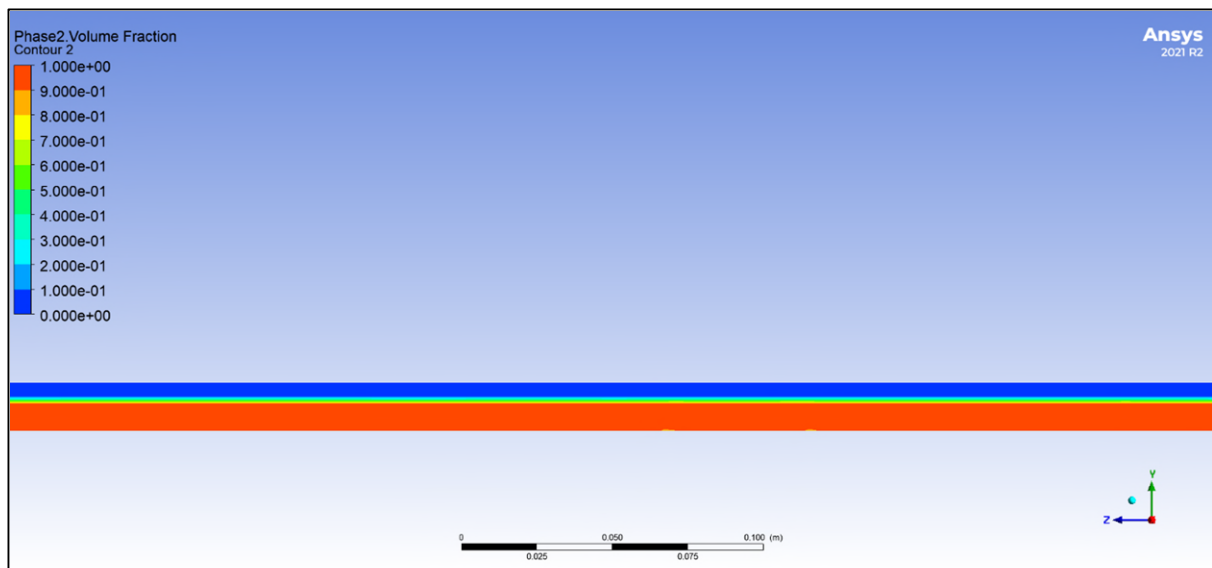


Figure 4-14 Water volume fraction contours with fine mesh (600000 hexahedral cells)

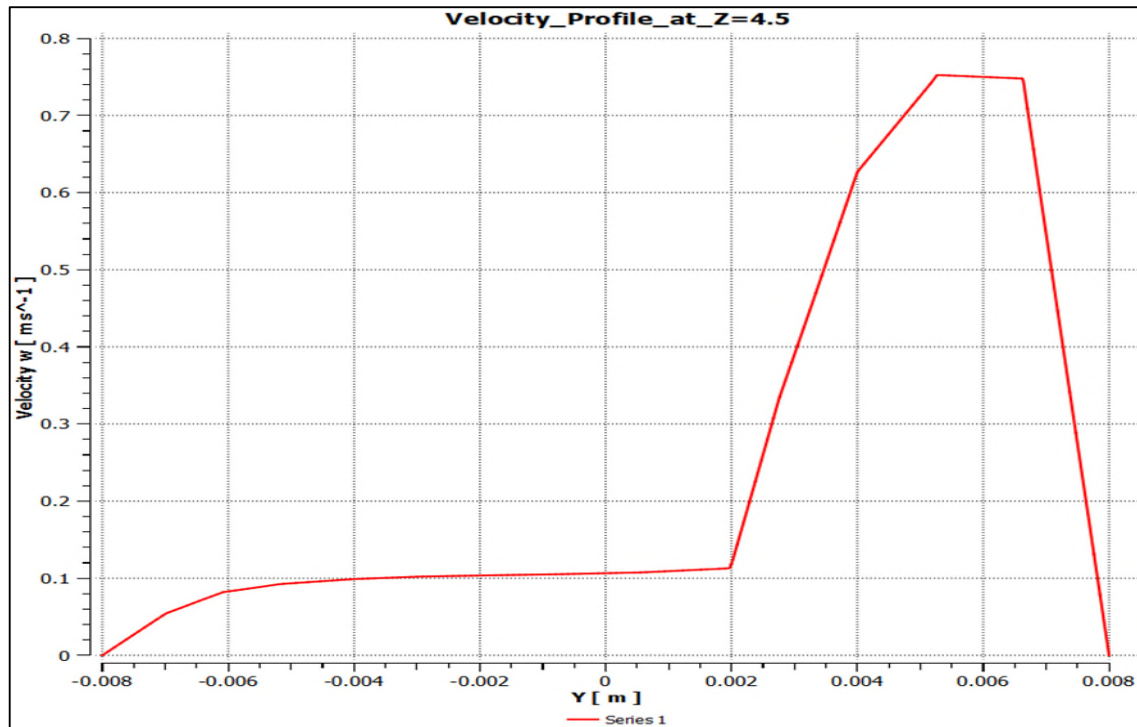


Figure 4-15 Velocity Profile at Z = 4.5 m with max velocity 0.75 m/s with coarse mesh (300000 hexahedral cells)

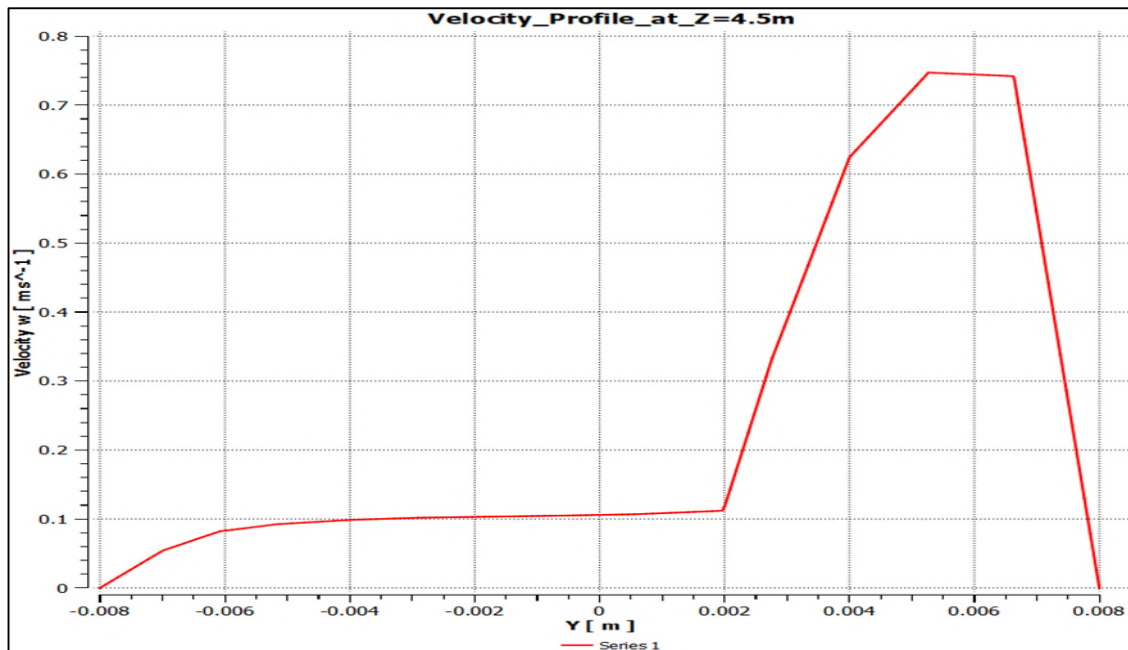


Figure 4-16 Velocity Profile at Z = 4.5 m with max velocity 0.75 m/s with fine mesh (600000 hexahedral cells)

4.2.3.2 Case 2

In case 2, the simulation is run with the superficial velocities of air and water respectively as follow $U_{SL} = 0.1m/s$ and $U_{SG} = 3.77m/s$. Pressure drop, liquid holdup and velocity profiles has been investigated at some time steps to follow the behaviour of the flow.

At $t = 0.02s$

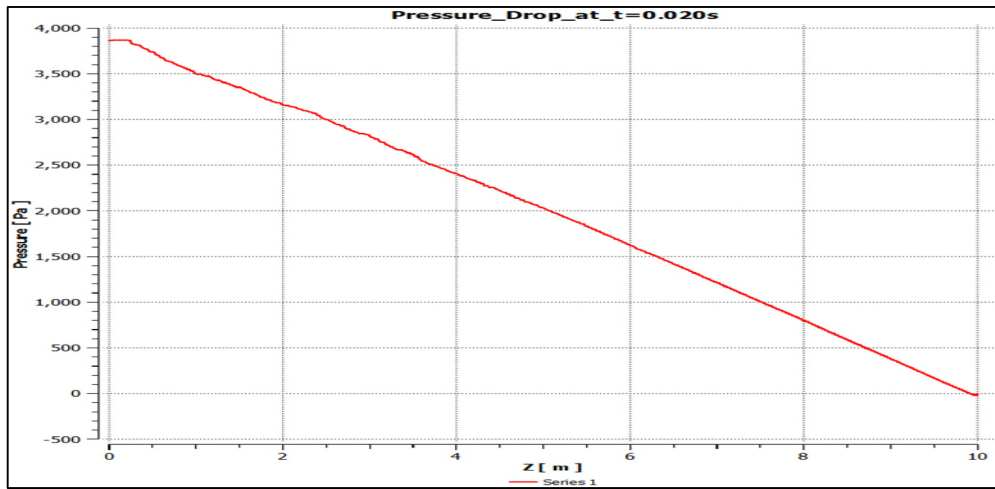


Figure 4-17 Pressure drops with fine mesh (500000 hexahedral cells) with max 3900 [Pa]

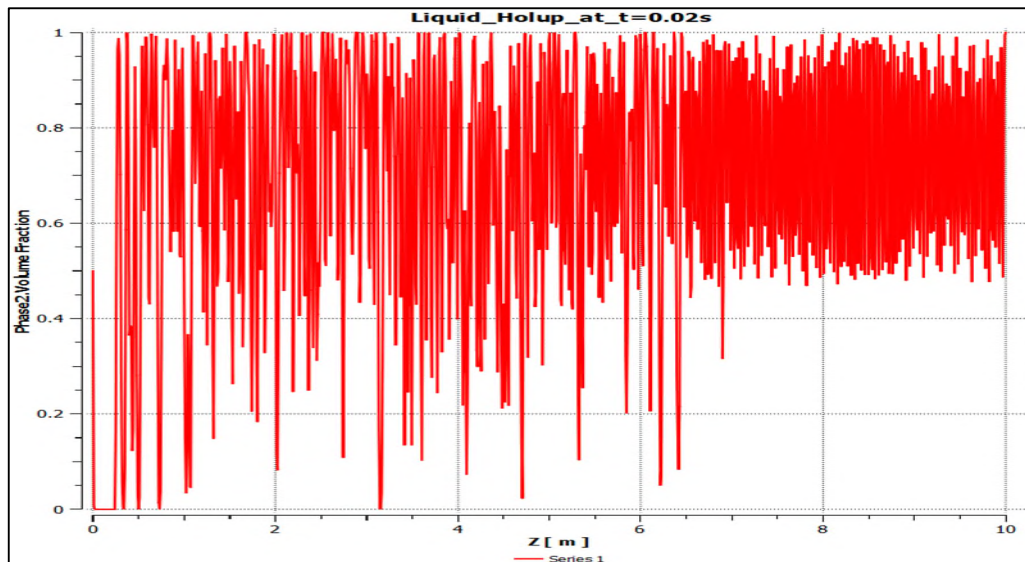


Figure 4-18 Liquid Holdup at pipeline with fine mesh (500000 hexahedral cells)

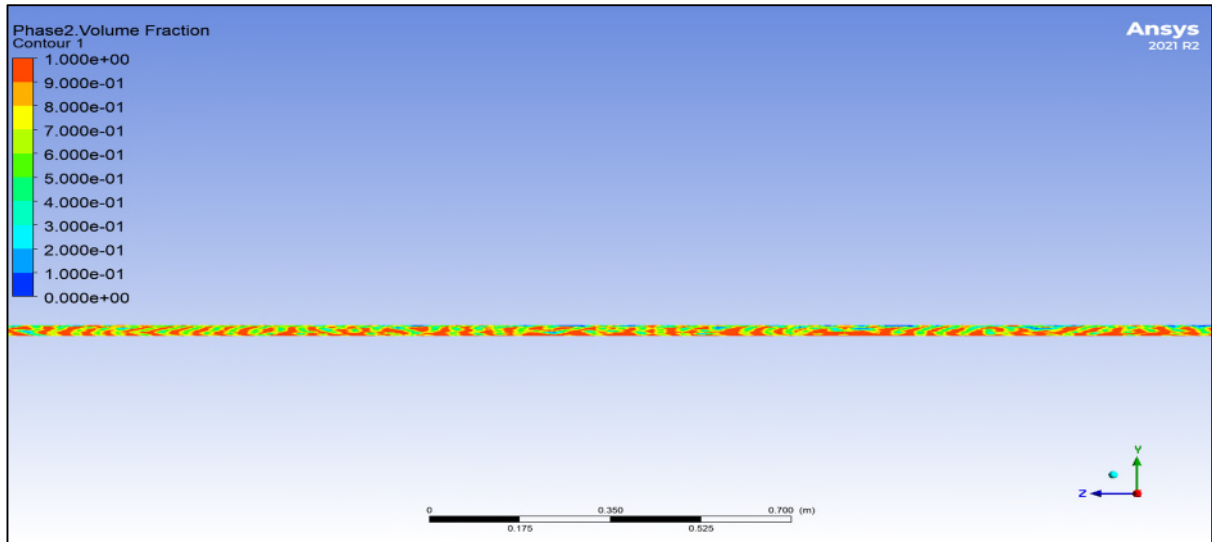


Figure 4-19 Water volume fraction contours (sluggish) with fine mesh (500000 hexahedral cells)

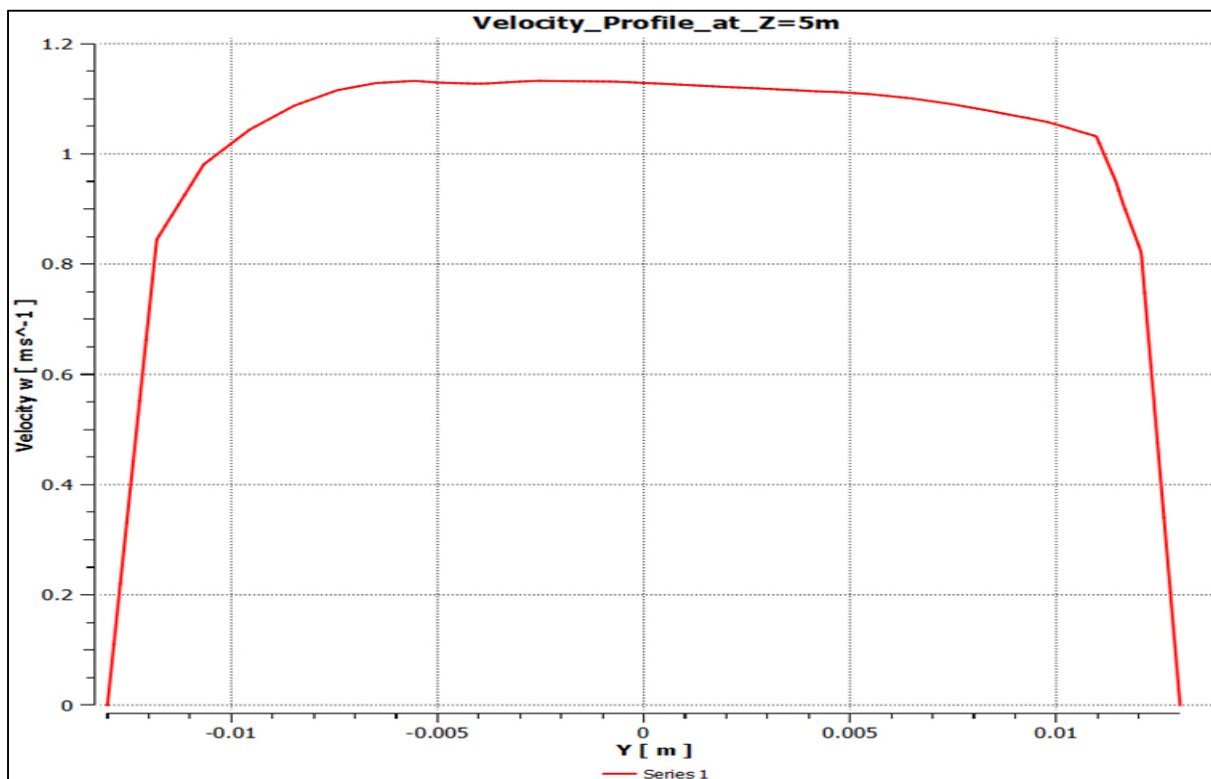


Figure 4-20 Velocity Profile at Z = 5 m with max velocity 1.10 m/s with coarse mesh (500000 hexahedral cells)

At $t = 0.04s$

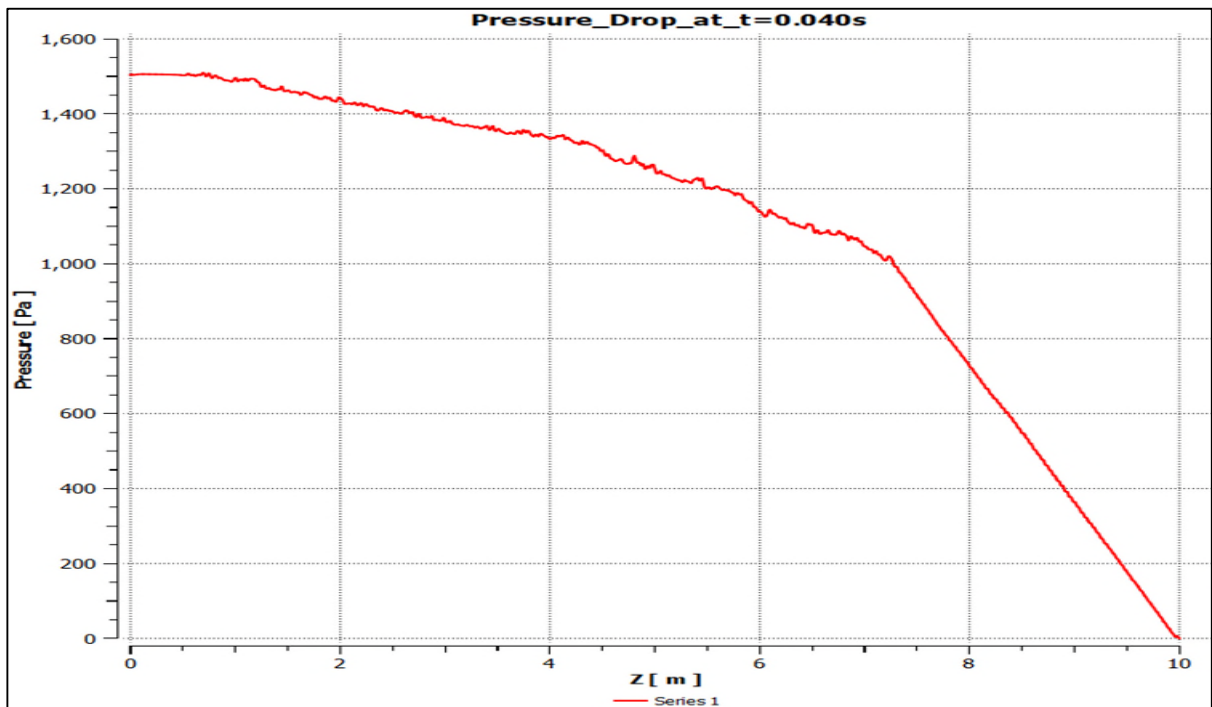


Figure 4-21 Pressure drops with fine mesh (500000 hexahedral cells) with max 1500 [Pa]

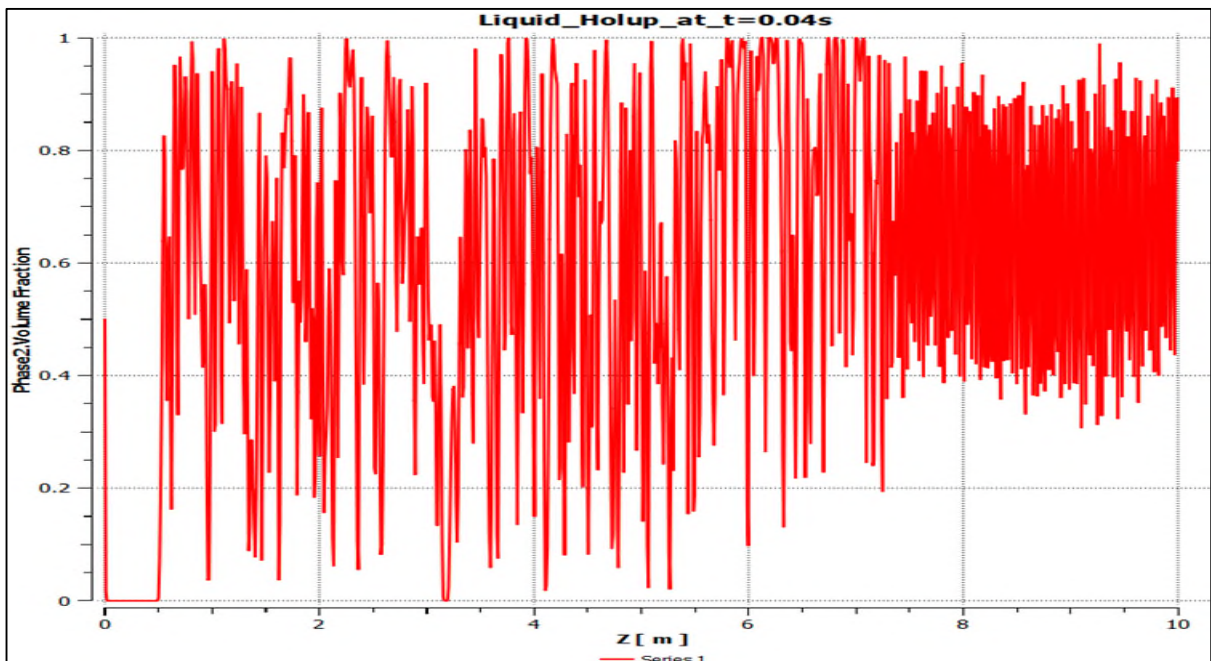


Figure 4-22 Liquid Holdup at pipeline with fine mesh (500000 hexahedral cells)

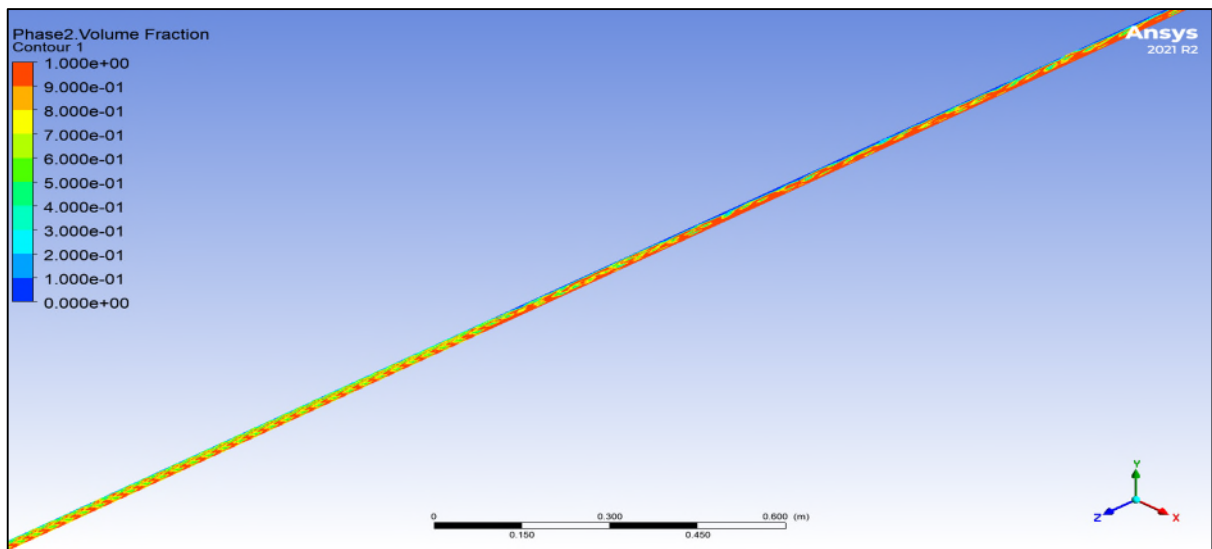


Figure 4-23 Water volume fraction contours (sluggish) with fine mesh (500000 hexahedral cells)

