

*Republic of Iraq*  
*Ministry of Higher Education*  
*and Scientific Research*  
*University of Babylon*  
*College of Engineering*  
*Mechanical Engineering Department*



# **The Effect of Pipe Shape on Pressure Drop in a Two-Phase Vertical Flow (Fuel-Water)**

*A Project Submitted to the*  
*College of Engineering /the University of Babylon in*  
*Partial Fulfillment of the Requirements for the Degree of Higher-Diploma*  
*in Engineering/Mechanical Engineering/Fuel and Power*

**By**

***Mays Hameid Abd-ulhussein Ibrahim***

***Supervised by***

***Prof. Dr. Riyadh S. Salah Al-Turaihi***

***2022 A.D.***

***1443 A.H***

## *Dedication*

*To one who gave me unconditional support,  
inspiration, and strength, my best friend  
Mustafa Hadi*

*Mays 2022*

## Acknowledgements

Full achievement of any project cannot be achieved without thanking those who made it possible. First, I would like to express my special gratitude towards “*Allah*” for giving me the strength to keep going.

I would like to express the deepest appreciation to my supervisor *Dr. Riyadh S. Al-Turaihi* for his dedicated involvement, assistance, and guidance that helped me during the research and writing of this project, I could not do this without his help.

My sincere thanks also go to *Dr. Saba Yasoub* at the University of Babylon who helpfully answered my questions related to the fluent program.

My appreciation goes to all staff members of the Department of Mechanical Engineering at the University of Babylon, Head of the Department *Assist. Prof. Dr. Samer Mohammed* for his constructive contribution to this study.

A special thanks to *my parents, sisters, and brothers* whose prayers for me were what sustained me this far.

I would also like to thank all who helped me directly or indirectly to complete this work.

*MAYS*

*2022*

## Abstract

The influence of different cross-section (circular, square and triangle) of a stainless steel pipe on pressure drop of gasoil-water two-phase upward vertical flow has been investigated numerically. The flow area and length of the pipe for different cross-sections were suggested as  $45.6 \times 10^{-4}$  (m<sup>2</sup>) and 1 (m) respectively. The optimum cross-section (which was a circular pipe) has been chosen as a case to predict the effect of different range of velocities which were (0.7-1.1) m/s for gasoil and (0.35-0.55) m/s for water on the pressure drop along the pipe by using CFD. The pipe was simulated by ANSYS FLUENT 17.2 and drawn by SOLIDWORK 2018.

The study was involved to predict the effect of exists an obstacle in the stream of fluids flow on the performance of the system and how the size of the obstacle can affect the pressure drop in the vertical upward flow. The results showed that the circular cross-section pipe has the lowest pressure drop in comparison with the other cross-sections. In addition, the gasoil-water superficial velocity, the rectangular obstruction, and the size of obstruction have a direct proportional influence on pressure drop along the pipe.

The conclusions drawn from this study were observed that the pressure drop increased by 35% as gasoil superficial velocity increased from 0.7 m/s to 0.9 m/s for a given water superficial velocity (0.35 m/s). The pressure drop increased by 4% as water superficial velocity increased from 0.35 m/s to 0.45 m/s for a given gasoil superficial velocity (0.7 m/s) at a distance 0.2 m.

The rectangular obstruction has a direct proportional influence on the pressure drop; the existence of obstruction inflow stream led to an increased the pressure drop to 68% for constant velocities of the gasoil in 0.7 m/s and the water in 0.55 m/s at 0.2 m. The size of obstruction affect pressure drop directly. The increase of obstruction size to 125%, led to an increase in the pressure drop by 18%, and decreasing obstruction size to 25%, led to a decreased in the pressure drop by 35% at a distance 0.2 m.

# Table of Contents

Acknowledgments .....	I
Abstract .....	II
Table of Contents .....	IV
Nomenclature .....	VI

## Chapter One – Introduction

1.1. General Concept .....	1
1.2. Flow Regimes .....	2
1.3. Vertical Flow Patterns .....	2
1.4. Pressure in Two-Phase Flow .....	4
1.5. Objectives of Present Work .....	5

## Chapter Two – Literature Review

2.1. Vertical and Horizontal Flow Studies without Obstruction .....	6
2.2. Vertical and Horizontal Flow Studies with Obstruction .....	10
2.3. Scope of Present Work .....	13

## Chapter Three – Numerical Analysis

3.1. Introduction .....	14
3.2 Geometrical Model .....	14
3.2.1 Geometry .....	14
3.2.1.1 Three Pipes Geometries .....	14
3.2.1.2 Circular Pipe Geometry with Obstruction .....	16
3.2.2 General Mesh Strategy .....	17
3.2.2.1 Three Pipes Mesh .....	17
3.2.2.2 Circular Pipe Mesh with Obstruction .....	18

3.3 Boundary Condition .....	19
3.4 Governing Equations .....	22
3.5 Turbulent Model Equations .....	24
3.6 Assumptions .....	24
3.7 Simulation Steps .....	25

#### **Chapter Four – Results and Discussions**

4.1. Program Validation .....	26
4.2 Effect of Pipe Shape on Pressure Drop without Obstruction .....	27
4.3 Effect of Velocity on Pressure Drop with Obstruction .....	28
4.4 Effect of Rib Length on Pressure Drop .....	33

#### **Chapter Five– Conclusions**

5.1. Conclusions .....	39
5.2. Recommendations .....	40

<b>References</b> .....	41
-------------------------	----

# Nomenclature

## *Latin Symbols*

<b>Symbol</b>	<b>Definition</b>	<b>Unit</b>
C	Constant	-
d	Diameter of the bed	m
F	Body force	N
g	Gravity acceleration	m/s <sup>2</sup>
H	Initial height of solid particles	m
K	Momentum exchange coefficient	-
L	Length of the pipe	m
n	Number of phases	-
P	Pressure	Pa
t	Time	s
v	Velocity	m/s

## *Greek Symbols*

<b>Symbol</b>	<b>Definition</b>	<b>Units</b>
$\partial$	Partial differential	-
$\epsilon$	Turbulent dissipation rate	(m <sup>2</sup> /s <sup>2</sup> )
$\sigma$	Surface tension	(kg/m)
$\rho$	Density	(kg/m <sup>3</sup> )
$\alpha$	Volume fraction	-
$\mu$	Viscosity	(N/m <sup>2</sup> .s)

## *Subscripts Symbols*

<b>Symbol</b>	<b>Definition</b>
m	Mixture
k	Phase
<i>dr</i>	Drift
w	Water
oil	Gasoil

## *Superscripts Symbols*

<b>Symbol</b>	<b>Definition</b>
n	Number of phases

## *Abbreviations*

CFD	Computational Fluid Dynamics
CWF	Constant Water Flow rate
CTF	Constant Total Flow rate
WC	Water Cut
BPD	Barrels per day
FPD	Frictional pressure drop
ID	Inner Diameter
VOF	Volume of Fluid
RNG	Renormalization Group

# **CHAPTER ONE**

## **INTRODUCTION**

# Introduction

## 1.1 General Concept

The term multiphase flow means the fluid flow consisting of two or more phases or components. A phase is actually can be one of the states of matter (gas, liquid, or solid). Considerable pumping power costs can be reduced by a systematic use of multiphase flow. The simplest and most common case of multiphase flow is two-phase flow and it is defined as a mixture of two phases flowing simultaneously through the system [1].

There are many types of two-phase flow, such as liquid-liquid flow, solid-liquid flow, gas-liquid flow, and gas-solid flow. Two-phase flow is more complex than single-phase flow and unsteady due to the difference between the material properties of each phase [2].

Two-phase flow is very common in many science and engineering fields, such as agricultural, biomedical, chemical, food science, and petroleum engineering. Some other applications of two-phase flow exist in condensers, evaporators, gas lift pumps, boiler tubes, oil and gas transportation pipelines, etc.

The presented study in this project focused on the pressure in liquid-liquid two phase-flow through pipelines. The pressure prediction in such systems is sophisticated because it depends on several parameters. An example of liquid-liquid two-phase flow is the flow of gasoil and water. The injection of water into the flowing gasoil in the pipeline reduces flow resistance and pressure gradient. As a result, large pumping units are not required. By accurately predicting void fraction and pressure, this technique would improve the safety and overall performance of the two-phase flow system.

## **1.2 Flow Regimes**

Flow patterns or flow regimes are the difference in interracial structures. One of the important parameters to describe two-phase flow is the flow regime, reflecting the pattern and inner structure of the flow, and two-phase flow regimes identification is becoming increasingly important in many industrial processes, such as petroleum, chemical, and physical industrial processes [3]. Depending upon the respective distribution of individual phases, a two-phase flow may change from nearly all liquid to all gas flow .

The vertical liquid-liquid streams regimes are droplet, elongated droplet, annular core, and churn. The regimes for horizontal liquid-liquid streams stratified with a wavy stream with droplets at the interface, scattering, and annular flow [4].

## **1.3 Vertical Flow Patterns**

The presence of an interface between the two phases complicates the description of two-phase flow in tubes. This interface exists in a wide variety of forms, depending on the geometry and inclination of the tube as well as the flow rates and physical properties of the phases.

Five main flow patterns may be distinguished for the particular case of upwards flow in vertical tubes. The main features are described below [5]. As shown in Figure (1-1).

1. Bubble flow: The liquid phase flows continuously in bubble flow, while the gas phase flows as discrete bubbles in a liquid continuum.. The bubbles are distorted spheres.
2. Slug flow: bubble coalescence occurs when the bubble concentration in bubble flow becomes high, and the largest bubbles are of the same

order of size as the tube diameter. Further intermingling results in deforming the bubble's shape into the bullet-shaped pocket of gas, which is characteristic of plug flow. Plug flow then consists of these pockets of gas, commonly called plugs or Taylor bubbles, separated by regions of bubbly flow, commonly called slugs. A thin liquid film, which flows vertically downwards, surrounds the plugs of gas.

3. Churn flow: Churn flow is a highly troubled flow regime in which the vertical motion of the liquid is oscillatory. Churn flow possesses some of the characteristics of plug flow, with the main differences being as follows:
  - a. The gas plugs become narrower and more irregular.
  - b. The regions of high gas concentration repeatedly destroy the continuity of the liquid in the slug.
  - c. Can no longer observe the thin falling film of liquid surrounding the gas plugs.
4. Annular flow: In annular flow, the gas flows along the center of the tube. The liquid flows partially as a film along the walls of the tube, and in the central gas, the core flows partially as droplets.
5. Mist flow: A mist flow has the gas (or vapor) as the continuous phase and numerous very small droplets entrained in it, whose presence can often be difficult to observe without the help of illumination and/or magnification.

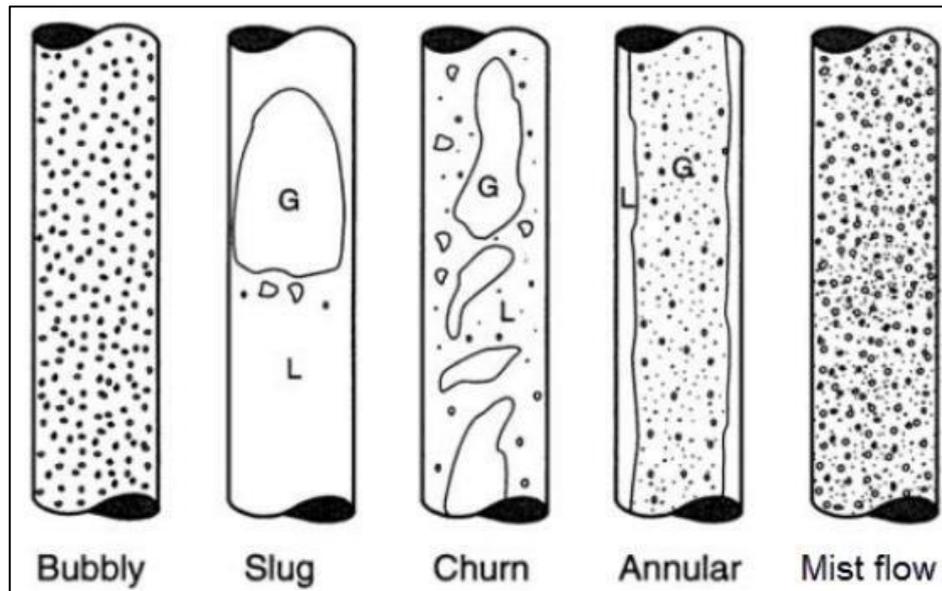


Figure (1-1): Two-phase flow patterns in a vertical flow [6].

