INVESTIGATING THE EFFECT OF FLOW REGIME ON SAND TRANSPORT IN MULTIPHASE FLOW USING COMPUTATION FLUID DYNAMICS SOFTWARE

BY

EMMANUEL IKPESU

ENG1503987

A PROJECT REPORT SUBMITTED TO THE DEPARTMENT OF
PETROLEUM ENGINEERING,

FACULTY OF ENGINEERING,
UNIVERSITY OF BENIN,

IN PARTIAL FULFILLMENT OF THE
REQUIREMENTS FOR THE AWARD OF BACHELOR OF
ENGINEERING (B.Eng.) DEGREE IN PETROLEUM
ENGINEERING.

JULY 2021

CERTIFICATION

This is to certify that this project titled “Investigating the effect of flow regime on sand transport in multiphase flow using CFD software” was carried out by **EMMANUEL IKPESU** with matriculation number **ENG1503987** in the department of Petroleum Engineering, Faculty of Engineering, University of Benin, Benin City, Edo State, Nigeria.

ENGR. BLESSING OTAMERE (PROJECT SUPERVISOR)	**DATE**
MR. TAIWO OLUWASEUN (PROJECT COORDINATOR)	**DATE**
DR. IKPONMWOSA OHENHEN (HEAD OF DEPARTMENT)	**DATE**
PROF. S.O. ISEHUNMA (EXTERNAL SUPERVISOR)	**DATE**

DEDICATION

This project is dedicated to God almighty who gave me the wisdom, knowledge, courage and sound health to successfully complete this thesis and my Granddad Late. Raphael Unoroh for his inspiration and encouragement while he was alive.

ACKNOWLEDEGEMENT

First and foremost, I would like to appreciate my supervisor, Engr. Blessing Otamere for his patience and timely contributions toward this thesis. It was his confidence in me that gave me the needed motivation to complete this project. It has been a great pleasure working with him.

I would also like to express my sincere gratitude to Professor K.I Idigbe, Dr. Ike, Engr. Benedict, Engr. Osagie, Engr. Sunny and Engr. Peter for their guidance and support while under taking my undergraduate study at the University of Benin.

I wish to acknowledge my loving parents, Mr. and Mrs. Ikpesu for the encouragement and financial support they gave to me during the course of this project.

I am grateful to the management and staff at Mobil Nigeria Production especially Mr. Esosa, Mr. Raphael, and Mrs. Damilola and Mrs. Ihuoma for the training they gave to me during my six months internship with them in 2019. That was what guided the selection of this project thesis. I would also like to appreciate some of my classmates among which special recognition goes to Rita Okwonkwo, Akin Isaac, Mathias and Fred for providing me with a conducive environment for research.

Also, I want to acknowledge the Eremeh and Wanogho Family for their hospitality and support during my time in school and internship respectively.

Finally, I thank God above all for granting me the strength, wisdom and grace to make an outstanding project work.

ABSTRACT

Multiphase flow is defined as the simultaneous flow of two or more phases (e.g., gas, oil, water, or solid). When operating petroleum production facilities such as pipelines, this is a normal flow. As a result of complexity, the physical phenomenon governing them than that of single-phase flow, a production engineer's ability to effectively conduct a research on this system would necessitate a thorough understanding of the system to aid in its optimal operation.

The study's objective is to create a CFD model using the ANSYS version 19.1 platform, validate the model with experimental data, and review studies and the employed model to estimate the critical velocity of a sand particle in a slurry flow and the particle's erosional effect for a pipe of a particular diameter (0.07m)

Based on literature reviews and comparative studies, the Eulerian model with Reynold Stress Model (RSM) turbulence closure was chosen as the best model to analyze multiphase fluid flow.

The research combines validation work in all feasible scenarios to evaluate the creation of the CFD model with a parametric analysis to look at the effects of various factors on particle deposition. Pipe diameters of 0.02 – 0.07m, continuous phase flow rates of 0.1-1 m/s, and other parameters were investigated.

In conclusion, ANSYS version 19.1 platform is a valid way of analyzing multiphase flow in pipelines, proven using historical experimental data. Laminal flow is suitable for suitable fine particle and yields minimal erosion when its velocity is above the particle critical velocity. While coarse particles are transmitted by turbulence flow, with reducing erosion as the velocity increases above the critical velocity.

TABLE OF CONTENTS

NOMENCLATURE

V_s	settling velocity
V_p	phase velocity
ρ_p	phase density
z_p	Volume fraction
$\not{V}$	Volume
$\bar{u}_i\ (x_i)$	average velocity
$\hat{u}_j\ (x_i, t)$	instantaneous value of velocity
T	integration time
$\bar{V}$	average velocity
v_z	axial velocity
τ	shear stress
$\frac{\partial u}{\partial y}$	velocity gradient
V	velocity
D	inner diameter of pipe
μ	dynamic viscosity

v_μ kinematic viscosity

$P(x,\ y)$ pressure

$\vec{V}(x,\ y)$ velocity field

$u(x,\ y)$ x-component of velocity

$v(x,\ y)$ y-component of velocity

LIST OF FIGURES

LIST OF TABLES

CHAPTER ONE: INTRODUCTION

1.1 BACKGROUND OF STUDY

Pipelines are connection of pipes used to transmit fluids from one place to another. They usually have a minimum diameter of 0.1 m and a minimum length of 1.6 km (Guha and Berrones,2008). In the oil and gas industry, pipeline major purposes are;

- Export (transportation) pipelines
- Water or chemical injection flowline
- Flowlines between facilities (platforms, subsea manifolds, etc.) etc.

While transporting oil and gas through pipelines, we are often faced with several problems such as liquid loading, wax deposition, sand depositions etc. In particular, the topic of sand transport has received scant attention in literatures over the years. In fact, the sands are produced alongside oil and gas from unconsolidated of the reservoirs with low formation strength. Other causes the sand production include;

- High reservoir fluid viscosity
- Formation pore pressure reduction
- Increase in water production (cut)

At the wellhead, horizontal pipelines with or without screens transport the residual sand in reservoir fluid from the feeder to the flow stations. The entailed sand may deposit on the wall of the pipes due to pressure drop, causing problems such as;

- Abrasion
- Corrosion
- pipe blockage
- reduction in flow area
- And most importantly, low output from the lines

In cases were sand production are expected, sand exclusion measures/techniques (gravel parking, sand screen and sand filters) are used to avoid the intrusion of sand into the pipeline from downhole, however, they cause significant loss in productivity. Moreover, the cleaning in pipeline with pig may be relatively simple but it's for small deposition. Thus, for the reasons, the development of management strategies has become a conventional method in the industry under a board subject called flow assurance.

Due to this, multiphase flows in pipelines or annuli are of great importance and widely employed in the industry. In particular solid-fluid (two or three) phase flow has become increasingly popular owing to its manifold applications. This kind of multiphase flow through pipeline is being studied since the 20th century with focus on the development of general solutions based on available experimental data for solid volumetric or mass concentration profiles, pressure drops, and slurry velocity profiles. O'Brien and Morrough (1933) and Rouse (1937) were two of the first experiments to look at slurry flows in an open channel with low solid concentrations. To predict the concentration distribution, they used a diffusion model. The pioneers in defining the frictional pressure losses in slurry flow are Durand (1951), Durand and Condolios (1952), and Newitt et al. (1955). The model proposed by Ling et al. (2003) for heterogeneous slurry and the correlations developed by Thomas (1965) and Krieger (1972) for homogeneous slurry added a new dimension to the analysis of pressure losses in this form of multiphase flow. Govier and

Aziz (1972), Vocadlo and Charles (1972), Aude et al. (1974), Aude et al. (1975), and Seshadri (1975) are some of the other significant works on empirical associations for slurry pressure losses (1982). Prior to the year 2000, most studies had a restricted application range, scope, and data (Lahiri and Ghanta, 2007).

Later, in an attempt to overcome these limitations, various computational fluid dynamics (CFD)-based models were proposed. Lin et al., 2003; Cornelissen et al., 2007; Hernández et al., 2008; Lin and Ebadian, 2008; Chen et al., 2009; Kaushal et al., 2013; Gopaliya and Kaushal, 2015; Kumar and Kaushal, 2016) are examples of such studies on slurry flow in a pipeline. The results of research on pipeline slurry flows, on the other hand, are not always applicable to identical flows in annuli. Annular slurry flows have not been analyzed as thoroughly as their pipeline counterparts. Özbelge and Köker (1996), Özbelge and Beyaz (2001), Eraslan and Özbelge (2003), Özbelge and Eraslan (2006), Camçi and Özbelge (2006), Kelessidis et al. (2007), and Özbelge and Ünal (2007) are several notable works on annular slurry flows (2008). Escudier et al. (2002) have provided a bibliographic collection of annular flow articles.

The two-phase flow of liquid and gas in a pipeline was well defined by Govier and Aziz (1972). The addition of solid particles to a two-phase pipeline flow was found to result in a smaller pump and a higher flow rate (Orell, 2007; Pouranfard et al., 2015). This type of three-phase device aids in the reduction of air pollution, noise, and injuries while also saving electricity. It has also been stated that adding or injecting air into a two-phase slurry system reduces pumping costs and improves bitumen recovery in oil-sand fields (Sanders et al., 2007). Numerous studies were conducted on three-phase pipeline flow (Scott and Rao, 1971; Toda et al., 1978, Hatate et al., 1986; Fukuda and Shoji, 1986; Kago et al., 1986; Gillies et al., 1997; Bello et al., 2005; Rahman et al., 2013; Li et al., 2015; Pouranfard et al., 2015).

Sand issue is a complex mechanical process in which the severity of the damage depends on wide varieties of parameters, such as multiphase flow regime, solid particle / fluid properties etc. Studies of sand particles in pipelines can be investigated experimentally via laboratory test or computationally (using computer simulations) via numerically methods.

After the invention of computer, computational approached were then employed in the industry. Furthermore, since experimental work is costly and cannot be a representation of industrial scale, many researchers have conducted significant numerical studies using Computational Fluid Dynamics (CDF) to carry-out analysis of sand particles in multiphase fluid flowing through pipeline.

1.2 STATEMENT OF PROBLEM

The purpose of pipeline is to provide flow path for transmitting produced reservoir fluids from the production platform to the process facilities and sales point. They are made up of different material which determines their physical and chemical properties.

The produced stream from the reservoir are usually a multiphase mixture of desired fluid (oil, gas or both) and undesired fluid (water etc.) and dispersed solid like sand due to drawdown and some geomechanics properties like low consolidation the reservoir formation. The flow of this solid particles along with the fluid can lead to the blockage of flow path, damage of device (including the pipeline creating flow path). For flow assurance, this calls for the need to investigate the system for the prevention, remediation and optimization of the problems arising due to this sand particle.

The study can be done experimentally or empirically but due to limitation of laboratory and available data, the result from these methods can be easily upscale to a large commercial process. These limitations, calls for the need of a CFD numerical approach to carry-out the study.

1.3 MOTIVATION OF STUDY

Many reservoirs from major oil and gas producing regions (such as the Niger Delta in Nigeria) are prone to the sand production due to their high unconsolidated formations and sand production potential during the well life. Phenomena such as sand deposition can lead to partial or total blockage of flow path, enhance pipe corrosion and pig trapping. This failure can lead to unplanned downtime and risk to equipment as well as personnel.

1.4 AIM AND OBJECTIVES OF STUDY

AIM

The aim of this study is to develop a CFD model of moderate accuracy that can be used to investigate the effect of multiphase flow regime with dispersed sand phase and identify optimum production management strategy for flow assurance.

OBJECTIVES

The set objectives of this project are:

1. To develop a CFD model for multiphase flow with sand particles as dispersed solid using ANSYS

2. To validate model with wide range of experimental and empirical data sets
3. To identify the different flow regimes encountered and their effect during multiphase flow in pipelines with sand in-situ
4. To conduct study with develop CFD model to predict erosion and settling condition through annular
5. To Identify major factors affecting sand transport in multiphase flow

1.5 SIGNIFICANCE OF STUDY

The importance of the study cannot be over emphasized as they are used to the production engineering in carrying out pipeline design, investigation and remedial study. Experimental and empirical methods can also be employed but they are limited the range of operations they can analysis, time consumed and cost.

Flow assurance in an important role of the production engineering, and this study aid to give a quick insight on the optimum way to carry-out multiphase fluid transmission in pipeline.

1.6 SCOPE OF STUDY

This project is focused validation ANSYS Fluent CFD software as a means of analyzing fluid flow, thereafter study the effect of flow regimes on multiphase flow with sand as disperse phase.

CHAPTER TWO: LITERATURE REVIEW

2.1 PRINCIPLE OF FLUID FLOW

The governing equations of fluid flow are based on the fundamental laws of fluid flow:

- Conservation of mass

  which states that the mass of any system closed to all transfers of matter and energy must remain constant over time, as the mass of the system cannot vary, and hence quantity cannot be added or removed.

  $m_{in} - m_{out} = m_{accumulated}$ --2.1

- Conservation of momentum

  also known as Newton's First Law or the Law of Inertia, states that the momentum of a system is constant if no external force are acting on the system.

  $-\vec{F} = m\,\vec{a}$ ------------------------------------2.2

- Conservation of energy

  It's based on the first law of thermodynamics and states that an isolated system's total energy remains constant over time; it is considered to be conserved

2.1.1 LAGRANGIAN DESCRIPTION OF FLUID FLOW

Also known as the differential representation of fluid flow, identify (or name) a fluid's material, track (or follow) it as it flows, and keep an eye on how its attributes change. In the flow field, the attributes may include velocity, temperature, density, mass, or concentration, among others.