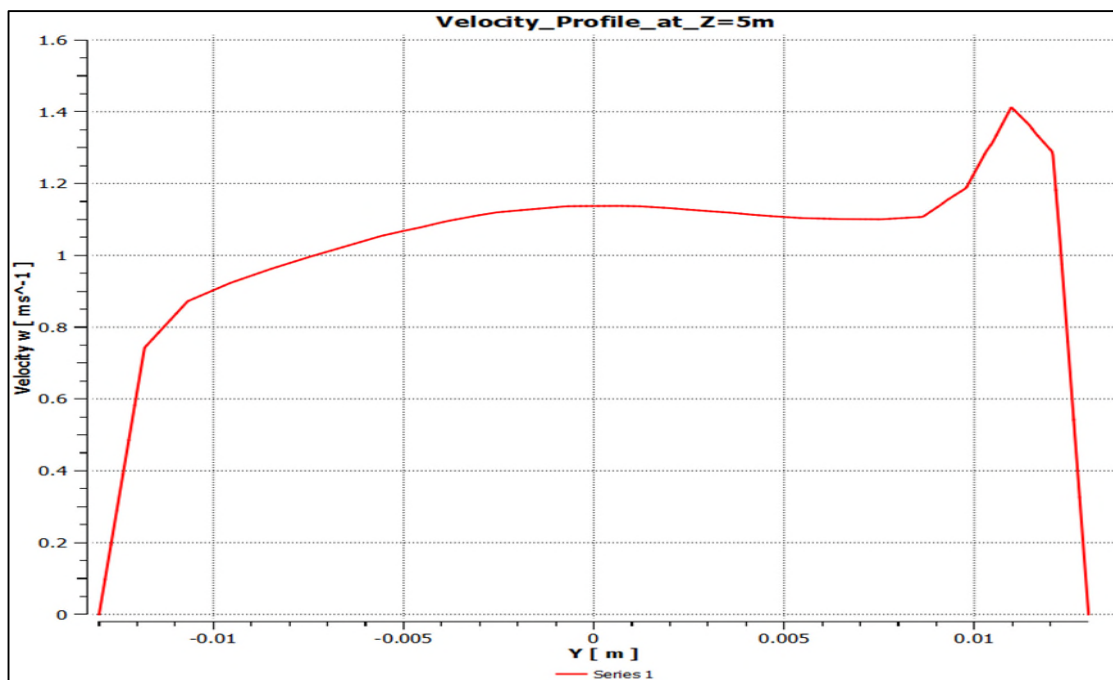


Figure 4-24 Velocity Profile at Z = 5 m with max velocity 1.4 m/s with coarse mesh (500000 hexahedral cells)

At $t = 0.06s$

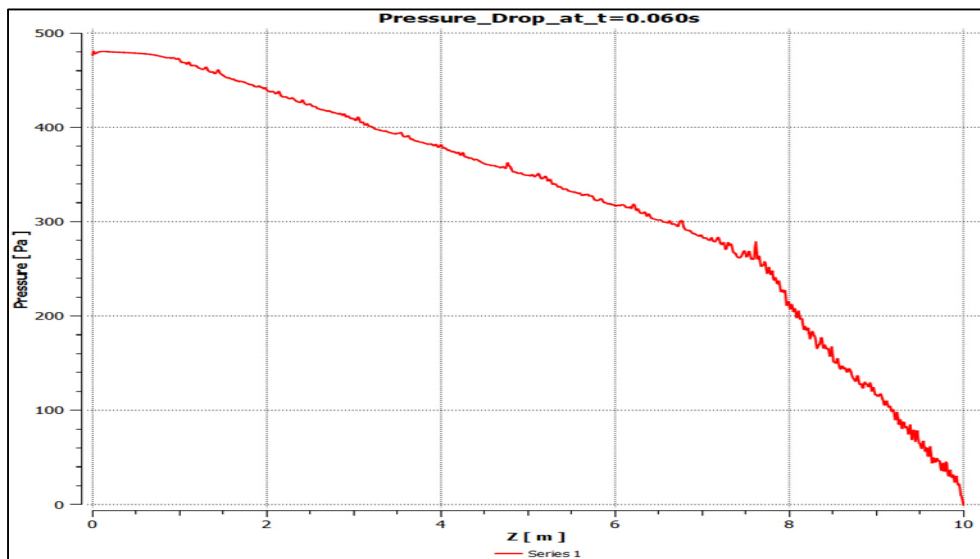


Figure 4-25 Pressure drops with fine mesh (500000 hexahedral cells) with max 490 [Pa]

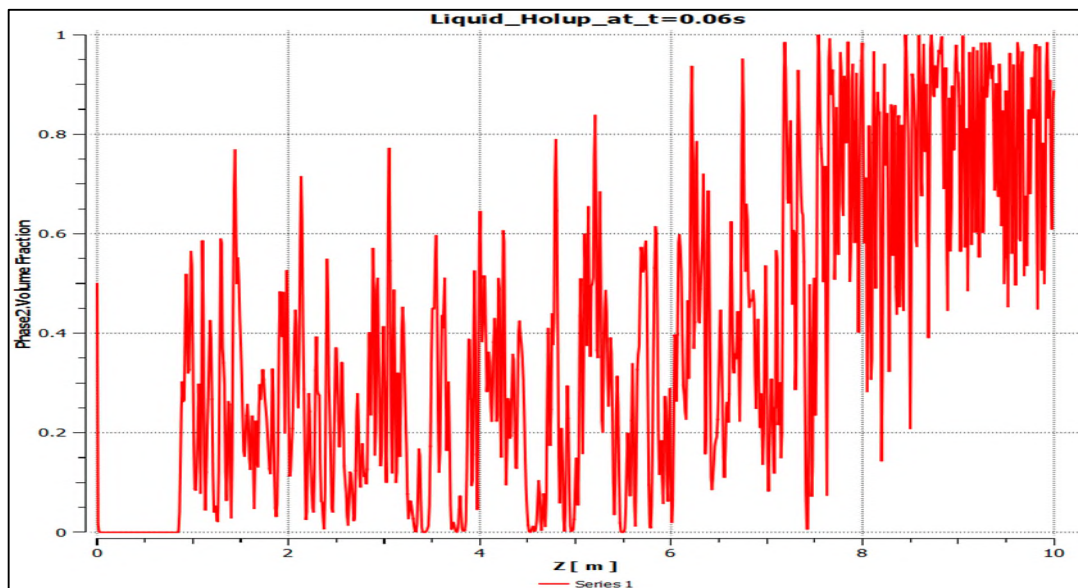


Figure 4-26 Liquid Holdup at pipeline with fine mesh (500000 hexahedral)

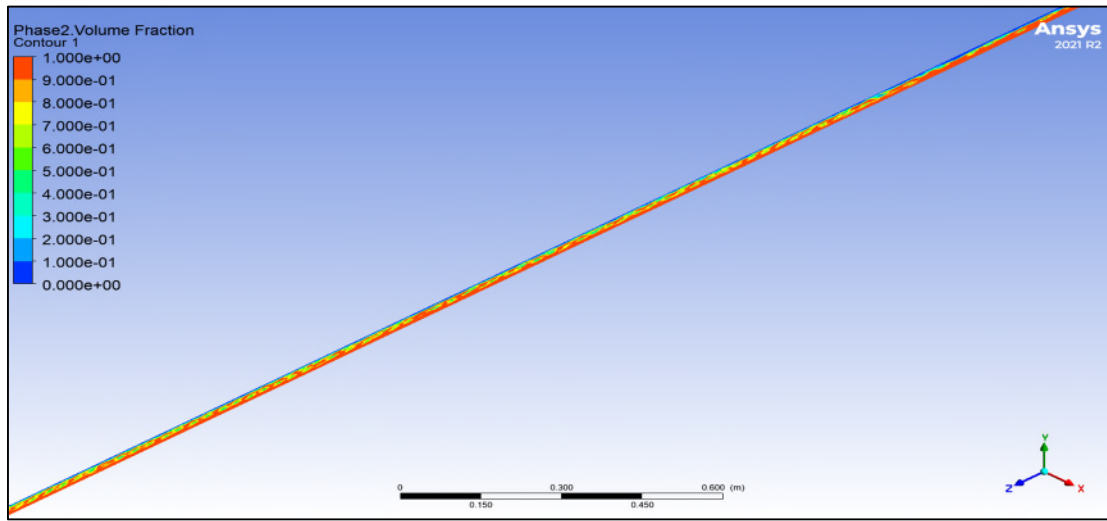


Figure 4-27 Water volume fraction contours (sluggish) with fine mesh (500000 hexahedral cells)

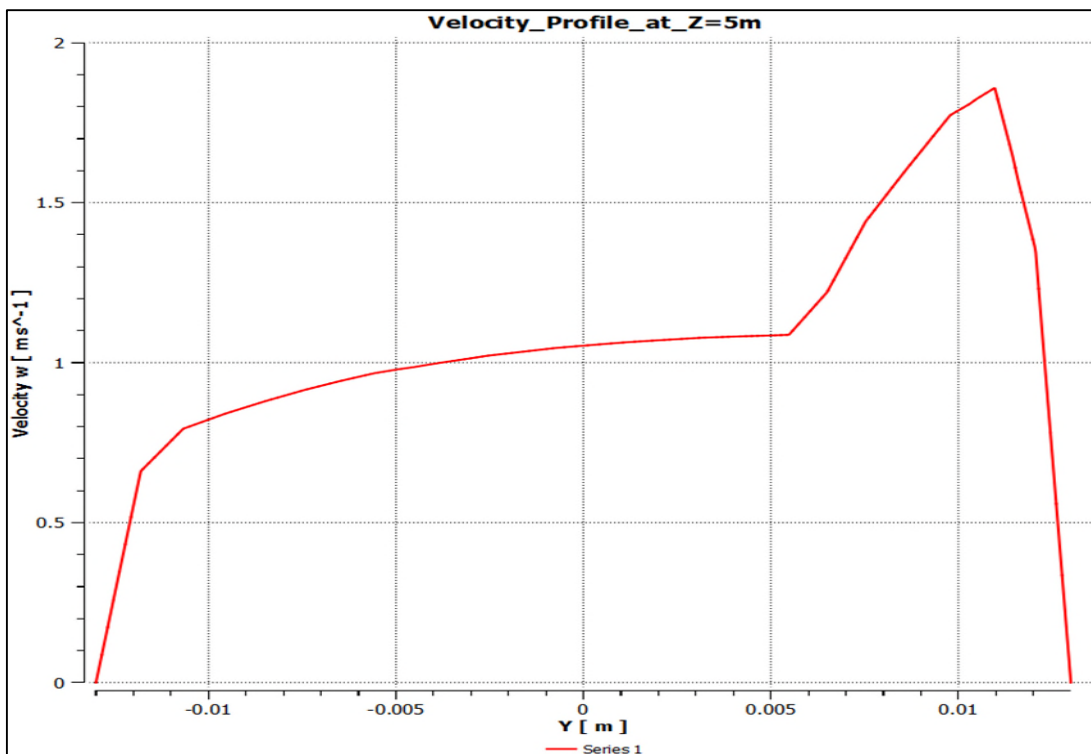


Figure 4-28 Velocity Profile at Z = 5 m with max velocity 1.85 m/s with coarse mesh (500000 hexahedral cells)

4.2.3.3 Case 3

In case 3, the simulation is run with the superficial velocities of air and water respectively as follow $U_{SL} = 0.075\text{m/s}$ and $U_{SG} = 3.77\text{m/s}$. Pressure drop, liquid holdup and velocity profiles has been investigated at some time steps to follow the behaviour of the flow.

At $t = 0.02\text{s}$

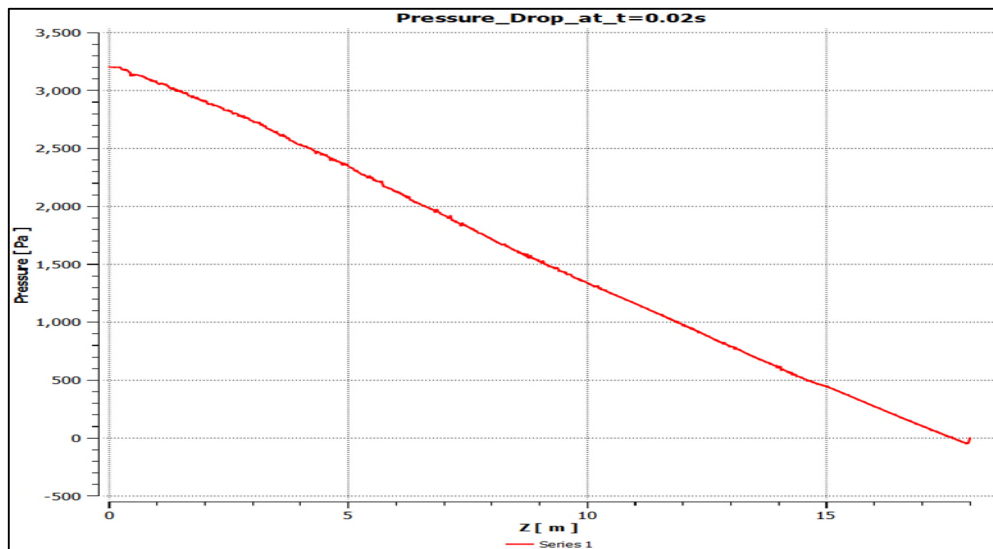


Figure 4-29 Pressure drops with fine mesh (600000 hexahedral cells) with max 3200 [Pa]

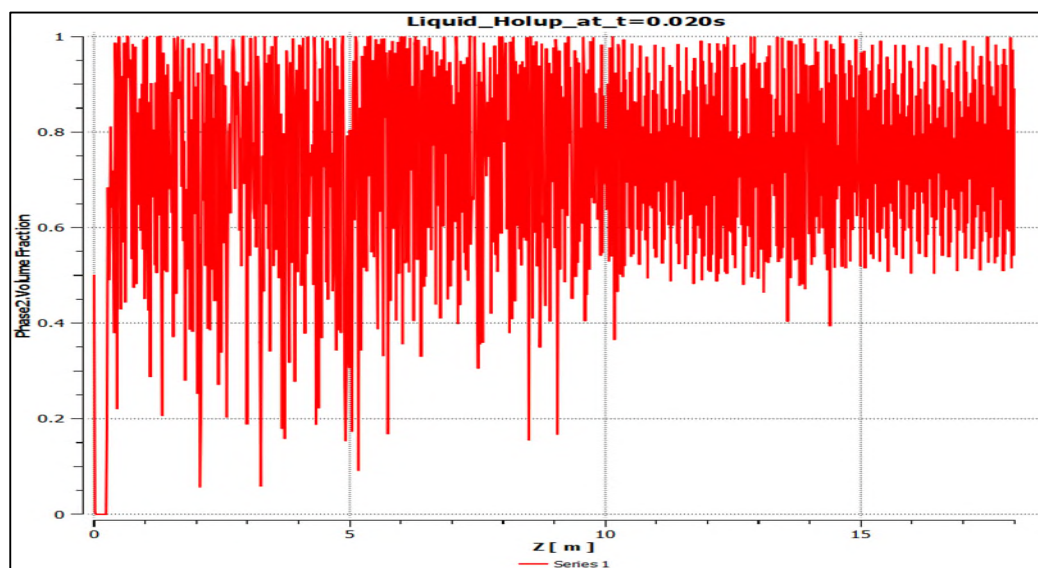


Figure 4-30 Liquid Holdup at pipeline with fine mesh (600000 hexahedral)

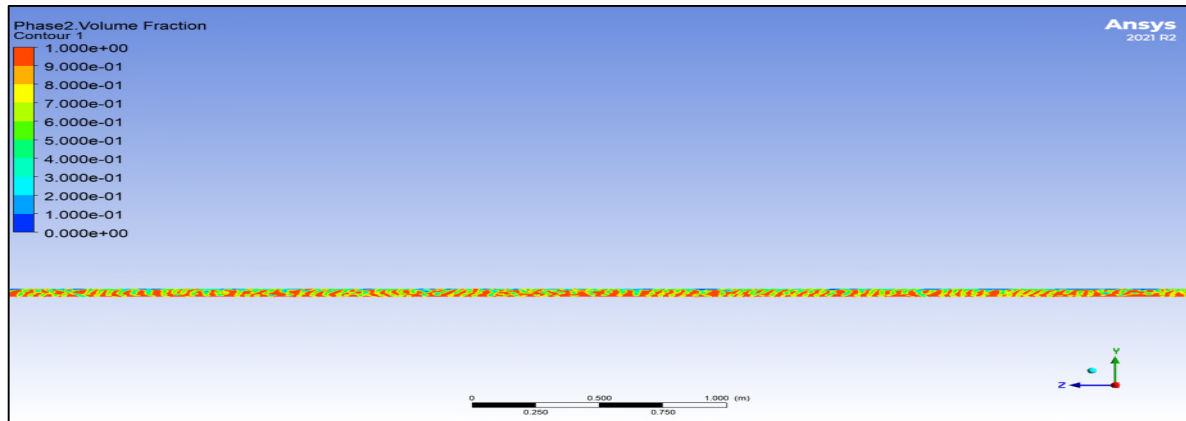


Figure 4-31 Water volume fraction contours (sluggish) with fine mesh (600000 hexahedral cells)

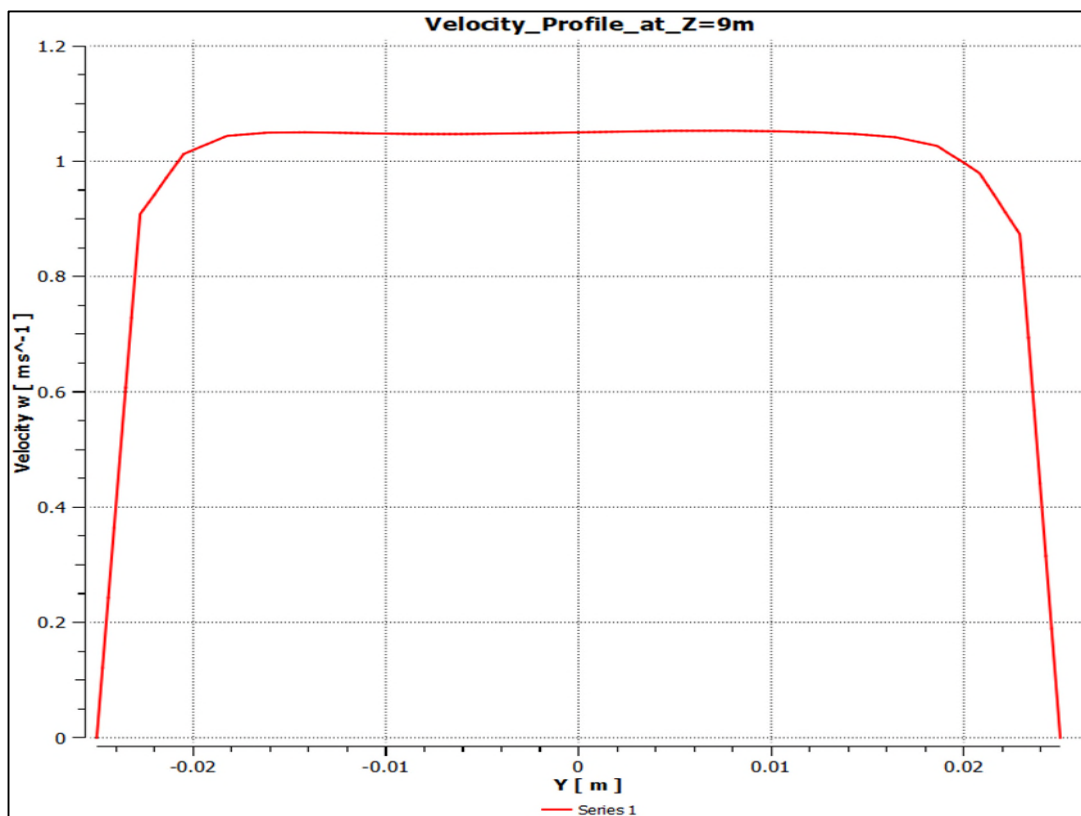


Figure 4-32 Velocity Profile at Z = 9 m with max velocity 1.05 m/s with coarse mesh (600000 hexahedral cells)

At $t = 0.03s$

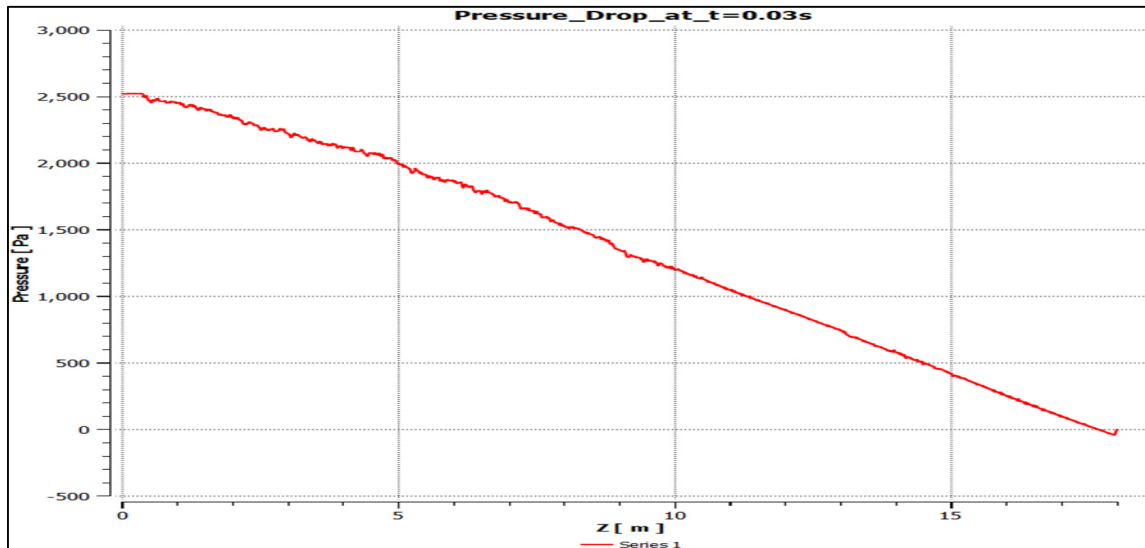


Figure 4-33 Pressure drops with fine mesh (600000 hexahedral cells) with max 2500 [Pa]