## 1.4 Pressure in Two-Phase Flow

Multiphase flow has become increasingly important in a wide variety of science and engineering systems such as power, heat transfer, and transport systems. In these systems, it is necessary to determine the pressure for optimum design and safe operations. The pressure gradient in two-phase flow can be derived from the momentum equation [7].

The information about pressure is usually necessary to dimension new devices, decide on the setup peripheries, such as pumps, valves, etc. [8]. The pressure gradient due to the friction in two-phase flows is the most complex and difficult one to predict, because of its dependency on pipe inclination, flow regime, and pipe roughness [7].

## **1.5 Objectives of Present Work**

The present work is interested in studying the pressure during two-phase flow in different shapes of pipes, with and without rectangular obstruction, via computational simulation.

The specific objectives of this research are:

1. To model and analysis of liquid-liquid, two-phase flow through vertical pipe (upward flow) with and without obstruction by using ANSYS FLUENT code.
2. To study three different shapes of pipes (circular, square, and triangle) with the same measurement of area, same material, and same boundary conditions.
3. To investigate how the change of velocity of gasoil as prime phase and water as the second phase can affect the pressure of the two-phase flow.
4. To investigate how the existence of rectangular obstruction in the pipe can affect the pressure of the two-phase flow.

**CHAPTER TWO**  
**LITERATURE REVIEW**

## Literature Review

The purpose of this chapter is to observe abbreviated of the prior researches, about the two-phase flow. In particular, the liquid-liquid flow type will be introduced. The flow behavior and pressure drop in smooth and ribbed channels have been researched by many researchers. The flow of earlier investigations in this chapter was categorized; vertical and horizontal flow without obstruction, and vertical and horizontal flow with obstruction.

### **2.1 Vertical and Horizontal Flow Studies without Obstruction**

The researcher (**Jana 2007**) [9] investigated experimentally the pressure drop and liquid holdup that occurs when two liquids flow through a vertical pipe. Kerosene and water were chosen as the liquids, velocity range (0.05 - 1.2) m/s for both liquids. The results showed that the pressure drop initially decreases, reaches a minimum value, and then increases as kerosene velocity increased at a constant water flow rate. As kerosene velocity increases, water holdup decreases.

(**Shannak 2008**) [10] conducted an experimental study of air-water two-phase flow frictional pressure drop of horizontal and vertical smooth and rough pipes. In this study, the frictional pressure drop was considered as a function of the roughness of the pipe, vapor quality, and mass flux. The results showed that at a given constant system pressure, the frictional pressure drop increased with higher mass flux, higher vapor quality, and higher relative roughness.

(Liu 2010) [11] attempted to quantify Pressure drops during the co-current flow of mechanical oil and tap water in a 40mm inner diameter horizontal pipe. Superficial velocity for oil ranged from 0.04 to 1.2m/s, and for water from 0.04 to 2.2m/s. The water volume fraction in the oil-water mixture varied from 0.05 to 0.976. The frictional pressure loss was investigated experimentally and theoretically. It was found that the pressure drop increased as the velocity of oil and water increased, and the model predictions agreed well with the experiment data.

Experimental research has been conducted by (Ismail 2014) [12] to study the pressure drop of the Malaysian waxy crude oil- at ambient conditions through the horizontal pipeline. Mixture velocities were varied from (0.1 – 0.8) m/s. The results showed that pressure drop related to flow rates, where the pressure drop was increased by increasing the flow rates.

The researcher (Xiong 2014) [13] focused on the pressure gradient for gas-liquid bubbly flow through a vertical pipeline with a length of 1.52 m and an inner diameter of 11.68 mm.by using a Computational Fluid Dynamics (CFD) numerical simulation. The results were compared with experimental data. Pressure gradients were predicted by the CFD model which showed a good agreement with the experimental data. It showed that the pressure gradient was very sensitive to liquid velocity since the pressure gradient increased sharply with the increase in liquid velocity, while the effect of gas velocity was relatively slight on the pressure gradient.

Numerical data were presented by (Sanati 2015) [14] for water and air velocity in two-phase upward flow through a vertical circular channel. The obtained results were compared with the experimental data and

showed that the void fraction increased with an increase of gas velocity and decreased with the increase of water velocity. The pressure drop decreased as the gas velocity increased and liquid velocity decreased. In addition, increasing the height was accompanied by decreasing the pressure drop.

(Ali 2017) [15] studied experimentally and numerically the effect of the volumetric quality on pressure drop of gas-liquid flow in horizontal pipes of different diameters. Three pipes were designed for the experiment with a diameter of 12.70, 19.05, and 25.40 (mm). For constant water flow rate (CWF), the measurements were performed by adding air to the water at a constant total flow rate (CTF) in which the flow rates for both phases were changed to give the same CTF. The results showed that the pressure drop increased with higher volumetric qualities for the cases of CWF but it decreased for higher volumetric qualities of CTF because of the change in the flow pattern.

(Cihang Lu 2018) [7] presented an experimental study of a frictional pressure drop in two-phase air-water flows in straight pipes diameters of 38.1 mm, 50.8 mm, and 101.6 mm. A systematic investigation of the impacts of flow orientation was carried out. The results showed that the frictional pressure gradient increased with the increase of the gas superficial velocity of air with constant water superficial velocity in all orientations (vertical (downward and upward), horizontal).

(Wordu 2019) [16] used a mathematical model, multiphase flow correlations, statistical model, and computer programming to predict the pressure drop. The overall pressure gradient for a total depth of 10,000 feet (3,048 m) was anticipated using Hagedorn & Brown, Griffith & Wallis correlations of models, which were compared to field data. The

measurement predicted pressure gradient was 0.320778 (psi/ft) (2.21 kPa), Graffith & Wallis 0.382649 (psi/ft) (2.64 kPa) , Hagedorn & Brown 0.382649 (psi/ft) (2.64 kPa), and the produced model 0.271514 (psi/ft) (1.87 kPa). These findings suggested that the model equation was created superior and produced a reasonably accurate pressure drop gradient prediction based on the measured pressure gradient.

Hagedorn & Brown

$$\frac{dp}{dL} = \frac{g}{g_c} \rho_m + \frac{f_{tp} \rho_f v_m^2}{2g_c d} \quad (2-1)$$

Griffith & Wallis

$$\frac{dp}{dL} = \frac{g}{g_c} \bar{\rho} + \frac{2f\rho_l v_l^2}{2g_c d} \quad (2-2)$$

Where,

$dp/dL$  = pressure drop, psia/ft

$\rho_m$  = mixture density Ibm/ft<sup>3</sup>

$V_{sg}$  = gas superficial velocity, ft/sec

$d$  = pipe diameter, ft

$f_{tp}$  = Two phase friction factor

$g_c$  = Conversion constant (32.17) Ibmft/Ibf

$g$  = acceleration due to gravity ft/sec<sup>2</sup>

$\rho_l$  = liquid density Ibm/ft<sup>3</sup>

$\bar{\rho}$  = mixture density Ibm/ft<sup>3</sup>

The effect of viscosity on pressure drop measurements of oil-water annular two-phase flow in a horizontal 6-inch (15.24 cm) diameter stainless steel pipe was investigated by (Basha 2020) [17]. Working fluids were included two different mineral oils, Exxsol D80 and Exxsol D130, as well as water. Experiments were conducted for various water cut (WC) ratios (0

to 100 percent). The oil-water flow rates ranged between 2000 and 12000 barrels per day (BPD). Frictional pressure drop (FPD) has been found to decrease from WC = 0 to WC 20% at a given flow rate. Further increase in WC led to an increase FPD in up to 40%. The decrease in FPD was not significant when WC is greater than 40%. The FPD of Oil (D130) was 7% which was higher than the FPD of Oil (D80) in a specific case with WC 40% (horizontal,  $\theta = 0^\circ$  case, flow rate = 10000 BPD).

(Liu 2021) [18] carried out air and viscous oil two-phase flow experiments for 60 mm diameter test section. The viscosity range of the oil was (100-480 mPa.s). The experimental results showed that there were a negative and a positive frictional pressure drops in the churn flow, based on the measured liquid holdup and pressure drop data of churn flow. The variation in frictional pressure drop with increasing superficial gas velocity was associated with superficial liquid velocity. The rate of change of the frictional pressure drop increased, with increasing the superficial gas velocity, as the viscosity increased.

## **2.2 Vertical and Horizontal Flow Studies with Obstruction**

(Mahood 2009) [19] Presented an analytical solution for pressure drop caused by obstructions in horizontal two-phase air-water flow. A variety of obstruction shapes and sizes were investigated. The results showed that obstructions intercepting the liquid phase would cause large pressure drop. It was found that the pressure drop for the 40% obstruction area has a higher value than 25%.

(Al-Turaihi 2013) [20] investigated 2-D numerical simulations and visualization experiments the two-phase flow phenomena and the effect of pressure difference in two-phase flow around a triangular-section cylinder

in a rectangular channel enlarged from assembly circular tube of the two phases. Varied air and water flow rates were taken. The results showed that high turbulence was appeared when air discharge increased which generated more waves and bubbles. The effect of existence a triangular-section cylinder was clear in dividing the two-phase flow, generating vortices, and finally enhancing the mixing of the two-phase flow. The mean pressure difference increased when air or water discharge increased.

The researcher (**Bai 2014**) [21] carried out an experimental study on swirling gas-liquid two-phase flow pattern and pressure drop inside a 62mm ID vertical pipe with a helical tape insert as shown in figure (2-1). The experimental results were obtained in a 7m vertical test section for different liquid and gas velocities. When the flow was annular, the pressure drop in the non-swirling flow was lower than that in the swirling flow at a low liquid velocity, while in the churn flow, the pressure drop in the swirling flow was greater than in the non-swirling flow.



Figure (2-1): Helical tape insert structure.

(**Al-Turaihi 2018**) [22] investigated the flow of two-phase in a ribbed convergent rectangular upward vertical duct. Water and air have been used as two-phase materials. A numerical study has been performed to test the influence of increased air and discharged water on the pressure drop through the convergent section. The discharge of the inlet water and air was (5-20) L/min and (5.833-16.666) L/min, respectively. The results showed that as the discharge of the water and air increased, the pressure along the test channel increased, and when the convergence angle increased, the pressure drop increased.

(**Habeeb 2019**) [23] investigated experimentally the two-phase flow through a vertical transparent pipe. The experimental rig has been designed to measure pressure drop for various combinations of phases through the obstruction. It was used a wide volumetric range of water (5 l/min - 20 l/min) and air (8.3334 L/min - 25 l/min) for a pipe with 50 mm internal diameter along 1 m length. The pressure has measured using four pressure sensors placed along the test pipe. The measured pressure values have been used for various air and water volumetric calculations. The results showed that the measuring of the pressure gradient in a distance along the rig pipe was inversely related to air volumetric. The pressure drop was higher in the region close to the obstruction area.

(**Kiran 2019**) [24] investigated experimentally and numerically two-phase flow characteristics in the annulus. The experimental study included two sets of five tests with increasing superficial gas velocity from (9.2 to 47.2 m/s) at a constant liquid flow rate. Using CFD simulation, pressure drop was determined. The results showed that the pressure gradient increased with superficial gas and liquid velocities.

The researcher (Majdi 2020) [25] carried out experimentally the pressure drop in a ribbed vertical channel for a two-phase (air and water) flow. Varied shapes of ribs (rectangle, semi-circular, and triangle) were used in this investigation. The air superficial velocities were (0.0555 - 0.1666 m/s), while the water superficial velocities were (0.0333 - 0.0888 m/s). The results showed that if the discharge of water or air increased, the average pressure difference increased. The triangular ribbed channel recorded the lowest pressure drop, which indicated that the finest shape among the studied ones was the triangular-shaped ribbed channel.

### **2.3 Scope of Present Work**

The research work is devoted to the horizontal and vertical two-phase flow. Some of these investigations have been done to analyze the two-phase flow behavior in inclined and vertical downward systems but the most investigations were limited to the study of flow patterns.

The previous studies have paid little attention to investigating the effect of pipe shape on the behavior of the two-phase flow in the vertical pipe and the importance of the major design parameters like flow rate, pipe elevation, and fluid velocity on the performance of the system.

In the present work, three pipes with the same flow area and different section shapes (circular, square, and triangle) have been used. It have been modeled employing the computational fluid dynamic to study the effect of pipe section shape on the pressure of liquid-liquid two-phase upward flow through a vertical pipe.

The study also involved the effect of exists an obstacle in the stream of fluids flow on the performance of the system and how the size of the obstacle can affect the pressure in the vertical upward flow.