This method is the most used in fluid mechanics, especially because of its mathematical simplicity and It applies laws to ‘infinitesimal fluid particle’ going through the flow domain

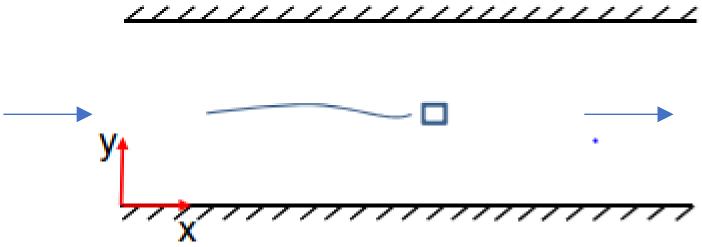


Fig.2. 1 Lagrangian Description of Fluid Flow

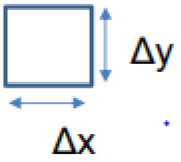


Fig.2. 2 Infinitesimal Fluid Particle

At time t_o, particle P is at (x_o, y_o)

- $\overrightarrow{V_p}(t_o) = \overrightarrow{V_o}$
- $\overrightarrow{x_p}(t_o) = \overrightarrow{x_o}$
- $\overrightarrow{y_p}(t_o) = \overrightarrow{y_o}$

The problem with is approach arise from the fact that we have an infinite number of infinitesimal fluid particles.

2.1.2 EULERIAN DESCRIPTION OF FLUID FLOW

The Integral representation of fluid flow is another name for it. Identify (or label) a fixed site in the flow field and track changes in its attribute as various materials pass past it. Instead of carrying-out study on a particle, we switch to following points and space. So, we will say;

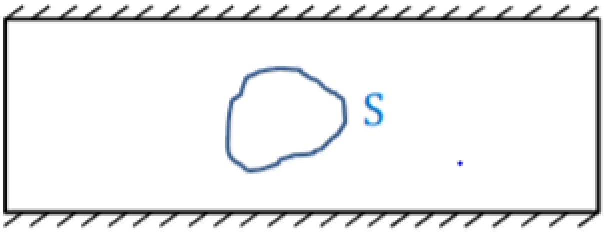


Fig.2. 3 Eularian Description of Fluid Flow

Figure 2 Apply laws to finite volume in flow domain

The velocity at (x_o, y_o) at time t_o is

- $\vec{V}(x_o, y_o, t_o) = \vec{V_o}$

At a point, if the velocity (or other flow properties) of any particle passing through it is constant, the flow is classified as a steady state flow. **Note:** The velocity of the ‘infinitesimal fluid particle’ changes with time and space. Also, the change of this properties at any point is zero (0). Whereas, if the properties of the fluid particles changes at a particular point, the flow is termed as unsteady flow.

Eulerian approach can also be used to track particle but we have to know the position of the particle at a particular time to do so. It was also the approach employed for the simulation study.

2.1.2.1 MASS CONSERVATION (EULERIAN APPROACH)

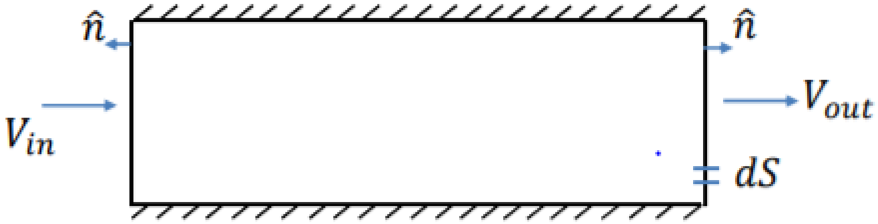


Fig.2. 4 Bounded Volume

If V_{in} represent the flow into and V_{out} the flow out of the system, and mass flow rate by $\dot{m}$, then for an unsteady flow the mass conservation equation can be written as;

$\dot{m}_{in} - \dot{m}_{out} = \dot{m}_{accumulated}$ ------------------------------2.3

Scenario 1: Uniform velocity

$\dot{m}_{in} = \rho V_{in} S_{in}$ $\dot{m}_{out} = \rho V_{out} S_{out}$ $\dot{m}_{accumulated} = \frac{d}{dt}(\rho \cancel{V})$

Scenario 2: Non-uniform velocity

$\dot{m}_{in} = \iint_{S_{in}} \rho(\overrightarrow{V}.\hat{n})dS$ $\dot{m}_{out} = \iint_{S_{out}} \rho(\overrightarrow{V}.\hat{n})dS$ $\dot{m}_{accumulated} = \frac{d}{dt}\iiint_{C.\cancel{V}} \rho d\cancel{V}$

$\iint_{S_{in}} \rho(\overrightarrow{V}.\hat{n})dS - \iint_{S_{out}} \rho(\overrightarrow{V}.\hat{n})dS = \frac{d}{dt}\iiint_{C\cancel{V}} \rho d\cancel{V}$ -------------------------2.4

Therefore

$\frac{d}{dt}\iiint_{C.\cancel{V}} \rho d\cancel{V} - \iint_{C.S} \rho(\overrightarrow{V}.\hat{n})dS = 0$ ------------------------------------2.5

2.1.2.2 MOMENTUM CONSERVATION (EULERIAN APPROACH)

The discussion is anchor in the same channel flow used for mass conservation. Momentum is simple;

Momentum = $m\vec{V}$ --------------------------------2.6

and newton second law says that for momentum to change, the rate of change of momentum is balanced by the forces on any particular body.

$\vec{F} = m\vec{a}$ ---------------------------------2.7

So, this is the form we apply to the infinitesimal fluid particle. In the integral form, we apply this in aggregate to the control volume and this can be any particular control volume similar to that in integral form of mass conservation.

Net mass flow outflow rate through S = $\iint_{S_{out}} \rho(\vec{V}.\hat{n})dS$ ----------------------------2.8

Net momentum outflow rate through S = $\iint_{S_{out}} \rho(\vec{V}.\hat{n})\vec{V}dS$ ------------------------2.9

For momentum conservation, momentum change is balanced by the forces. The forces are pressure and the viscous force.

For the pressure force,

$\overrightarrow{F_P} = \iint_S -P\hat{n}dS$ --2.10

The direction is in $-\hat{n}$ because pressure acts along the inward normal.

For viscous force, $\overrightarrow{F_v}$, more details will be discussed in later chapter. It would be related to the velocity grated and the coefficient of viscosity (assuming Newtonian flow). Therefore;

$$\iint_S \rho(\vec{V}.\hat{n})\vec{V}dS = -\iint_S -P\hat{n}dS + \overrightarrow{F_v} \text{ ----------------------------------2.11}$$

This is the form of momentum conservation that is used by ANSYS fluent solver.

2.2 FLOW REGIME IN PIPE

Based on a dimensionless number known as Reynold number, Re, and the amount of perturbation flow can be classified as laminar or turbulent flow (Liu Henry 2003). The Reynolds number is defined as;

$$Re = \frac{\rho VD}{\mu} = \frac{VD}{v_\mu} \text{ --------------------------------------2.12}$$

Reynolds number is the ratio of inertial forces to viscous force within a fluid which is subjected to relative internal movement due to different fluid velocity. It helps to predict the flow pattern in different fluid situations. The flow regime is determined by the **critical Reynolds number**, Re_c.

2.2.1 LAMINAR FLOW

If the Reynolds number is below the **critical Reynolds number**, Re_c, usually 2100 (Liu Henry 2003), the flow is termed to be laminal. In laminar flow, individual particles of fluid follow paths that do not cross those of neighboring particles (Bernard Massey et la 2001). It occurs at low enough velocities for viscosity forces to outnumber inertia forces, if any individual particle tries to deviate from its specified route, viscosity firmly restrains it, and the ordered procession of fluid particles continues. When there is relative movement between neighboring fluid particles, viscous stresses are developed, and these tensions tend to eliminate the relative movement. Newton described the fundamental law of viscous resistance in 1687 as;

$$\tau = \mu \frac{\partial u}{\partial y} \text{----------------------------} 2.13$$

2.2.1.1 STEADY LAMINAR FLOW IN CIRCULAR PIPES: THE HAGEN-POISEUILLE LAW

The law governing laminar flow in circular pipes was first studied by two men independently. They conducted experimental examinations of flow in straight pipes with circular cross-sections around 1840. G. H. L. Hagen (1797–1884), a German engineer, produced the initial results, which were published in 1839.

He'd experimented with the flow of water through small brass tubes, and his calculations revealed that the loss of head experienced by the water as it flowed through a given length of tube was directly proportional to the rate of flow, and inversely proportional to the fourth power of the tube's diameter.

At the same time, J. L. M. Poiseuille (1799–1869), a French physician, was conducting a series of meticulous and precise tests with the goal of examining blood flow via veins. He used water in fine glass capillary tubes in his studies and came to the same conclusions as Hagen (albeit he couldn't figure out why there was an apparent disagreement with extremely short tubes). In 1840, his first findings were made public. The law regulating laminar flow in circular pipes was then theoretically derived.

It's worth noting that, while Poiseuille's results were invaluable in pointing the way to the theory of laminar flow in circular tubes, they aren't really applicable to the flow of blood in veins; for one thing, vein walls aren't rigid, and blood isn't a Newtonian fluid, meaning it doesn't have a constant viscosity even at a constant temperature.

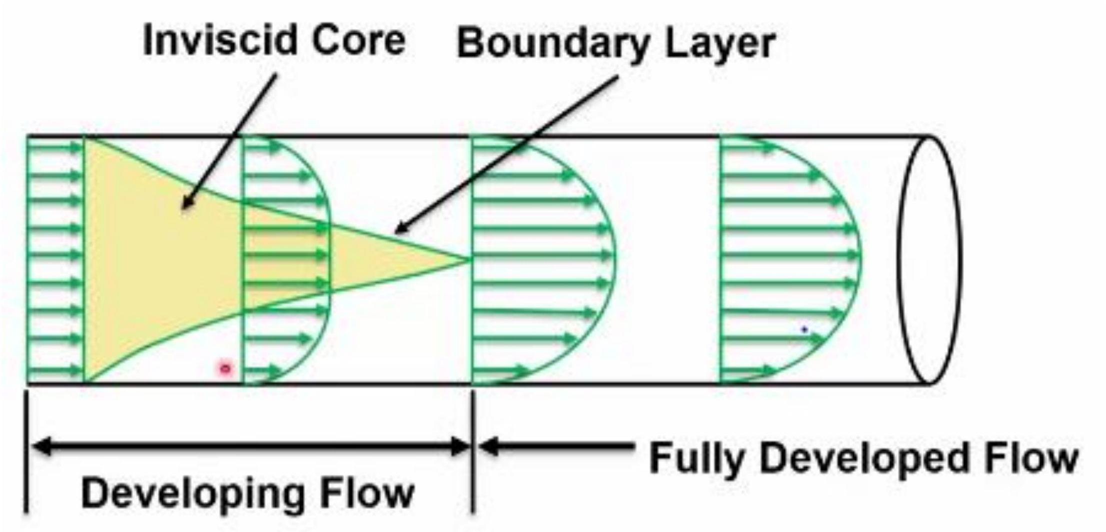


Fig.2. 5 Laminal flow velocity profile

The expected trends are shown in the figure above. So, the flow has a uniform velocity coming in, and as the flow moves into the pipe, the flow at the wall has to stick to the pipe, that’s a no-slip condition. As a result of that, the flow near the wall gets decelerated by viscous friction or viscous shear. The region that get affected by viscosity increases as you move down stream and the region that is affected by the viscosity is called the boundary layer, as shown in the figure.

Outside of the boundary layer the flow has not yet been affected by viscosity, so we have an inviscid core. Due to this, the flow tends to decelerate, therefore, that of the inviscid core has to accelerate to keep the same mass flow going through. And ultimately, the boundary layer would grow and then the boundary layers from the opposite walls will merge. And then, in that region you would get the classic parabolic profile. And this is called the fully developed region.

In the fully developed region, the velocity profile doesn't change, therefore, any location downstream in this region still have the same parabolic profile. The particle movement can be thought of as flowing the same profile with a constant velocity in a straight line, therefore it has no acceleration in the fully develop region.

In the fully developed region there's an analytical solution and parabolic profile, while in the developing region there is no analytical solution but the length of the developing region known as the entrance length, L_e, can be predicted.

$$\frac{L_e}{D} \approx 0.06\,Re_D \text{ --}2.14$$

This is something we check valid our numerical solution from the ANSYS Fluent solver. We can also predict what the trend of the velocity at the axis of the centerline velocity has to be.

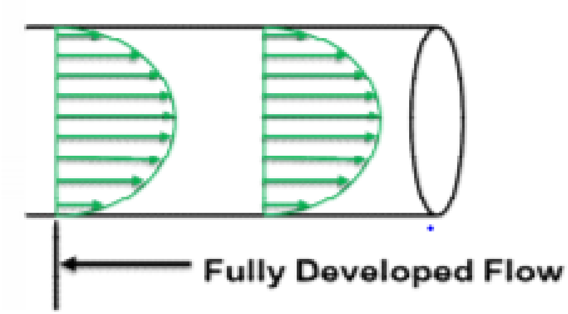


Fig.2. 6 Velocity Profile of Developed Laminar Flow

For the fully developed region, as shown in the figure above, continuity can be represented as;

$\frac{1}{r}\frac{\partial(rv_r)}{\partial r} + \frac{\partial v_z}{\partial r} = 0$ ---------------------------------2.15

The axial velocity is no longer changing in the axial direction, so $\frac{\partial v_z}{\partial r}$ is zero automatically making $\frac{1}{r}\frac{\partial(rv_r)}{\partial r}$ equal to zero. The implication is that the radial velocity can have no gradient in the radial direction, i.e. the radial velocity has to be constant along a line like that. But the radial velocity is zero at the wall. Therefore;

$$v_r = 0$$

The conservation of momentum in radial direction can be represented by;

$\rho\left(v_r\frac{\partial v_r}{\partial r} + v_z\frac{\partial v_r}{\partial z}\right) = -\frac{\partial P}{\partial r} + \mu\left(\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial v_r}{\partial r}\right) - \frac{v_r}{r^2} + \frac{\partial^2 v_r}{\partial z^2}\right)$ -------------------------------2.16

Since the radial velocity, v_r, is zero, it means the pressure gradient in the radial direction is equal to zero, i.e. pressure is only a function of the axial coordinate ($P = P(z)$)

In the axial direction, momentum conservation is given as;

$\rho\left(v_r\frac{\partial v_z}{\partial r} + v_z\frac{\partial v_z}{\partial z}\right) = -\frac{\partial P}{\partial r} + \mu\left(\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial v_z}{\partial r}\right) - \frac{v_r}{r^2} + \frac{\partial^2 v_z}{\partial z^2}\right)$ ----------------------------2.17

Recall, $v_r = 0$ and also the particle is moving in a straight line at a constant velocity, the equation is then reduced to;

$0 = -\frac{\partial P}{\partial r} + \mu\left(\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial v_z}{\partial r}\right)\right)$ -------------------------------------2.18

So, if an infinitesimal fluid particle is considered, and set this equation to a constant and integrate it, the parabolic profile would be obtained.