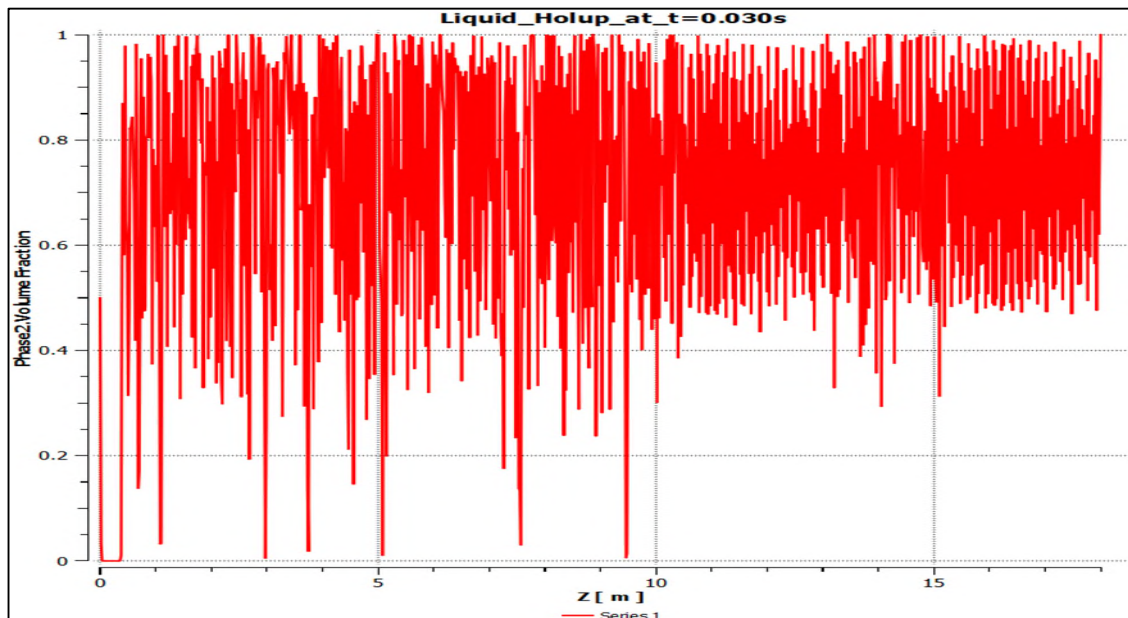


Figure 4-34 Liquid Holdup at pipeline with fine mesh (600000 hexahedral)

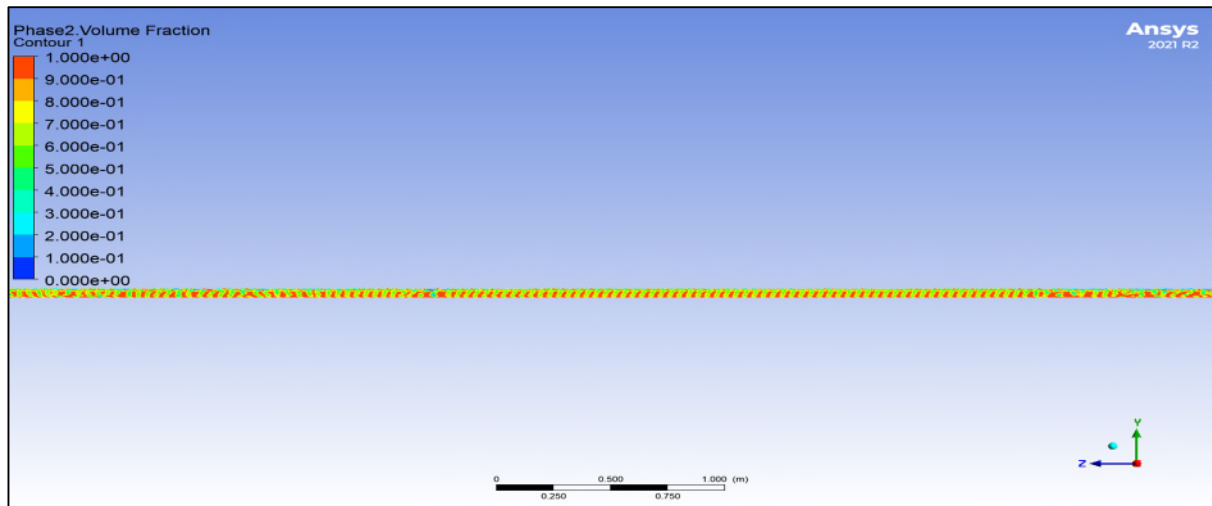


Figure 4-35 Water volume fraction contours (sluggish) with fine mesh (600000 hexahedral cells)

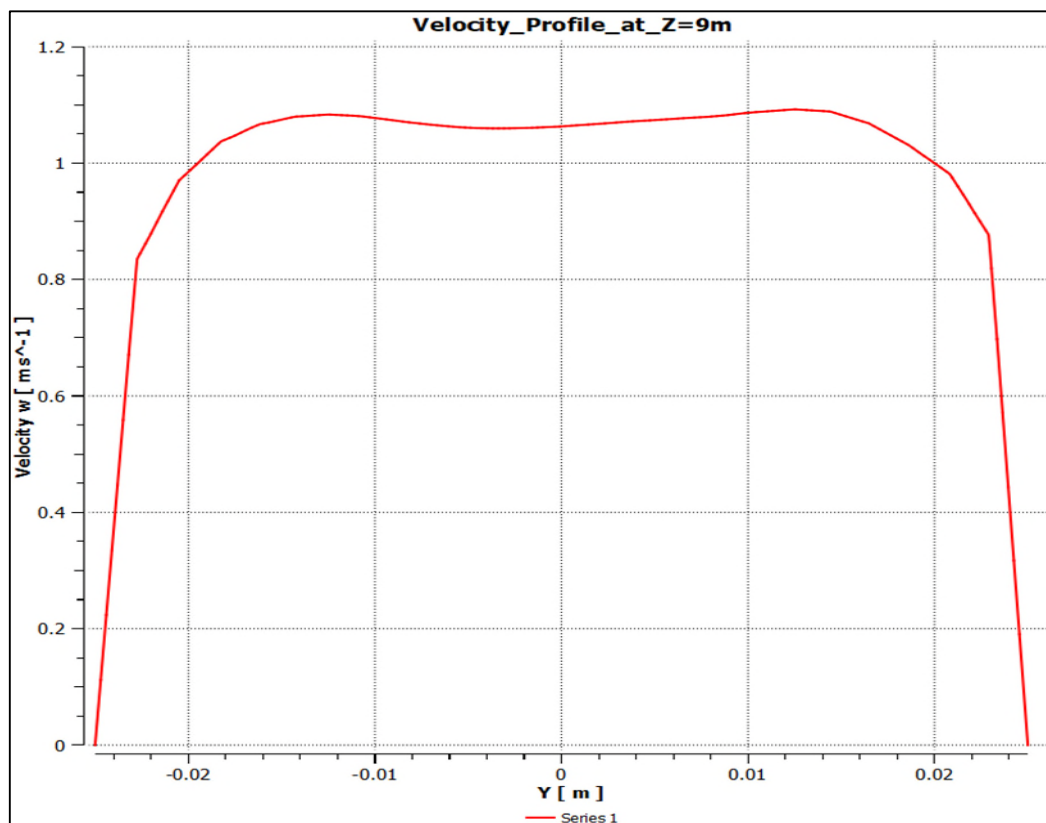


Figure 4-36 Velocity Profile at Z = 9 m with max velocity 1.1 m/s with coarse mesh (600000 hexahedral cells)

At $t = 0.05s$

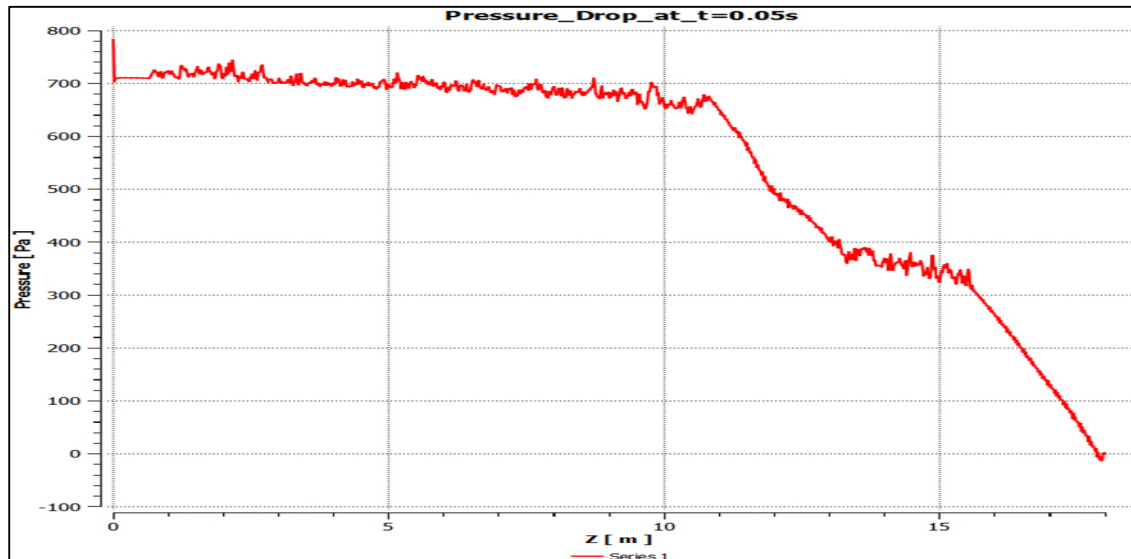


Figure 4-37 Pressure drops with fine mesh (500000 hexahedral cells) with max 790 [Pa]

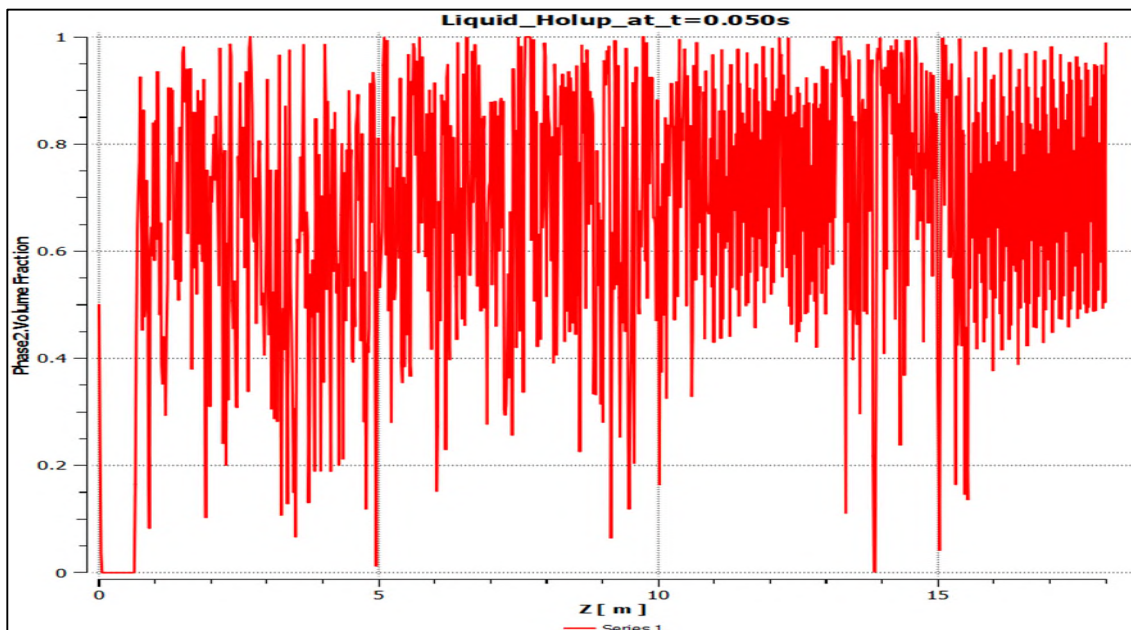


Figure 4-38 Liquid Holdup at pipeline with fine mesh (600000 hexahedral)

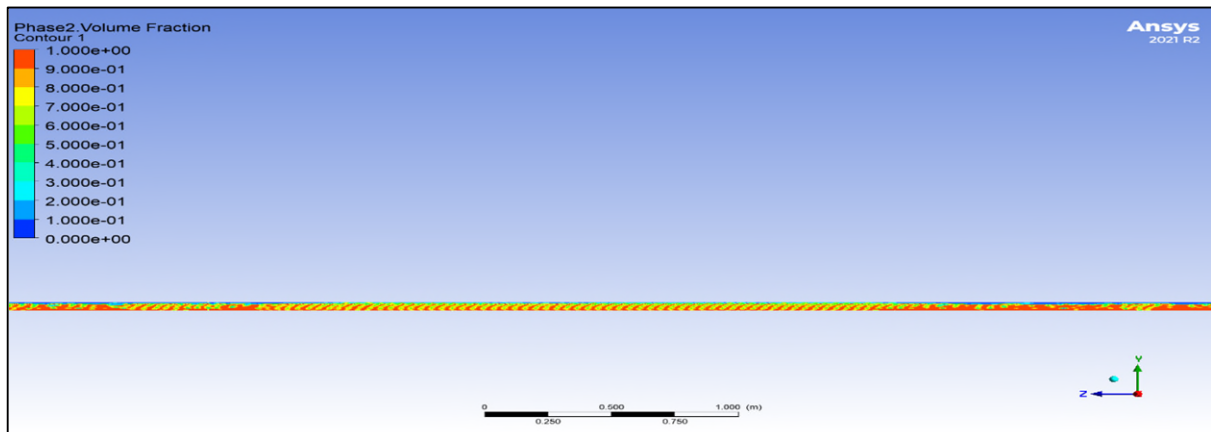


Figure 4-39 Water volume fraction contours (sluggish) with fine mesh (600000 hexahedral cells)

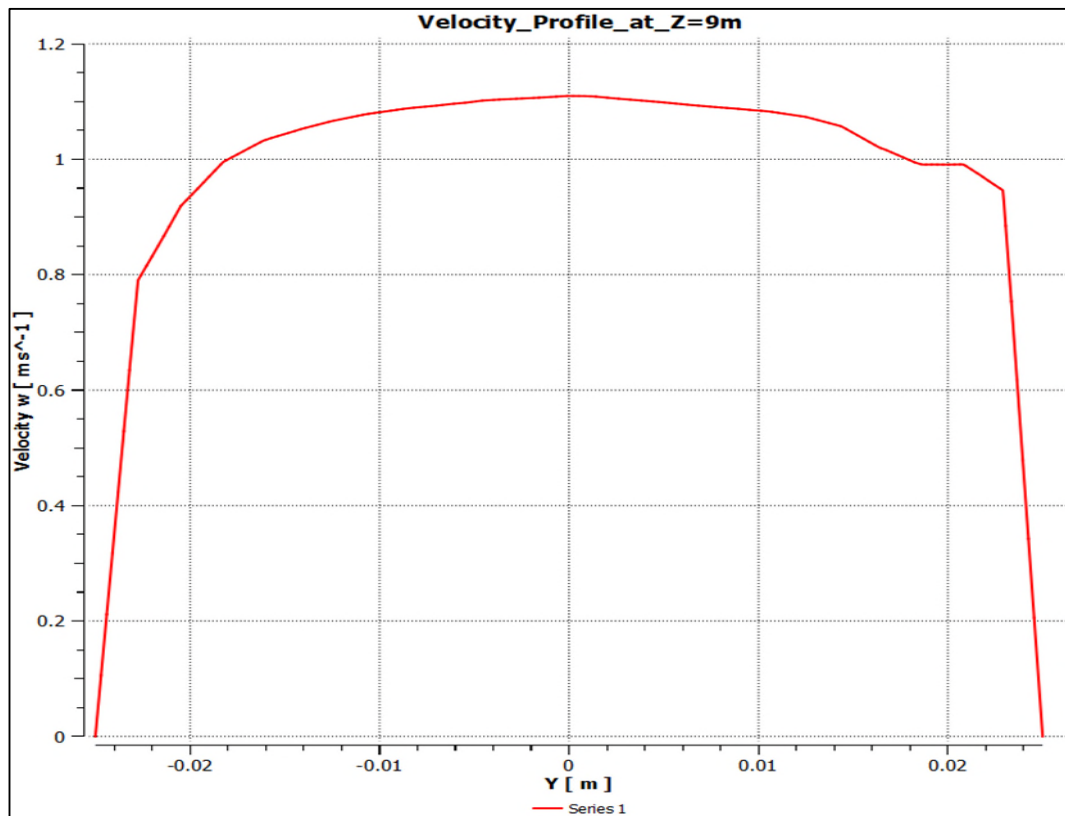


Figure 4-40 Velocity Profile at Z = 9m with max velocity 1.1 m/s with coarse mesh (600000 hexahedral cells)

4.3 Validation

4.3.1 Validation of the multiphase Isotropic model

The validation is done by using **case 1** that represents the pipe with diameter 16 mm and length 9 m, and particularly with $U_{SL} = 0.075m/s$ and $U_{SL} = 0.94m/s$ see picture below (Fig. 4.45) from experimentation. From this image, the data are extracted with Web-plot-Digitizer software and compared with data from numerical simulation on a spreadsheet see below (Fig. 4.43). Our first task is the Error analysis study that is made of absolute error and relative error.

$$\text{Absolute error} = |\text{Numerical value} - \text{Experimental value}| \quad (4-1)$$

$$\text{Relative error} = \frac{\text{Absolute error}}{\text{Experimental value}} \times 100 = [\quad]\% \quad (4-2)$$

Three tables have been created. In Table 1, absolute and relative errors are evaluated between times $t_1 = 15.70s$ and $t_2 = 15.78s$. In Table 2, absolute and relative errors are evaluated between $t_1 = 43.13s$ and $t_2 = 43.24s$ and finally in Table 3, absolute and relative errors are evaluated between $t_1 = 47.10s$ and $t_2 = 47.21s$.

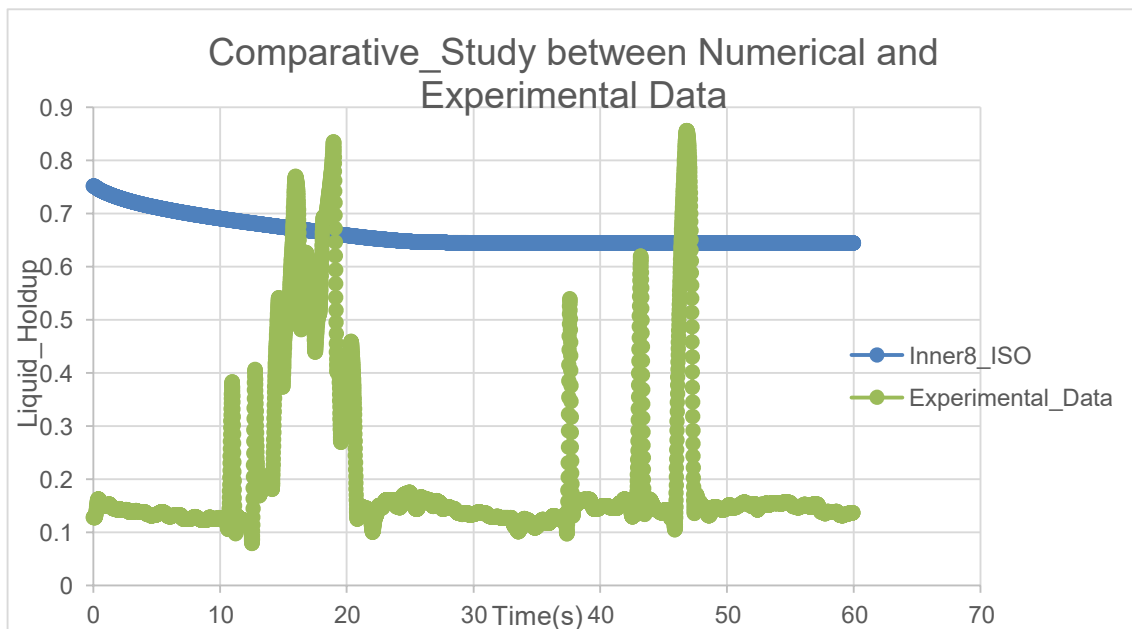


Figure 4-41 Comparison graph bet Numerical and Experimental data case 1

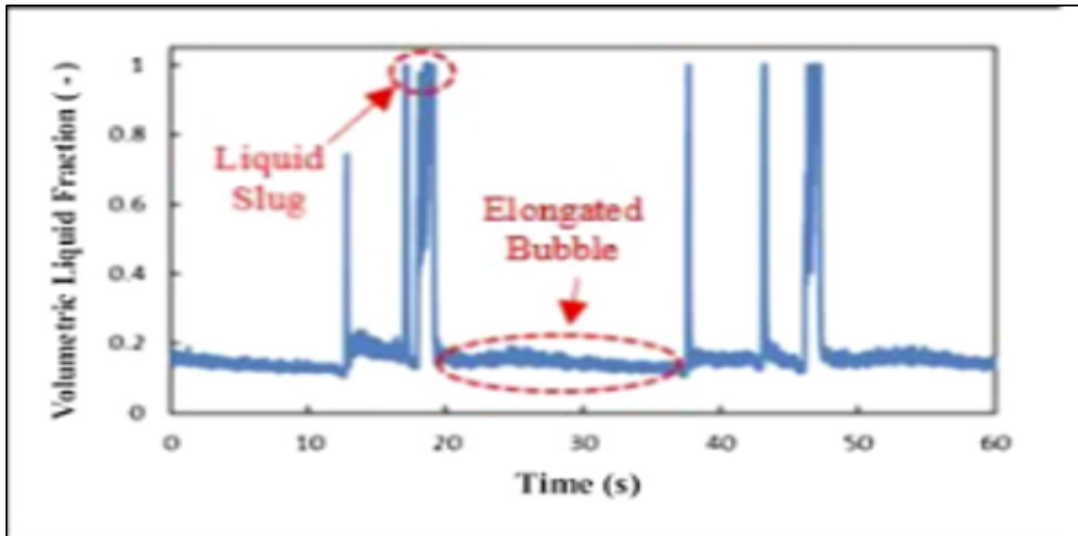


Figure 4-42 Graph of experimental data for case 1

Table 4-1 Data recorded between $t = 15.7s$ and $t = 15.78s$

Time (s)	Numerical data	Experimental data	Absolute Error	Relative Error (%)
15.7	0.671590472	0.642817417	0.028773055	4.476085102
15.71	0.671559919	0.647595879	0.02396404	3.700462133
15.72	0.671529358	0.652665078	0.01886428	2.890346127
15.73	0.671498785	0.658065835	0.01343295	2.041277506
15.74	0.671468218	0.663837961	0.007630256	1.149415456
15.75	0.671437649	0.669977336	0.001460313	0.217964497
15.76	0.671407491	0.67641984	0.005012349	0.741011469
15.77	0.671376876	0.683097037	0.011720161	1.71573884
15.78	0.671346275	0.68994049	0.018594214	2.695046132

Table 4-2 Data recorded between t = 43.13s and t = 43.24s

Time (s)	Numerical data	Experimental data	Absolute Error	Relative Error (%)
43.13	0.64411734	0.573558721	0.070558619	12.30189974
43.14	0.644117482	0.585973669	0.058143813	9.922598315
43.15	0.644117622	0.596530137	0.047587485	7.977381448
43.16	0.644117763	0.605177996	0.038939767	6.434432058
43.17	0.644117895	0.611867114	0.032250781	5.270879948
43.18	0.644118034	0.616547361	0.027570673	4.471785131
43.19	0.644118165	0.619168607	0.024949558	4.029525741
43.2	0.644118286	0.619680721	0.024437565	3.943573546
43.21	0.644118408	0.618033574	0.026084834	4.220617628
43.22	0.644118522	0.614177034	0.029941488	4.875058196
43.23	0.644118652	0.608060971	0.036057681	5.929944941
43.24	0.644118791	0.599635255	0.044483536	7.41843242

Table 4-3 Data recorded between t = 47.1s and 47.21s

Time(s)	Numerical data	Experimental data	Absolute Error	Relative Error (%)
47.1	0.644084822	0.749641579	0.105556757	14.08096349
47.11	0.644084628	0.738574511	0.094489883	12.79354773
47.12	0.64408443	0.726556324	0.082471894	11.35106686
47.13	0.644084235	0.713521924	0.069437689	9.731682616
47.14	0.644084039	0.699406214	0.055322175	7.909877513
47.15	0.644083833	0.684144098	0.040060266	5.855530391
47.16	0.644083629	0.667670482	0.023586852	3.532708595
47.17	0.64408343	0.649920269	0.005836838	0.898085282
47.18	0.644083229	0.630828363	0.013254866	2.101184289
47.19	0.644083027	0.610329669	0.033753358	5.530348581
47.2	0.644082826	0.588359092	0.055723734	9.471041501
47.21	0.644082626	0.564851706	0.079230921	14.02685339

4.3.1.1 Error Analysis

In Table 4-1, the relative error is decreasing from 4% to 0.21% between t = 15.70s and t = 15.75s and particularly at t = 15.75s and t = 15.76s, the errors are 0.21% and 0.74% with respectively 0.671437649 for numerical value and 0.669977336 for experimental value at t = 15.75s, and 0.671407491 for numerical value and 0.67641984 for experimental value at t = 15.76s.