# **CHAPTER THREE**

## **NUMERICAL ANALYSIS**

## Numerical Analysis

### **3.1 Introduction**

In this study, ANSYS FLUENT 17.2 is used to simulate and analyze the flow characteristics of two-phase flow (gasoil-water) through the test channel. The effect of pipe section shape, gasoil and water discharge on the pressure are studied. Computational fluid dynamics is used to understand such a system because it is an effective tool for predicting flow behavior.

The solution in FLUENT is divided into four parts: physical model geometry, mesh generation, physical model setup, and post-processing. In ANSYS FLUENT 17.2, the multiphase Euler-Euler models are classified into three types: volume of fluid (VOF), mixture, and Eulerian (**FLUENT User`s Guide**) [26]. In the present work, two-phase flow through three sections of the pipe and one rectangular rib are modeled using a mixture model and K- $\epsilon$  turbulence model with RNG.

### **3.2 Geometrical Model**

#### **3.2.1 Geometry**

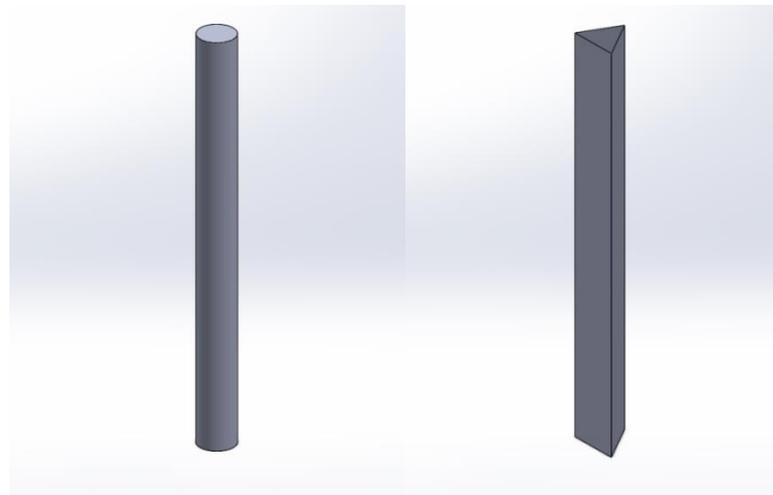
##### **3.2.1.1 Three Pipe Geometries**

A SOLIDWORK 2018 is used to draw the geometry of the system as a three-dimensional structure for a numerical simulation of liquid-liquid flow through the section. The first geometry, is the circular pipe which was drawn with a diameter 0.0762 m and 1 m length. The second pipe was a square section with side length 0.0675 m and 1 m length, and the last pipe was a triangle section with side length 0.01026 m and 1 m length as shown

in Table (3-1). The section area was  $45.6 \times 10^{-4} \text{ m}^2$  for all pipes. Figure (3-1) shows the structure of three pipes.

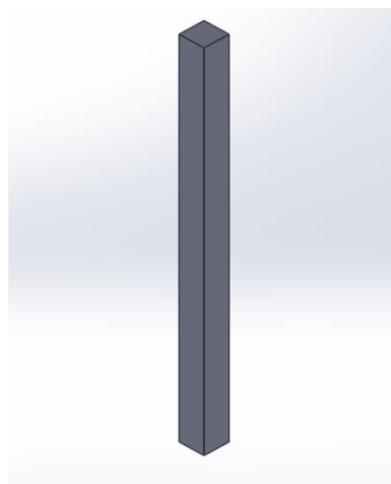
Table (3-1): Pipe Section Shape Diameters.

Pipe Section Shape	Diameter (m)
Circular	0.0762
Square	0.0675
Triangle	0.01026



(a)

(b)



(c)

Figure (3-1): (a) Circular (b) Triangle, and (c) Square pipe sections.

### **3.2.1.2 Circular Pipe Geometry with Obstruction**

The circular pipe is modeled in a two-dimensional structure with dimensions 0.0762 m diameter and 1 m length with rectangular rib inside of dimensions 0.01905×0.0127 m (  $\frac{1}{2}$  R of the circular pipe), located at 0.25L from the inlet, which is simulated by using SOLIDWORK 2018, as shown in Figure (3-2).

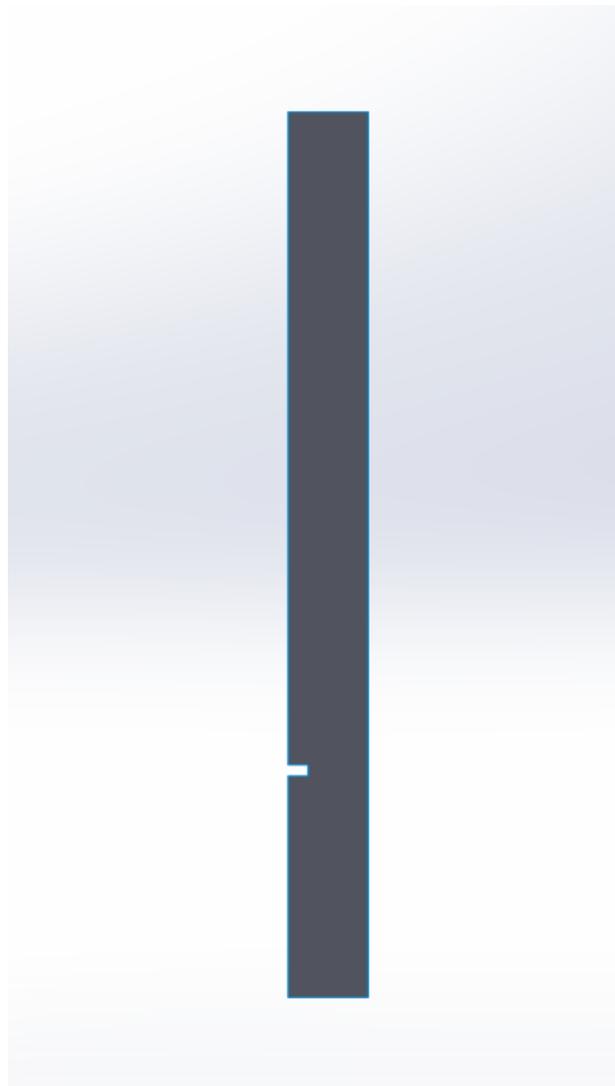
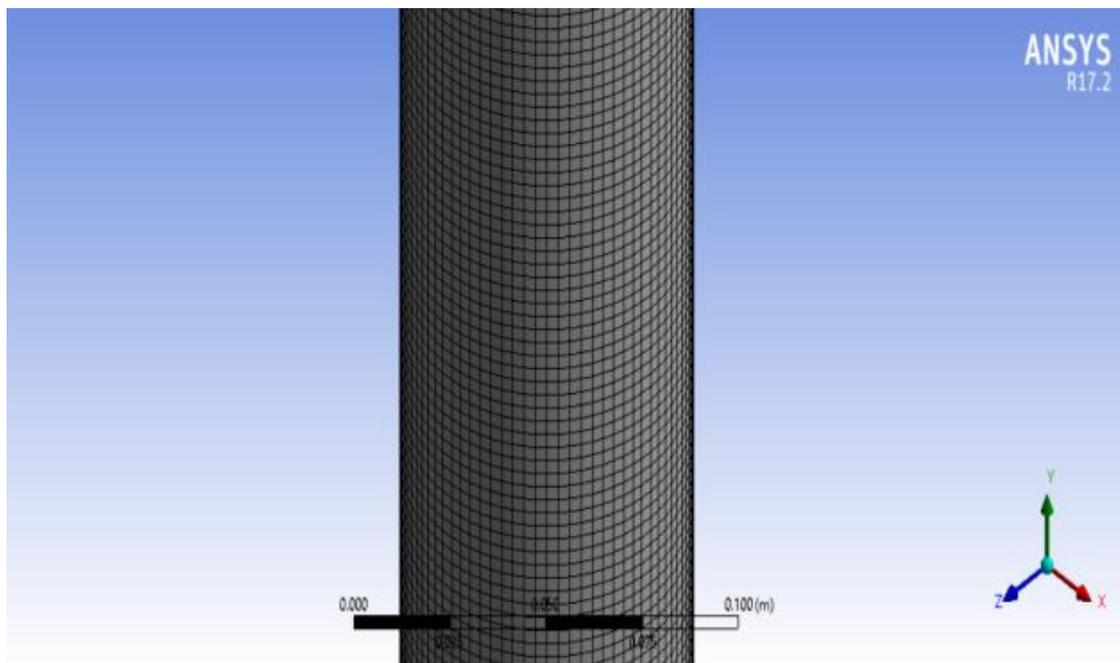


Figure (3-2): Circular section with rectangular obstruction.

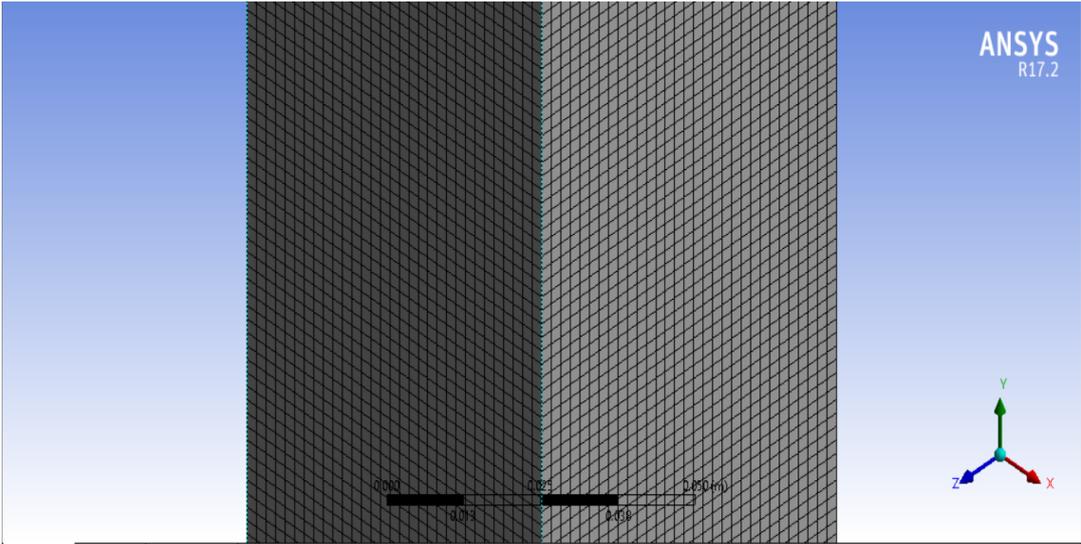
## **3.2.2 General Mesh Strategy**

### **3.2.2.1 Three Pipes Mesh**

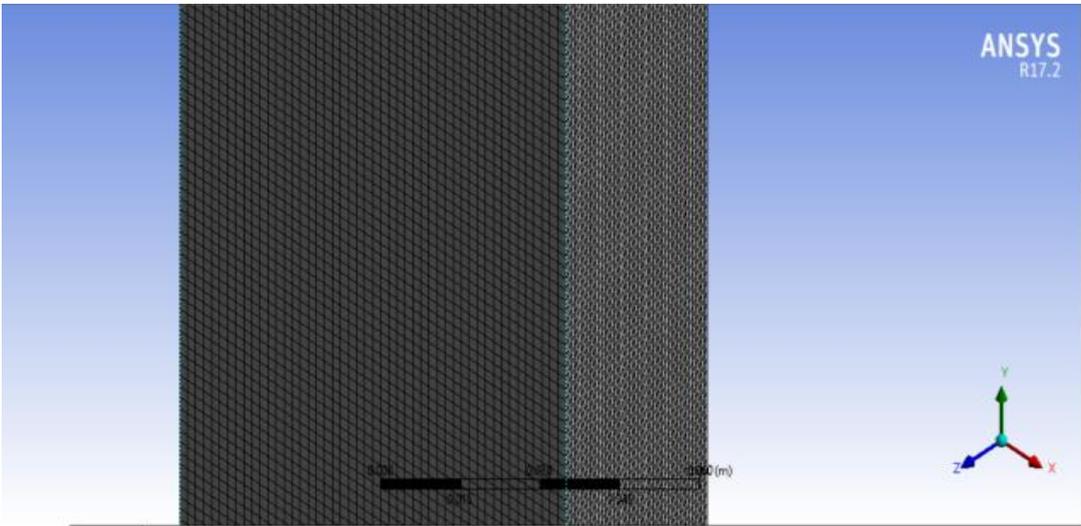
Meshing is a critical step in the numerical simulation process. The mesh has an impact on the solution's accuracy, convergence, and speed (FLUENT User's Guide (2006)). Meshing was performed in a 3-D structure using the ANSYS Workbench 17.2 and Quadrilateral dominant method. The geometry is divided into small square elements with a size of 0.003cm for maximum and minimum discretization, as show in Figure (3-3). The results are fixed at these mesh size values (number of nodes and elements for test section is 404880 and 391280, respectively) because this mesh satisfy the accuracy of the numerical procedure. Figures (3-4) represent the convergence of model.