In the fully developed region, because of the simplicity of the equation, an analytical solution can be obtained that is valid only in that region. The velocity profile comes out to be parabolic and can be written in the form;

$$\frac{v_z}{\overline{V}} = 2\left[1-\left(\frac{r}{R}\right)^2\right] \text{---------------------------------}2.19$$

If r = 0, it implies that the axial velocity is twice the average velocity. The analytical solution also gives the skin friction coefficient which is a normalized wall shear, and can be written in the form:

$$C_f = \frac{\tau_w}{0.5\,\rho\,\overline{V}^2} = \frac{16}{Re_D} = 0.16 \text{ (if } Re_D = 100 \text{) ------------------------------}2.20$$

It basically gives the friction at the wall and It is related to;

$$\tau_w = \mu\left.\frac{\partial u_z}{\partial y}\right|_w \text{------------------------------------}2.21$$

From the velocity profile, we can differentiate it and calculate, then substitute it back to the skin factor coefficient. Note: comparison of analytical solution and that of fluent numerical solution might be different because of additional assumptions.

2.2.2 TURBULENT FLOW

When the Reynolds number is above the **critical Reynolds number**, Re_c, usually 2100 (Liu Henry 2003), the flow is termed to be turbulent. In fluid dynamics, turbulence or turbulent flow is fluid motion characterized by erratic variations in pressure and flow velocity. It is in contrast to a laminar flow, which happens when a fluid flows in parallel layers, with no interruption between those layers.

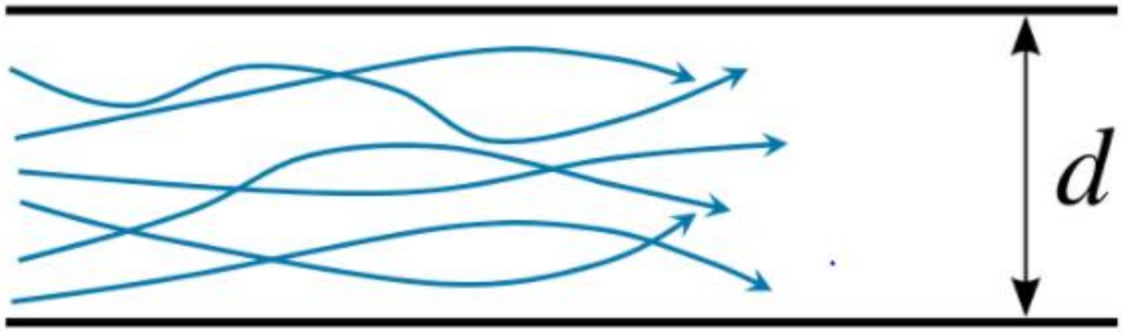


Fig.2. 7 Turbulent Flow Particles Movement Profile

Here acceleration forces are large in comparison with the viscous force. Turbulence is characterized by the following features:

- **Irregularity**

Turbulent flows are notorious for being exceedingly erratic. Turbulence problems are typically tackled statistically rather than deterministically as a result of this. The flow of turbulence is chaotic. Not all chaotic flows, however, are turbulent.

- **Diffusivity**

The readily available supply of energy in turbulent flows tends to accelerate the homogenization (mixing) of fluid mixtures. The characteristic which is responsible for the enhanced mixing and increased rates of mass, momentum and energy transports in a flow is called "diffusivity

Turbulent diffusion is usually described by a turbulent diffusion coefficient. This turbulent diffusion coefficient is defined in a phenomenological sense, by analogy with the molecular diffusivities, but it does not have a true physical meaning, being dependent on the flow conditions, and not a property of the fluid itself. In addition, the turbulent diffusivity concept assumes a constitutive relation between a turbulent flux and the gradient of a mean variable similar to the relation between flux and gradient that exists for molecular transport. In the best case, this assumption is only an approximation. Nevertheless, the turbulent diffusivity is the simplest approach for quantitative analysis of turbulent flows, and many models have been postulated to calculate it.

On introducing into a turbulent flow field, a velocity sensor which is capable of measuring the local instantaneous velocity, measured results of velocity dependence on time as shown in figure;

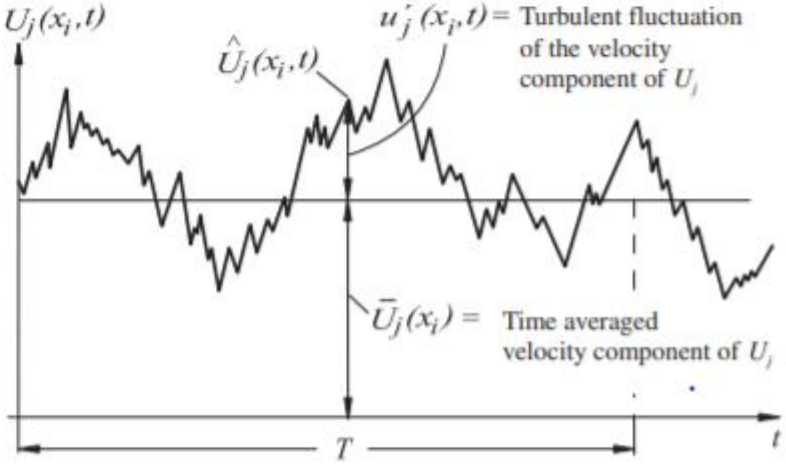


Fig. 18.3 Time velocity path at a point x_i within a turbulent flow field

Fig.2. 8 Time velocity at a point x, within a turbulent flow field

Hence the average velocity is defined as followed:

$\bar{u}_i\ (x_i) = \lim_{T\to\infty}\frac{1}{T}\int_0^T \hat{u}_i\ (x_i,t)\ dt$ ----------------------------------2.22

The instantaneous velocity, $\hat{u}_i\;(x_i,t)$, can be decomposed into a time-average part, $\bar{u}_i\;(x_i)$, and a fluctuating part, $u'_i\;(x_i,t)$. The average velocity equation can be rewritten as;

$\underbrace{T\bar{u}_i\;(x_i)}_{\substack{\text{rectangular}\\ \text{area}}} \;=\; \underbrace{\lim_{T\to\infty}\int_0^T \hat{u}_i\;(x_i,t)\;dt}_{\substack{\text{integral over}\\ \text{time-dependent signal}}}$ ---------------------------------2.23

When considering this definition of the time mean value of velocity, then the quantity $u'_i\;(x_i,t)$, designated as turbulent velocity fluctuation, the following equation holds:

$u'_i\;(x_i,t) = \hat{u}_i\;(x_i,t)\;-\;\bar{u}_i\;(x_i)$ ------------------------------2.24

Applying to this relationship the operator $\lim\limits_{T\to\infty}\frac{1}{T}\int_0^T(\,)\;dt,$ the following can be carried out:

$\lim\limits_{T\to\infty}\frac{1}{T}\int_0^T u'_i\;(x_i,t)\;\;dt = \lim\limits_{T\to\infty}\frac{1}{T}\int_0^T[\,\hat{u}_i\;(x_i,t)\;-\;\bar{u}_i\;(x_i)]\;dt$ ------------------------2.25

$= \underbrace{\lim\limits_{T\to\infty}\frac{1}{T}\int_0^T\;\hat{u}_i\;(x_i,t)\;dt}_{=\bar{u}_i} - \underbrace{\lim\limits_{T\to\infty}\frac{1}{T}\int_0^T\;\bar{u}_i\;(x_i)\;dt}_{=\bar{u}_i}$ --------------------------2.26

The two integrals on the RHS of the equation are equal, and their difference equals 0, implying that the following holds for the time average turbulent velocity fluctuation:

$\lim\limits_{T\to\infty}\frac{1}{T}\int_0^T u'_i\;(x_i,t)\;dt = \overline{u'_\imath\;(x_\imath,t)} = 0$ ----------------------------2.27

And the overbar on $u'_i\;(x_i,t)$ reflects a simplified manner of stating the time averaging that was done. When the turbulent velocity variations are designated with $u'_i\;(x_i,t)$ (or simply u'_i), the it can be said that the time average of the turbulent velocity fluctuations u'_j is equal to zero per definition. As a result, there is a way to express turbulence in local, time-varying quantities in such a way that the turbulent fluctuations of all the variables are represented. The flow quantities that are used in the calculations show a time frame. It has a zero-mean value.

For the fluctuating velocity quantity u'_j moments of higher order can also be defined:

$\overline{u'^{n}_{j}} = \lim_{T\to\infty} \frac{1}{T}\int_0^T u'^{n}_{j} \; dt,$ -----------------------------2.28

which, in general, display values other than zero. The following holds true, particularly for the rms value of turbulent velocity fluctuations:

$\sigma_i = \sqrt{\overline{u'^{2}_{j}}} = \sqrt{\lim_{T\to\infty} \frac{1}{T}\int_0^T \overline{u'^{2}_{j}} \; dt}$ -----------------------------2.29

This can be used for the definition of the turbulent intensity:

$T_u = \frac{\sqrt{\frac{1}{2}\overline{u_i u_i}}}{\bar{U}_{tot}} = \frac{\sqrt{\frac{1}{2}\left(\overline{u_1^2} + \overline{u_2^2} + \overline{u_3^2}\right)}}{\bar{U}_{tot}}$ -------------------------------2.30

This quantity represents a measure of the intensity of the turbulent fluctuations of the velocity components with respect to the local mean value $\bar{U}_{tot}$.

2.2.3 MULTIPHASE MODEL

The multiphase model in this CFD study is a Eulerian model based on the Euler-Euler method (Fluent, 2019). This is due to the fact that this research involves solid–liquid–gas three-phase flows, which contain both granular (fluid–solid) and non-granular (fluid–fluid) flows. The Eulerian approach is well-known for its ability to successfully address many types of couplings using individual momentum and continuity equations (Anderson and Jackson, 1967).

The rules of mass and momentum are satisfied by each phase independently, while volume fractions represent the space occupied by each phase. The conservation equations can be determined by averaging the local instantaneous balance for each phase (Anderson and Jackson, 1967) or by applying the mixture theory approach (Anderson and Jackson, 1967). (Bowen, 1976).

The volume of the p, $\cancel{V_p}$, is defined by

$\cancel{V_p} = \int z_p \, d\cancel{V}$----------------------------------2.31

And the sum of the volume fraction is equal to 1, i.e.;

$\sum_{q=1}^{n} z_p = 1$ -----------------------------2.32

For a uniform velocity, the conservation of mass for the mixture is as below:

$\frac{d}{dt}\left(z_p \rho_p \cancel{V_p}\right) + \nabla.\left(z_p \rho_p V_p S_p\right) = 0$ ------------------------------- 2.33

The conservation of momentum for a fluid phase p is:

$\frac{d}{dt}\left(z_p \rho_p \cancel{V_p}\right) + \nabla.\left(z_p \rho_p V_p V_p\right) = -z_p \nabla \mathrm{p} + \nabla.\bar{\bar{\tau}}_p + z_p \rho_p g_p + \sum_{q=1}^{n}\{K_{pq}\,(V_{p2} - V_{p1}) +$

$\dot{m}_{p1} V_{p1} - \dot{m}_{p2} V_{p2}\} + (\vec{F}_p + \vec{F}_{lift,p} + \vec{F}_{vm,p})$ -----------------------2.34

A multi-fluid granular model is used to characterize the flow behavior of a fluid-solid mixture, based on the work of Alder and Wainwrigh (1960), Chapman and Cowling (1970), and Syamlal et al. (1993).

The conservation of momentum for solid phase is:

$$\frac{d}{dt}(z_s \rho_s \cancel{V_s}) + \nabla.(z_s \rho_s V_s V_s) = -z_s \nabla \mathrm{p} - \nabla \mathrm{p}_s + \nabla.\bar{\bar{\tau}}_p + z_s \rho_s g + \sum_{l=1}^{n} \{K_{ls}\,(V_l - V_s) + \dot{m}_{ls} V_{ls} - \dot{m}_{sl} V_{sl}\} + (\vec{F}_s + \vec{F}_{lift,s} + \vec{F}_{vm,s}) \text{ --------------------------------2.35}$$

2.2.4 SAND TRANSMISSION IN PIPELINE

While the solid particles in a liquid are very fine (say, in the 0.1- to l-m range) and much denser than the liquid, they may settle out by gravity over time, but when the fluid is flowing, even if the flow is laminar, they may become uniformly suspended (homogeneous). Although laminar flow does not include turbulence, the velocity difference across the pipe can cause particles to rotate and take an uneven route, comparable to Brownian motion but on a larger scale. In a laminar flow with a relatively high velocity, such random motion of very fine particles might allow the particles to be suspended in a homogenous condition, but the particles may settle out when the flow is interrupted or the velocity of the laminar flow is too low.

The terminal velocity (settling velocity) of the denser-than-fluid particles in the 1- to 10-m range may be sufficient to induce the particles to settle out of laminar flow, yet turbulent flow uniformly disperses or suspends them.