In Table 4-3, the relative error is decreasing from 14% to 0.89% between t = 47.10s and t = 47.17s and particularly at t = 47.16s and 47.18s the errors are 3% and 2% respectively. At t = 47.17s, the error becomes 0.89% with respectively 0.64408343 for the numerical value and 0.649920269 for the experimental value.

In Table 4-2, the absolute error is decreasing slowly by 1 from 12% to 3% between t = 43.13s to 43.20s and particularly remains around 4% between t = 43.18 and 43.22s. At t = 43.20s the absolute error becomes the lowest value

3.94% with respectively 0.644118286 for the numerical value and 0.619680721 for the experimental value.

From the above analysis, we can conclude that at some time steps, numerical and experimental data values are very closed, showing a good validation.

4.3.2 Validation of the new Anisotropic multiphase model

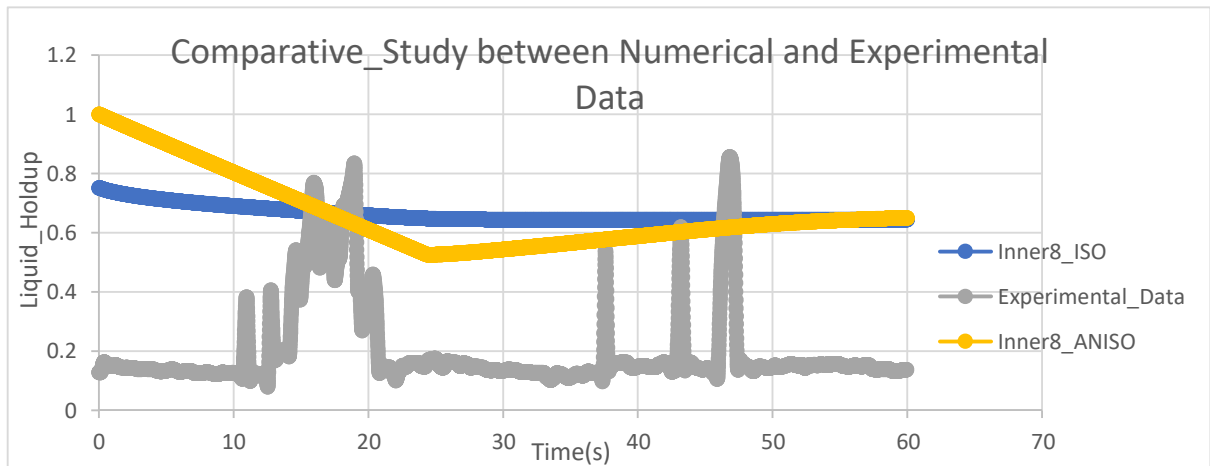


Figure 4-43 Comparison graph between Anisotropic and Experimental data case 1

Table 4-4 Data recorded between t = 43.13s and t = 43.25s

Time(s)	Numerical Data ANISO	Experimental Data	Absolute Error	Relative Error (%)
43.13	0.602156482	0.573558721	0.02859776	4.986021338
43.14	0.602200845	0.585973669	0.016227176	2.76926702
43.15	0.602245214	0.596530137	0.005715076	0.958053248
43.16	0.602289633	0.605177996	0.002888362	0.477274846
43.17	0.602333994	0.611867114	0.009533119	1.558037551
43.18	0.602378372	0.616547361	0.014168989	2.298118532
43.19	0.602422811	0.619168607	0.016745796	2.704561578
43.2	0.602467259	0.619680721	0.017213463	2.777795451
43.21	0.602511752	0.618033574	0.015521822	2.511485275
43.22	0.602556188	0.614177034	0.011620846	1.892100353
43.23	0.602600644	0.608060971	0.005460327	0.897990105
43.24	0.602645127	0.599635255	0.003009872	0.50195055
43.25	0.602689573	0.588849755	0.013839818	2.350313921

Table 4-4. that represents data recorded between t = 43.13s and t = 43.25s has been chosen because this timing is where the graph in the experimental data has a sluggish behaviour. Between t = 43.13 and t = 43.25s, the relative error is very low (4% to 0.47%). At some time steps the relative errors are, for (t = 43.16s) with value of 0.47%, showing 0.602289633 and 0.605177996 respectively as numerical and experimental data values and for (t = 43.24) with the value of 0.50%, showing 0.602645127 and 0.599635255 respectively as numerical and experimental data values. We can confirm from the above analysis, the anisotropic data values from the numerical simulations are good and validated.

4.4 Conclusion

Numerical studies have been conducted on the characteristics of the liquid film during the transition during to slug of air-water two-phase flow in horizontal pipes. The tested pipe diameters are 16 mm, 26 mm and 50 mm. The superficial velocities of air and water are in the range of $U_{SL} = 0.03 - 0.30 \text{ m/s}$ and $U_{SL} = 0.7 - 10 \text{ m/s}$ respectively. The behaviour of the interface is visualized by the plotting of the liquid Holdup, the water volume fraction contours and velocity profiles at different time steps.

From our studies through simulations, different flow patterns such as stratified Fig. (4-14.) and Fig. (4-15), pseudo slug Fig.(4-23.), slug Fig. (4-5) and Fig. (4-8) have been obtained.

Case 1.

The mesh dependency is done with Case 1. Two simulations are run with two different cells numbers (300000 cells) and (600000 cells) using $U_{SL} = 0.075 \text{ m/s}$ and $U_{SG} = 0.7 \text{ m/s}$ as respectively water and air superficial velocities. The results have been displayed in section 4.3.3.1 as mesh dependency.

From the results, two time-steps are considered ($t = 0.03 \text{ s}$) and ($t = 50 \text{ s}$) to see the flow behaviour including the pressure drop, the liquid Holdup, the velocity profiles, and the water volume fraction contours earlier and later in the simulation.

At $t = 0.03 \text{ s}$, the max pressure drops in Fig. (4-16) and (4-17) representing respectively the results for (300000 cells) and (600 000 cells) are respectively 500 [pa] and 250 [pa].

The water volume fraction contours in Fig. (4-18) and (4-19) representing respectively the results for (300000 cells) and (600000 cells) are both sluggish.

The max velocity profiles in Fig. (4-20) and (4-21) representing respectively the results for (300000 cells) and (600000 cells) are 0.27 m/s and 0.26 m/s respectively and the two velocity profiles are both parabolic, showing that the flow occupies the whole pipe in both cases (sluggish).

The liquid holdup in Fig. (4-22) and (4-23) representing respectively the results for (300000 cells) and (600000 cells) are showing huge fluctuations and reaching the max that is 1 in both cases.

At $t = 50s$, the max pressure drops in Fig. (4-24) and (4-25) representing respectively the results for (300000 cells) and (600 000 cells) are respectively 130 [pa] and 130 [pa]. they are both equal.

The water volume fraction contours in Fig. (4-26) and (4-27) representing respectively the results for (300000 cells) and (600000 cells) are both stratified.

The max velocity profiles in Fig. (4-28) and (4-29) representing respectively the results for (300000 cells) and (600000 cells) are 0.75 m/s and 0.75m/s respectively and the two velocity profiles are both parabolic, showing that the flow occupies the whole pipe in a stratified way, air on the top with max air velocity of 0.75 m/s and water at the bottom with max water velocity of 0.1 m/s.

From the above analysis, we can retain that even with different mesh configurations such as coarse or fine, the results do not vary or change too much. So, we can carry on working by choosing either coarse or fine mesh.

Case 2.

In case 2., we have a pipe with 26 mm of diameter and 10 m of length. The simulations are run with (500000 hexahedral cells) using $U_{SL} = 0.1m/s$ and $U_{SG} = 3.77m/s$ as respectively water and air superficial velocities. The results have been displayed in section 4.3.3.2 as Case 2.

From the results, three time-steps are considered ($t = 0.02s$) , ($t = 0.04s$) and ($t = 0.06s$) to see the flow behaviour including the pressure drop, the liquid Holdup and the velocity profiles following the results from the visual experimental data processing Fig. 7.

At ($t = 0.02s$), ($t = 0.04s$) and ($t = 0.06s$) the max pressure drops in Fig. (4-30), (4-34) and (4-38) representing respectively the results (500 000 cells) of the simulations are 3900 [pa], 1500 [pa] and 490 [pa].

At $t = 0.02s$, the flow behaviour is sluggish from Fig. (4-31), (4-32) and (4-33), liquid holdup showing huge fluctuations, sometime reaching 1 Fig.(4-31), water volume fraction contours changing rapidly Fig.(4-32) and the velocity profile Fig. (4-33) being parabolic, showing that pipe is full.

At $t = 0.04s$, the flow behaviour is still sluggish but pseudo slug from Fig. (4-35), (4-36) and (4-37), liquid holdup showing huge fluctuations, sometime reaching 1 Fig.(4-35), water volume fraction contours changing rapidly with aeration at some places Fig.(4-36) and the velocity profile Fig. (4-37) still parabolic, showing the growth of the air or stratification in pipe.

At $t = 0.06s$, the flow behaviour is still sluggish but highly aerated slug from Fig. (4-39), (4-40) and (4-41), liquid holdup showing fluctuations, sometime between 0 and 0.8 Fig. (4-39), water volume fraction contours changing rapidly with huge aeration everywhere Fig. (4-40) and the velocity profile Fig. (4-41), showing a huge growth of the air or stratification in pipe.

Case 3.

In case 3., we have a pipe with 50 mm of diameter and 18 m of length. The simulations are run with (600000 hexahedral cells) using $U_{SL} = 0.075m/s$ and $U_{SG} = 3.77m/s$ as respectively water and air superficial velocities. The results have been displayed in section 4.3.3.3 as Case 3.

From the results, three time-steps are considered ($t = 0.02s$), ($t = 0.03s$) and ($t = 0.05s$) to see the flow behaviour including the pressure drop, the liquid Holdup and the velocity profiles following the results from the visual experimental data processing Fig. 9.

At $t = 0.02s$, the flow behaviour is sluggish from Fig. (4-43), (4-44) and (4-45), liquid holdup showing huge fluctuations, sometime reaching 1 Fig.(4-43), water volume fraction contours changing rapidly Fig.(4-44) and the velocity profile Fig. (4-45) being parabolic, showing that pipe is full.

At $t = 0.03s$, the flow behaviour is still sluggish from Fig. (4-47), (4-48) and (4-49), liquid holdup showing huge fluctuations, sometime reaching 1 Fig.(4-47), water volume fraction contours changing rapidly to show successive slugs Fig.(4-48) and the velocity profile Fig. (4-49) still parabolic, showing no stratification in pipe.

At $t = 0.05s$, the flow behaviour is still sluggish from Fig. (4-51), (4-52) and (4-53), liquid holdup showing huge fluctuations, sometime reaching 1 Fig.(4-51), water volume fraction contours changing rapidly to show successive slugs with some aeration Fig.(4-52) and the velocity profile Fig. (4-53) still parabolic, showing the beginning of stratification in pipe.

In these three cases, the initialisation of slug flows is done earlier at $t = 0.02s$, $0.03s$, $0.04s$, $0.05s$ and $0.06s$. The flow can become either stratified, stratified wavy, pseudo slug and slug depending on the time steps.

Chapter 5: Conclusion and recommendations for future work

5.1 Conclusions

The overall aim of our project was a theoretical and a computational study or investigation of hydrodynamic slug flows in horizontal pipes. A particular focus was made on multiphase flow instabilities and turbulence models leading to creation of a new anisotropic multiphase flow mathematical model.

Chapter 2 conducts a review of the multiphase flow models. As a result, these multiphase models can be divided into six classes, including hybrid models, moving grid methods, interface tracking methods, quasi-multiphase models, and multi-fluid models. It will be beneficial for researchers to choose a model carefully, based on the governing phenomena relevant to their objectives and the availability of computer resources, as each model has pros and cons.

Another review on hydrodynamic slug flow modelling in chapter 2, reveals that slug flow mechanisms are made of interface waves formation, waves growth and pipe bridging events, slug growth, gas entrainment and stratified layer formation. Kelvin-Helmholtz instability and slug flow initiation criterion are very important in the development of slug flows. Due to inherent intermittency behaviour of the slug flow, its modelling has become a real challenge as you must take into account dispersed gas bubbles and large gas bubble region. Two main slug modelling approaches such as steady state models Wallis (1969) and transient models Taitel & Duckler (1976) have been adopted.

A review of turbulence modelling was done in chapter 2. As a result, turbulent modelling approaches can be viewed or classified as Direct Numerical Simulation (DNS), Reynolds-Averaged Navier-Stokes (RANS) models and Large-Eddy Simulation models (LES). Each approach has its advantages and disadvantages, allowing researchers the choice, based on the problem to be solved, and the computational resources available.

A further investigation about the Reynolds averaged Navier-Stokes (RANS) models shows that this group can be divided into isotropic and anisotropic

models. Isotropic turbulence models postulate that Reynolds stresses are proportional to mean velocity gradients, or, put another way, that the deviatoric Reynolds stress is exactly proportional to the local mean rate of strain. Most researchers including Taylor (1915), Prandtl (1925) and Von Karman used the isotropic models known as eddy viscosity-based RANS to develop their work. Models including mixing length, Spalart – Allmaras, K- ω and k- ε are all isotropic turbulence models. However, isotropic turbulence RANS models have some deficiencies for turbulent flows such as hydrodynamic slug in horizontal pipelines that display intermittent, swirling, and 3D behavior. As a result, the use of their own dynamic transport equations to explicitly determine Reynolds stresses is preferred. This new turbulent model is known as anisotropic turbulent model. A new term $\mu_\theta \theta G$ is added to the isotropic turbulent description to depict the anisotropic turbulent model as suggested by Laszlo Könözy (2019; 2021).

Chapter 3 conducts a review of the new approach for modeling multiphase anisotropic turbulence. In this new approach, a new term is added to the Reynolds stress tensor τ_m^{RA} in the momentum equation of the VOF model. This new term is known as the anisotropic modification $\mu_{\theta_m} \rho_m C_\mu \frac{k^2}{\varepsilon} \cdot |\nabla \times \mathbf{u}_m| \cdot \underline{\underline{G}}$, and the Reynolds stress tensor will become the new anisotropic multiphase mixture Reynolds stress tensor Eq (3-3). To complete the modelling of the new approach, two equations have been added. The turbulent kinetic energy equation k for anisotropic multiphase turbulent mixture flows Eq. (3-8) and the dissipation of turbulent kinetic energy transport equation Eq (3.10).

Chapter 4 focuses on the validation of the work. Numerical solutions have been used to run the simulations with experimental data from a published paper (See Appendix D). ANSYS Fluent 2022R2 computer code has been used for the design, meshing, set up and running different simulations. Two types of simulations are run. One for isotropic turbulence model with VOF and another one for anisotropic turbulence model where I add a new term to the VOF model by writing a user-defined function UDF C programming (See Appendix B). From our studies through simulations, different flow patterns such as stratified Fig. (4-14.) and Fig. (4-15), pseudo slug Fig. (4-23.), slug Fig. (4-5) and Fig. (4-8) have

been obtained. For the three cases considered, the pressure drop, liquid holdup and velocity profiles are plotted and compared. The results show that the transition to slug flow formation happens early during the simulations at $t = 0.02s$ on Fig. (4-31), Fig. (4-32) and Fig. (4-33). Also, at $t = 0.04s$ on Fig. (4-35), Fig. (4-36) and Fig. (4-37).

5.2 Recommendations and Future work

Further work and simulations should take place to improve the results accuracy regarding the velocity profiles, the pressure drop and liquid holdup. As our simulations are run on hexahedral mesh, it would be good to run some simulations on tetrahedral see to see the results.

A further parametric study of the multiphase anisotropic deviatoric similarity tensor G includes in the new hypothesis $(\mu_{\theta_m} \theta_m \cdot \underline{G})$ should be carried out for other multiphase flow problems relevant to industrial applications such as slurries.

The new hypothesis on the anisotropic multiphase Reynolds stress tensor proposed in this PhD thesis can be further extended to other multiphase RANS models.

More validation test cases can be performed for different fluid flow problems where experimental data is available in the literature.

Bibliography

- A. Theodorakakos and G. Bergeles: Metall. Mater. Trans. B, 29 (1998), 1321.
- A. Gosman and E. Loannides: J. Energy, 7 (1983), 482
- A. Tomiyama, N. Shimada and H. Asano: 4th ASME JSME Joint Fluids Engineering Conf., ASME, New York, (2003), 501.
- A. Tiwari and J. Abraham: Phys. Rev. E, 74 (2006), 056701.
- Abdul-Majeed G.H., 2000. Liquid slug hold-up in horizontal and slightly inclined two-phase slug flow. *Journal of Petroleum Science and Engineering* 27, 27-32.
- Andreussi, P., Paglianti, A., Vatiatas, N., Minervini, A., and Bendiksen, K., 1988. Analysis of slug flow in near horizontal and horizontal pipes. *European two-phase flow Group meeting, Brussels, Belgium, 30 May to 1 June. Paper B2.*
- Andreussi, P., Bendiksen, K., Nydal, O.J., 1993, Void distribution in slug flow. *Int. J. Multiphase Flow* 19, 817-828.
- Ansari, M.R., 1998. Dynamical behaviour of slug initiation generated by short waves in two-phase air-water stratified flow. *ASME HTD* 361, 289-295.
- Aris R (1962) Vectors tensors and basic equations of fluid mechanics. Dover Publications Inc, New York. ISBN 0-486-66110-5
- B. G. Thomas, X. Huang and R. C. Sussman: Metall. Mater. Trans. B, 25 (1994), 527.
- Baker, A.J., *Finite element computational fluid mechanic*, McGraw-Hill, New York, 1983.
- Baliga, R.B. Control-volume finite element method for 2D fluid flow and heat transfer, in W.J. Minkowycz, E.M. Sparrow (eds.), *Advances in Numerical Heat Transfer*, Chap. 3, pp. 97-135, Taylor and Francis, New York, 1997.
- Baliga, R.B., Pantaker, S.V., A control-volume finite element method for 2D fluid flow and heat transfer. *Numer. Heat Transfer*, 6, 245-261 (1983).
- Bardina, J. ferziger, J.H. and Reynolds, W.C. (1980). Improved Subgrid-scale models for large eddy simulation. AIAA Paper 80-1357.
- Barnea, D., Brauner, N., 1985. Hold-up of the liquid slug in two-phase intermittent flow. *Int. J. Multiphase Flow* 11, 444-453.
- Bendiksen, K.H., Espedal, M., 1992. Onset on slugging in horizontal gas-liquid pipe flow. *Int. J. Multiphase Flow* 18, 237-247.
- Bendiksen, K.H., Brandt, I., Jacobsen, K.A., Pauchon, C., 1987. Dynamic simulation of multiphase transportation systems. In: *Multiphase Flow Technology and Consequences for Field Development Forum, Stavanger, Norway*
- Bendiksen, K.H., Malnes, D., Moe, R., Nuland, S., 1991. The dynamic two-fluid model OLGA: theory and application. *SPE Prod. Eng.* 6, 171-180
- Bendiksen, K.H., Malnes, D., Nydal, O.J., 1996. On the modelling of slug flow. *Chem. Eng. Commun.* 141-142, 71-102.
- Benjamin T.B. (1968). "Gravity currents and related phenomena." *J. Fluid Mech.* **31, part 2**,: 209-248.
- Black, P.S., Daniels, L.C., Hoyle, N.C., Jepson, W.P., 1990. Studying transient multiphase flow using the pipeline analysis code (PLAC). *J. Energy Resour. Technol.* 112, 25-29

- Bonizzi, M., Issa, R.I., Kempf, M.H.W., 2001. Modelling of gas entrainment in horizontal slug flow. *In: International Conference on Multiphase flow, New Orleans, USA.*
- Boure, J., Reocreux, M., 1972. General equations of two phase flow. *In: All Union Heat Mass Transfer Conference. Minsk, USSR.*
- Boussinesq J (1877) Théorie de l'Écoulement tourbillant. Mem Présentés par Divers Savants Acad Sci Inst Fr 23:46–50
- Bradshaw P (1967) The turbulence structure of equilibrium boundary layers. *J Fluid Mech* 29(Part 4):625–645.
- Bradshaw P, Ferriss DH, Atwell NP (1967) Calculation of boundary-layer development using the turbulent energy equation. *J Fluid Mech* 28(Part 3):593–616.
- Brill, J.P., Schmidt, Z., Coberly, W.A., Herring, J.D., Moore, D.W., 1981. Analysis of two-phase tests in large-diameter flow lines in Prudhoe Bay Fields. *SPE Journal, June 1981, 363-378.*
- C. T. Crowe, J. D. Schwarzkopf, M. Sommerfeld and Y. Tsuji: *Multiphase Flows with Droplets and Particles*, CRC Press, Boca Raton, FL, (2011).
- C. Pfeiler, M. Wu and A. Ludwig: *Mater. Sci. Eng. A*, 413–414 (2005), 115.
- C. W. Hirt, A. A. Amsden and J. L. Cook: *J. Comput. Phys.*, **14** (1974), 227.
- C. W. Hirt and B. D. Nichols: *J. Comput. Phys.*, **39** (1981), 201.
- Chaudhary, R., B. Rietow and B. G. Thomas: *Materials Science and Technology Conf. and Exhibition 2009, MS&T, AIST, Warrendale, PA, (2009), 1090.*
- Chung, T.J., *Finite element computational fluid mechanic*, McGraw-Hill, New York, 1978
- Chung, T. (2010). *Computational Fluids Dynamics*. 2nd Ed, Cambridge University Press.
- Cleary, P.W., M. Prakash, J. Ha, N. Stokes and C. Scott: *Prog. Comput. Fluid Dyn.*, **7** (2007), 70.
- Czibere T (2001) Three dimensional stochastic model of turbulence. *J Comput Appl Mech* 2(1):7–20.
- Czibere T (2006) Calculating turbulent flows based on a stochastic model. *J Comput Appl Mech* 7(2):155–188.
- D. Geng, H. Lei and J. He: *Metall. Mater. Trans. B*, 41 (2010), 234.
- D. Lycett-Brown and K. H. Luo: *Phys. Rev. E*, 91 (2015), 023305.
- Davis, R.M., Taylor, G.I., 1950. The mechanics of large bubbles rising through extended liquids and through liquids in tubes. *Proc Roy. Soc.*, *200A*, 375-390.
- Davidson PA (2004) *Turbulence. An introduction for scientists and engineers.* Oxford University Press Inc, New York. ISBN 978-0-19-852949-1
- Davies S.R. (1992). *Studies of two-phase intermittent flow in pipelines.* Department of Chemical Engineering and Chemical Technology, Imperial College London.
- De Henau, V., Raithby, G.D., 1995b. A transient two-fluid model for the simulation of slug flow in pipelines: II. Validation. *Int. J. Multiphase Flow* **21**, 351-363.
- Dukler, A.E., Hubbard, M.G., 1975. A model for gas-liquid slug flow in horizontal and near horizontal tubes. *Ind. Eng. Chem. Fund.* **14**, 337-345.