(a)



(b)



(c)

Figure (3-3): 3D structure mesh of (a) circular (b) triangle, and (c) square pipe sections

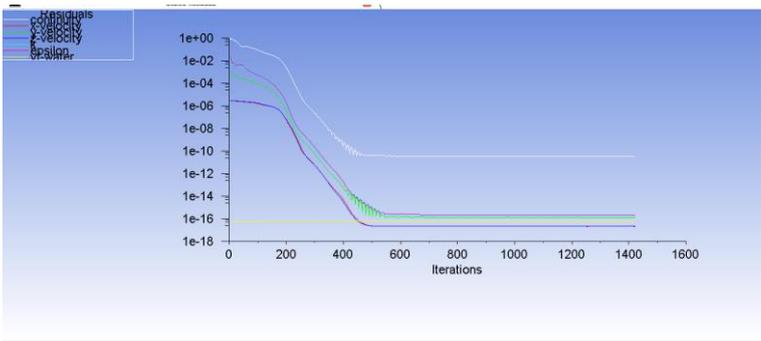


Figure (3-4): Two Phases Residuals Plot for 3D structures.

### **3.2.2.2 Circular Pipe Mesh with Obstruction**

Meshing was performed in 2-D structure using the ANSYS Workbench 17.2 and Quadrilateral dominant method. The geometry divided into small square elements with a size of 0.001cm for maximum and minimum discretization, as show in Figure (3-5). The results are fixed at these mesh size values (number of nodes and elements for test section is 75361 and 74266, respectively) due to satisfy the accuracy of the numerical procedure.

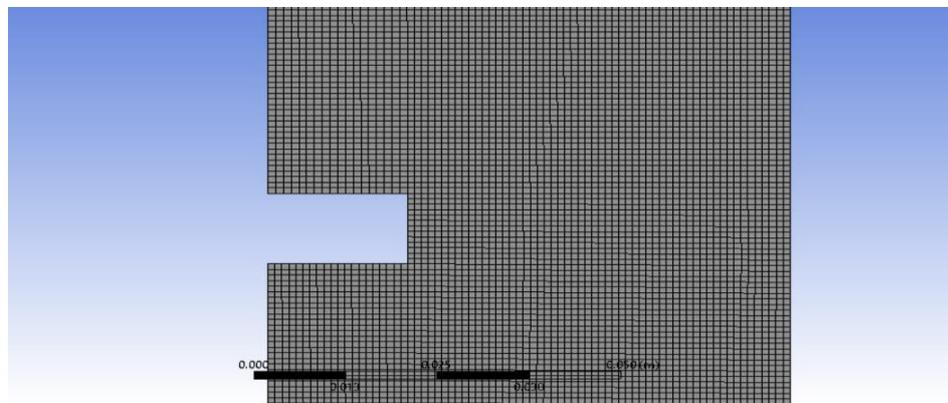
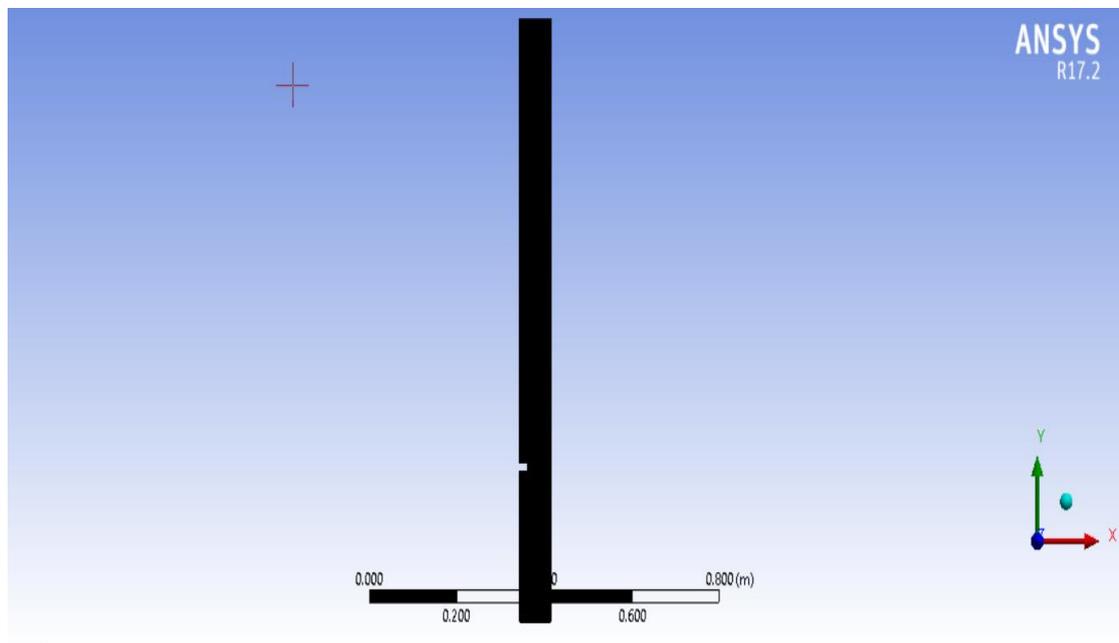


Figure (3-5): 2-D structure mesh.

### **3.3 Boundary Conditions (B.C)**

The bottom of the 3D structure was set as the inlet of the fully developed mixture and the top set as the outlet. All sides of the structure are designed to be as adiabatic walls, as shown in Figure (3-6).

The bottom of the 2-D structure divided into 13 elements. The side elements is considered the entry point of the primary phase, which is gasoil while the middle elements is considered the entry point of the secondary phase, which is water and the top edge as the outlet, as shown in Figure (3-7). All sides of the structure set as adiabatic walls. Velocities taken in study and physical properties shown below in tables (3-2), (3-3) and (3-4).

Table (3-2): The physical properties of gasoil and water.[26]

<b>Property</b>	<b>Water</b>	<b>Oil</b>
Density (kg/m <sup>3</sup> )	998.2	830
Dynamic viscosity (kg/m-s)	0.001003	0.00332

Table (3-3): Superficial velocities of water and gasoil.

<b>Case 1</b>	<b>Gasoil velocity= 0.7 m/s</b>		
<b>Water velocity m/s</b>	0.35	0.45	0.55
<b>Case 2</b>	<b>Gasoil velocity= 0.9 m/s</b>		
<b>Water velocity m/s</b>	0.35	0.45	0.55
<b>Case 3</b>	<b>Gasoil velocity= 1.1 m/s</b>		
<b>Water velocity m/s</b>	0.35	0.45	0.55

Table (3-4): Model relaxation

Variables	Relaxation factors
Pressure	0.3
Density	1
Slip velocity	0.1
Momentum	0.5
Turbulent kinetic energy	0.8
Turbulent dissipation energy	0.8
Turbulent viscosity	1

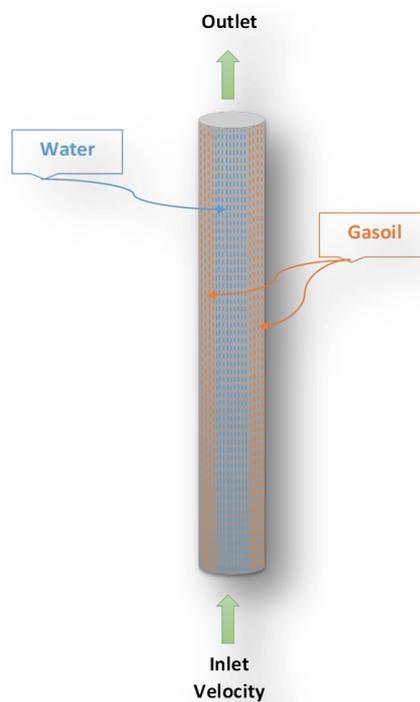


Figure (3-6): 3-D Circular pipe boundary conditions

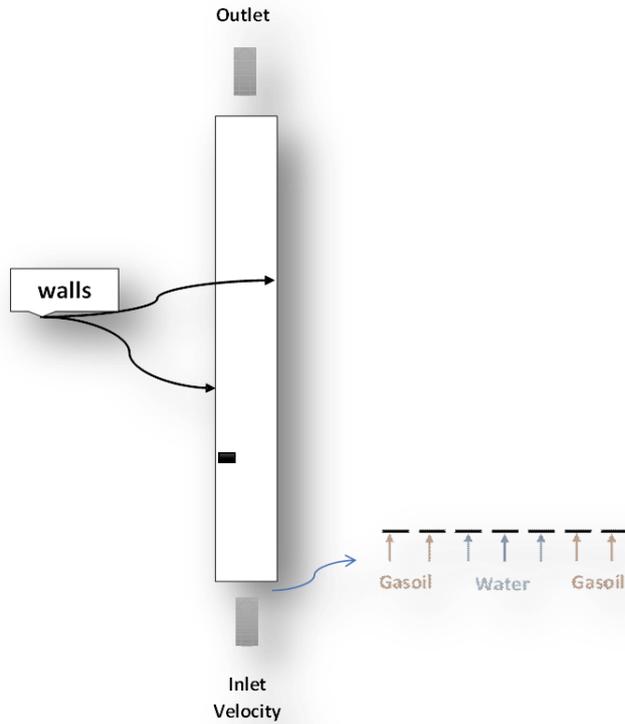


Figure (3-7): 2-D circular pipe boundary conditions.

### **3.4 Governing Equations**

For two-phase flow simulations, the mixture model is implemented. A blend model is used to solve the continuity, conservation and momentum equations for each phase. This model phases travel at different velocities. The volume fraction of the phase calculated using the continuity equation. As a result, the volume fractions of the primary and secondary phases for a control volume can be selected in any value in the range (0 - 1) [26], depending on the occupied space by the primary and secondary phases.

The governing equations of the mixture multiphase model are [20]:

**Equation of continuity represented by:**

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

Where,

$(v_m)$ : Mass-averaged velocity and can be written in this form:

$$\vec{v}_m = \frac{\sum_{k=1}^n \alpha_k \rho_k \vec{v}_k}{\rho_m} \dots \dots \dots (3 - 2)$$

$\rho_m$ : Mixture density

$$\rho_m = \sum_{k=1}^n \alpha_k \rho_k \dots \dots \dots (3 - 3)$$

$\alpha_k$ : Phase (k) volume fraction

**Momentum equation represented by:**

The general shape of such equation is given by:

$$\begin{aligned} \frac{\partial}{\partial t} (\rho_m \vec{v}_m) + \nabla \cdot (\rho_m \vec{v}_m \vec{v}_m) \\ = -\nabla P + \nabla \cdot [\mu_m (\nabla_{\vec{v}_m} + \nabla_{\vec{v}_m} T)] + \rho_m \vec{g} + \vec{F} \\ + \nabla \cdot \left( \sum_{k=1}^n \alpha_k \rho_k \vec{v}_{dr, k} \vec{v}_{dr, k} \right) \dots \dots \dots (3 - 4) \end{aligned}$$

Where,

n: Phases number

$\vec{F}$ : Body force

$\mu_m$ : Viscosity of mixture is given by:

$$\mu_m = \sum_{k=1}^n \alpha_k \mu_k \dots \dots \dots (3 - 5)$$

Where,

$\vec{v}_{dr}$ : Velocity of drift for the secondary phase (k), and

$$\vec{v}_{dr} = \vec{v}_k - \vec{v}_m \quad \dots \dots \dots (3 - 6)$$

### **3.5 Turbulent Model Equations**

For this computational study, the k-epsilon turbulent model with RNG mixture model are used. The general equations for this model are [30]:

$$\frac{\partial}{\partial t} (\rho_m k) + \nabla \cdot (\rho_m \vec{v}_m k) = \nabla \cdot \left( \frac{\mu_{t,m}}{\sigma_k} \nabla k \right) + G_{k,m} - \rho_m \epsilon \quad \dots \dots \dots (3 - 7)$$

$$\frac{\partial}{\partial t} (\rho_m \epsilon) + \nabla \cdot (\rho_m \vec{v}_m \epsilon) = \nabla \cdot \left( \frac{\mu_{t,m}}{\sigma_\epsilon} \nabla \epsilon \right) + \frac{\epsilon}{k} (C_{1\epsilon} G_{k,m} - C_{2\epsilon} \rho_m \epsilon) \quad \dots \dots \dots (3 - 8)$$

Where,

C: Constant

k: Momentum exchange coefficient

$\epsilon$ : Turbulent dissipation rate ( $m^2/s^2$ )

$\sigma$ : Surface tension (kg/m)

### **3.6 Assumptions**

The model is assumed to be:

- 1- Steady flow.
- 2- Incompressible flow.
- 3- Adiabatic.
- 4- Turbulent flow.
- 5- Pressure-based solver.