The flow regimes in slurry transport were divided into four categories by Turian and Yuan (1977). The extended pressure drops correlation method observed in slurry transport was used to develop these four correlations stated below:

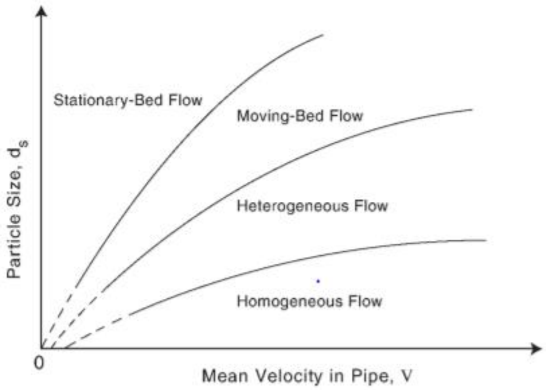
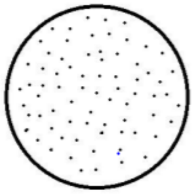


Fig.2. 9 Graphical Representation of Slurry Flow Classification

2.2.4.1 PSEUDO-HOMOGENEOUS

When the particle size is tiny and the pipe flow velocity is high, this happens, resulting in comparatively fine particles in a very turbulent flow.



$$f - f_w = 0.8444\, C^{0.5024}\, f_w^{1.428}\, C_D^{0.1516} \left[\frac{v^2}{Dg(s-1)}\right]^{-0.3531}$$

Fig.2. 10 Pseudo-homogeneous Flow

2.2.4.2 HETEROGENEOUS

This occurs when particles are fully suspended but not uniformly distributed nonhomogeneous. This happens when either the velocity is somewhat smaller or the particle size is somewhat larger than in the previous case. Heterogeneous flow can be subdivided into two categories: symmetric or asymmetric. Symmetric mixture flow exists when the concentration profile of the solid in the flow is symmetric or approximately symmetric about the centerline of the pipe, though the concentration may not be uniform or homogeneous in the radial direction across the pipe.

$$f - f_w = 0.5513\, C^{0.8687}\, f_w^{1.200}\, C_D^{-0.1677} \left[\frac{v^2}{Dg(s-1)}\right]^{-0.6938}$$

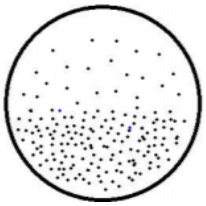


Fig.2. 11 Heterogeneous Flow

2.2.4.3 MOVING-BED (SALTATION) FLOW

Particles settle out of the flow and form a bed. The particles in the bed move in the flow direction by sliding, rolling, or saltation. This happens when either the velocity of the flow is less or the particle size is larger than in the previous case. Saltation refers to the phenomenon that some particles on the surface of the bed layer move intermittently in frog leaps.

$$f - f_w = 0.9857\, C^{1.018}\, f_w^{1.046}\, C_D^{-0.4213} \left[\frac{v^2}{Dg(s-1)}\right]^{-1.354}$$

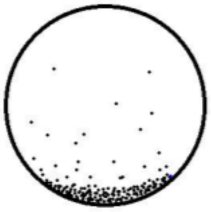


Fig.2. 12 Moving-Bed Flow

2.2.4.4 STATIONARY-BED FLOW

Particles settle out on the bed and they do not move in the bed. This happens with very coarse particles or very low velocity in pipes.

$$f - f_w = 0.4036\, C^{0.7389}\, f_w^{0.7717}\, C_D^{-0.4054} \left[\frac{v^2}{Dg(s-1)}\right]^{-1.096}$$

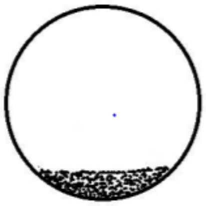


Fig.2. 13 Stationary-Bed Flow

In 1971, Wasp et al. [1] proposed using the following equation for classification of slurry flows:

$$\log\frac{C_T}{C_A} = -1.8\frac{V_s}{ku_i}$$

2.2.5 CRITICAL VELOCITY

In the study of sediment transport by liquid flow in pipes, the key sediment property is the settling velocity, V_s also known as critical or terminal velocity, is the velocity at which the sediment particle settles under gravity in the fluid when the fluid is at rest. It was determined by measuring the flow rate at which the solid particles begin to drop out when the particles were initially in suspension. When fluid is in turbulent flow, the upward movement must be greater than the settling velocity.

When the fluid is moving in the pipe as turbulent flow, the vertical component of the turbulent velocity fluctuations, u'_i, during its upward movement, must be greater than the settling velocity (namely, u'_i>V_s), before the turbulent flow is capable to suspend the sediment. The sediment is suspended in a horizontal turbulent flow under the balance of two forces: gravity that causes the sediment to fall at V_s, and turbulent diffusion due to the turbulence and the existence of a vertical concentration gradient of the sediment particles in the flow.

CHAPTER THREE: METHODOLOGY

3.1 RESEARCH METHODOLOGY

As the aim of the research was to investigated the effect of flow regime on sand transmission in pipeline, numerical and exploratory method of research were employed.

The ANSYS Fluent R19.1 CFD software was employed in this research for building simulation model and analyzing data. ANSYS is an engineering simulation and 3D design software for product modeling solutions with unmatched scalability and a comprehensive Multiphysics. It simulates computer models of structures, electronics component for analyzing strength, toughness, elasticity, temperature distribution, electromagnetism, fluid flow, and other attributes.

Versatile in it use, Fluent is the specific software in the ANSYS package that is capable of simulating fluid flow. It working principle for covering the mathematical models governing the fluid flow as discussed in the preceding chapter, to numerical solution to render simulation calculation is called the **Finite Volume Method (FVM).**

Learning ANSYS Fluent software was a milestone, which began with an online course titled '*A Hands-on Introduction to Engineering Simulation*' at EDX, watching several tutorial videos on YouTube and read serval simulation community articles and chat.

The process used for achieving the objectives are outlined below:

1. Build Pipe Geometry
2. Create Meshing / Label Boundary
3. Define Governing Equations

4. Define Boundary Conditions / Initialize

5. Run Calculation / Post-processing

In this chapter, each of the steps in the methodology will be discussed in detail and verification / validation was employ when necessary.

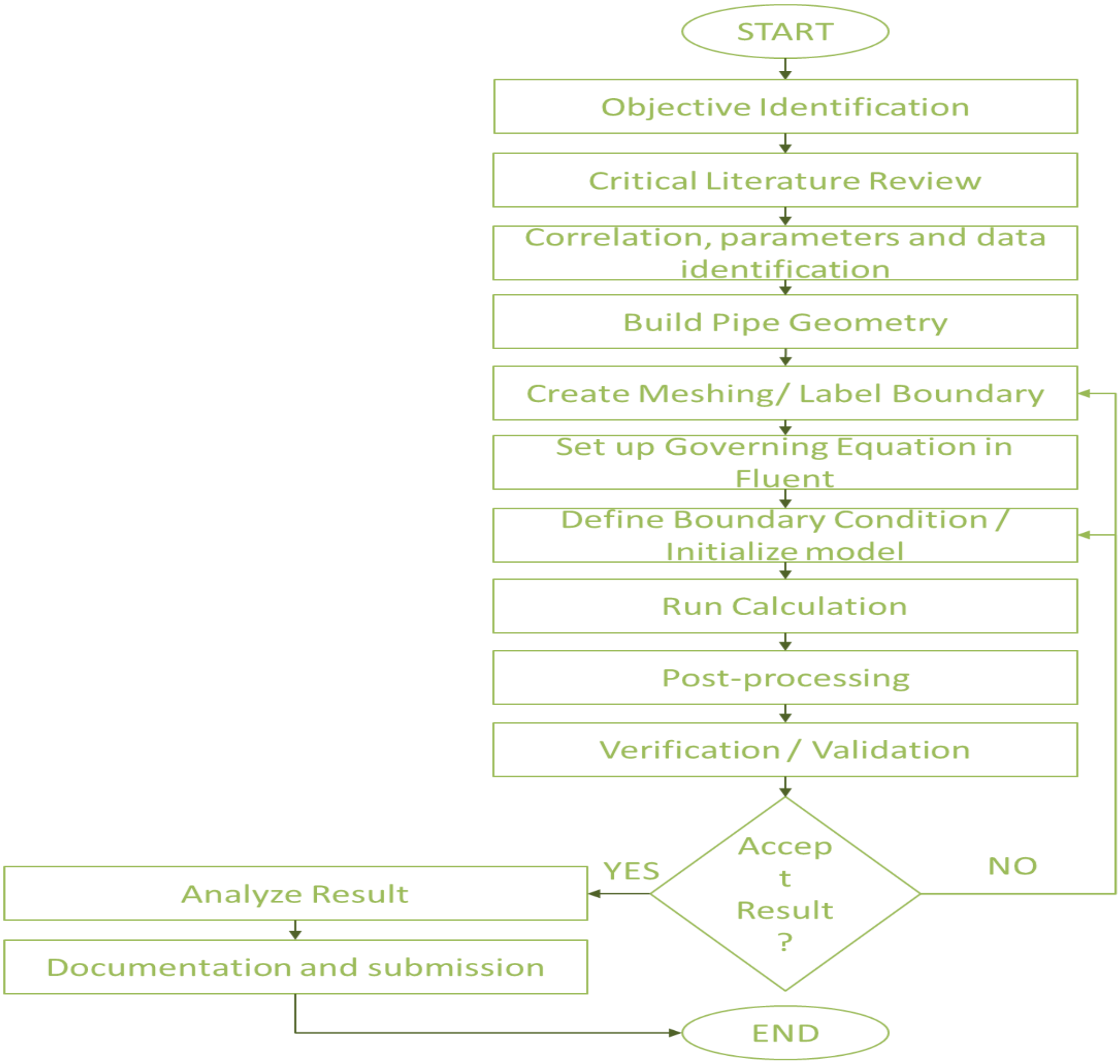


Fig.3. 1 Flow Chart of Research Methodology

3.2 BUILDING PIPE GEOMETRY

Space Claim which is one CAD incorporated in ANSYS was used for building the pipe geometry. The sets taken are explained below.

3.2.1 PIPE CROSS-SECTION SKETCHING

It involves developing of the cross section of the pipe to the desired size. For our pipeline system, a cylindrical was used, therefore, making the cross-section circles. While validating Ansys with experimental data and literatures, two circles were sketched, one to represent the inner diameter (0.027m) of the pipe and the other to represent the external diameter (0.031 m) of the pipeline.

Nonetheless, for the research study, inner diameter of 0.07 m and outer diameter of 0.073 m was employed.

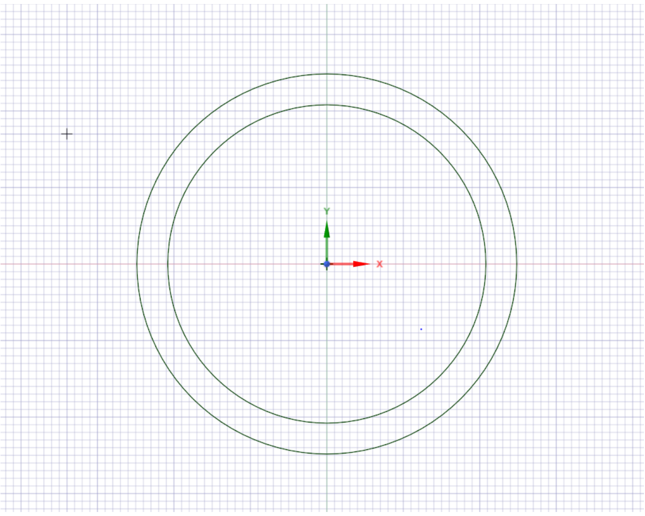


Fig.3. 2 Pipe Cross-section Sketching

3.2.2 EXTRUDING OF CROSS-SECTION SKETCHING

Extruding sketch was converted to a three-dimensional geometry, after which the inner cylinder was deleted to give a hollow cylindrical pipe. The pipe was extended to the desired line using Ansys pull icon (feature). The length of the flow domain was considered long enough to achieve the fully developed flow. Minimum entrance length considered for the flow development and the equation for finding entry length are given by:

$$L_{E,laminar} = 0.06R_eD$$

$$L_{E,turbulent} = 4.4R_e^{\frac{1}{6}}D$$

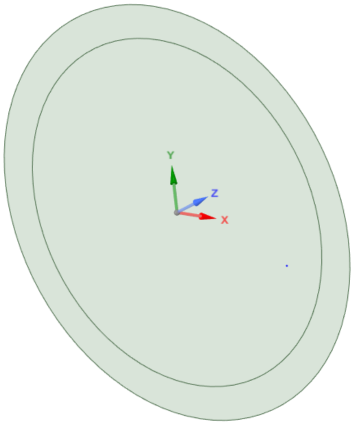


Fig.3. 3 Extrusion with inner circle

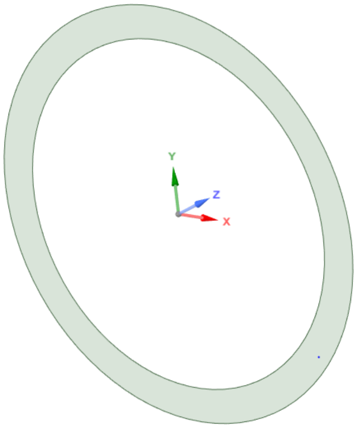


Fig.3. 4 Extrusion without inner circle

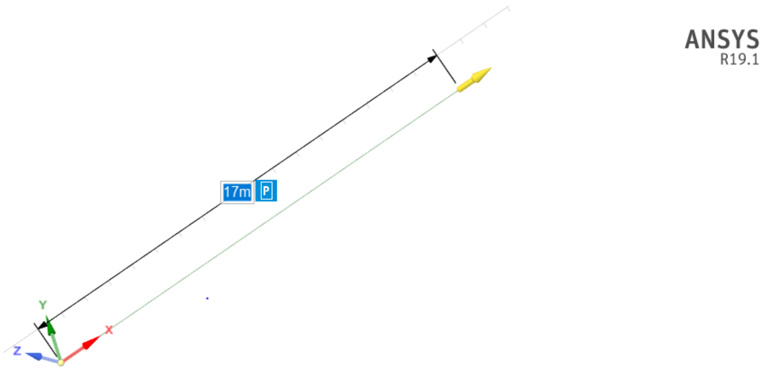


Fig.3. 5 Extrusion extraction to full length

Simulation result were found to be independent on length after full flow development from the inlet for the ANSYS Fluent validation model case study (Rasel A Sultan et al)