- Dukler, A.E., Moalem, M.D., Brauner, N., 1985. A physical model for predicting the minimum slug length. *Chem. Eng. Sci.*, 40, 1379-1385.
- Dukler, A.E., Fabre, J., 1992. Gas-liquid slug flow-knots and loose ends. In: *3rd International Workshop Two-phase Flow Fundamentals, Imperial College*.
- E. Aulisa, S. Manservigi, R. Scardovelli and S. Zaleski: *J. Comput. Phys.*, 225 (2007), 2301.
- Espedal, M., Bendiksen, K.H., 1989. Onset of instabilities and slugging in horizontal and near-horizontal gas-liquid flow. *European Two-Phase Flow Group Meeting, Paris, May 9-June 1, paper G4, pp. 1-30*.
- F. H. Harlow and A. A. Amsden: *J. Comput. Phys.*, 17 (1975), 19.
- Fabre, J., Line, A., 1992. Modelling of two-phase slug flow. *Ann. Rev. Fluid Mech.* 24, 21-46.
- Fabre, J., Ferschneider, G., Masbernat, L., 1983. Intermittent gas liquid flow modelling in horizontal or weakly inclined pipes. In: *International Conference on Physical Modelling for Multiphase Flow, Coventry, pp. 233-254*.
- Fabre, J., Line, A., Peresson, L., 1992. Two fluid/two flow pattern model for transient gas liquid flow in pipes. In: *4th International Conference on Multiphase Flow, Nice, pp. 269-284*.
- Fagundes Netto, J.R., Fabre, J., Peresson, N.L., 1999. Shape of long bubbles in horizontal slug flow. *Int. J. Multiphase Flow* 25, 1129-1160.
- Fan, Z., Lusseyran, F., Hanratty, T.J., 1993a. Initiation of slug in horizontal gas-liquid flows. *AIChE J.* 39, 1741-1753.
- Fan, Z., Ruber, Z., Hanratty, T.J., 1993b. Pressure profiles for slugs in horizontal pipes. In: *J. Multiphase Flow* 19, 421-437.
- Ferre, D., 1979. Ecoulements diphasiques a poches en conduite horizontale. *Rev. Inst. Fr. Pet.*, 113-142.
- Ferschneider, G., 1983. Ecoulements gaz-liquide a poches et a bouchon en conduite. *Rev. Inst. Fr. Pet.* 38, 153-182.
- Ferziger, J.H. and Peric, M., *Computational Methods for Fluids Dynamics*, 3rd ed., Springer, Berlin, (2002).
- Fletcher, C.A.J., *Computational techniques for fluid dynamics*, Vol. I., Springer, Berlin, 1991.
- G. A. Panaras, A. Theodorakakos and G. Bergeles: *Metall. Mater. Trans. B*, 29 (1998), 1117.
- G. Tryggvason, B. Bunner, A. Esmaeeli, D. Juric, N. Al-Rawahi, W. Tauber, J. Han, S. Nas and Y. J. Jan: *J. Comput. Phys.*, 169 (2001), 708.
- G. H. Yeoh and J. Tu: *Computational Techniques for Multiphase Flows*, Elsevier, Amsterdam, (2009).
- Gardner G.C. (1979). "Onset of Slugging in Horizontal Ducts." *International Journal of Multiphase Flow* 5(1): 201.
- Germano, M., Piomelli, U., Moin, P. and Cabot, W.H. (1991). A Dynamic Subgrid-scale eddy viscosity model. *Physics Fluids A*, 3, 1760-1765.
- Gomez, L.E., Shoham, O., Schmidt, Z., Chokshi, R.N., Brown, A., Northug, T., 1999. A unified Mechanistic model for steady state two-phase flow in wellbores and pipelines. In: *SPE Annual Technological Conference Exhibition, Houston, Texas, USA, SPE 56520, pp. 307-320*.

- Gomez, L.E., Shoham, O., Taitel, Y., 2000. Prediction of slug liquid hold-up: horizontal to upward vertical flow. *Int. J. Multiphase Flow*, 26, 517-521.
- Gregory, G.A., Scott, D.S., 1969. Correlation of fluid slug velocity and frequency in horizontal co-current gas-liquid slug flow. *AIChE Journal*, 15(6), 933-935.
- Gregory, G.A., Nicholson, M.K., Aziz, K., 1978. Correlation of the liquid volume fraction in the slug for horizontal gas-liquid slug flow. *Int. J. Multiphase Flow*, 4, 33-39.
- Greskovish, E.J., Shrier, A.L., 1971. Pressure drop and liquid hold-up in horizontal slug flow. *AIChE Journal*, 17(5), 1214-1219.
- H. Bai and B. G. Thomas: Metall. Mater. Trans. B, 32 (2001), 707.
- H. Yu and M. Zhu: ISIJ Int., 48 (2008), 584.
- Hale C.P. (2000). slug formation, growth and collapse. Department of Chemical Engineering and Chemical Technology, Imperial College London.
- Henderson, R.D., Karniadakis, G.E., Unstructured spectral element method for simulation of turbulent flows, *J. Comput. Phys.*, 122, 191-217 (1995).
- Heywood, N.I., Richardson, J.F., 1979. Slug flow of air-water mixtures in horizontal pipe: determination of liquid hold-up by γ -ray absorption. *Chem. Eng. Sci.*, 34, 17-30.
- Hill, T.J., Wood, D.G. 1990. A new approach to the prediction of slug frequency. *65th Annual Technical Conference and Exhibition of the Society of Petroleum Engineers, September 23-26, New Orleans, USA, SPE 20629, 141-149.*
- Ishii, M., 1975. Thermo-Fluid Dynamic Theory of two-phase. *Eyrolles, Paris*.
- Ishii, M., Mishima, K., 1984. Two-fluid model and hydrodynamic constitutive relations. *Nucl. Eng. Des.*, 107-126.
- Ishii M. (1982). Wave Phenomena and Two-Phase Flow Instabilities. Handbook of Multiphase Systems, Hemisphere Publishing Corp.
- Issa, R.I., Woodburn, P., 1998. Numerical prediction of instabilities and slug formation in horizontal two-phase flows. *In: 3rd International Conference on Multiphase flow, ICMF98, Lyon, France.*
- J. Smagorinsky: Mon. Weather Rev., 91 (1963), 99.
- J. U. Brackbill, D. B. Kothe and C. Zemach: *J. Comput. Phys.*, 100 (1992), 335.
- Jepson, W.P., 1989. Modelling the transition to slug flow in horizontal conduit. *Can. J. Chem. Eng.* 67, 731-740.
- K. Jin, P. Kumar, S. P. Vanka and B. G. Thomas: *Phys. Fluids*, 28 (2016), 093301.
- K. E. Wardle and H. G. Weller: *Int. J. Chem. Eng.*, 2013 (2013), 1.
- K. Takatani, Y. Tanizawa, H. Mizukami and K. Nishimura: *ISIJ Int.*, 41 (2001), 1252.
- Karamcheti K (1967) Vector analysis and cartesian tensors with selected applications. Holden-Day, San Francisco, California, USA
- Khosla, P.K., Rubin, S.G., A diagonally dominant second-order accurate implicit scheme, *Computers Fluids*, 2, 207-209 (1974).
- Kordyban, E.S., Ranov, T., 1970. Mechanism of slug flow formation in horizontal tubes. *J. Basic Eng. TASME*, 857-864.
- Kordyban E.S. (1977). "The Transition to Slug Flow in the Presence of Large Waves." *International Journal of Multiphase Flow* 3: 603.

- Kordyban E. (1985). "Some details of developing slugs in horizontal 2-phase flow." *AIChE J.* **31**: 802–806.
- Krepper, D. Lucas, T. Frank, H.-M. Prasser and P. J. Zwart: *Nucl. Eng. Des.*, **238** (2008), 1690.
- László Könözsy, (2019) "A New Hypothesis on the Anisotropic Reynolds Stress Tensor for Turbulent Flows, Volume I: Theoretical Background and Development of an Anisotropic Hybrid k-omega Shear-Stress Transport/Stochastic Turbulence Model", Fluid Mechanics and Its Applications Series, Vol. 120, *Springer International Publishing, Springer Nature Switzerland AG*, 2019, Hardcover ISBN: 978-3-030-13542-3, DOI: 10.1007/978-3-030-13543-0
- László Könözsy, "A New Hypothesis on the Anisotropic Reynolds Stress Tensor for Turbulent Flows, Volume II: Practical Implementation and Applications of an Anisotropic Hybrid k-omega Shear-Stress Transport/Stochastic Turbulence Model", Fluid Mechanics and Its Applications Series, Springer International Publishing, Volume 125, 2021, Hardcover ISBN: 978-3-030-60602-2, DOI: 10.1007/978-3-030-60603-9.
- Launder, B. E., & Sharma, B. I. (1974). Application of the energy-dissipation model of turbulence to the calculation of flow near a spinning disc. *Letters in Heat and Mass Transfer*, **1**, 131–138.
- Lesieur, M. (2008), *Turbulence* 4th Ed. Sriinger, Berlin.
- L. B. Lucy: *Astron. J.*, **82** (1977), 1013.
- L. Zhang, Y. Wang and X. Zuo: *Metall. Mater. Trans. B*, **39** (2008), 534.
- Lilek, Z., Peric, M., A fourth-order finite volume method with co-located variable arrangement, *Computers Fluids*, **24**, 239-252 (1995).
- Lin, P.Y., Hanratty, T.J., 1986. Prediction of the initiation of slugs with linear stability theory. *Int. J. Multiphase Flow* **12**, 79-98.
- Line, A., 1983. Ecoulement intermittent de gas et liquide en conduite verticale. *These Institute Nationale Polytechnique, Toulouse*.
- Lunde, O., Asheim, H., 1989. An experimental study of slug stability in horizontal flow. *In: 4th International Multiphase Flow Conference, Nice France, 19-21 June, pp. 419-430*.
- M. Ishii, T. C. Chawla and N. Zuber: *AIChE J.*, **22** (1976), 283.
- M. Ishii and T. Hibiki: *Thermo-fluid Dynamics of Two-phase Flow*, Springer Science+Business Media, New York, (2006).
- M. R. Maxey: *Phys. Fluids*, **30** (1987), 1915.
- M. R. Swift, W. R. Osborn and J. M. Yeomans: *Phys. Rev. Lett.*, **75** (1995), 830.
- M. Sussman and E. G. Puckett: *J. Comput. Phys.*, **162** (2000), 301.
- M. van Sint Annaland, N. G. Deen and J. A. M. Kuipers: *Chem. Eng. Sci.*, **60** (2005), 2999.
- Malnes, D., 1982. Slug flow in vertical, horizontal and inclined pipes. *Report IFE/KR/E-83/002, Inst. For Energy Technology. Kjeller, Norway*.
- Masson et al., Co-located equal-order control-volume finite element method for 2D axisymmetric incompressible fluid flow, *Int. J. Numer. Methods fluids*, **18**, 1-26 1994.
- Minato A., Ikeda T. & Naitoh M. (1986). "Mechanistic Model of Slugging Onset in Horizontal Circular Tube." *Journal of Nuclear Science and Technology* **23**: 761-768.

- Manolis, I.G., 1995. High Pressure gas-liquid slug flow. *PHD thesis, Department of Chemical Engineering and Chemical Technology, Imperial College of Science, Technology & Medicine, UK.*
- Marcano, R., Chen, T.X., Sarica, C., Brill, J.P., 1998. A study of slug characteristics for two-phase horizontal flow. *SPE*. 39856, 213-219.
- Menter FR (1992) Improved two-equation $k-\omega$ turbulence models for aerodynamic flows. NASA Tech Memo 103975:1–31.
- Menter FR (1994) Improved two-equation eddy-viscosity turbulence models for engineering applications. *AIAA J* 32(8):1598–1605.
- Menter FR, Kuntz M, Langtry R (2003) Ten years of industrial experience with the SST turbulence model. In: 4th Internal symposium, turbulence, heat and mass transfer, 2003, Antalya, Turkey, vol 4, Begell House, Inc., New York, Wallingford, pp 625–632. ISBN 1567001963.
- Mishima, K., Ishii, M., 1980. Theoretical prediction of onset of horizontal slug flow. *J. Fluids Eng.* 102, 441-444.
- Moin P. and K. Mahesh: *Annu. Rev. Fluid Mech.*, **30** (1998), 539.
- Moissis, R., Griffith, P., 1962. Entrance effects in two-phase slug flow. *J. Heat Transfer* 84, 29-39.
- Muzaferija,
- N. Bessho, R. Yoda, H. Yamasaki, T. Fujii, T. Nozaki and S. Takatori: *ISIJ Int.*, 31 (1991), 40.
- N. Kubo, T. Ishii, J. Kubota and T. Ikagawa: *ISIJ Int.*, 44 (2004), 556.
- N. A. Patankar and D. D. Joseph: *Int. J. Multiph. Flow*, 27 (2001), 1685.
- Nicholson, M.K., Aziz, K., Gregory, G.A., 1978. Intermittent two-phase flow in horizontal pipes: Predictive models. *Can. J. Chem. Eng.*, 56, 653-663.
- Nicklin, D.J., Wilkes, J.O., Davidson, J.F. 1962. Two-phase flow in vertical tubes. *Trans. Inst. Chem. Eng.* 40, 61-68.
- Nydal, O.J., Andreussi, P., 1991. Gas entrainment in long liquid slug advancing in a near horizontal pipe. *Int. J. Multiphase Flow* 17, 179-189.
- Nydal, O.J., Pintus, S., Andreussi, P., 1992. Statistical characterization of slug flow in horizontal pipes. *Int. J. Multiphase Flow*. 18(3), 439-453.
- Nydal, O.J., Banerjee, S., 1996. Dynamic slug tracking simulation for gas-liquid flow in pipelines. *Chem. Eng. Commu.* 141-142, 13-39.
- O. J. Ilegbusi and J. Szekely: *ISIJ Int.*, 34 (1994), 943.
- Oden, J.T., *Finite elements of non-linear continua*, McGraw-Hill, New York, 1972.
- Patankar, S.V., *Numerical heat transfer and fluid flow*, McGraw-Hill, New York, 1980.
- P. A. Cundall and O. D. L. Strack: *Géotechnique*, 29 (1979), 47
- P. E. R. Lopez, P. N. Jalali, J. Björkvall, U. Sjöström and C. Nilsson: *ISIJ Int.*, **54** (2014), 342.
- P. Kumar and S. Vanka: *Int. J. Multiph. Flow*, **77** (2015), 32.
- Pope, S.B. (2000), *Turbulence. Flows*. Cambridge.
- Prakash, M., P. W. Cleary, J. Grandfield, P. Rohan and V. Nguyen: *Prog. Comput. Fluid Dyn.*, **7** (2007), 101.
- Prandtl, L. (1925). Bericht über Untersuchungen zur ausgebildeten Turbulenz. *ZAMM*, 5, 136.

- Q. Yuan, T. Shi, S. P. Vanka and B. G. Thomas: Computational Modeling of Materials, Minerals and Metals, The Minerals, Metals, and Materials Society, Warrendale, PA, (2001), 491.
- R. Clift, J. R. Grace and M. E. Weber: Bubbles, Drops, and Particles, Dover Publication, Minesota, NY, (2005).
- R. Liu: Ph.D. thesis, University of Illinois at Urbana Champaign, (2014), <https://www.ideals.illinois.edu/handle/2142/72988>, (accessed 2017-05-27).
- R. H. Pletcher, J. C. Tannehill and D. Anderson: Computational Fluid Mechanics and Heat Transfer, CRC Press, Boca Raton, FL, (2012).
- R. D. Groot and P. B. Warren: J. Chem. Phys., 107 (1997), 4423.
- R. Liu, B. G. Thomas, J. Sengupta, S. D. Chung and M. Trinh: ISIJ Int., 54 (2014), 2314.
- R. Clift, J. R. Grace and M. E. Weber: Bubbles, Drops, and Particles, Dover Publicat
- Rodi, W. Turbulence Models and Their Application in Hydraulics, CRC Press, Boca Raton, FL, (1993).
- Ruder, Z., Hanratty, P.J., Hanratty, T.J., 1989. Necessary conditions for the existence of stable slugs. *Int. J. Multiphase Flow* 15, 209-226.
- S.-M. Cho, B. G. Thomas and S.-H. Kim: ISIJ Int., 58 (2018), 1443.
- S. A. Orszag and G. Patterson, Jr.: Phys. Rev. Lett., 28 (1972), 76.
- S. Pirker, D. Kahrmanovic and S. Schneiderbauer: Metall. Mater. Trans. B, 46 (2015), 953.
- S. Lo: Application of Population Balance to CFD Modelling of Bubbly Flow via the MUSIG model, AEA Technology, AEAT-1096, (1996).
- S. O. Unverdi and G. Tryggvason: J. Comput. Phys., 100 (1992), 25.
- S. Natsui, R. Nashimoto, T. Kumagai, T. Kikuchi and R. O. Suzuki: Metall. Mater. Trans. B, 48 (2017), 1792
- Schneider, G.E., Raw, M.J., Control-volume finite-element method for heat transfer and fluid flow using collocated variables . 1. Computational procedure, *Numer. Heat Transfer*, 11. 363-390 (1987).
- Scott, S.L., Shoham, O., Brill, J.P., 1986. Prediction of slug length in horizontal large diameter pipes. *Paper presented at the SPE 56th Annual California Regional Meeting, Oakland, California, April 1986*.
- Singh, V., S. K. Dash, J. S. Sunitha, S. K. Ajmani and A. K. Das: *ISIJ Int.*, **46** (2006), 210.
- Smagorinsky, J. (1963). General Circulation Experiments with primitive equations. I. The basic experiment, *Mon. Weather Rev*, 91, 99-164.
- Stapelberg, H.H., Mewes, D., 1994. The pressure Loss and slug frequency of liquid-liquid-gas slug flow in horizontal pipes. *Int. J. Multiphase Flow*, 20(2), 285.
- T. Zhang, Z. Luo, H. Zhou, B. Ni and Z. Zou: ISIJ Int., 56 (2016), 116.
- T. Zhang, Z. G. Luo, C. L. Liu, H. Zhou and Z. S. Zou: Powder Technol., 273 (2015), 154.
- T. Inamuro, T. Ogata, S. Tajima and N. Konishi: J. Comput. Phys., 198 (2004), 628.
- Taitel, Y., Barnea, D., 1990. Two-phase slug flow. *Adv. Heat Transfer* 20, 83-132.

- Taitel, Y., Barnea, D., 1998. Effect of gas compressibility on a slug tracking model. *Chem. Eng. Sci.* 53, 2089-2097.
- Taitel, Y., Dukler, A.E., 1976. A model for predicting flow regime transitions in horizontal and near horizontal gas-liquid flow. *AIChE J.* 22, 47-55.
- Taitel, Y. & Dukler A. (1977). "A model for slug frequency during gas-liquid flow in horizontal and near horizontal pipes." *International Journal of Multiphase Flow* 3(6): 585-596.
- Taitel, Y., Sarica, C., Brill, J.P., 2000. Slug flow modelling for downward inclined pipe flow: theoretical considerations. *Int. J. Multiphase Flow* 26, 833-844.
- Taylor, G. I. (1915). Eddy motion in the atmosphere. *Philosophical Transactions of the Royal Society*, A215, 1–26.
- Theron, B., 1989. Ecoulements diphasiques instationnaires en conduite horizontale. *PHD thesis, Institut National Polytechnique de Toulouse, France.*
- Tronconi, E., 1990. Prediction of slug frequency in horizontal two-phase flow. *AIChE Journal*, 36(5), 701-709.
- Vernier, P., Delhay, J., 1968. General two-phase flow equations applied to the thermodynamics of boiling nuclear reactors. *Energ. Primarie* 4, 1-43.
- Versteeg, H.K. and Malalasekera, W.M.G. (2007). *Introduction to Computational Fluids Dynamics: The finite volume method*. 2nd Ed (Indian Reprint) Pearson Education.
- Von Kármán, Th. (1930). Mechanische Ähnlichkeit und Turbulenz. In *Proceedings of the 3rd International Congress on Applied Mechanics*, Stockholm, Pt. 1, 85.
- von Kármán T (1931) Mechanical similitude and turbulence. National Advisory Committee for Aeronautics (NACA) technical memorandum no 611, Washington, USA; English Translation by Vanier, J pp 1–21
- W. P. Jones and B. E. Launder: *Int. J. Heat Mass Transf.*, 15 (1972), 301.
- W. Lou and M. Zhu: *Metall. Mater. Trans. B*, 44 (2013), 762.
- Wallis, G.B., 1969. *One dimensional Two-phase Flow*. McGraw-Hill, New York.
- Wallis G.B. & Dobson J.E. (1973). "The Onset of Slugging in Horizontal Stratified Air-Water Flow." *International Journal of Multiphase Flow* 1: 173-193.
- Wang, Y. H. Li, L. Guo, X. Yu and B. Zhou: *Steel Res. Int.*, 86 (2015), 732.
- Weber, M.E., 1981. Drift in intermittent two-phase flow in horizontal pipes. *Can. J. Chem. Eng.*, 59, 398-399.
- Woods, B.D., Hanratty, J.T., 1996. Relation of slug stability to shedding rate. *Int. J. Multiphase Flow*, 22(5), 809-828.
- Woods, B.D., Hurlburt, E.T., Hanratty, T.J., 2000. Mechanism of slug formation in downwardly inclined pipes. *Int. J. Multiphase Flow* 26, 977-998.
- X. Shan and H. Chen: *Phys. Rev. E*, 47 (1993), 1815.
- X. Zhang and H. Shen: *J. Hydrodyn.*, 18 (2006), 431.
- Y. Wang and L. Zhang: *ISI J. Int.*, 50 (2010), 1777.
- Z. Chen, Z. Zong, M. B. Liu, L. Zou, H. T. Li and C. Shu: *J. Comput. Phys.*, 283 (2015), 169.
- Z. Liu, F. Qi, B. Li and M. Jiang: *J. Iron Steel Res. Int.*, 21 (2015), 1081.
- Z. Q. Liu, F. S. Qi, B. K. Li and S. C. P. Cheung: *Int. J. Multiph. Flow*, 79 (2016), 190.

Zeng, G., Brill, J.P., Taitel, Y., 1994. Slug flow behaviour in a hilly terrain pipeline.
Int. J. multiphase Flow 20, 63-79.

Appendix A: Two-phase Flow Terminology

Many concepts and nomenclature are specifically in two-phase gas-liquid flow hydrodynamics. A brief description of the two-phase flow concepts and terms is necessary before discussing methods to predict pressure drop and liquid holdup in two-phase flow modeling.

1. Velocity

Different fluid velocity terms are used to describe the flow of gas and liquid in the pipe.

1.1 Superficial velocity

The superficial liquid velocity is defined as the velocity of the liquid when only the liquid-phase fraction of the fluid flows in the entire inside pipe area. The same being true for the gas superficial velocity. Many of the two-phase pressure drop correlations apply the concept of superficial velocities.

The gas superficial velocity is defined as:

$$V_{GS} = \frac{Q_G}{A} \quad (\text{A.1})$$

The liquid superficial velocity is defined as:

$$V_{LS} = \frac{Q_L}{A} \quad (\text{A.2}) \quad \text{Where}$$

A Pipe internal cross-sectional area (m^2)

Q_L Liquid-phase volumetric flow rate (m^3/s)

Q_G Gas-phase volumetric flow rate (m^3/s)

V_{LS} Liquid-phase superficial velocity (m/s)

V_{GS} Gas-phase superficial velocity (m/s)

1.2 Mixture velocity

Mixture velocity is defined as the total volumetric flow (That is the liquid plus the gas) per unit area of the pipe inside cross sectional area.

$$V_m = \frac{Q_T}{A} \quad (\text{A.3}) \quad \text{where}$$

Q_T Fluid mixture flow rate $Q_T = Q_L + Q_G$ (m^3/s)

V_m Mixture velocity (m/s)

Alternatively,

$$V_m = \frac{Q_L + Q_G}{A} \quad (\text{A.4})$$

Or

$$V_m = \frac{Q_L}{A} + \frac{Q_G}{A} = V_{LS} + V_{GS} \quad (\text{A.5})$$

Therefore, we can say the mixture velocity is the summation of the liquid and gas superficial velocities.

1.3 Gas and Liquid true velocities

The gas and liquid will travel at different velocities, depending on the gas mass split fraction and physical properties of each phase. These velocities are known as the true velocities. Gas-phase travels faster in most cases due to relatively gas density compared to the liquid-phase density. Thus

$$V_G \geq V_L$$

This phenomenon is known as phase slippage. The slippage ration is defined as:

$$SR = \frac{V_G}{V_L} \quad (\text{A.6})$$

Where SR is the slippage ratio, dimensionless.

When a no slip is assumed, this will indicate that the liquid travels at the same gas velocity.

$$V_L = V_G. \quad (\text{A.7})$$

The assumption of slippage or a no slippage between phases will result in different definitions and values of the liquid holdup, which will be defined in the next section. The cross sectional average true velocities are determined as follows:

$$u_G = \frac{Q_G}{A_G} = \frac{Q_G}{H_G A} \quad (\text{A.8})$$

$$u_L = \frac{Q_L}{A_L} = \frac{Q_L}{(1-H_G)A} \quad (\text{A.9})$$

Where

$u_G, u_L, Q_G, Q_L, A_G, A_L, H_G, (1 - H_G)$ are respectively gas and liquid true velocities, gas and liquid volumetric flow rates, the cross-sectional areas occupied by each phase and liquid and gas fractions.