### **3.7 Simulation Steps**

For two-phase flow modeling by using a dynamic computer fluid model, the following steps are implemented:

- 1- SOLIDWORK is used to configure the model geometry.
- 2- ANSYS Workbench 17.2 is used to generate the mesh of two-phase field.
- 3- Choosing a model (mixture).
- 4- Specify the definition of material.
- 5- Choosing phases and boundary conditions.
- 6- Obtain the initial solution.
- 7- The solution is run with a maximum iteration (1400) for 3D structures and (2500) for 2D structure.
- 8- The results of the pressure and the air volume fraction are extracted.

# **CHAPTER FOUR**

## **RESULTS AND DISCUSSION**

## Results and Discussion

This chapter presents the results and discussion of the numerical work for the two-phase flow performed in this study. A comparison between the present work and other research`s work has been made.

Models are used for three pipe sections in different shapes with the same area of the flow section: circular, square, and triangle sections. The effect of changing the shape of the pipe for constant velocities on the pressure drop is studied. The effect of increasing the gasoil velocity from (0.7) to (1.1) m/s, the water velocity from (0.35) to (0.55) m/s for 1 m long circular pipe on the pressure drop is investigated.

### 4.1. Program Validation

To verify the numerical code, the calculated pressure drop values compared to the numerical results of **Sarah Oleiwi's** [27]. Multi-phase Flow-through Fluidized Bed Column for water velocity is 0.33 m/s and the ratio of initial height of solid particles to the diameter of the bed (H/D ratio) 1.18 are used.

The numerical results of the pressure drop from this study and the numerical results of **Sarah Oleiwi** [27] were found to be in good agreement with a percentage error of (4%) as shown in Figure (4-1).

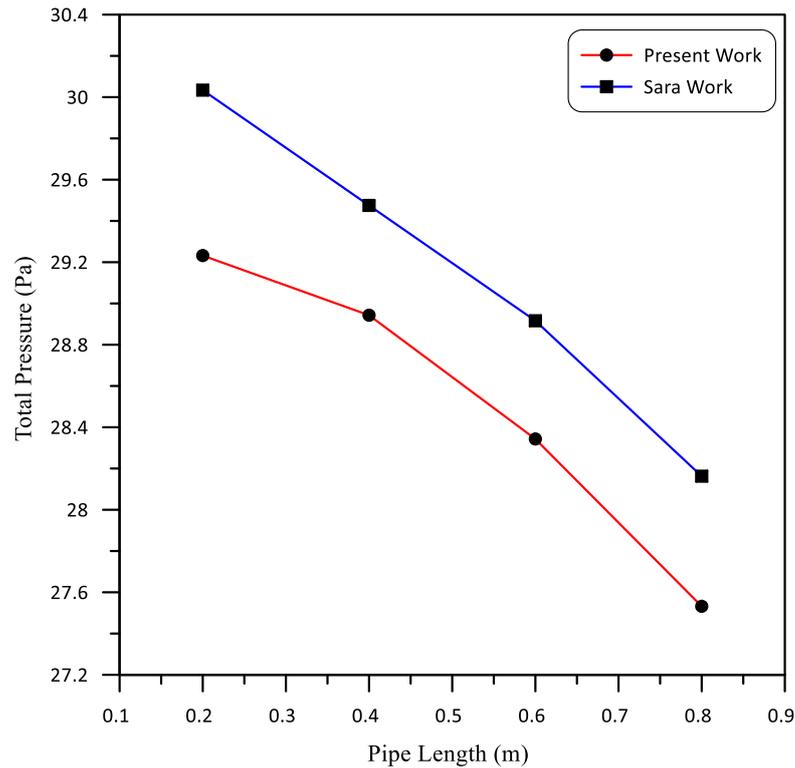


Figure (4-1): The validation of numerical code with numerical results of **Sarah Oleiwi [27]**.

## **4.2 Effect of Pipe Shape on Pressure Drop without Obstruction**

Figure (4-2) shows the effect of pipe shape on pressure drop with constant velocities and the same cross-section area. It can be noted that the circular pipe section has the lowest pressure drop at 0.2 m, which is 164.7741 Pa, while it is 170.2216 Pa and 176.5483 Pa for the square and triangle pipes section, respectively. This means they have the highest pressure drop. The reason of the pressure drop being in a low value in the circular pipe section is that it has no sharp and right angles and that allows the force of fluid on the pipe to be distributed uniformly along the whole pipe. Thus, this leads to a reduction in the pressure drop-resistance of circular cross-section compared to other pipe section.

The results show that the circular pipe is the most efficient. Therefore, it is appropriate for reducing flow losses and pressure drop during fluid transfer.

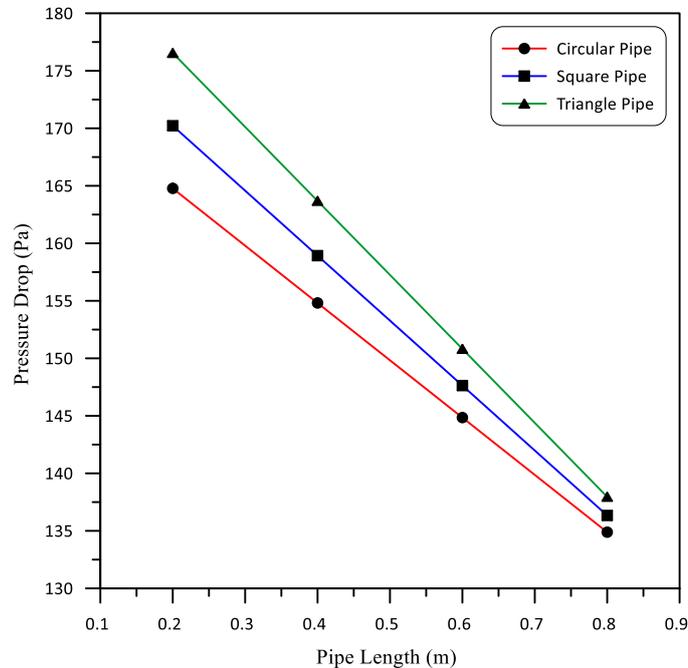


Figure (4-2): Effect of pipe shape on the pressure drop with constant velocities.

### **4.3 Effect of Velocity on Pressure Drop with Obstruction**

A numerical analysis is conducted to find out the effect of changing the fluid velocity on the pressure drop in the most efficient pipe, which is the circular pipe section.

Figures (4-3) and (4-4) show the effect of increasing the velocity of gasoil from 0.7 m/s to 1.1 m/s and velocity of water from 0.35 m/s to 0.55 m/s on the pressure drop at four different points along the circular pipe with rectangular rib located at 0.25L from the inlet.

When gasoil velocity increases from 0.7 m/s to 1.1 m/s at 0.2 m, the pressure drop increase from 325.7946 Pa to 608.443 Pa at a constant water velocity of 0.35 m/s. In addition, when water velocity increases from 0.35

m/s to 0.55 m/s for the same point and constant gasoil velocity of 0.7 m/s, the pressure drop increases from 325.7946 Pa to 344.0797 Pa. This means the pressure drop increases with increasing the gasoil and water velocities. The influence of gasoil velocity on pressure drop is higher than the water velocity. These results show good agreements with [21] and [28].

In addition, the existence of obstruction influences the pressure drop, due to the turbulence caused by the sudden changes in a cross-sectional area [19]. Figure (4-5) shows a closer look to illustrate pressure drop behavior in the obstruction region from 0.2 m to 0.4 m. It is found that before the obstruction at (0.2 m), for a given velocity of water (0.35 m/s) and constant gasoil velocity of (0.7 m/s), the pressure drop is 325.7946 Pa and at 0.28 m is 125.2844 Pa. This means it decreases, while downstream the obstruction at 0.3 m and 0.4 m the pressure drop is 141.2286 Pa and 170.7788 Pa, respectively, which means it increases and that behavior is adverse to first on.

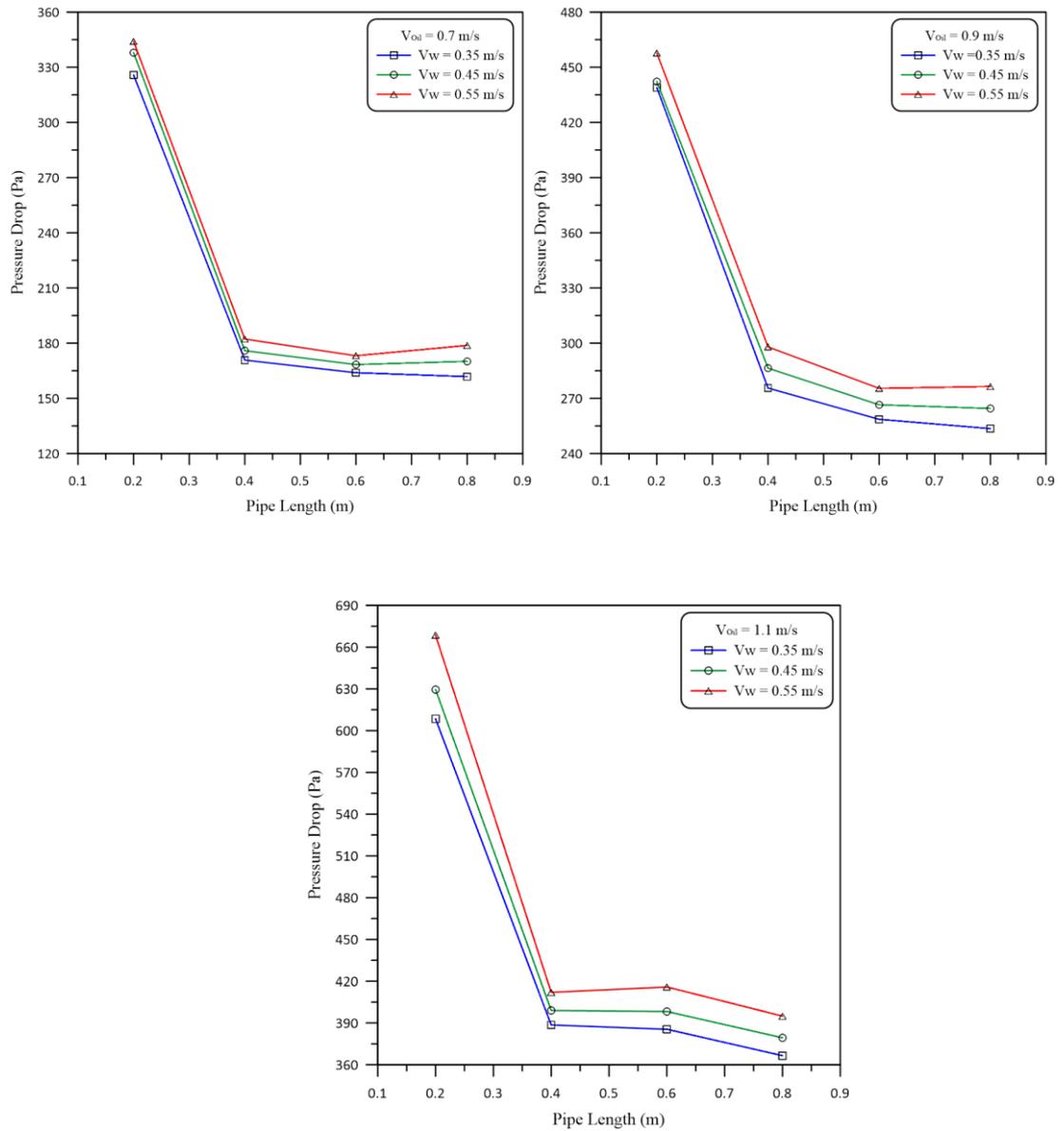


Figure (4-3): The effect of gasoil velocities on pressure drop.