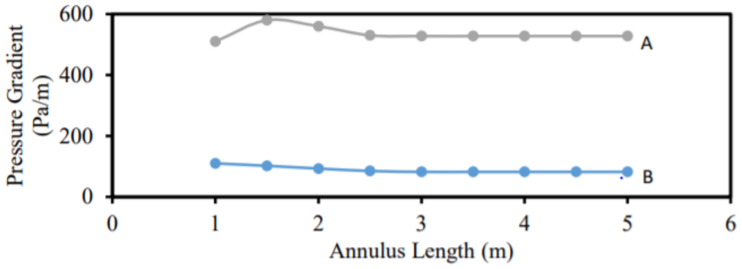
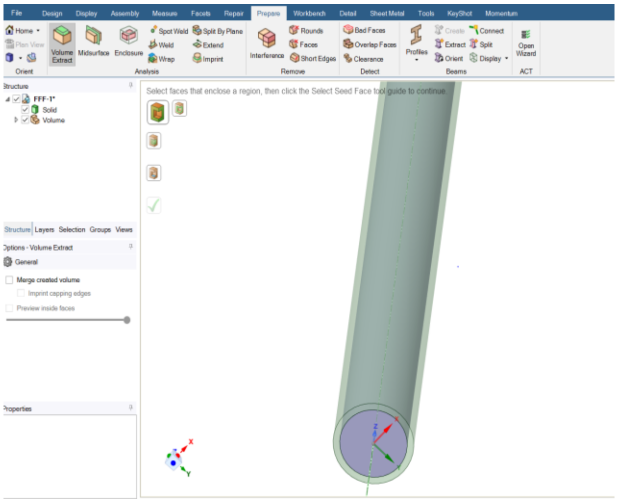


Fig.3. 6 Length independence analysis

3.2.3 PREPARATION OF FLOW VOLUME

The flow path in the hollow pipe is developed by creating a flow path using the volume extract icon. Thereafter, the pipe wall is suppressed from the physic in order to carry-out the study just on the flow path.



Select faces that enclose a region, then click the Select Seed Face tool guide to continue.

☐ Merge created volume
☐ Imprint capping edges
☐ Preview inside faces

Fig.3. 7 Flow Volume with Pipe Wall

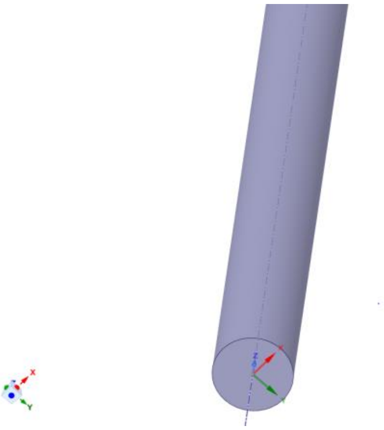
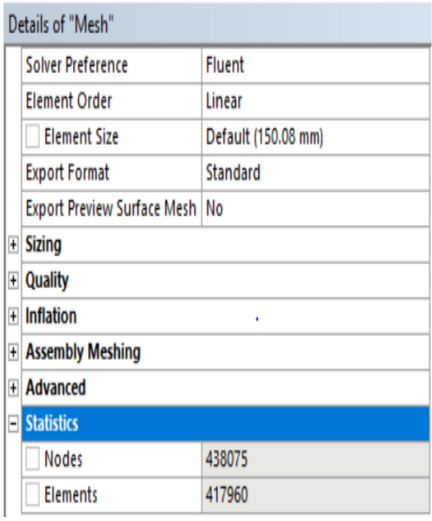


Fig.3. 8 Flow Volume without Pipe Wall

3.3 CREATE MESHING / LABEL BOUNDARY

ANSYS Fluent R19.1 was used to create the computational grids for horizontal pipes and annuli. After thoroughly confirming the mesh independency of the simulation results using data from Fukuda and Shoji's (1986) work, meshing was completed (Rasel A Sultan, 2018). With an increase in the number of nodes above a specific threshold (150000), the output 17 pressure drop became practically constant, as indicated in the figure. The minimal number of nodes required to establish mesh independence for pipelines was 135000, and the comparable figure for annuli was 540000, according to simulations of many other similar data points.



Details of "Mesh"

Solver Preference	Fluent
Element Order	Linear
☐ Element Size	Default (150.08 mm)
Export Format	Standard
Export Preview Surface Mesh	No
⊞ **Sizing**	
⊞ **Quality**	
⊞ **Inflation**	
⊞ **Assembly Meshing**	
⊞ **Advanced**	
⊟ **Statistics**	
☐ Nodes	438075
☐ Elements	417960

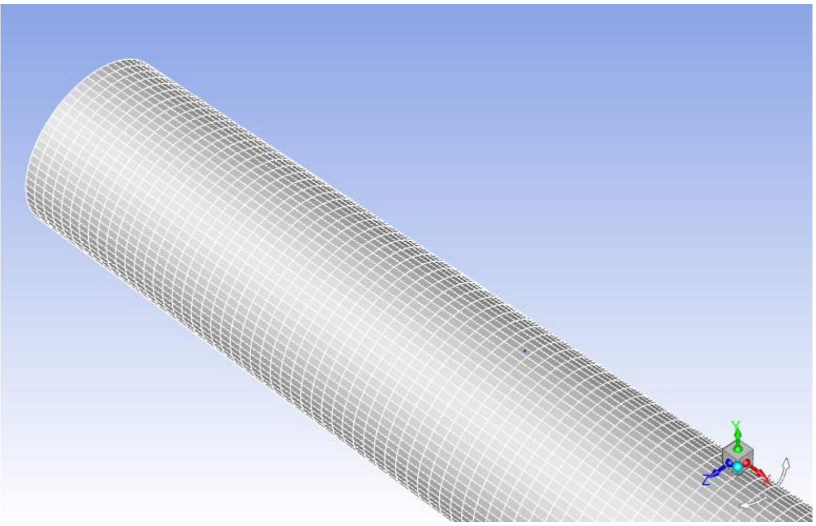


Fig.3. 9 Mesh Details

Fig.3. 10 Pipe Wall Meshing

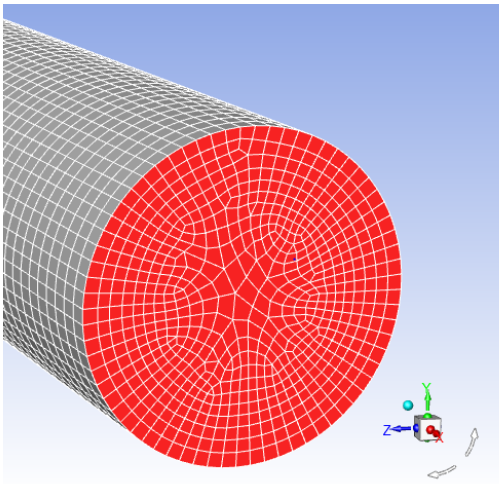


Fig.3. 11 Pipe Cross-section Meshing

Regarding the study, further meshing analysis was carried out to note the influence of the cross-section on the sand particle tracking in the post-processing session of ANSYS Fluent. Prior to finalizing the meshing, the geometry session: Inlet, outlet and wall, were labelled.

3.4 DEFINING GOVERNING EQUATIONS

Fluent solver was setup to display mesh after reading and workbench color scheme, use double precision and employ parallel processing using 4 processes and 1 GPU (computer specification: HP Pavilion Gaming 15, Intel i5-8300H 2.3GHz, 16.0 GB RAM).

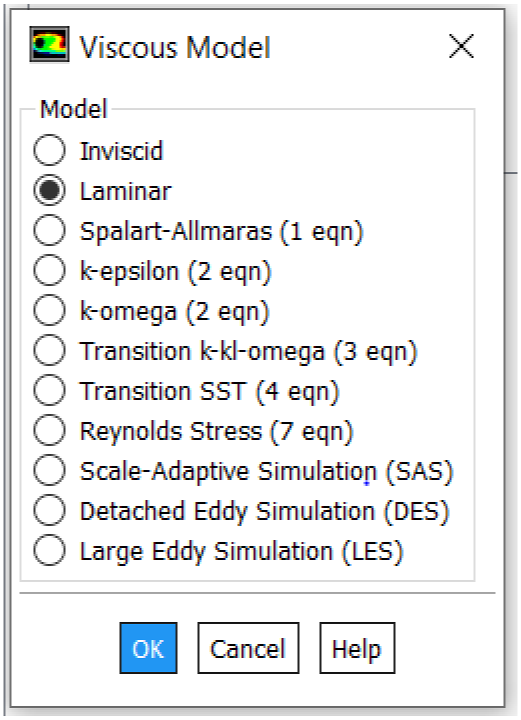
In General, Fluent was setup to solve steady state flow problem, and for the multiphase flow, a Eulerian model based on the Euler-Euler method was employed, were water was used as the continuous phase as sand as the discrete phase. Thereafter, viscous model was then set up in order to properly account for different flow regimes

Phase Properties	
Continuous phase (water)	
Density, kg/m^3	998
Diameter, µm	1.003×10^{-3}
Discrete phase (sand)	
Density, kg/m^3	2650

Table3. 1 Materials Properties

3.4.1 TURBULENCE MODEL

For a CFD problem, the choice of turbulence model depends on the physics of the flow, the degree of accuracy required, and the time required for the solution. It is vital to understand the pro and cons of each model in order to make an informed choice. Regarding this, turbulence model presented in Ansys are summarized as



Viscous Model

Model

○ Inviscid
● Laminar
○ Spalart-Allmaras (1 eqn)
○ k-epsilon (2 eqn)
○ k-omega (2 eqn)
○ Transition k-kl-omega (3 eqn)
○ Transition SST (4 eqn)
○ Reynolds Stress (7 eqn)
○ Scale-Adaptive Simulation (SAS)
○ Detached Eddy Simulation (DES)
○ Large Eddy Simulation (LES)

Fig.3. 12 ANSYS Fluent Turbulence Models

DNS: Direct implementation of fluctuated values into the Naiver-Stokes equation without any turbulence model

LES: A hybrid of DNS and RANS that uses filtered Naiver-Stokes equations for large-scale eddies. To solve small-scale eddies, it is preferable to use an appropriate model.

RANS: A mathematical model for both steady-state and dynamic flows based on average values of variables (unsteady for URANS). The numerical simulation is controlled by an arbitrarily chosen turbulence model to determine the influence of turbulence fluctuation on the mean fluid flow.

RANS methods and sub-models are widely used for many computational fluid dynamics issues because they require less hardware, computational time, and human effort. The use of LES is uncommon, however it is viable in some circumstances where significantly more processing power is required to defeat RANS.

Turbulence model selection

Large eddy simulation (LES) of steady fluid flow through pipelines or annuli has been compared to the performance of popular Reynolds-averaged Naiver–Stokes (RANS) turbulence models such as the k–model, k–model, and Reynolds stress model (RSM) in the open literature (e.g., Markatos 1986; Vijiapurapu and Cui 2010). Vijiapurapu and Cui (2010) compared the experimental results of Zagarola and Smits (1997) and Nourmohammadi, Hopke, and Stukel with the results obtained using different turbulence models (1985). In the comparison, RSM was optimum for turbulence flow through pipelines in terms of both cost and computation time.

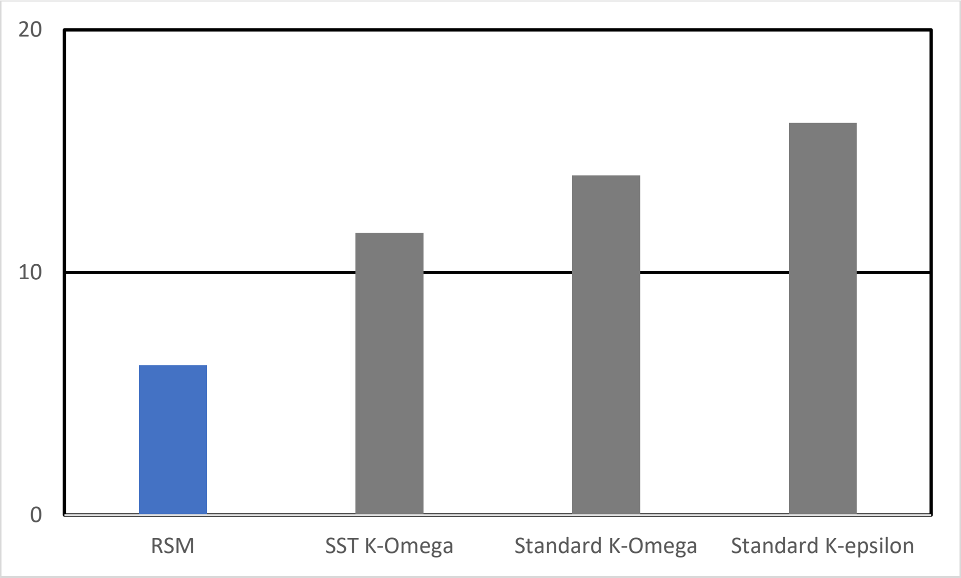


Fig.3. 13 Average Error of Turbulence Model Comparison

Using RSM, 0.82 was proposed as the adjustable constant by applying the gradient-diffusion model to the diffusion term of the RANS equation (Lien and Leschziner 1994).

The pressure-based solver does not compute it by default, but it can be activated in the Viscous Model dialog box. The turbulent Prandtl number has a default value of 0.85. In the Viscous Model dialog box, you can alter the value of Pr_t.

Turbulent mass transfer is processed in the same way, with a turbulent Schmidt number of 0.7 as the default. The Viscous Model dialog box allows you to adjust the default value.

3.4.2 DISCRETE PHASE MODEL

Interaction with continuous phase was set on and DPM iteration interval on 20. For particle tracking, max. number of steps was 50000, whereas, erosion/accretion, saffman lift and virtual mass force were selected under physical model with virtual mass factor of 0.5.