2. Liquid Holdup

Liquid holdup is defined as the volume fraction of the pipe occupied or filled by liquid. Liquid holdup of gas and liquid are related as:

$$H_L + H_G = 1 \quad (\text{A.10})$$

Where

H_L Liquid Hold-up fraction

H_G Gas Hold-up fraction

Hold-up is defined by the following equation.

$$H_L = \frac{V_L}{V_P} \quad (\text{A.11})$$

Where

V_L Time averaged liquid volume per unit length of pipe

V_P Pipe volume per unit length of pipe

In most cases the gas-phase holdup is defined in terms of liquid-phase holdup ($H_G = 1 - H_L$). The liquid holdup is a function of many parameters such as fluid

physical properties, pipe dimensions and angle inclination, liquid and gas flow rates and system pressure. Liquid holdup is a very important parameter in two-phase pressure drop prediction, sizing of process and control equipments such as slug catchers, separator, two-phase pumps and control valves. In most cases, liquid holdup fraction will change progressively along the pipeline length due to the change in fluid temperature and pressure. The holdup variable and the velocity terms may be related as

$$V_L = \frac{V_{LS}}{H_L} \quad (\text{A.12})$$

And

$$V_G = \frac{V_{GS}}{H_G} = \frac{V_{GS}}{1-H_L} \quad (\text{A.13})$$

When a no slippage flow is assumed ($V_L = V_G$), the above equations can be grouped as:

$$\frac{V_{LS}}{H_L} = \frac{V_{GS}}{(1-H_L)} \quad (\text{A.14})$$

The above equation can be rearranged to solve for the liquid holdup in terms of superficial velocities as

$$H_L = \frac{V_{LS}}{V_{LS}+V_{GS}} = \frac{V_{LS}}{V_m} = \frac{Q_L}{Q_T} \quad (\text{A.15})$$

When a flow slippage is assumed, which is the actual flow behaviour in most cases, one of the empirical correlations must be used to predict the liquid holdup. Under slippage conditions, the liquid velocity is less than the liquid velocity at no slippage conditions, and liquid holdup is higher than that of liquid holdup at no slippage conditions.

3. Quality

Quality in two-phase flow is defined as the mass ratio of the gas phase to the total fluid mass in a particular pipe segment; quality is different from liquid holdup since it is mass not volume fraction. It formulated as:

$$x_G = \frac{M_G}{M_L + M_G} \quad (\text{A.16})$$

Where

x_G Quality, dimensionless

M_G Gas-phase mass at position in the pipe, (kg)

M_L Liquid-phase mass at position in the pipe, (kg)

Quality is a strong function of temperature and pressure and is an important parameter for applications with fluid vaporization or condensation. Quality is related to volume fractions (holdups) with the following relation:

$$H_L = \left[1 + SR \left(\frac{1-x_G}{x_G} \right) \frac{\rho_L}{\rho_G} \right]^{-1} \quad (\text{A.17})$$

Where

SR Slip ratio

ρ_L Liquid-phase density kg/m^3

ρ_G Gas-phase density kg/m^3

Appendix B: User Defined Functions

User Defined Functions (UDF) and their Usage in ANSYS Fluent

UDF is a C/C++ function that can be dynamically loaded with the ANSYS Fluent solver to enhance its standard features. The enhancement might be done to customize boundary conditions, customize material properties such as temperature, viscosity, density, to include additional source terms, to solve new scalar transport equation, to control chemical reaction rate, to compute value at the end of iteration, and to alter or improve the existing CFD models such as turbulence models and multiphase models.

1. Structure of an UDF

There are two types of UDF files that can either be interpreted or compiled. Interpreted UDF files are used to write few small functions without accessing to solver dynamic data while compiled UDF files are used for accessing solver memory and perform desired tasks.

Every UDF file has a name ending with the extension (.C) or (.Cpp) such as boundary_profile.C or boundary_profile.Cpp and additional header files can be included depending upon the demand (`#include "sq.h"`, `#include "flow.h"`, `#include "unsteady.h"`).

2. Basic Mesh Terminology

Mesh can be either structured or unstructured. Structured mesh is the mesh with number of neighbour cells remaining unchanged at any place and unstructured mesh is the one with number of neighbours keep changing from place to place.

- A cell is a control volume into which domain is broken up.
- A cell centre is the point where information about cell is stored.
- A node is a grid point.
- An edge is a boundary of a face.
- A face is a boundary of a cell.
- A face centre is the point where information about the face is stored.

3. Data Structures in ANSYS Fluent

Thread: A thread is the collection of certain entities. It represents a structured data type that stores data common to group of cells or faces.

Domain: A domain is the highest level of data structure in Fluent. It stores all the data associated with a collection of nodes, faces, and cells threads in a mesh.

For single phase applications, there is only a single domain structure.

Domain* d;

d = Get_Domain(1);

For multiphase applications, there are domain structures for each phase, the interaction between phases, as well as for mixture.

Inside the domain, data are organized in threads.

Node thread: A structure data type that stores data associated with a mesh point (grouping of nodes).

Face thread: A collection of grouping of faces.

face_t: An integer data type that identifies a particular face withing a face thread.

- a. Boundary face thread
- b. Interior face thread

All faces that form a specific boundary are grouped together in one face thread (boundary face thread).

Cell thread: A collection or grouping of cells.

cell_t: An integer data type that identifies a particular cell within a cell thread.

4. Why C Programming language

ANSYS Fluent is written in C language because of its computational speed, low level capabilities such as mathematical functions and memory operations, high level operations including graphics and networking.

It provides linking the extra functionality using “shared object” in unix and “Dynamic linked Library” in windows system.

4.1 Basic Concept of C Programming

Return_type function_name (parameter list)

```
{  
  
    Body of the function  
  
    Return output  
  
}
```

/ This is how I put a comment in my C program*/*

Data Types in C program:

Data types can

- Int: integer number.
- Long: integer number of increased range.
- Float: floating point (real) number.
- Double: double precision floating point (real) number.
- Char: single byte of memory, enough to hold a character.
- Constant: value remains unchanged throughout the UDF (#define g 9.81)
- Variable: value keeps changing (real volume;)

Local variables:

void function_1()

```
{  
  
    Int a, b ;
```

```
        a=5;          /* You can use a and b within braces only */
        b=20;

    }
```

Global variables:

```
Int a, b = 10;
```

```
Void function_1 ()
```

```
{
    Int c;

    c=a+b;
}
```

/* You can use a and b within any braces */

```
Void function_2 ()
```

```
{
    Int d;

    d=a-b;
}
```

Control Structures:

```
If (logical expression)
```

```
{ statement }
```

```
Else          /* used for logical operations */
```

```
{statement}
```

```
Int i, j, n <=10;
```

```
for (i=1; i<=n; i++)
```

```
/* Loops used for repeated tasks */
```

```
{
```

```
    j=i*i;
```

```
    printf("%d %d\n", i,j);
```

```
/* print integers 1-10 and their squares */
```

```
}
```

Arithmetic Operators:

- =; assignment
- + addition
- - subtraction
- * multiplication
- / division
- % modulo reduction
- ++ increment
- -- decrement

Logical Operators:

- < less than
- <= less than or equal to
- > greater than
- >= greater than or equal to
- == equal to
- != not equal to

5 UDF data Handling Concept

Some mesh and solver data types have a capital letter. These are data structure in C programming language and usually passed between functions as pointers to these structures.

Type	Example	Declaration
Domain	*d;	d is a pointer to the Domain
Thread	*t;	t is a pointer to the Thread
Cell_t	c;	c is cell index (integer)
Face_t	f;	f is face index (integer)
Node	*node	node is a pointer to the Node structure

5.1 Macros to write (UDF) functions

These functions are defined in udf.h library and start with the word DEFINE_. For some macros, the solver by default provides the thread pointer once the UDF is hooked to a particular face. Ex: DEFINE_PROFILE (name, thread, position).

```
#include "udf.h"
```

```
DEFINE_PROFILE (Pressure_Profile,t,i)    |* Example of Macros program *|
```

```
{
    real X[ND_ND];
    real Y;
    face_t f;
    begin_f_loop(f,t)
    {
        F_CENTROID (x,f,t);
        y = X[1];
```

```

        F_PROFILE (f,t,i) = 1.1e5 - y*y / (0.0745 * 0.0745) * 0.1e5
    }

    end_f_loop(f,t)
}

```

Some macros to write UDF are as follows:

```

DEFINE_ADJUST
DEFINE_DELTAT
DEFINE_EXECUTE_AT_END
DEFINE_EXECUTE_AT_EXIT
DEFINE_EXECUTE_FROM_GUI
DEFINE_EXECUTE_ON_LOADING
DEFINE_EXECUTE_AFTER_CASE/DATA
DEFINE_INIT
DEFINE_ON_DEMAND
DEFINE_OUTPUT_PARAMETER
DEFINE_RW_FILE

```

Looping Macros in UDF:

```

thread_loop_f(t,d)
{
    /* Loop over all face threads in domain d */
}

thread_loop_c(t,d)

```

```

{
    /* Loop over all cell threads in domain d */
}

face_t f;
begin_f_loop(f,t)
{
    /* Loop over all faces in a given thread */
}

end_f_loop(f,t)


cell_t c;
begin_c_loop(c,t)
{
    /* Loop over all cells in a given thread */
}

End_c_loop(c,t)

```

5.2 Accessing solver Data and variables

Macro	Arguments type	Returns
C_VOLUME(c,t)	Cell_tc, thread*t	Real cell volume for 2D or 3D real cell volume/2 for axisymmetric
C_R(c,t)	Cell tc, thread*t	Density
C_P(c,t)	Cell_tc, thread*t	Pressure
C_U(c,t)	Cell_tc, thread*t	U velocity
C_V(c,t)	Cell_tc, thread*t	V velocity
C_W(c,t)	Cell_tc, thread*t	W velocity
C_T(c,t)	Cell_tc, thread*t	Temperature
C_H(c,t)	Cell_tc, thread*t	Enthalpy
C_K(c,t)	Cell_tc, thread*t	Tur, kinetic energy
C_NU(c,t)	Cell_tc, thread*t	Turbulent viscosity for Sparlat-Allmaras
C_D(c,t)	Cell_tc, thread*t	Turbulent kinetic energy dissipation rate
C_O(c,t)	Cell_tc, thread*t	Specific dissipation rate
C_YI(c,t,i)	Cell_tc, thread*t,int i	Species mass fraction
C_CENTROID(x,c,t)	real X[ND_ND],Cell_tc, thread*t	X (cell centroid)

5.3 Accessing Solver Data variables (Gradient)

Cell gradients are defined in mem.h. They are as follows:

C_R_G(c,t) suffix_G in variables access the gradient

C_R_G(c,t)[0] gradient in X direction

C_R_RG(c,t) reconstruction gradients

C_R_M1(c,t) previous time step

C_R_M2(c,t) previous to previous time step

5.4 The ND and NV Macros

The ND Macros:

The constant ND_ND is defined as 2 for 2D and 3 for 3D. it can be used to build a matrix in 2D or a matrix in 3D. Example:

```
real A[ND_ND] [ND_ND];  
for (i=0; i<ND_ND; i++)  
{  
    for (j=0; j<ND_ND; ++j)  
    {  
        A[i][j] = f(i,j);  
    }  
}
```

- The utility ND_SUM computes the sum of ND_ND arguments.

ND_SUM(x,y,z)

2D: $x + y$;

3D: $x + y + z$;

- The utility ND_SET generates ND_ND assignment statement.

real U [ND_ND][ND_ND], V[ND_ND][ND_ND], W[ND_ND][ND_ND];

ND_SET (U, V, W, C_U(c,t), C_V(c,t), C_W(c,t));

U = C_U(c,t);

V = C_V(c,t);

W = C_W(c,t);

The NV Macros:

The NV macros have the same purpose as ND macros, but they operate on vectors (that is arrays of length ND_ND) instead of separate components.

- The utility NV_V performs an operation on two vectors.

NV_V(a, =, x);

a[0] = x[0]; a[1] = x[1];

- The utility NV_VV performs operations on vector elements. The operation that is performed on the elements depends upon what symbol (-, /, *) is used as an argument in place of the sign + in it.

NV_VV (a, =, x, +, y);

2D: $a[0] = x[0] + y[0]$, $a[1] = x[1] + y[1]$;

3D: $a[0] = x[0] + y[0]$, $a[1] = x[1] + y[1]$, $a[2] = x[2] + y[2]$;

- The utility NV_V_VS adds a vector to another vector which is multiplied by a scalar.

NV_V_VS (a, =, x, +, y, *, 0.5);

2D: $a[0] = x[0] + (y[0]*0.5)$, $a[1] = x[1] + (y[1]*0.5)$;

- The utility NV_VS_VS adds a vector to another vector which are each multiplied by a scalar.

NV_VS_VS (a, =, x, *, 2.0, +, y, *, 0.5);

2D: $a[0] = (x[0]*2.0) + (y[0]*0.5)$, $a[1] = (x[1]*2.0) + (y[1]*0.5)$;

- The utility NV_MAG computes the magnitude of a vector. This is considered as the square root of the sum of squares of the vector components.

NV_MAG(x)

2D: $\text{sqrt}(x[0]*x[0] + x[1]*x[1])$;

3D $\text{sqrt}(x[0]*x[0] + x[1]*x[1] + x[2]*x[2])$;

- The utility NV_MAG2 computes the sum of squares of vector components.

NV_MAG2(x)

2D: $(x[0]*x[0] + x[1]*x[1] + x[2]*x[2])$;

- The following utilities compute the dot product of two sets of vector components.

NV_DOT (x, y, z, U, V, W)

2D: $(x*U + y*V);$

3D: $(x*U + y*V + z*W);$

NV_DOT (x,U)

2D: $(x[0]*U[0] + x[1]*U[1]);$

3D: $(x[0]*U[0] + x[1]*U[1] + x[2]*U[2]);$

NV_DOT (x, U, V, W)

2D: $(x[0]*U + x[1]*V);$

3D: $(x[0]*U + x[1]*V + x[2]*W);$

Appendix C: UDF C Code

```
/*-----*/  
  
/* UDF C CODE: ANISOTROPIC HYBRID k-epsilon SST/STM */  
  
/* CLOSURE MODEL IN THE ANSYS-FLUENT ENVIRONMENT */  
  
/*-----*/  
  
/* ABBREVIATIONS - SST: SHEAR-STRESS TRANSPORT, */  
  
/* STM: STOCHASTIC TURBULENCE MODEL */  
  
/*-----*/  
  
/* TO RUN THE CODE, TYPE IN THE ANSYS-FLUENT COMMAND LINE:  
*/  
  
/* solve/set/expert (PRESS ENTER) */  
  
/* SET: Keep temporary solver memory from being freed? [yes] */  
  
/*-----*/  
  
/* ORIGINAL SINGLE-PHASE CODE WRITTEN BY Dr. Dr. László Könözy '  
2012-2019 */  
  
/*-----*/  
  
/* MULTIPHASE VOF MODIFICATION WRITTEN BY Mr. Claude Kouadjo ' 2020-  
2023 */  
  
/*-----*/  
  
#include "udf.h"  
  
#include <math.h>
```

```

/*-----*/

/* TOLERANCE VALUE TO CONTROL AND AVOID DIVISION BY ZERO */

/*-----*/

#define EPS 1.0e-10

/*-----*/

/* DOMAIN ID OF THE LIQUID PHASE */

/*-----*/

#define domain_ID 1

/*-----*/

/* NUMBER OF INITIAL ITERATION STEPS WHEN THE SOURCE TERMS OF
*/

/* THE ANISOTROPIC HYBRID TURBULENCE MODEL ARE SWICHTED OFF
*/

/*-----*/

#define NUMBER_OF_INITIAL_ITERATIONS 2

/*-----*/

/* DEFINE THE MODELLING APPROACH OF */

/* THE FLUCTUATING COORDINATE SYSTEM (e1',e2',e3') */

/*-----*/

```

```

#define MODELLING_APPROACH 2

#define URANS 0

#define RANS1 1

#define RANS2 2

/*-----*/

/* DEFINE THE CONSTANTS OF THE k-Epsilon SST MODEL */

/*-----*/

#define beta_star 0.09

#define gamma1 0.5531666

#define gamma2 0.4403546

#define sigma_omega2 0.856

/*-----*/

/* DEFINE THE CONSTANTS OF THE ANISOTROPIC SIMILARITY TENSOR
(STM) */

/*-----*/

#define alpha_cz -1.8802

#define beta_cz -0.6267

#define gamma_cz -0.9401

#define mu_cz 0.4093

#define theta_cz 0.0520

```

```

/*-----*/

/* WEIGHT PARAMETER OF THE ANISOTROPIC HYBRID k-omega SST/STM
MODEL */

/*-----*/

#define mu_THETA -0.025


/*-----*/

/* VON KÁRMÁN CONSTANT */

/*-----*/

#define kappa 0.40704


/*-----*/

/* BRADSHAW CONSTANT */

/*-----*/

#define a1 0.3


/*-----*/

/* DEFINE USER-DEFINED MEMORY (UDM) INDEXES */

/*-----*/

#define UDM_length_scale 0


#define UDM_dTHETA_G11dx1 1

```

#define UDM_dTHETA_G12dx2 2

#define UDM_dTHETA_G13dx3 3

#define UDM_dTHETA_G21dx1 4

#define UDM_dTHETA_G22dx2 5

#define UDM_dTHETA_G23dx3 6

#define UDM_dTHETA_G31dx1 7

#define UDM_dTHETA_G32dx2 8

#define UDM_dTHETA_G33dx3 9

#define UDM_production_k 10

#define UDM_production_omega 11

#define UDM_ARST11 12

#define UDM_ARST22 13

#define UDM_ARST33 14

#define UDM_ARST12 15

#define UDM_ARST13 16

#define UDM_ARST23 17

#define UDM_u1nm1 18

```

#define UDM_u2nm1 19

#define UDM_u3nm1 20

/*-----*/

/* DEFINE USER-DEFINED SCALAR (UDS) INDEXES */

/*-----*/

#define UDS_THETA_G11 0

#define UDS_THETA_G12 1

#define UDS_THETA_G13 2


#define UDS_THETA_G21 3

#define UDS_THETA_G22 4

#define UDS_THETA_G23 5


#define UDS_THETA_G31 6

#define UDS_THETA_G32 7

#define UDS_THETA_G33 8


/*-----*/

/* INITIALISATION OF FIELD VARIABLES */

/*-----*/

DEFINE_INIT(initialisation_field_variables, domain)
{

```

```

cell_t cell_index;

Thread *cell_thread;

/*-----*/

/* INITIALISATION OF FIELD VARIABLES IN THE ZEROth ITERATION STEP
*/

/*-----*/

/* SPECIFICATION OF INITIAL CONDITIONS OF THE PHYSICAL PROBLEM
*/

/*-----*/

thread_loop_c(cell_thread,domain)
{
    begin_c_loop(cell_index,cell_thread)
    {
        C_UDMI(cell_index,cell_thread,UDM_length_scale) = 0.0;

        C_UDMI(cell_index,cell_thread,UDM_u1nm1) = 0.0;
        C_UDMI(cell_index,cell_thread,UDM_u2nm1) = 0.0;
        C_UDMI(cell_index,cell_thread,UDM_u3nm1) = 0.0;

        C_UDMI(cell_index,cell_thread,UDM_dTHETA_G11dx1) = 0.0;
        C_UDMI(cell_index,cell_thread,UDM_dTHETA_G12dx2) = 0.0;
        C_UDMI(cell_index,cell_thread,UDM_dTHETA_G13dx3) = 0.0;
    }
}

```

```

C_UDMI(cell_index,cell_thread,UDM_dTHETA_G21dx1) = 0.0;
C_UDMI(cell_index,cell_thread,UDM_dTHETA_G22dx2) = 0.0;
C_UDMI(cell_index,cell_thread,UDM_dTHETA_G23dx3) = 0.0;

C_UDMI(cell_index,cell_thread,UDM_dTHETA_G31dx1) = 0.0;
C_UDMI(cell_index,cell_thread,UDM_dTHETA_G32dx2) = 0.0;
C_UDMI(cell_index,cell_thread,UDM_dTHETA_G33dx3) = 0.0;

C_UDMI(cell_index,cell_thread,UDM_production_k) = 0.0;
C_UDMI(cell_index,cell_thread,UDM_production_omega) = 0.0;

C_UDMI(cell_index,cell_thread,UDM_ARST11) = 0.0;
C_UDMI(cell_index,cell_thread,UDM_ARST22) = 0.0;
C_UDMI(cell_index,cell_thread,UDM_ARST33) = 0.0;

C_UDMI(cell_index,cell_thread,UDM_ARST12) = 0.0;
C_UDMI(cell_index,cell_thread,UDM_ARST13) = 0.0;
C_UDMI(cell_index,cell_thread,UDM_ARST23) = 0.0;
} end_c_loop(cell_index,cell_thread)
}

Message("INITIALISATION IS COMPLETED BY THE USER-DEFINED
FUNCTION!\n");

```

```

} /* END OF THE DEFINE_INIT FUNCTION */

/*-----*/

/* DEFINE_ADJUST FUNCTION FOR COMPUTING ITERATIVE FIELD
VARIABLES */

/*-----*/

DEFINE_ADJUST(compute_field_variables_mixture, domain)
{
    cell_t cell_index;

    Thread *cell_thread;

    /*-----*/

    /* DECLARATION OF VARIABLES */

    /*-----*/

    real a, b;

    real rho, mu, nu, mu_t, nu_t;

    real u1, u2, u3, k, omega;

    real du1dx1, du2dx1, du3dx1;

    real du1dx2, du2dx2, du3dx2;

    real du1dx3, du2dx3, du3dx3;

```

real OMEGA1, OMEGA2, OMEGA3, abs_OMEGA;

real dkdx1, dkdx2, dkdx3;

real u1tm1, u2tm1, u3tm1;

real u1nm1, u2nm1, u3nm1;

real u1ref, u2ref, u3ref;

real delta_u1, delta_u2, delta_u3, abs_delta_u;

real lambda;

real E11, E12, E13;

real E21, E22, E23;

real E31, E32, E33;

real lambda_star_11, lambda_star_22, lambda_star_33;

real alpha_star_cz, beta_star_cz, gamma_star_cz;

real G11, G12, G13;

real G22, G23, G33;

```
real THETA;
```

```
real dTHETA_G11dx1, dTHETA_G12dx2, dTHETA_G13dx3;
```

```
real dTHETA_G21dx1, dTHETA_G22dx2, dTHETA_G23dx3;
```

```
real dTHETA_G31dx1, dTHETA_G32dx2, dTHETA_G33dx3;
```

```
if(N_ITER >= NUMBER_OF_INITIAL_ITERATIONS)
```

```
{
```

```
domain = Get_Domain(domain_ID);
```

```
/*=====*/
```

```
/* DEFINE_ADJUST LOOP 1: COMPUTE ALL FIELD VARIABLES */
```

```
/*=====*/
```

```
thread_loop_c(cell_thread, domain)
```

```
{
```

```
begin_c_loop(cell_index, cell_thread)
```

```
{
```

```
/* DENSITY OF THE FLUID [kg/m^3] */
```

```
rho = C_R(cell_index, cell_thread);
```

```
/* DYNAMIC VISCOSITY OF THE FLUID [Pa*s] */
```

```
mu = C_MU_L(cell_index, cell_thread);
```

```
/* KINEMATIC VISCOSITY OF THE FLUID [m^2/s] */
```

```
nu = mu/rho;
```

```
/*-----*/
```

```
/* SCALAR COMPONENTS OF THE VELOCITY VECTOR */
```

```
/*-----*/
```

```
u1 = C_U(cell_index, cell_thread);
```

```
u2 = C_V(cell_index, cell_thread);
```

```
u3 = C_W(cell_index, cell_thread);
```

```
/* TURBULENT KINETIC ENERGY PER UNIT MASS [m^2/s^2] */
```

```
k = C_K(cell_index, cell_thread);
```

```
/*-----*/
```

```
/* ELEMENTS OF THE VELOCITY GRADIENT TENSOR [1/s] */
```

```
/*-----*/
```

```
du1dx1 = C_DUDX(cell_index, cell_thread);
```

```
du1dx2 = C_DUDY(cell_index, cell_thread);
```

```
du1dx3 = C_DUDZ(cell_index, cell_thread);
```

```
du2dx1 = C_DVDX(cell_index, cell_thread);
```

```
du2dx2 = C_DVDY(cell_index, cell_thread);
```

```

du2dx3 = C_DVDZ(cell_index, cell_thread);

du3dx1 = C_DWDX(cell_index, cell_thread);
du3dx2 = C_DWDY(cell_index, cell_thread);
du3dx3 = C_DWDZ(cell_index, cell_thread);

/*-----*/
/* SCALAR COMPONENTS OF THE VORTICITY VECTOR [1/s] */
/*-----*/

OMEGA1 = du3dx2-du2dx3;
OMEGA2 = du1dx3-du3dx1;
OMEGA3 = du2dx1-du1dx2;

/*-----*/
/* ABSOLUTE VALUE (MAGNITUDE) OF THE VORTICITY VECTOR [1/s] */
/*-----*/

abs_OMEGA =
sqrt((OMEGA1*OMEGA1)+(OMEGA2*OMEGA2)+(OMEGA3*OMEGA3));

/*-----*/
/* COMPUTE THE TURBULENT LENGTH SCALE FUNCTION */