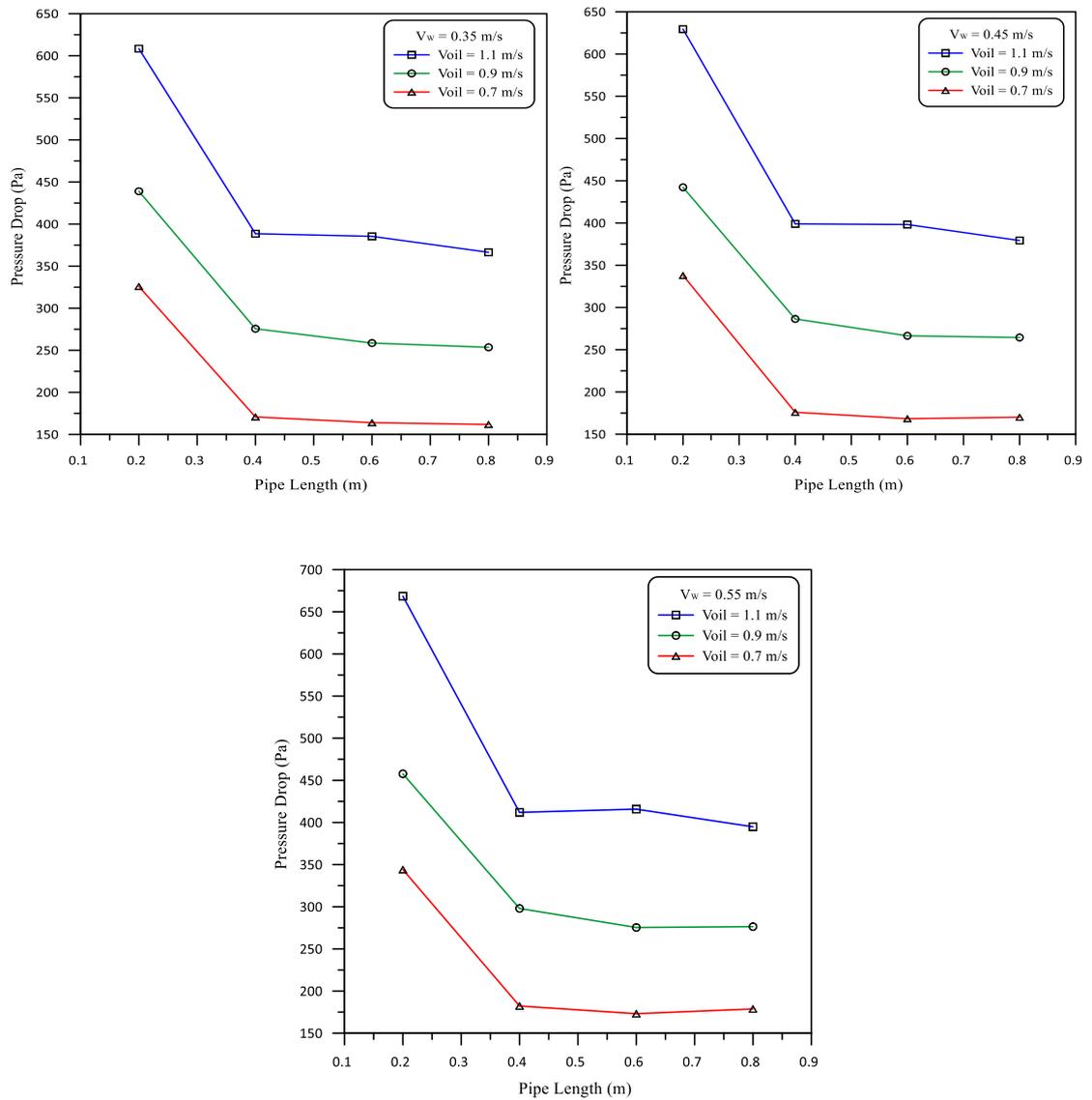


Figure (4-4): The effect of water velocities on pressure drop.

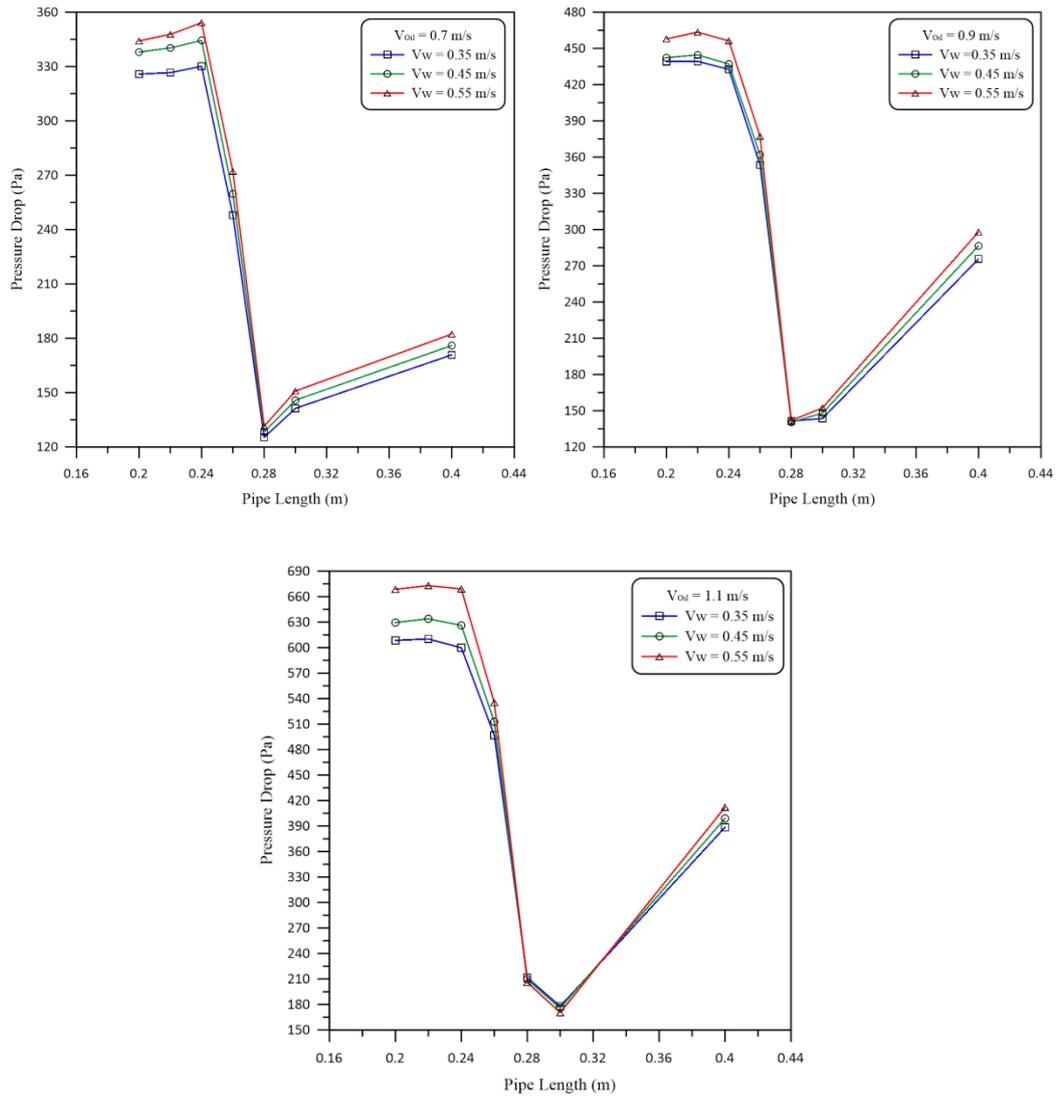


Figure (4-5): Closer look for the obstruction region shows the effect of increasing velocity of gasoil and water on pressure drop.

## 4.4 Effect of Rib Length on Pressure Drop

In comparison between two-phase flow in smooth pipe and corrugated pipes, it shows a very different behavior at the same flow conditions. Figure (4-6) shows the effect of rib length on pressure drop. Different sizes of rib had taken (25-125) % of the original rib. At point 0.2 m when the size of the rib increases from 100% to 125%, an obvious increase in the pressure drop can be noted from 344.0797 Pa to 407.5627 Pa, then the pressure drop reaches the regular change with the length of the pipe. This is due to the increase in obstruction area of the stream flow that tends to intercept a much amount of fluid flowing and of course caused a high- pressure drop. These results show good agreements with [29].

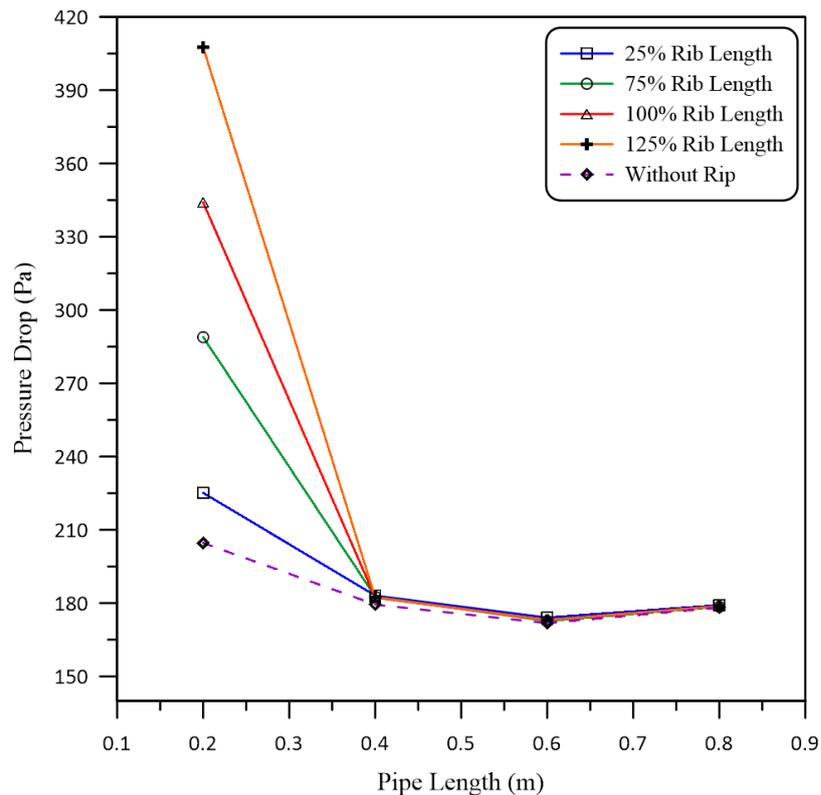
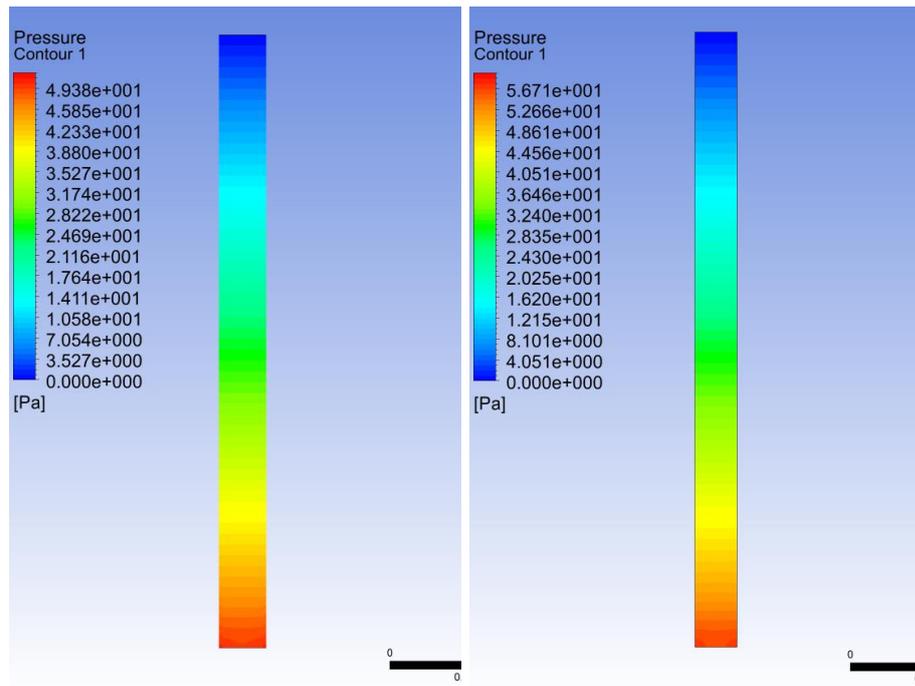


Figure (4-6): The effect of rib length on pressure drop.