The sand particle was injected from the intel surface at a direction normal to it using constant total flow rate of 1.61e-05 kg/s. based on the classification of sand particles, several particles sizes was analyzed, which are listed below;

Class	Minimum size (mm)	Maximum Size (mm)	Average (mm)
Very fine	1/16	1/8	1/12
Fine	1/8	1/4	1/6
Medium	1/4	1/2	1/3
Coarse	1/2	1	3/4
Very coarse	1	2	0.5

Table3. 2 Particle Size Classification

3.5 DEFINE BOUNDARY CONDITIONS / INITIALIZE

The boundary conditions of the system were;

Boundary Conditions	
Inlet	
Water velocity, m/s	0.01-0.1 (laminar flow) 0.1-1.0 (turbulent flow)
Particle velocity, m/s	0
Hydraulic diameter, m	0.07
Turbulent intensity, %	3.97 – 5.23
Outlet	
Gauge pressure, Pa	0
Hydraulic diameter, m	0.07
Turbulent intensity, %	3.97 – 5.23
Wall	
Phase-1 (water)	No-slip
Phase-2 (sand)	Reflect

Table3. 3 Boundary Conditions

Model was always initialized before running any simulation calculation. The initial conditions where;

3.6 RUN CALCULATION / POST-PROCESSING

After initialization of the model, the simulation calculation was carried-out using 1000 number of iterations. In addition to this, a stop critical of 10e-5 convergence factor was also set.

In ANSYS, post-processing is the process for visualizing result from the calculated model. Charts, planes, line and contours were created in order to get visual understanding of the result.

KEY MILESTONES AND PROJECT ACTIVITIES	**DURATION**																											
	April 2020 – JANUARY 2021														**JANUARY 2021 – JULY 2021**													
	1	**2**	**3**	**4**	**5**	**6**	**7**	**8**	**9**	**10**	**11**	**12**	**13**	**14**	**1**	**2**	**3**	**4**	**5**	**6**	**7**	**8**	**9**	**10**	**11**	**12**	**13**	**14**
FYP I																												
Title selection / proposal		🟩	🟩	🟩	🟩																							
Literature review					🟩	🟩	🟩	🟩	🟩	🟩	🟩	🟩	🟩															
Methodology													🟩	🟩	🟩													
Information gathering for documentation															🟩	🟩	🟩	🟩	🟩	🟩	🟩	🟩	🟩	🟩	🟩			

FYP II																												
CFD Software training			🟩	🟩	🟩	🟩	🟩	🟩	🟩	🟩	🟩	🟩	🟩		🟩	🟩	🟩	🟩	🟩	🟩	🟩		🟩	🟩				
Boundary conditions and parameters identification based on the data gathered																							🟩	🟩				
Modelling and Simulation																									🟩	🟩	🟩	🟩
Data analysis																												🟩

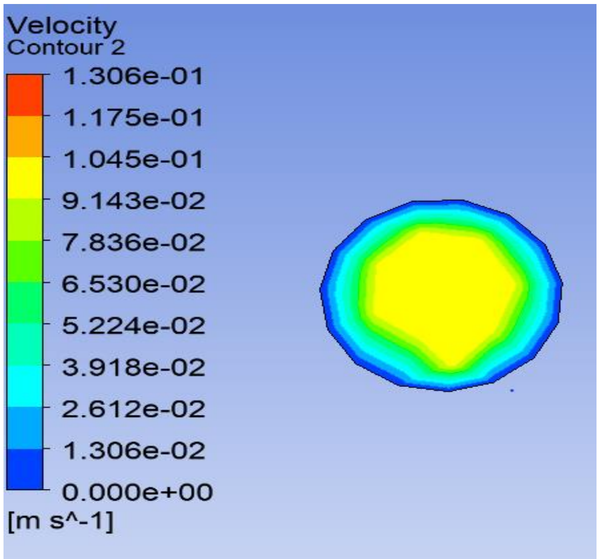
Table3. 4 Project Gantt Chart (MODIFIED from Azrie)

CHAPTER FOUR: RESULT AND DISCUSSION

4.1 RESULT

In this chapter the result obtained from the post-processing of ANSYS Fluent would be presented and discussed.

4.1.1 OUTLET CROSS-SECTION VELOCITY DISTRIBUTION



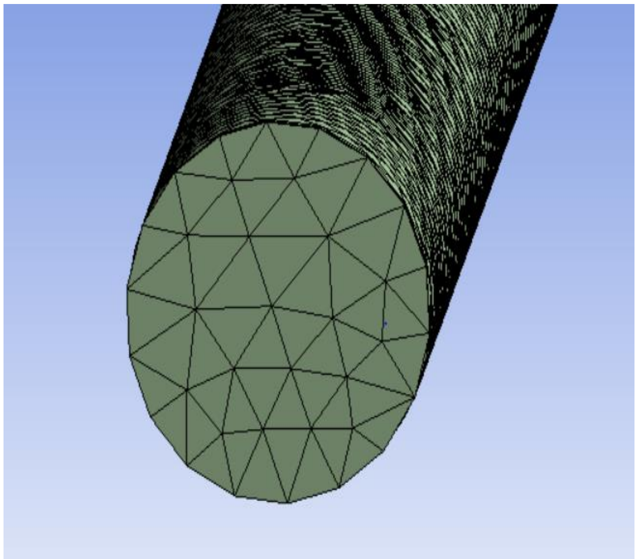
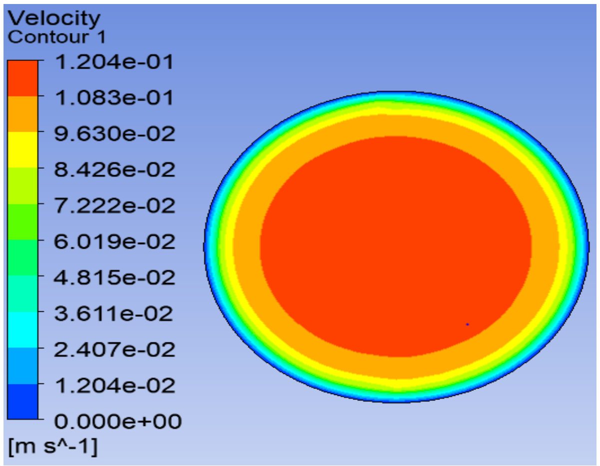


Fig.4. 1 Velocity Distribution for Coarse Meshing Fig.4. 2 Coarse Cross-section Meshing



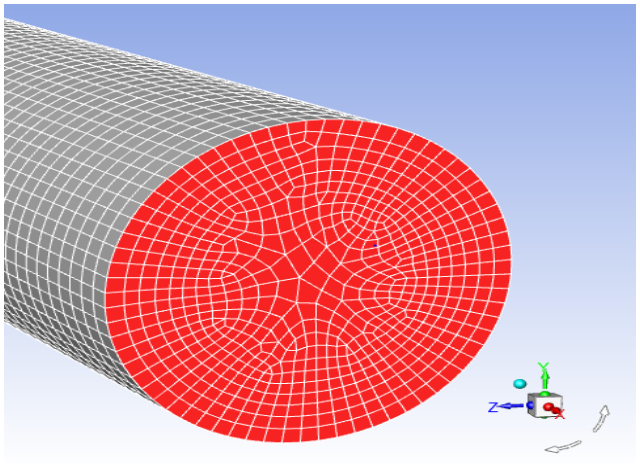


Fig.4. 3 Velocity Distribution for Fine Meshing   Fig.4. 4 Fine Cross-section Meshing