```

```

/*-----*/

C_UDMI(cell_index,cell_thread,UDM_length_scale) =
sqrt((a1*k)/((kappa*kappa)*(abs_OMEGA*abs_OMEGA)));

/*-----*/

/* SCALAR COMPONENTS OF THE GRADIENT THE TURBULENT KINETIC
ENERGY */

/*-----*/

dkdx1 = C_K_G(cell_index, cell_thread)[0];
dkdx2 = C_K_G(cell_index, cell_thread)[1];
dkdx3 = C_K_G(cell_index, cell_thread)[2];

/* DYNAMIC EDDY VISCOSITY OF THE FLUID [Pa*s] */
mu_t = C_MU_T(cell_index, cell_thread);

/* KINEMATIC EDDY VISCOSITY OF THE FLUID [m^2/s] */
nu_t = mu_t/rho;

/*-----*/

/* COMPUTE THE VELOCITY DIFFERENCE */

/* IN THE DEFINITION OF THE FLUCTUATING e2' UNIT VECTOR */

```

```

/*-----*/

if(MODELLING_APPROACH == URANS)
{
    u1tm1 = C_U_M1(cell_index, cell_thread);
    u2tm1 = C_V_M1(cell_index, cell_thread);
    u3tm1 = C_W_M1(cell_index, cell_thread);

    /* VELOCITY DIFFERENCE BETWEEN TWO CONSECUTIVE TIME STEPS
    (URANS) */

    delta_u1 = u1-u1tm1;
    delta_u2 = u2-u2tm1;
    delta_u3 = u3-u3tm1;
}

if(MODELLING_APPROACH == RANS1)
{
    u1nm1 = C_UDMI(cell_index,cell_thread,UDM_u1nm1);
    u2nm1 = C_UDMI(cell_index,cell_thread,UDM_u2nm1);
    u3nm1 = C_UDMI(cell_index,cell_thread,UDM_u3nm1);

    /* VELOCITY DIFFERENCE BETWEEN TWO CONSECUTIVE ITERATION
    STEPS */

```

```

delta_u1 = u1-u1nm1;

delta_u2 = u2-u2nm1;

delta_u3 = u3-u3nm1;

}


if(MODELLING_APPROACH == RANS2)

{

u1ref = 0.0;

u2ref = 0.0;

u3ref = 0.0;


/* VELOCITY DIFFERENCE RELATIVE TO THE REFERENCE VELOCITY */

delta_u1 = u1-u1ref;

delta_u2 = u2-u2ref;

delta_u3 = u3-u3ref;

}


/*-----*/

/* ABSOLUTE VALUE (MAGNITUDE) OF THE VELOCITY DIFFERENCE */

/* IN THE FLUCTUATING e2' UNIT VECTOR */

/*-----*/


abs_delta_u = sqrt((delta_u1*delta_u1)

```

```

        +(delta_u2*delta_u2)
        +(delta_u3*delta_u3));

/*-----*/

/* COMPUTE THE ELEMENTS OF THE TRANSFORMATION TENSOR */

/*-----*/

if((abs_delta_u*abs_OMEGA) <= EPS)
{
    lambda = 0.0;

    E11 = 0.0;
    E12 = 0.0;

    E21 = 0.0;
    E22 = 0.0;

    E31 = 0.0;
    E32 = 0.0;
}
else
{
    a = abs_delta_u*abs_OMEGA;

```

```

lambda
((delta_u1*OMEGA1)+(delta_u2*OMEGA2)+(delta_u3*OMEGA3))/a;

b = 1.0/(sqrt(1.0-lambda*lambda));

E11 = b*((delta_u1/abs_delta_u)-lambda*(OMEGA1/abs_OMEGA));
E12 = b*((delta_u2*OMEGA3-delta_u3*OMEGA2)/a);

E21 = b*((delta_u2/abs_delta_u)-lambda*(OMEGA2/abs_OMEGA));
E22 = b*((delta_u3*OMEGA1-delta_u1*OMEGA3)/a);

E31 = b*((delta_u3/abs_delta_u)-lambda*(OMEGA3/abs_OMEGA));
E32 = b*((delta_u1*OMEGA2-delta_u2*OMEGA1)/a);
}

if(abs_OMEGA <= EPS)
{
E13 = 0.0;
E23 = 0.0;
E33 = 0.0;
}
else

```

```

{
    E13 = -(OMEGA1/abs_OMEGA);
    E23 = -(OMEGA2/abs_OMEGA);
    E33 = -(OMEGA3/abs_OMEGA);
}

/*-----*/
/* ANISOTROPIC SCALE FACTORS MUST BE ADJUSTED */
/* BASED ON EXPERIMENTAL DATA FOR A GIVEN PROBLEM */
/*-----*/

/* lambda_star_11 = -0.575; */
/* lambda_star_22 = -0.595; */
/* lambda_star_33 = -0.775; */

lambda_star_11 = 1.0;
lambda_star_22 = 1.0;
lambda_star_33 = 1.0;

/*-----*/
/* COMPUTE THE DIAGONAL ELEMENTS (COMPONENTS) OF */
/* THE ANISOTROPIC DEVIATORIC SIMILARITY TENSOR */
/*-----*/

```

```

alpha_star_cz = ((2.0*alpha_cz)-beta_cz-gamma_cz)/3.0;

beta_star_cz = ((2.0*beta_cz)-alpha_cz-gamma_cz)/3.0;

gamma_star_cz = ((2.0*gamma_cz)-alpha_cz-beta_cz)/3.0;


/*-----*/

/* COMPUTE THE DOMINANT TURBULENT SHEAR STRESS */

/*-----*/


THETA = mu_t*abs_OMEGA;


/*-----*/

/* COMPUTE THE ELEMENTS OF THE TRANSFORMED DIMENSIONLESS
*/

/* ANISOTROPIC DEVIATORIC SIMILARITY TENSOR */

/*-----*/


G11 = lambda_star_11*alpha_star_cz*(E11*E11)

      +lambda_star_22*beta_star_cz*(E12*E12)

      +lambda_star_33*gamma_star_cz*(E13*E13)

      +2.0*((E11*E12)+mu_cz*(E11*E13)+theta_cz*(E12*E13));


G22 = lambda_star_11*alpha_star_cz*(E21*E21)

```

$$\begin{aligned}
& +\text{lambda_star_22}*\text{beta_star_cz}*(\text{E22}*\text{E22}) \\
& +\text{lambda_star_33}*\text{gamma_star_cz}*(\text{E23}*\text{E23}) \\
& +2.0*((\text{E21}*\text{E22})+\text{mu_cz}*(\text{E21}*\text{E23})+\text{theta_cz}*(\text{E22}*\text{E23}));
\end{aligned}$$

$$\begin{aligned}
\text{G33} &= \text{lambda_star_11}*\text{alpha_star_cz}*(\text{E31}*\text{E31}) \\
& +\text{lambda_star_22}*\text{beta_star_cz}*(\text{E32}*\text{E32}) \\
& +\text{lambda_star_33}*\text{gamma_star_cz}*(\text{E33}*\text{E33}) \\
& +2.0*((\text{E31}*\text{E32})+\text{mu_cz}*(\text{E31}*\text{E33})+\text{theta_cz}*(\text{E32}*\text{E33}));
\end{aligned}$$

$$\begin{aligned}
\text{G12} &= \text{lambda_star_11}*\text{alpha_star_cz}*(\text{E11}*\text{E21}) \\
& +\text{lambda_star_22}*\text{beta_star_cz}*(\text{E12}*\text{E22}) \\
& +\text{lambda_star_33}*\text{gamma_star_cz}*(\text{E13}*\text{E23})+(\text{E11}*\text{E22})+(\text{E12}*\text{E21}) \\
& +\text{mu_cz}*((\text{E11}*\text{E23})+(\text{E13}*\text{E21}))+\text{theta_cz}*((\text{E12}*\text{E23})+(\text{E13}*\text{E22}));
\end{aligned}$$

$$\begin{aligned}
\text{G13} &= \text{lambda_star_11}*\text{alpha_star_cz}*(\text{E11}*\text{E31}) \\
& +\text{lambda_star_22}*\text{beta_star_cz}*(\text{E12}*\text{E32}) \\
& +\text{lambda_star_33}*\text{gamma_star_cz}*(\text{E13}*\text{E33})+(\text{E11}*\text{E32})+(\text{E12}*\text{E31}) \\
& +\text{mu_cz}*((\text{E11}*\text{E33})+(\text{E13}*\text{E31}))+\text{theta_cz}*((\text{E12}*\text{E33})+(\text{E13}*\text{E32}));
\end{aligned}$$

$$\begin{aligned}
\text{G23} &= \text{lambda_star_11}*\text{alpha_star_cz}*(\text{E21}*\text{E31}) \\
& +\text{lambda_star_22}*\text{beta_star_cz}*(\text{E22}*\text{E32}) \\
& +\text{lambda_star_33}*\text{gamma_star_cz}*(\text{E23}*\text{E33})+(\text{E21}*\text{E32})+(\text{E22}*\text{E31}) \\
& +\text{mu_cz}*((\text{E21}*\text{E33})+(\text{E23}*\text{E31}))+\text{theta_cz}*((\text{E22}*\text{E33})+(\text{E23}*\text{E32}));
\end{aligned}$$

```

/*-----*/

/* STORE THE ELEMENTS OF THE (G) TENSOR IN USER-DEFINED
SCALARS */

/*-----*/

C_UDSI(cell_index,cell_thread,UDS_THETA_G11)      =
mu_THETA*THETA*G11;

C_UDSI(cell_index,cell_thread,UDS_THETA_G12)      =
mu_THETA*THETA*G12;

C_UDSI(cell_index,cell_thread,UDS_THETA_G13)      =
mu_THETA*THETA*G13;


C_UDSI(cell_index,cell_thread,UDS_THETA_G21)      =
mu_THETA*THETA*G12;

C_UDSI(cell_index,cell_thread,UDS_THETA_G22)      =
mu_THETA*THETA*G22;

C_UDSI(cell_index,cell_thread,UDS_THETA_G23)      =
mu_THETA*THETA*G23;


C_UDSI(cell_index,cell_thread,UDS_THETA_G31)      =
mu_THETA*THETA*G13;

C_UDSI(cell_index,cell_thread,UDS_THETA_G32)      =
mu_THETA*THETA*G23;

C_UDSI(cell_index,cell_thread,UDS_THETA_G33)      =
mu_THETA*THETA*G33;

```

```

/*-----*/

/* COMPUTE THE LOCAL VALUE OF THE ADDITIONAL PRODUCTION
TERM OF */

/* THE TURBULENT KINETIC ENERGY TRANSPORT EQUATION */

/*-----*/

```

```

C_UDMI(cell_index,cell_thread,UDM_production_k) =
mu_THETA*THETA*(((G11*du1dx1)+(G22*du2dx2)+(G33*du3dx3))
+(G12*(du1dx2+du2dx1))
+(G13*(du1dx3+du3dx1))
+(G23*(du2dx3+du3dx2)));

```

```

/*-----*/

/* COMPUTE THE ELEMENTS (SCALAR COMPONENTS) OF */

/* THE ANISOTROPIC REYNOLDS STRESS TENSOR (ARST) */

/*-----*/

```

```

C_UDMI(cell_index,cell_thread,UDM_ARST11) =
(2.0*rho*nu_t*du1dx1)+(mu_THETA*THETA*G11)-(2.0/3.0)*rho*k;

C_UDMI(cell_index,cell_thread,UDM_ARST22) =
(2.0*rho*nu_t*du2dx2)+(mu_THETA*rho*a1*k*G22)-(2.0/3.0)*rho*k;

```

```

C_UDMI(cell_index,cell_thread,UDM_ARST33) =
(2.0*rho*nu_t*du3dx3)+(mu_THETA*rho*a1*k*G33)-(2.0/3.0)*rho*k;

C_UDMI(cell_index,cell_thread,UDM_ARST12) =
(rho*nu_t*(du2dx1+du1dx2))+(mu_THETA*THETA*G12);

C_UDMI(cell_index,cell_thread,UDM_ARST13) =
(rho*nu_t*(du3dx1+du1dx3))+(mu_THETA*THETA*G13);

C_UDMI(cell_index,cell_thread,UDM_ARST23) =
(rho*nu_t*(du3dx2+du2dx3))+(mu_THETA*THETA*G23);

/*-----*/
/* STORE THE COMPONENTS OF THE VELOCITY VECTOR (u1,u2,u3) */
/* IN THE (Nth) ITERATION STEP IN USER-DEFINED MEMORIES */
/*-----*/

C_UDMI(cell_index,cell_thread,UDM_u1nm1) = u1;
C_UDMI(cell_index,cell_thread,UDM_u2nm1) = u2;
C_UDMI(cell_index,cell_thread,UDM_u3nm1) = u3;
}

} end_c_loop(cell_index,cell_thread) /* END OF LOOP 1 */

```

```

/*=====
=====*/

/* DEFINE_ADJUST LOOP 2:                                     */

/* COMPUTE AND STORE THE SPATIAL DERIVATIVES OF THE FIELD
VARIABLES */

/*=====
=====*/

domain = Get_Domain(domain_ID);

thread_loop_c(cell_thread, domain)

{
    begin_c_loop(cell_index, cell_thread)

    {
        /*-----*/

        /* COMPUTE THE SPATIAL DERIVATIVES OF THE ELEMENTS OF */

        /* THE ANISOTROPIC (THETA*G) TENSOR                      */

        /*-----*/

        dTHETA_G11dx1 = C_UDSI_G(cell_index,cell_thread,UDS_THETA_G11)[0];
        dTHETA_G12dx2 = C_UDSI_G(cell_index,cell_thread,UDS_THETA_G12)[1];

```

```

dTHETA_G13dx3 = C_UDSI_G(cell_index,cell_thread,UDS_THETA_G13)[2];

dTHETA_G21dx1 = C_UDSI_G(cell_index,cell_thread,UDS_THETA_G21)[0];
dTHETA_G22dx2 = C_UDSI_G(cell_index,cell_thread,UDS_THETA_G22)[1];
dTHETA_G23dx3 = C_UDSI_G(cell_index,cell_thread,UDS_THETA_G23)[2];

dTHETA_G31dx1 = C_UDSI_G(cell_index,cell_thread,UDS_THETA_G31)[0];
dTHETA_G32dx2 = C_UDSI_G(cell_index,cell_thread,UDS_THETA_G32)[1];
dTHETA_G33dx3 = C_UDSI_G(cell_index,cell_thread,UDS_THETA_G33)[2];

/*-----*/
/* STORE THE SPATIAL DERIVATIVES OF THE ELEMENTS OF      */
/* THE ANISOTROPIC (THETA*G) TENSOR IN USER-DEFINED MEMORIES */
*/

/*-----*/

C_UDMI(cell_index,cell_thread,UDM_dTHETA_G11dx1) = dTHETA_G11dx1;
C_UDMI(cell_index,cell_thread,UDM_dTHETA_G12dx2) = dTHETA_G12dx2;
C_UDMI(cell_index,cell_thread,UDM_dTHETA_G13dx3) = dTHETA_G13dx3;

C_UDMI(cell_index,cell_thread,UDM_dTHETA_G21dx1) = dTHETA_G21dx1;
C_UDMI(cell_index,cell_thread,UDM_dTHETA_G22dx2) = dTHETA_G22dx2;
C_UDMI(cell_index,cell_thread,UDM_dTHETA_G23dx3) = dTHETA_G23dx3;

```

```

C_UDMI(cell_index,cell_thread,UDM_dTHETA_G31dx1) = dTHETA_G31dx1;
C_UDMI(cell_index,cell_thread,UDM_dTHETA_G32dx2) = dTHETA_G32dx2;
C_UDMI(cell_index,cell_thread,UDM_dTHETA_G33dx3) = dTHETA_G33dx3;
}

} end_c_loop(cell_index,cell_thread) /* END OF LOOP 2 */

} /* END IF */

} /* END OF THE DEFINE_ADJUST FUNCTION */


/*-----*/

/* SOURCE TERM OF THE REYNOLDS MOMENTUM EQUATION-U1 */

/*-----*/

DEFINE_SOURCE(momentum_equation_u1_3d, cell_index, cell_thread, dS,
eqn)
{
real source = 0.0;

real dTHETA_G11dx1, dTHETA_G12dx2, dTHETA_G13dx3;

dTHETA_G11dx1 = C_UDMI(cell_index,cell_thread,UDM_dTHETA_G11dx1);
dTHETA_G12dx2 = C_UDMI(cell_index,cell_thread,UDM_dTHETA_G12dx2);
dTHETA_G13dx3 = C_UDMI(cell_index,cell_thread,UDM_dTHETA_G13dx3);

```

```

if(N_ITER >= NUMBER_OF_INITIAL_ITERATIONS)

{

    source = dTHETA_G11dx1+dTHETA_G12dx2+dTHETA_G13dx3;

}


dS[eqn] = 0.0;


return(source);

} /* END OF THE DEFINE_SOURCE FUNCTION (MOMENTUM EQUATION-U1)
*/


/*-----*/

/* SOURCE TERM OF THE REYNOLDS MOMENTUM EQUATION-U2 */

/*-----*/

DEFINE_SOURCE(momentum_equation_u2_3d, cell_index, cell_thread, dS,
eqn)
{

    real source = 0.0;


    real dTHETA_G21dx1, dTHETA_G22dx2, dTHETA_G23dx3;


    dTHETA_G21dx1 = C_UDMI(cell_index,cell_thread,UDM_dTHETA_G21dx1);
    dTHETA_G22dx2 = C_UDMI(cell_index,cell_thread,UDM_dTHETA_G22dx2);

```

```
dTHETA_G23dx3 = C_UDMI(cell_index,cell_thread,UDM_dTHETA_G23dx3);
```

```
if(N_ITER >= NUMBER_OF_INITIAL_ITERATIONS)
```

```
{
```

```
    source = dTHETA_G21dx1+dTHETA_G22dx2+dTHETA_G23dx3;
```

```
}
```

```
dS[eqn] = 0.0;
```

```
return(source);
```

```
} /* END OF THE DEFINE_SOURCE FUNCTION (MOMENTUM EQUATION-U2)
```

```
*/
```

```
/*-----*/
```

```
/* SOURCE TERM OF THE REYNOLDS MOMENTUM EQUATION-U3 */
```

```
/*-----*/
```

```
DEFINE_SOURCE(momentum_equation_u3_3d, cell_index, cell_thread, dS,  
eqn)
```

```
{
```

```
    real source = 0.0;
```

```
    real dTHETA_G31dx1, dTHETA_G32dx2, dTHETA_G33dx3;
```

```

dTHETA_G31dx1 = C_UDMI(cell_index,cell_thread,UDM_dTHETA_G31dx1);
dTHETA_G32dx2 = C_UDMI(cell_index,cell_thread,UDM_dTHETA_G32dx2);
dTHETA_G33dx3 = C_UDMI(cell_index,cell_thread,UDM_dTHETA_G33dx3);

if(N_ITER >= NUMBER_OF_INITIAL_ITERATIONS)
{
    source = dTHETA_G31dx1+dTHETA_G32dx2+dTHETA_G33dx3;
}

dS[eqn] = 0.0;

return(source);
} /* END OF THE DEFINE_SOURCE FUNCTION (MOMENTUM EQUATION-U3)
*/

/*-----*/
/* SOURCE TERM OF THE TURBULENT KINETIC ENERGY EQUATION (k) */
/*-----*/

DEFINE_SOURCE(turbulent_kinetic_energy, cell_index, cell_thread, dS, eqn)
{
    real source = 0.0;

    if(N_ITER >= NUMBER_OF_INITIAL_ITERATIONS)

```

```

{
    /* ADDITIONAL ANISOTROPIC PRODUCTION TERM FOR k */
    source = C_UDMI(cell_index,cell_thread,UDM_production_k);
}

dS[eqn] = 0.0;

return(source);
} /* END OF THE DEFINE_SOURCE FUNCTION (TURBULENT KINETIC
ENERGY) */

/* END OF THE USER-DEFINED FUNCTIONS (UDFs) C CODE */

```

Appendix D: Experimentation

The experiments were carried out in the Horizontal Two-Phase Flow facility (HORTOFF) of the Fluid Mechanics Laboratory, Department of Mechanical and Industrial Engineering, Gadjah Mada University, Indonesia. A schematic of the tools used in the current research and activity is shown in Figure D-1. It consists of pipes, a high-speed camera, a high-impedance amplifier, reservoir and storage tanks, a separator, a compressor, valves, flowmeters, pumps, a computer or laptop, and a mixer.

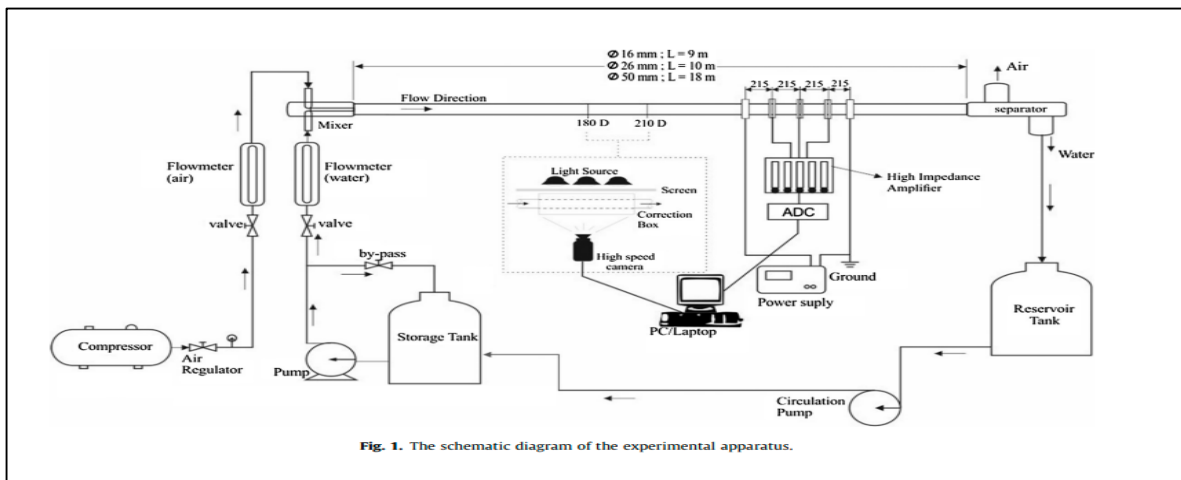


Figure D-1 The schematic diagram of the experimental apparatus

The pipes being tested are composed of acrylic and have respective inner diameters of 16 mm, 26 mm, and 50 mm. The pipes have relative lengths of 9 m, 10 m, and 18 m for the appropriate inner diameters. The experiments are done in a temperature-neutral environment.

Both air and water are employed as fluids in the experiment. A centrifugal compressor provides the air, and two Dwyer gas rotameters with maximum capacities of 200 and 600 standard cubic/minute regulate the airflow rate. A centrifugal pump provides the water, while two omega water rotameters with maximum capacity of 10 GPM and approximately 2% accuracy regulate the waterflow rates.