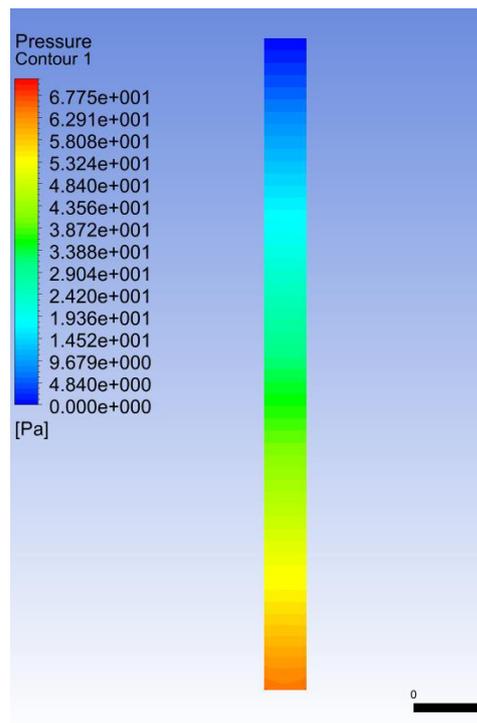
The values of the pressure drop that are represented by contours are shown in Figure (4-7) by the color rule of the gasoil velocity 0.7 and water velocity 0.55 for three pipe sections (circular, square, and triangle).

Figures (4-8), (4-9) show the contour of pressure drop for the different velocities, by using rainbow color to illustrate the pressure drop variation where the blue color represent the lowest pressure drop in the pipe and the red color represent the highest pressure drop.



(a)

(b)



(c)

Figure (4-7): Pressure drop contour for gasoil velocity 0.7 and water velocity 0.55 (a) circular (b) square, and (c) triangular pipe sections

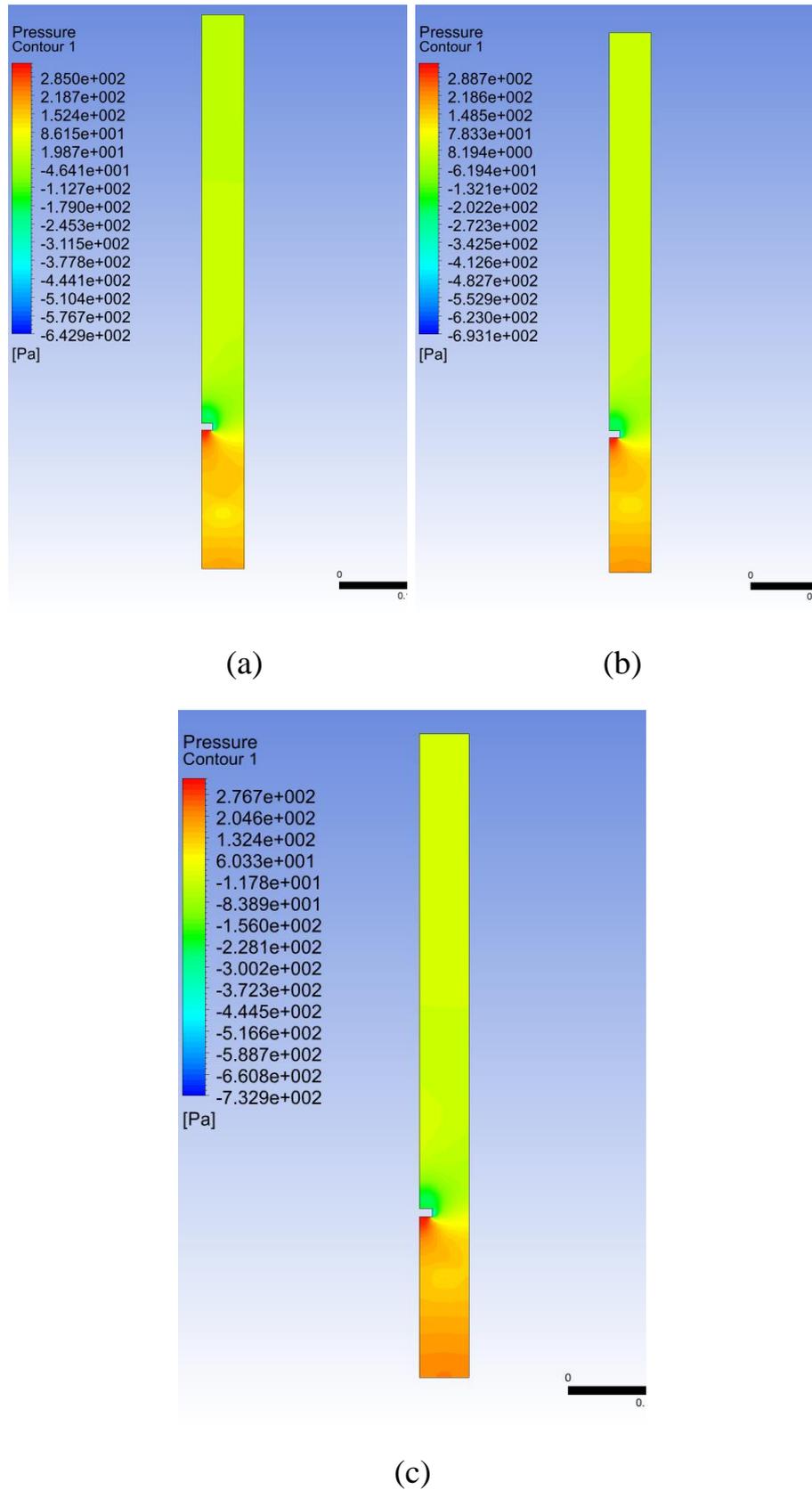
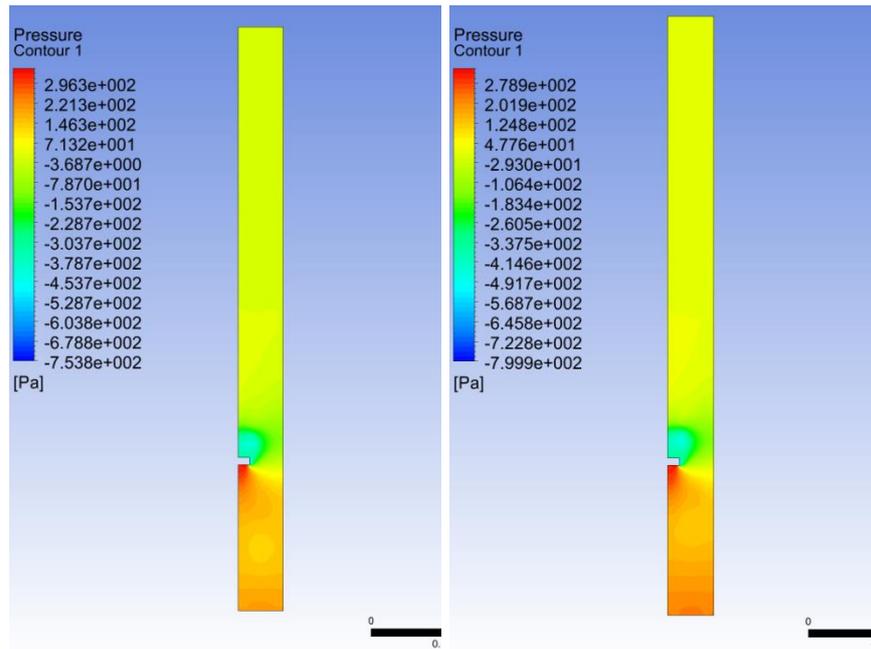
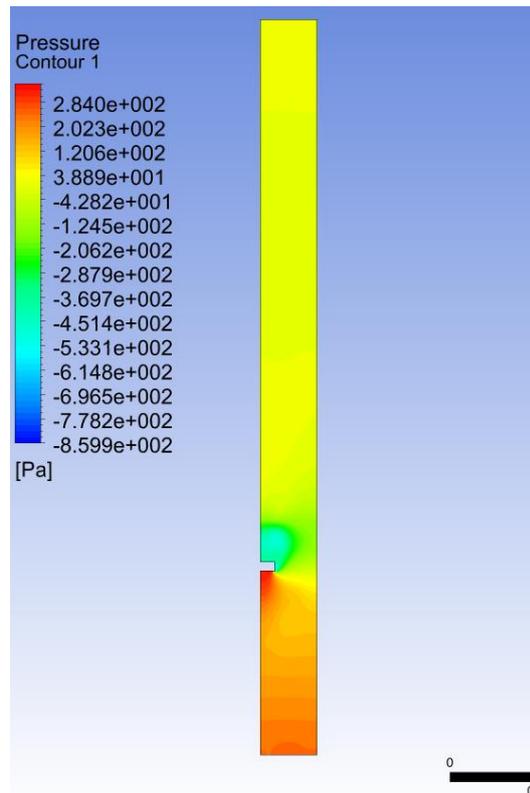


Figure (4-8): Pressure drop contour when gasoil velocity 0.7 as water velocity is (a) 0.35 (b) 0.45, and (c) 0.55



(a)

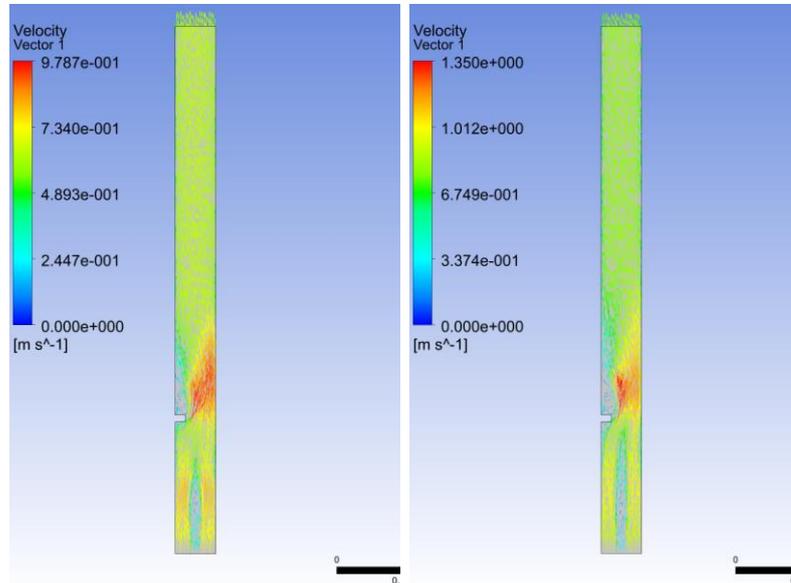
(b)



(c)

Figure (4-9): Pressure drop contour when gasoil velocity 0.9 as water velocity (a) 0.35 (b) 0.45, and (c) 0.55

Figure (4-10) shows the velocity vector to illustration the directions of flow, where the vortex take a place directly above the rib due to the obstruction of the rectangular rib.



(a)

(b)

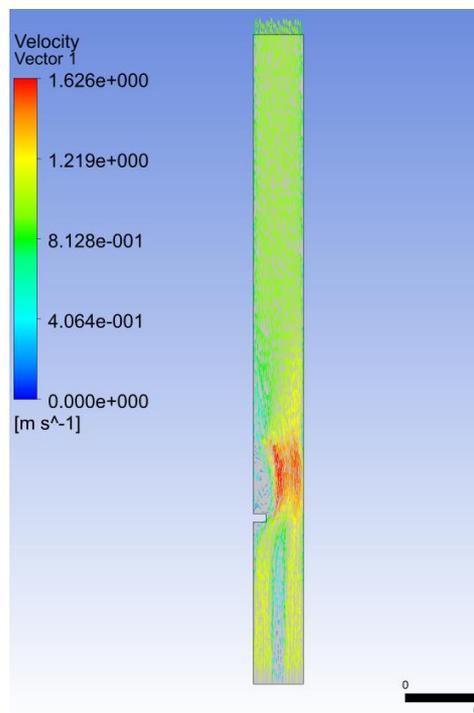


Figure (4-10): Velocity vector when water velocity 0.35 as gasoil velocity (a) 0.7 (b) 0.9, and (c) 1.1.