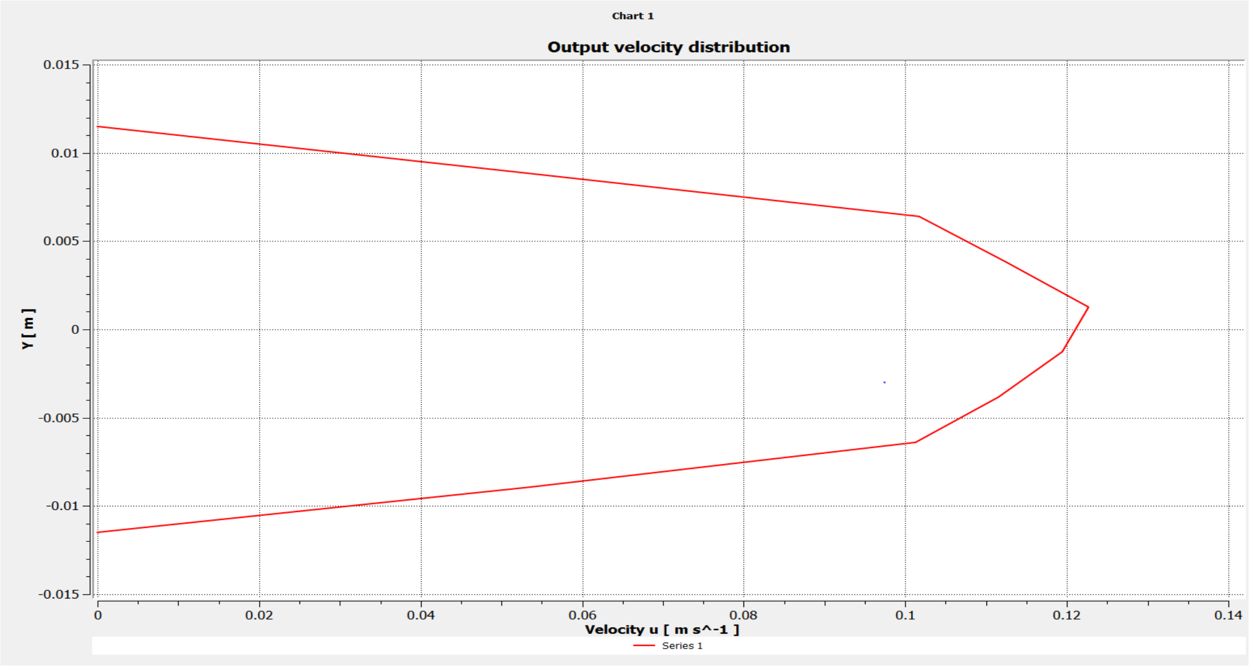


Fig.4. 5 Axial Velocity Distribution for Coarse Meshing at Cross-section

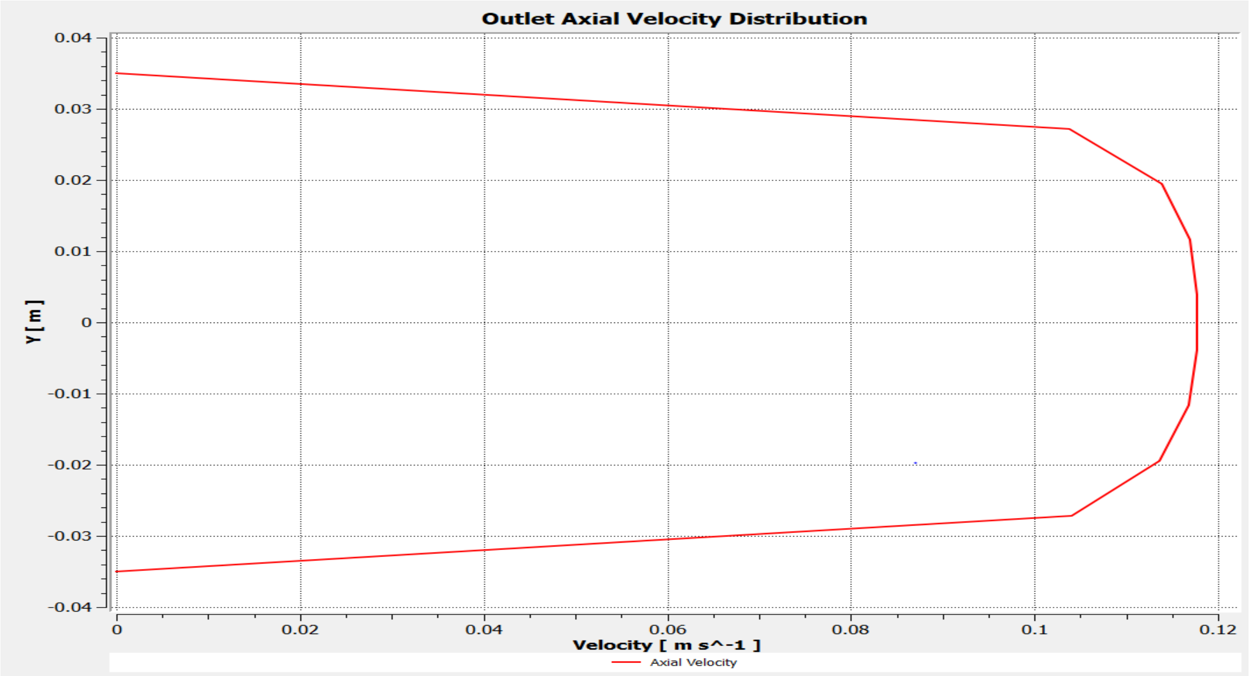


Fig.4. 6 Axial Velocity Distribution for Fine Meshing at Cross-section

4.1.2 SAND TRANSPORTATION AND WALL EROSION (Particle size of 200µm)

4.1.2.1 VELOCITY OF 0.1 M/S

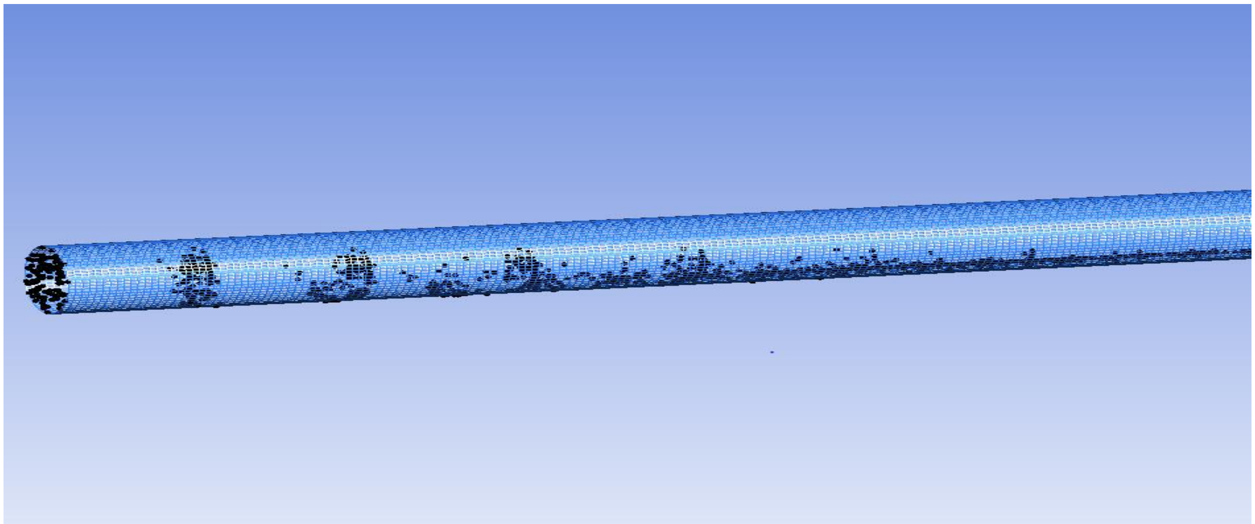


Fig.4. 7 Sand Transport at 0.1 m/s

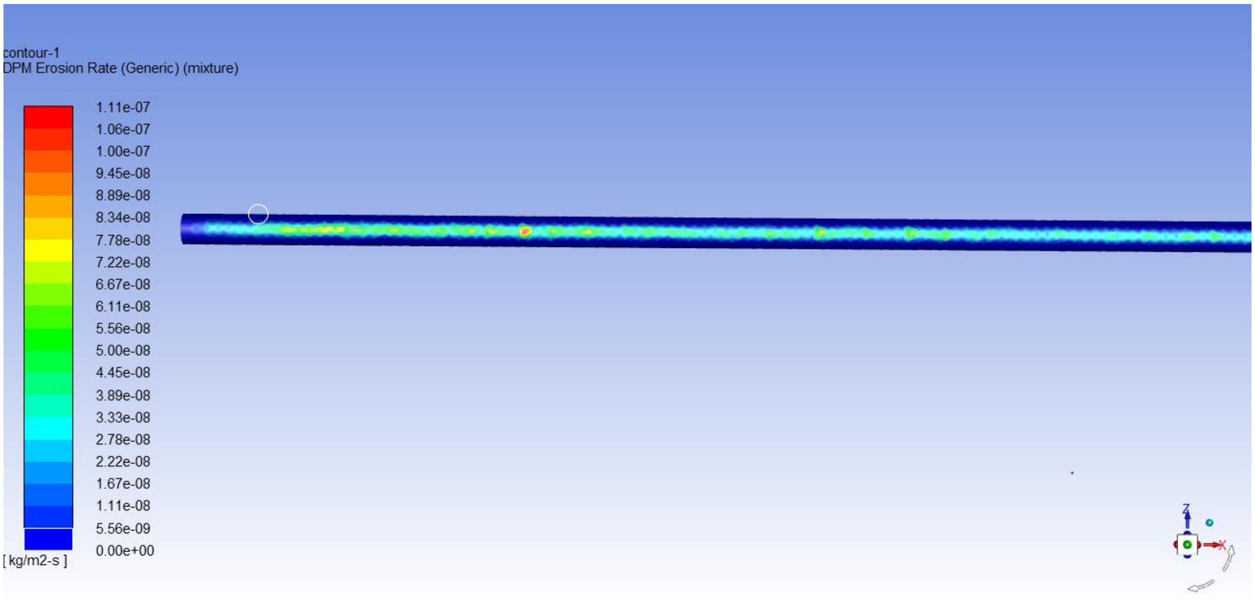


Fig.4. 8 Bottom view of Erosion Distribution at 0.1 m/s

4.1.2.2 VELOCITY OF 0.2 M/S

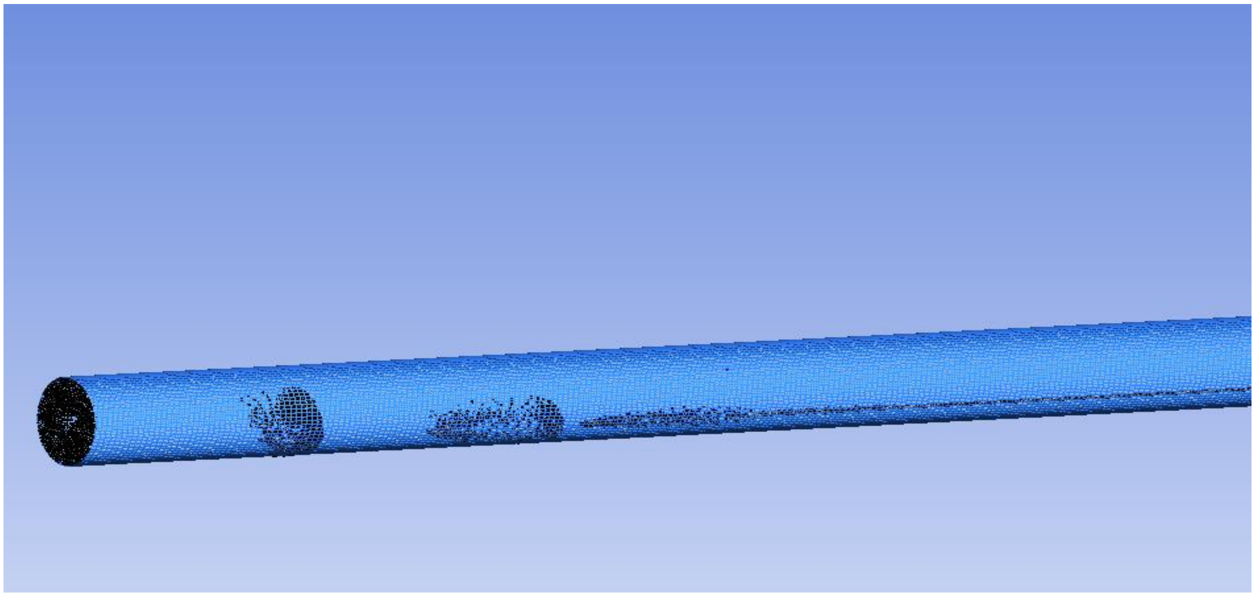


Fig.4. 9 Sand Transport at 0.2 m/s

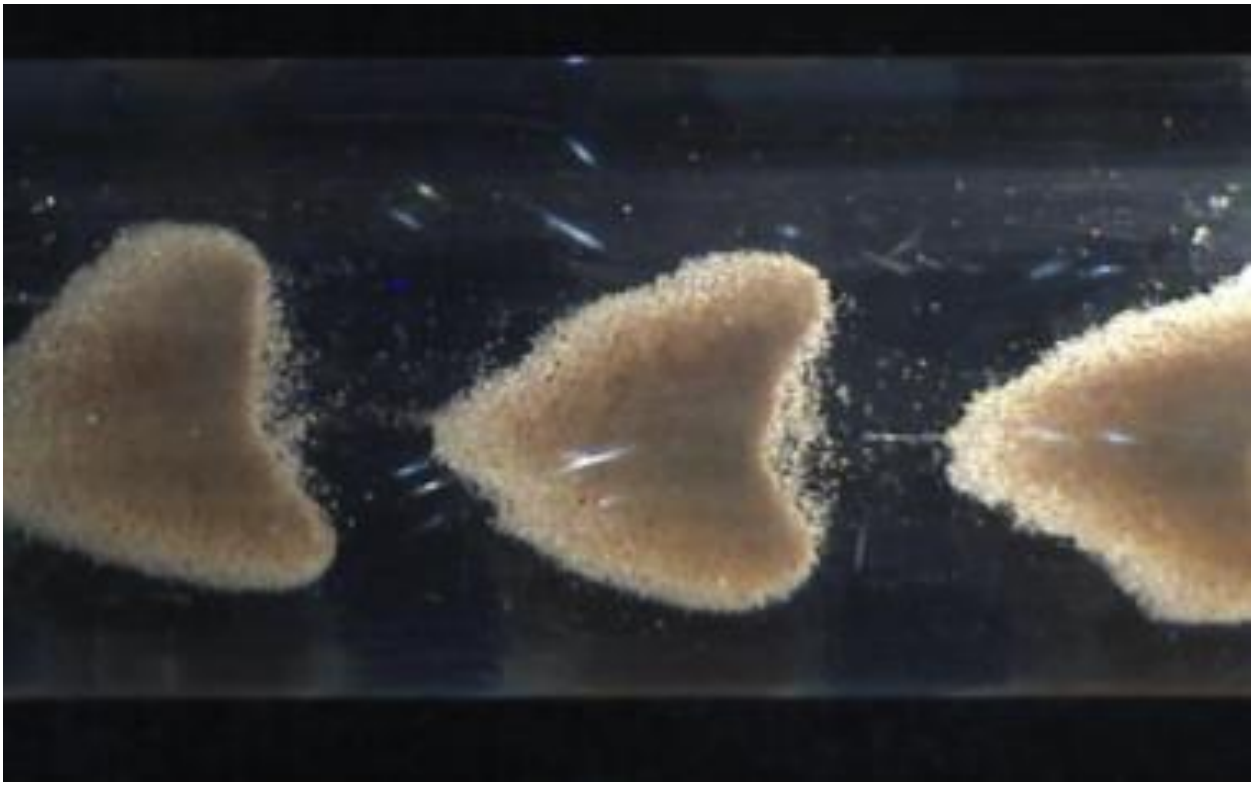


Fig.4. 10 Water velocity at 0.2 m/s, Al-lababidi (2012)