A high-speed video camera with a maximum resolution of 3000 x 4000 pixels and a frame rate of 1200 fps monitors the behaviour of the air-water interface along the pipe. The illumination of a number of LED lights supports the visualisation study, and the correction box minimises the difference in refractive indices between the fluids and the tube wall material to reduce picture distortion of air-water flows inside a tube.

At the developed flow test area, about 5 metres from the water and gas entrance, a rectangular transparent acrylic box is installed to reduce picture distortion during the observations. To improve the visualisation observation, water with a similar refractive index to acrylic is placed inside the correction box [12]. This would accentuate and correctly begin the peculiarities of the slug flow behaviour and the phenomenon occurring in the nearby wall.

Images and video from the recording are extracted to create an image sequence. The binary approach, developed by Montoya et al. [21], is an image processing technique that is used to process the raw images. In order to minimise inaccuracy, a technique developed by Widyatama et al. [27] is utilised to remove the small bubbles that are present near the gas-liquid interface. The gas-liquid interface is so precisely recognised and determined. A specific configuration is used to implement the air and water inlet, keeping air and water flowing separately into the test section as shown in Fig D.2. The air is supplied to the inlet from the top section and water is entering from the bottom section.

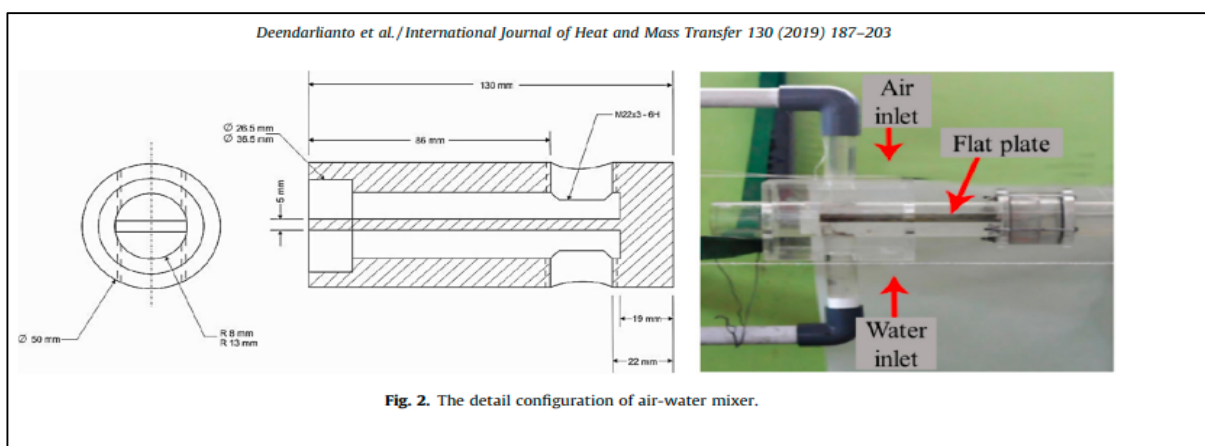


Figure D-2 The detail configuration of air-water mixer

In the actual experimental research, the instantaneous data of the local volumetric liquid fraction and the height of the liquid are collected using the constant electric current method (CECM). In this case, the volumetric liquid fraction is determined by dividing the cross-sectional area of the liquid phase by the cross-sectional area of the inner pipe. While the liquid height is the distance between the lowest point of the inner pipe and the highest level of liquid at the designated reference point as measured from the longitudinal section view.

Figures (a) and (b) show the schematic diagram of the CECM used in the experiment and the typical CECM photo used in the test rig respectively.

Based on the differential in electric resistance between the gas and liquid phases, the CECM method operates. Five sensors, each having a pair of 1 mm thick brass electrodes spaced 5 mm apart and flush affixed to the test tube inner surface, are positioned with an axial spacing of 215 mm. They are precisely manufactured to prevent flow disturbance from the electrodes. The sensors' job is to transmit signals to an A/D converter floating amplifier with high input impedance in a personal computer. At 0.5 kHz, the volumetric liquid fraction data are gathered. There are more specifics on the CECM measurement system and guiding principles in Fukano [16].

For all tested inner pipe sizes, matrix tests are compiled in Table D-1. Each inner pipe diameter has undergone 42 testing runs, each of which was performed more than three times. The range of water and air superficial velocities are $J_L = 0.03 - 0.30 \text{ m/s}$ and $J_G = 0.7 - 10.0 \text{ m/s}$.

Table D-1 Experimental matrix test

Table 1
Experimental matrix test.

J_L (m/s)	J_G (m/s)						
	0.7	0.94	1.88	2.83	3.77	6.2	10
0.03	#1	#2	#3	#4	#5	#6	#7
0.05	#8	#9	#10	#11	#12	#13	#14
0.075	#15	#16	#17	#18	#19	#20	#21
0.1	#22	#23	#24	#25	#26	#27	#28
0.2	#29	#30	#31	#32	#33	#34	#35
0.3	#36	#37	#38	#39	#40	#41	#42

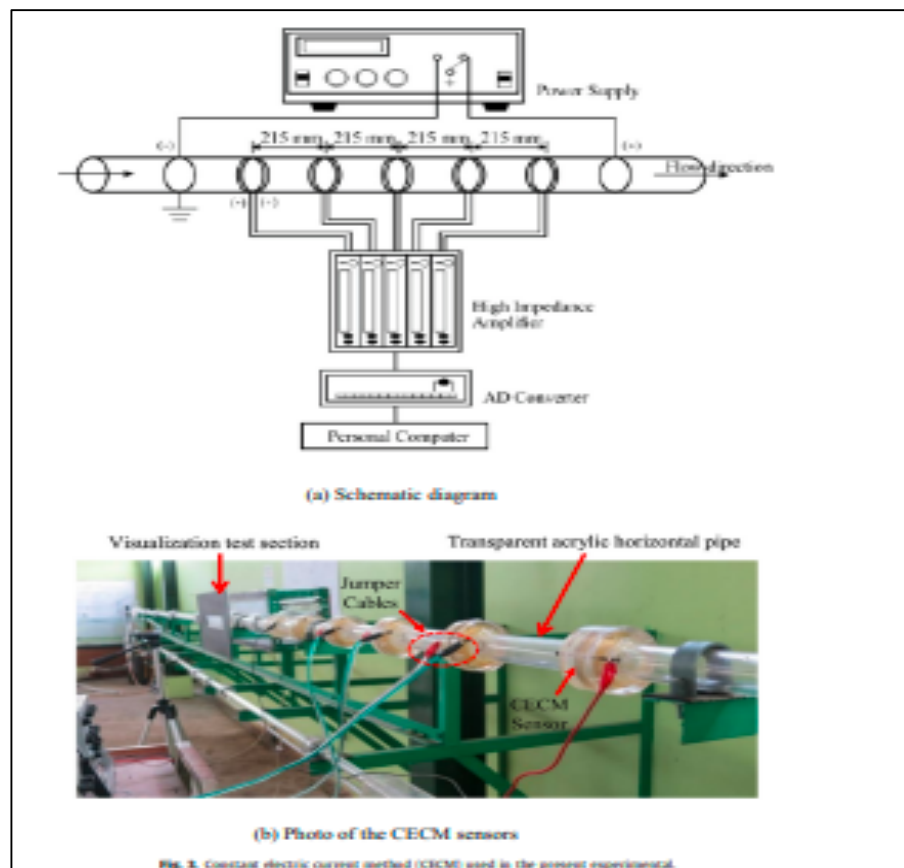


Figure D-3 Constant electric current method (CECM) used in the present experimental

Experimental Results

Three cases have been considered for our work.

Case 1.

A horizontal pipeline of length 9 m and diameter 16 mm has been used for the transient simulation. The flow is assumed to three dimensional and adiabatic. Initial and boundary conditions have been set up. The following input data have been used. See Table D-2 below.

- Superficial velocity of liquid U_{SL} and gas U_{SG}
- Density and viscosity of liquid (ρ_L and μ_L) and gas (ρ_G and μ_G)
- Pipe diameter (D)
- Surface tension 0.072

Table D-2 Case 1

Air superficial velocity	0.7	0.94	1.88
Water superficial velocity	0.075		
Air viscosity	1.7894e-05		
Air density	1.225		
Water viscosity	0.001003		
Water density	998.2		
Pipe diameter	16 mm		
Pipe length	9 m		
Air volume fraction	0.25 (Inner 8 mm)		
Water volume fraction	0.75		
Time	60 s		

Time step size	0.01
Surface tension	0.072

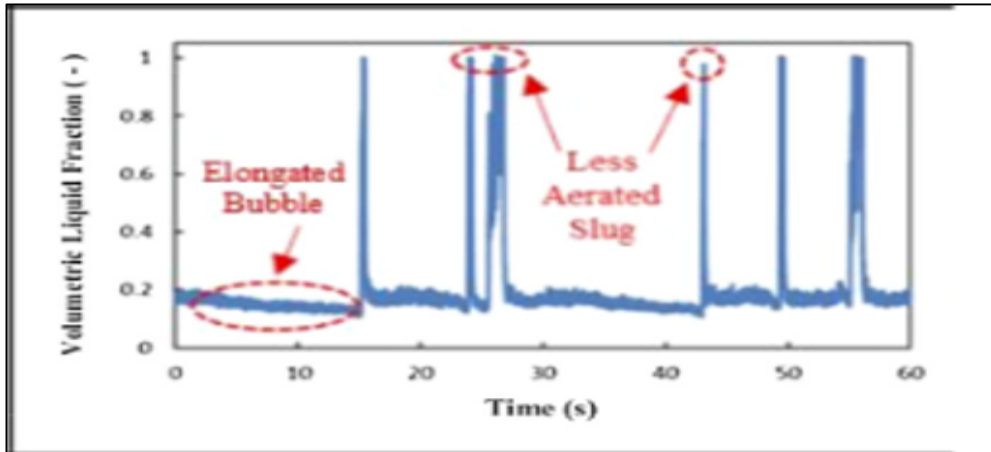


Figure D-4 Case 1 with $U_{SL} = 0.075 \text{ m/s}$ and $U_{SG} = 0.7 \text{ m/s}$

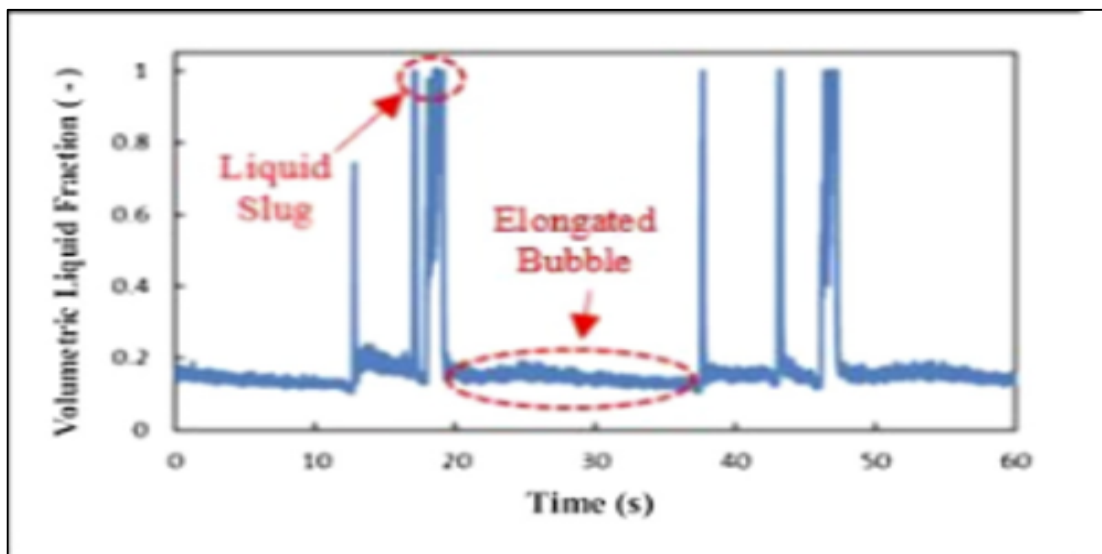


Figure D-5 Case 1 with $U_{SL} = 0.075 \text{ m/s}$ and $U_{SG} = 0.94 \text{ m/s}$

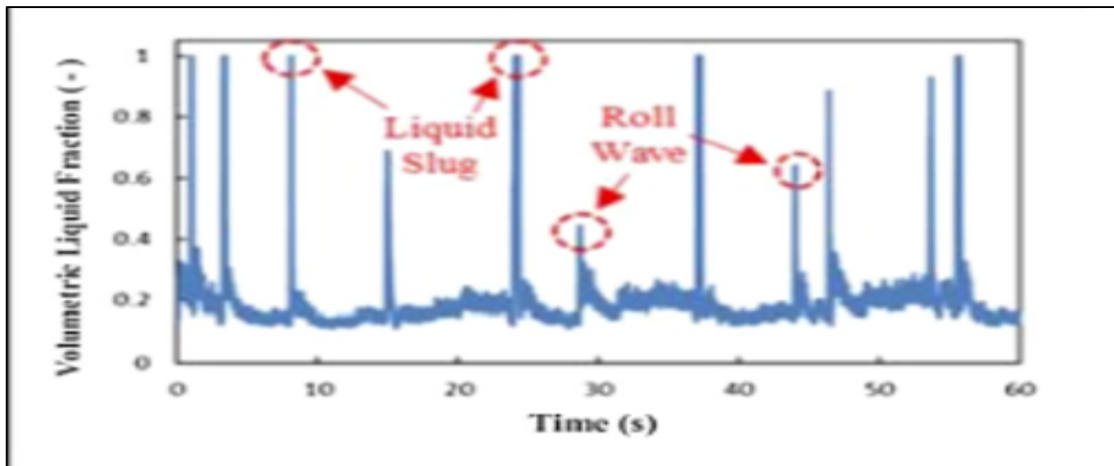


Figure D-6 Case 1 with $U_{SL} = 0.075 \text{ m/s}$ and $U_{SG} = 1.88 \text{ m/s}$

Case 2.

A horizontal pipeline of length 10 m and diameter 26 mm has been used for the transient simulation. The flow is assumed to three dimensional and adiabatic. Initial and boundary conditions have been set up. The following input data have been used. See Table D-3 below.

- Superficial velocity of liquid U_{SL} and gas U_{SG}
- Density and viscosity of liquid (ρ_L and μ_L) and gas (ρ_G and μ_G)
- Pipe diameter (D)
- Surface tension 0.072

Table D-3 Case 2

Air superficial velocity	3.77		
Water superficial velocity	0.1	0.2	0.3
Air viscosity	1.7894e-05		
Air density	1.225		
Water viscosity	0.001003		
Water density	998.2		
Pipe diameter	26 mm		
Pipe length	10 m		
Air volume fraction	0.25 (Inner 13 mm)		
Water volume fraction	0.75		
Time	60 s		
Time step size	0.01		
Surface tension	0.072		

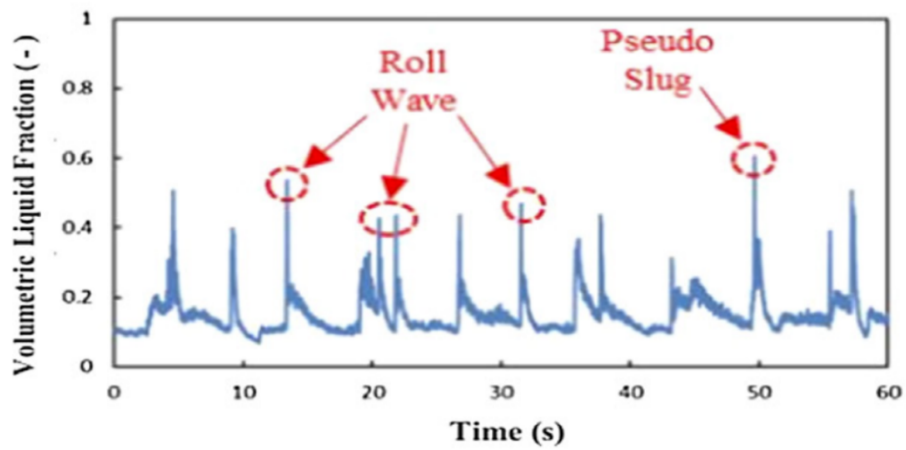


Figure D-7 Case 2 with $U_{SL} = 0.1 \text{ m/s}$ and $U_{SG} = 3.77 \text{ m/s}$

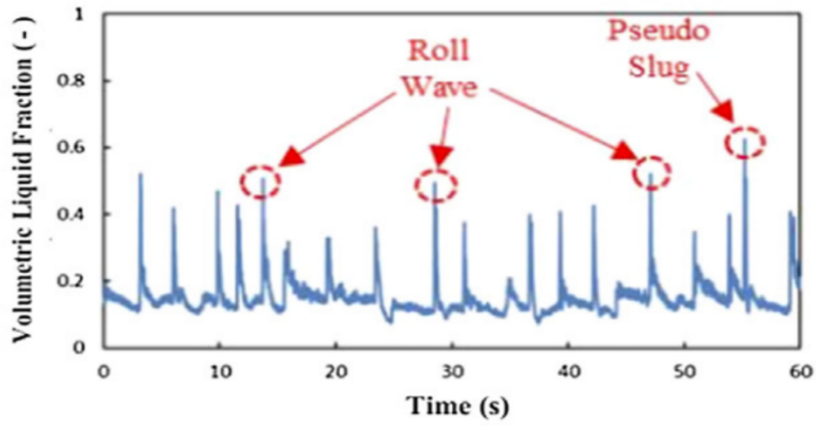


Figure D-8 Case 2 with $U_{SL} = 0.2 \text{ m/s}$ and $U_{SG} = 3.77 \text{ m/s}$

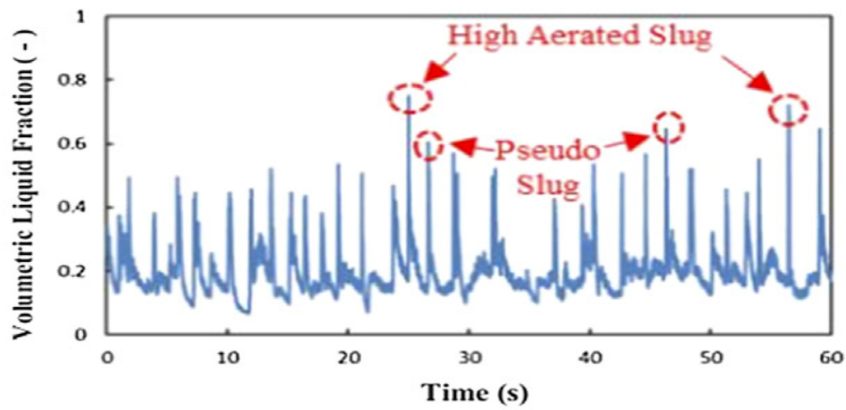


Figure D-9 Case 2 with $U_{SL} = 0.3 \text{ m/s}$ and $U_{SG} = 3.77 \text{ m/s}$

Case 3.

A horizontal pipeline of length 18 m and diameter 50 mm has been used for the transient simulation. The flow is assumed to three dimensional and adiabatic. Initial and boundary conditions have been set up. The following input data have been used. See Table D-4 below.

- Superficial velocity of liquid U_{SL} and gas U_{SG}
- Density and viscosity of liquid (ρ_L and μ_L) and gas (ρ_G and μ_G)
- Pipe diameter (D)
- Surface tension 0.072

Table D-4 Case 3

Air superficial velocity	0.7		3.77
Water superficial velocity	0.075	0.1	0.2
Air viscosity	1.7894e-05		
Air density	1.225		
Water viscosity	0.001003		
Water density	998.2		
Pipe diameter	50 mm		
Pipe length	18 m		
Air volume fraction	0.25 (Inner 8 mm)		
Water volume fraction	0.75		
Time	50 s	60 s	
Time step size	0.01		
Surface tension	0.072		

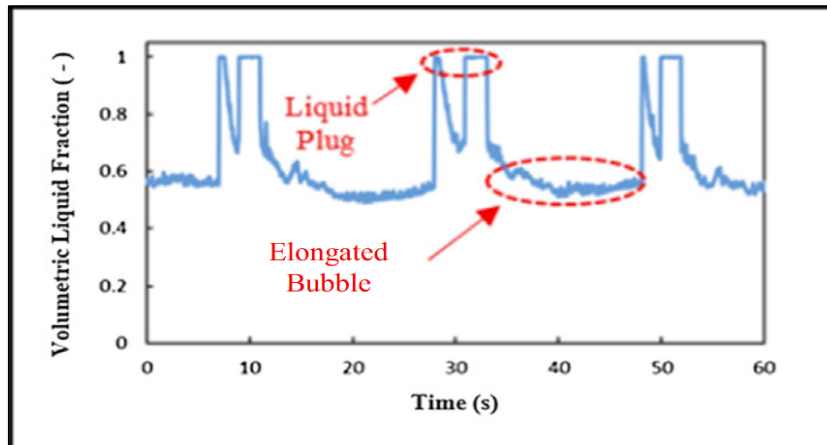


Figure D-10 Case 3 with $U_{SL} = 0.2 \text{ m/s}$ and $U_{SG} = 0.7 \text{ m/s}$

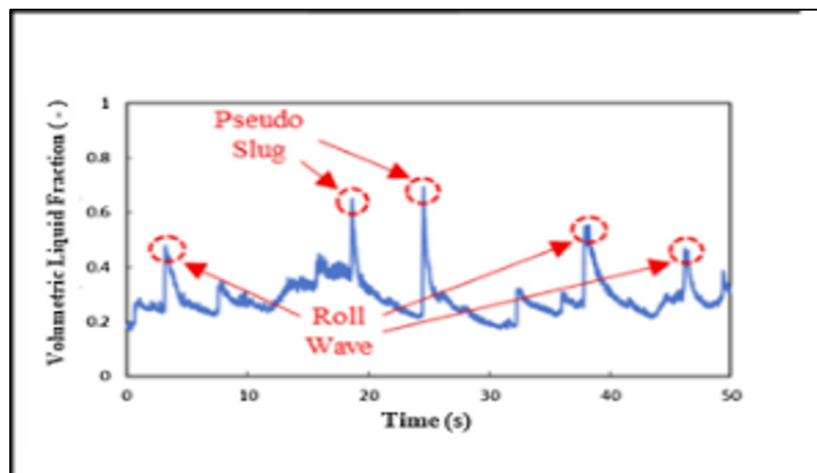


Figure D-11 Case 3 with $U_{SL} = 0.075 \text{ m/s}$ and $U_{SG} = 3.77 \text{ m/s}$

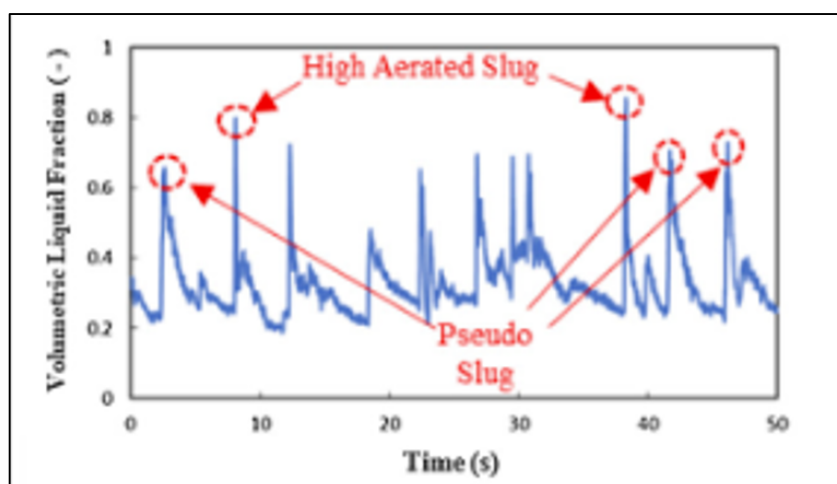


Figure D-12 Case 3 with $U_{SL} = 0.2 \text{ m/s}$ and $U_{SG} = 3.77 \text{ m/s}$