**CHAPTER FIVE**  
**CONCLUSIONS AND**  
**RECOMMENDATIONS**

## **Conclusions**

### **5.1 Conclusions**

In this study, the effect of pipe section shape on pressure drop has been investigated numerically. Three different pipe sections are taken (circular, square, and triangular cross-section). The most efficient pipe chosen, which is the circular cross-section, is to study the influence of changing the superficial velocity of gasoil and water on pressure drop. In addition, this study investigates the effect of rectangular obstruction on pressure drop. SOLIDWORK 2018 is used to draw the structures and ANSYS Fluent 17.2 is used to simulate this study. The following conclusions drawn from this work are:

1. A circular cross-section pipe has the lowest pressure drop in comparison with the other two pipe sections. And this is what makes it most suitable to be used for long-distance transportation purposes.
2. Gasoil superficial velocity and water superficial velocity have a direct proportional influence on pressure drop along the pipe. Increasing gasoil superficial velocity from 0.7 m/s to 0.9 m/s and water superficial velocity from 0.35 m/s to 0.45 m/s lead to an increase in pressure drop by 35% and 4%, respectively at 0.2 m.
3. The rectangular obstruction has a direct proportional influence on pressure drop. This was due to the turbulence caused by the sudden changes in the cross-sectional area. When the rib has been added to the pipe, it is led to increasing the pressure drop 68% at constant velocities of gasoil 0.7 m/s and water 0.55 m/s at 0.2 m.
4. The size of obstruction affects pressure drop directly. The increase of obstruction size to 125%, leads to an increase in the pressure drop by

18%, and decreasing obstruction size to 25%, leads to a decrease in the pressure drop by 35% at point 0.2 m.

## **5.2 Recommendations**

Followings are recommendations for future works:

1. Using other types of fluids such as crude oil or kerosene and compare between them to show the influence of viscosity on pressure drop in vertical upward two-phase flow.
2. Studying the effect of obstructions shape (triangular- circular) with different sizes on pressure drop in vertical upward two-phase flow.

# REFERENCES

## References

- [1] Abdul Hussein H. M., Ahmed S. T., and Habeeb Laith J., “Two-Phase Flow for Gas-Liquid, Gas-Solid, Liquid-Solid, and Liquid-Liquid in A Horizontal Smooth and Turbulator Conduit - A Review”, *Journal of Mechanical Engineering Research and Developments*, Vol. 43, No. 7, PP. 26-50, 2020.
- [2] Awad M. M., “Two-Phase Flow Modeling in Circular Pipes”, Ph.D. Thesis, Department of mechanical engineering, University of Newfoundland, Canada, 2007.
- [3] Dong F., Liu X., Deng X., Lijun Xu, and Ling-an Xu, “Identification of Two-Phase Flow Regimes in Horizontal, Inclined and Vertical Pipes”, *Measurement Science and Technology*, Vol. 12, No. 8, PP. 1069–1075, 2001.
- [4] Sabreen A., Mohammed A., and Al-Mudhafar M. A., “Experimental Study of (Water-Oil) Flow Patterns and Pressure Drop in Vertical and Horizontal Pipes”, *International Journal of Air-Conditioning and Refrigeration*, Vol. 26, No. 4, PP. 1-9, 2018.
- [5] McQuillan K. W., and Whalley P. B., “Flow Patterns in Vertical Two-Phase Flow”, *Int. J. Multiphase Flow*, Vol. 11, No. 2, PP. 161-175, 1985.
- [6] Christopher E. Brennen, “Fundamentals of Multiphase Flow”, 1st edition, Cambridge University Press, New York, 2005.
- [7] Cihang Lu, Kong R., Qiao S., Larimer J., Kim S., Bajorek S., Tien K., Hoxie C., “Frictional Pressure Drop Analysis for Horizontal and Vertical Air-Water Two-Phase Flows in Different Pipe Sizes”, *Nuclear Engineering and Design*, Vol. 323, PP. 147-161, 2018.

- [8] Ładosz A., Rudolf von Rohr P., “Pressure Drop of Two-Phase Liquid-Liquid Slug Flow in Square Microchannels”, *Chemical Engineering Science*, Vol. 191, PP. 398-409, 2018.
- [9] Jana A. K., Ghoshal P., Das G., and Das P. K., “An Analysis of Pressure Drop and Holdup for Liquid-Liquid Upflow through Vertical Pipes”, *Chemical Engineering & Technology*, Vol. 30, No. 7, PP. 920-925, 2007.
- [10] Shanna B. A., “Frictional Pressure Drop of Gas-Liquid Two-Phase Flow in Pipes”, *Nuclear Engineering and Design*, Vol. 238, PP. 3277-3284, 2008.
- [11] Liu W. H., and Guo L. J., “Pressure Drop for Oil-Water Two-Phase Flow in Horizontal Pipe”, *AIP Conference Proceedings*, Vol. 1207, PP. 665-673, 2010.
- [12] Ismail I., and Misnan S., “Pressure Drop and Water Holdup of Malaysian Crude Oil and Water Two-Phase Flow in Pipes”, *Advanced Materials Research*, Vols. 931-932, PP. 1248-1252, 2014.
- [13] Xiong X., Rahman M.A., and Zhang Y., “Pressure Gradient Estimation for Two-Phase Gas/Liquid Flow in a Vertical Pipe Using CFD”, *2014 Oceans - St. John's*, PP. 1-6, 2014.
- [14] Sanati A., “Numerical Simulation of Air-Water Two-Phase Flow in Vertical Pipe Using k- $\epsilon$  Model”, *International Journal of Engineering & Technology*, Vol. 4, No. 1, PP. 61-70, 2015.
- [15] Ali Z., Hamad F.A., Faraji F., Santim C.G.S., and Basha N., “Investigation of Pressure Drop in Horizontal Pipes with Different Diameters”, *International Journal of Multiphase Flow*, Vol. 91, PP. 120-129, 2017.

- [16] Wordu D. N., Ideriah Felix J. K., and Nkoi B., “A Study of Pressure Gradient in Multiphase Flow in Vertical Pipes”, *European Journal of Engineering Research and Science*, Vol. 4, No. 1, PP. 54-59, 2019.
- [17] Basha M., Shaahid S. M., and Alhems L. M., “Effect of Viscosity on Pressure Drop of Oil-Water Two-Phase Flow in 6” Horizontal and Inclined Stainless Steel Annulus Pipe”, *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences*, Vol. 69, No. 2, PP. 156-167, 2020.
- [18] Zilong Liu, Yubin Su, Ming Lu, Zilong Zheng, and Ruiquan Liao, “Frictional Pressure Drop and Liquid Holdup of Churn Flow in Vertical Pipes with Different Viscosities”, *Geofluids*, Vol. 2021, PP. 1-8, 2021.
- [19] Mahood H. B., “Effect of Flow-Obstruction Geometry on Pressure Drops in Horizontal Air-Water Two-Phase Flow”, *Al-Qadisiya Journal For Engineering Sciences*, Vol. 2, No. 3, PP. 641-653, 2009.
- [20] Al-Turaihi Riyadh S., and Habeeb Laith J., “Experimental Study and CFD Simulation of Two-Phase Flow Around Triangular Obstacle in Enlarging Channel”, *International Journal of Engineering Research and Applications*, Vol. 3, No. 4, PP. 2036-2048, 2013.
- [21] Bai B., and Liu W., “Two-Phase Flow Patterns and Pressure Drop inside a Vertical Pipe Containing a Short Helical Tape Insert”, in *Proc. of ASME-FEDSM 2014*, Chicago, IL, 2014.
- [22] Al-Turaihi Riyadh S., Shareef Abbas S., and Abdalaimma A., “A Numerical Study of Two-Phase Flow through a Rectangular Duct with a Convergence Ribs”, *International Journal of Engineering & Technology*, Vol. 7, No. 4.19, PP. 969-974, 2018.
- [23] Habeeb Laith J., Isam Mejbel Abed, and Rafel Hekmat Hameed, “Experimental Analysis of Two-Phase Flow through Cylindrical

Obstruction in Vertical Pipe”, *International Journal of Mechanical Engineering and Technology*, Vol. 10, No. 01, PP. 74-88, 2019.

[24] Kiran R., Ahmed R., and Salehi S., “Experiments and CFD Modelling for Two-Phase Flow in A Vertical Annulus”, *Chemical Engineering Research and Design*, Vol. 153, PP. 201-211, 2020.

[25] Majdi Hasan S., and Habeeb Laith J., “Experimental and Numerical Studies of Two-Phase Flow in Ribbed Periodic Vertical Channel”, *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences*, Vol. 68, No. 1, PP. 34-53, 2020.

[26] ANSYS, “ANSYS Fluent User's Guide”, ANSYS, Inc., Release 15.0, November 2013.

[27] Sarah Hasan Oleiwi, “Experimental and Numerical Investigation of Multi-phase Flow in Fluidized Bed Column”, M.Sc. Thesis, Department of Mechanical Engineering, University of Babylon, 2016.

[28] Ammar Abdulkadhim, Dhuha Radhi, and Ali Mohammed Hussein Mohsen, “Experimental Investigation of Two-phase Fluid Flow over a Rectangular Obstructions Located Inside Enlarged Rectangular Channel”, *Mathematical Modelling of Engineering Problems*, Vol. 6, No. 2, PP. 183-187, 2019.

[29] Coker A. Kayode., “Ludwig’s Applied Process Design for Chemical and Petrochemical Plants”, Volume 1, 4th edition, Gulf Professional Publishing, 2007.

[30] Ali Abdalaimma Hassan AL-Genaby, “Experimental and Numerical Study of Two-Phase Flow in the Ribbed Divergent\Convergent Rectangular Duct”, M.Sc. Thesis, Department of Mechanical Engineering, University of Kerbala, 2019.

## الخلاصة

يبحث هذا العمل نظرياً تأثير ثلاثة انواع من الأنابيب بنفس مساحة التدفق  $0.00456 \text{ م}^2$ ، وطول 1 م، وبأشكال مختلفة (دائري، مربع، ومثلث) على الضغط الكلي خلال تدفق زيت وماء داخل الأنبوب. تم اختيار الأنابيب الأمثل من بين هذه الأنواع (وهو الدائري) لدراسة تأثير نطاق مختلف من السرعات يتراوح بين (0.7-1.1) م/ث ( $0.35-0.55$ ) م/ث لزيت الغاز والماء على التوالي، على انخفاض الضغط على طول الأنبوب. حيث تمت المحاكاة بواسطة برنامج ANSYS FLUENT 17.2 وتم رسمه بواسطة SOLIDWORK 2018.

تضمنت الدراسة أيضاً تأثير وجود عائق في تدفق السوائل على أداء النظام وكيف يمكن أن يؤثر حجم العائق على الضغط في التدفق الرأسي التصاعدي. أظهرت النتائج أن الأنبوب الدائري لديه أقل ضغط إجمالي مقارنة بالأنبوبين الآخرين، وللسرعة السطحية لزيت الغاز والماء، والعائق المستطيل، ولحجم العائق تأثير طردي مباشر على الضغط الكلي على طول الأنبوب.

الاستنتاجات المستخلصة من هذه الدراسة هي أن الضغط الكلي يزداد بنسبة 35% مع زيادة السرعة السطحية لزيت الغاز من 0.7 م/ث إلى 0.9 م/ث لسرعة سطحية للماء بمقدار (0.35 م/ث)، ويزداد بنسبة 4% عندما تزداد السرعة السطحية للماء من 0.35 م/ث إلى 0.45 م/ث لسرعة سطحية معينة لزيت الغاز بمقدار (0.7 م/ث) عند 0.2 م من المدخل.

للعائق تأثير نسبي مباشر على الضغط الكلي، يؤدي وجود عائق إلى زيادة الضغط الكلي بنسبة 68% عند سرع ثابتة لزيت الغاز 0.7 م/ث والماء 0.55 م/ث عند 0.2 م. حجم العائق له تأثير مباشر على الضغط الكلي ايضاً، حيث تؤدي زيادة حجم العائق بنسبة 125% إلى زيادة الضغط الكلي بنسبة 18%، وتقليل حجم العائق إلى 25%، يؤدي إلى انخفاض إجمالي الضغط بنسبة 35% عند النقطة 0.2 م.



جمهورية العراق  
وزارة التعليم العالي والبحث العلمي  
كلية الهندسة / جامعة بابل  
قسم الهندسة الميكانيكية

## تأثير شكل الانبوب على هبوط الضغط خلال جريان رأسي ثنائي الطور (وقود-ماء)

رسالة

مقدمة إلى كلية الهندسة – جامعة بابل وهي جزءاً من متطلبات نيل درجة الدبلوم  
العالي في الهندسة/الهندسة الميكانيكية/وقود وطاقة

من قبل

ميس حميد عبد الحسين ابراهيم

بإشراف

أ.م. د. رياض صباح الطريحي