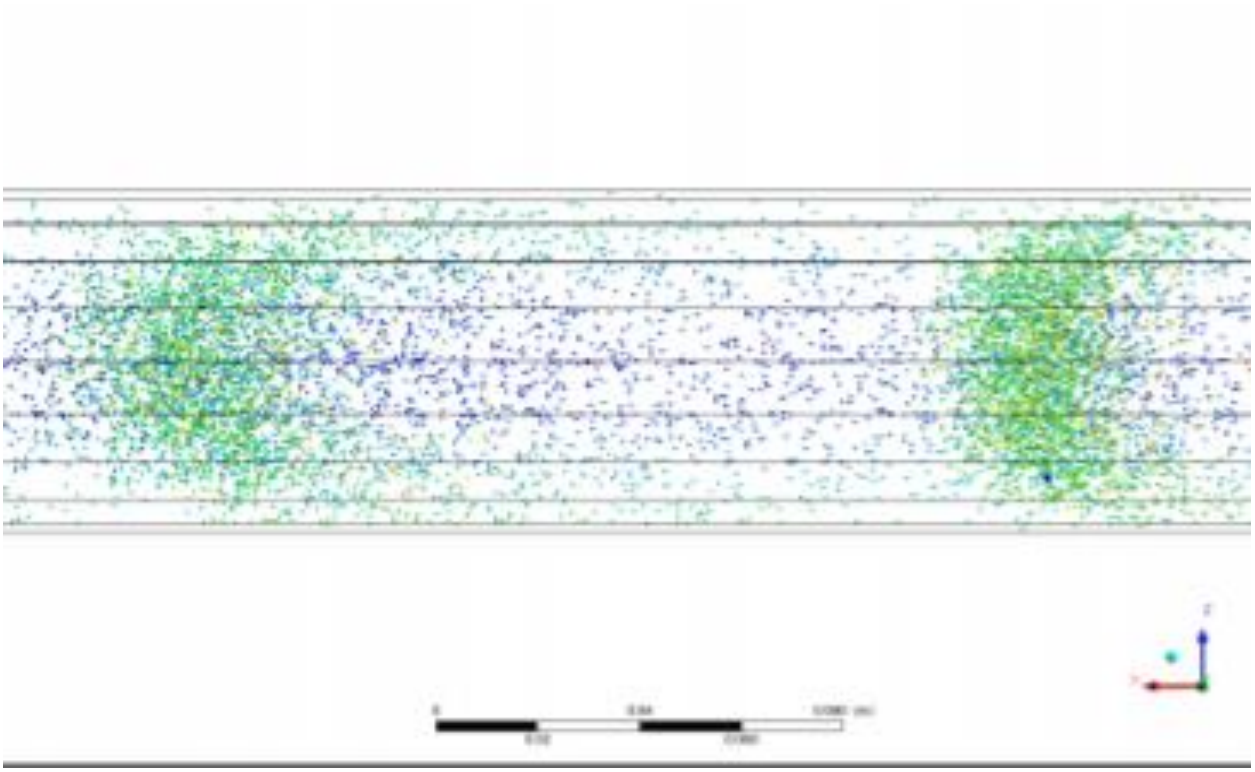


Fig.4. 11 Top view of Particle Distribution at 0.2 m/s

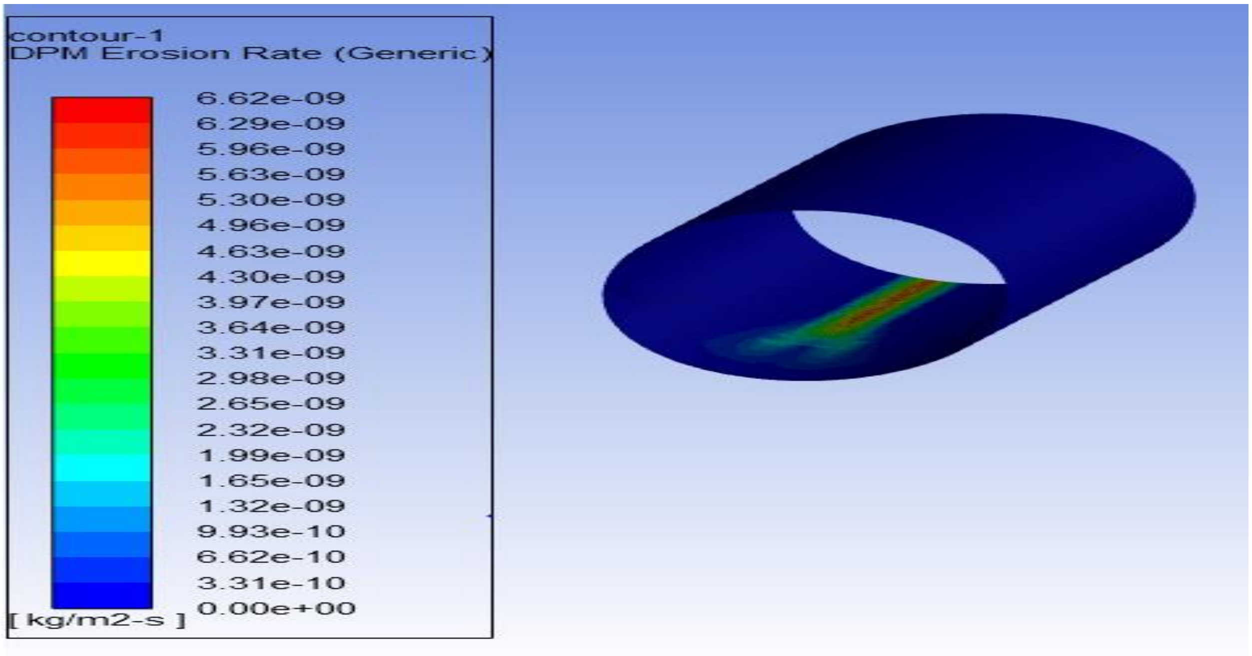


Fig.4. 12 Erosion Distribution at 0.2 m/s

4.1.2.3 VELOCITY OF 0.5 M/S

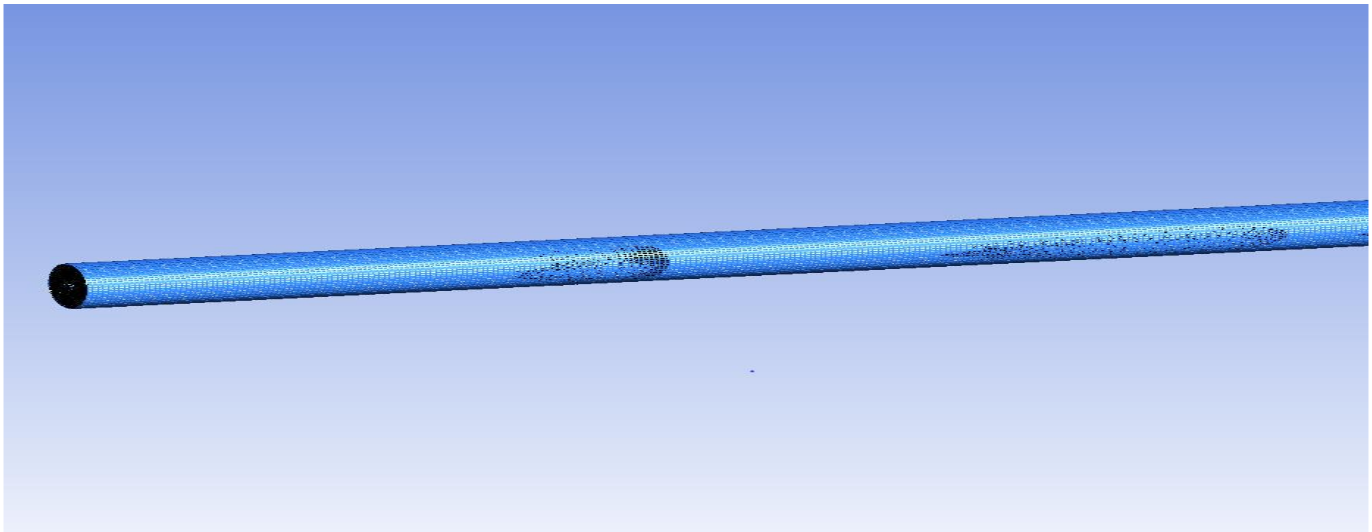


Fig.4. 13 Sand Transport at 0.5 m/s

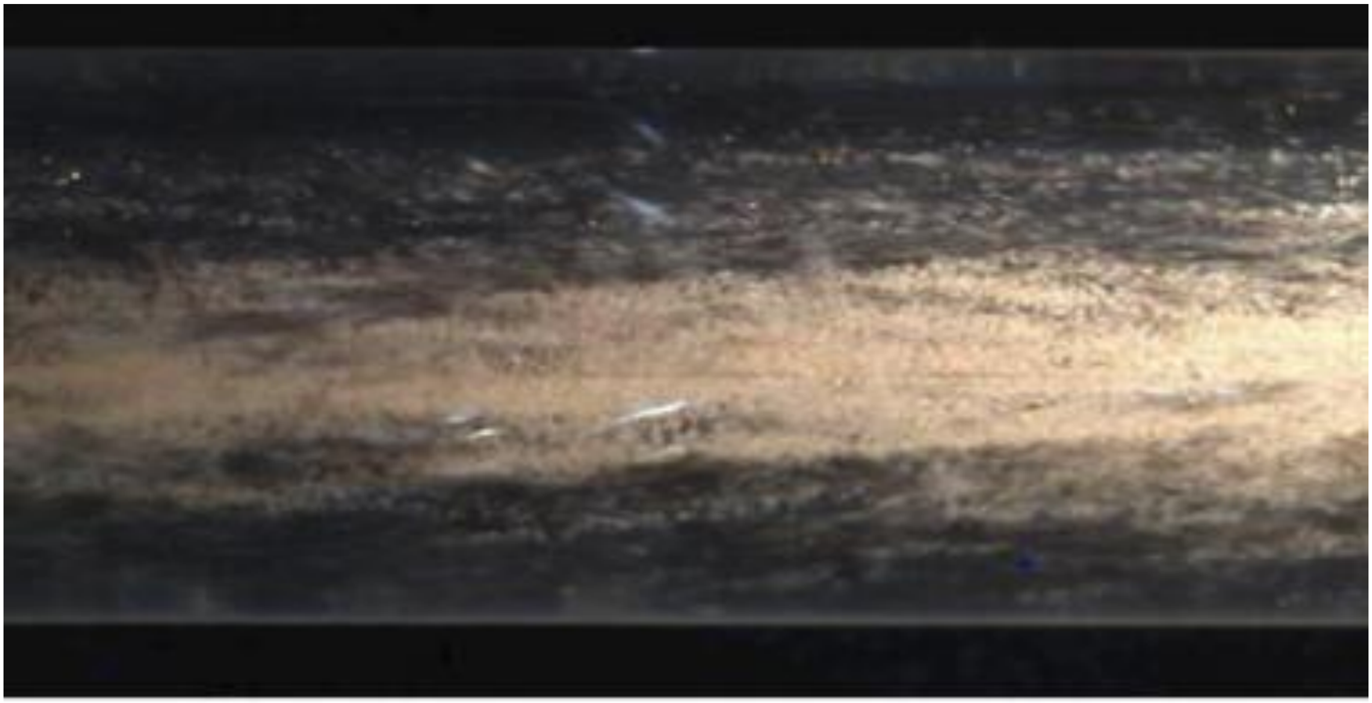


Fig.4. 14 Water velocity at 0.5 m/s, Al-lababidi (2012)

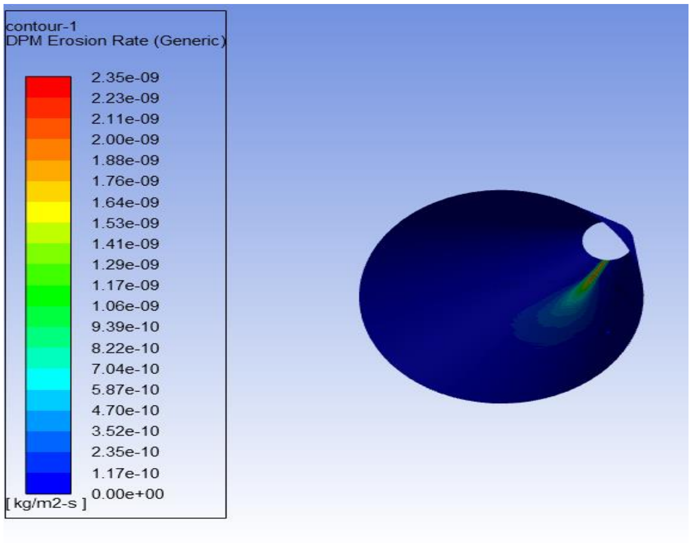


Fig.4. 15 Erosion Distribution at 0.5 m/s

‘

4.1.2.4 VELOCITY OF 100 M/S

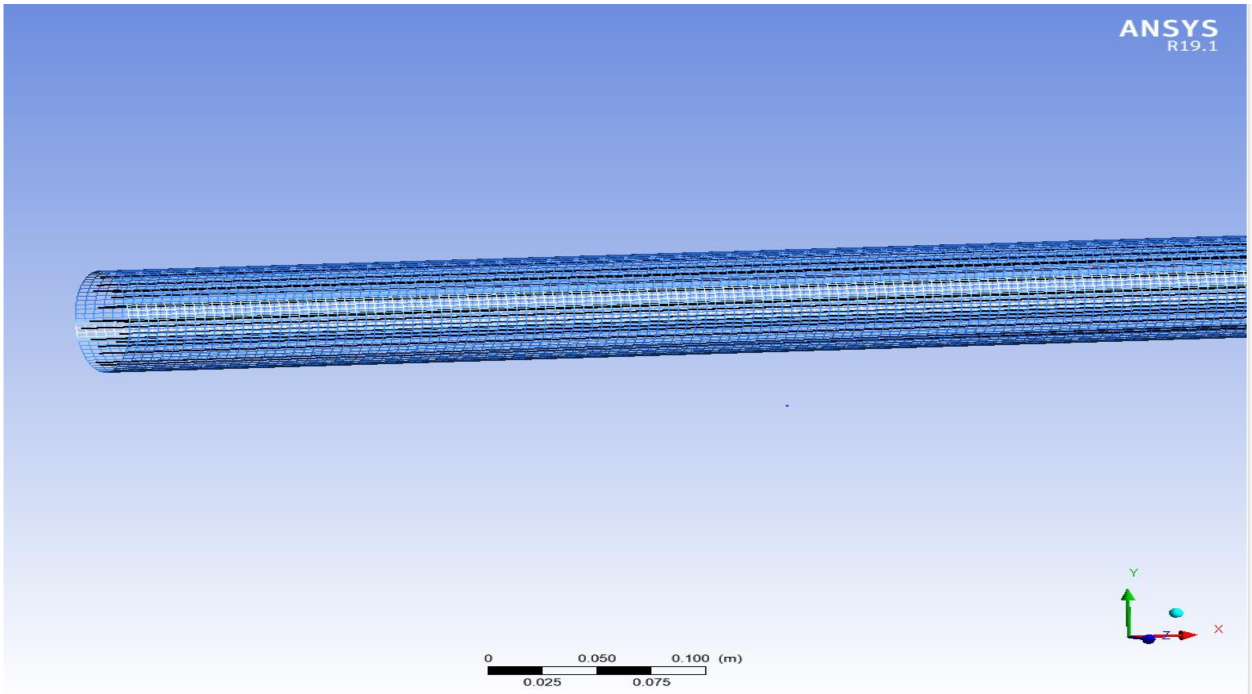


Fig.4. 16 Sand Transport at 100 m/s

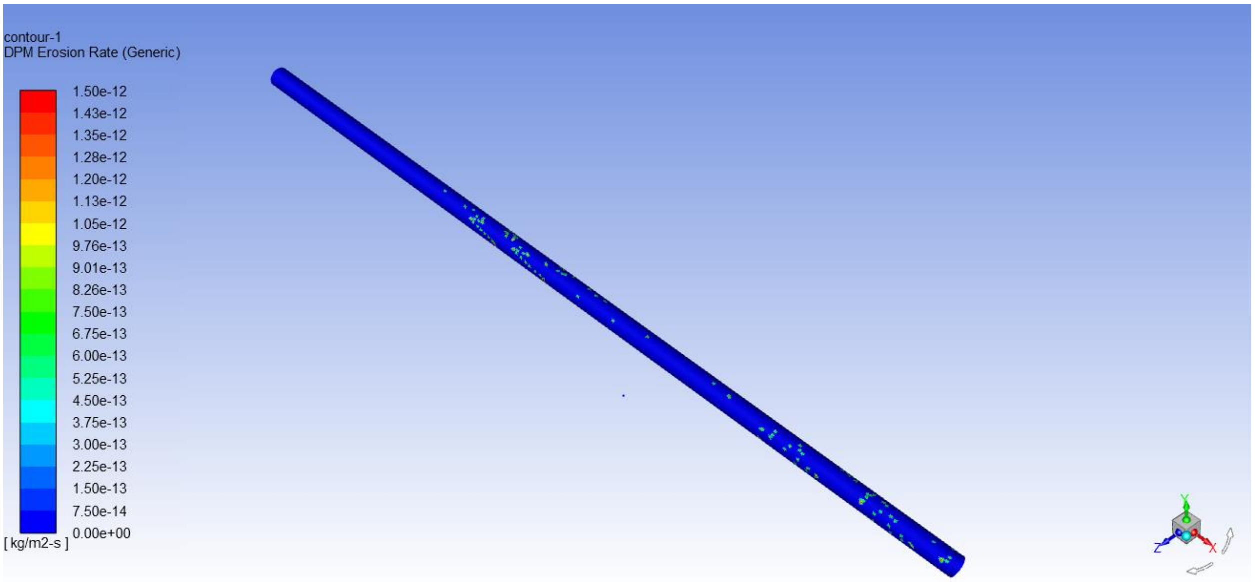


Fig.4. 17 Erosion Distribution at 100 m/s

4.2 DISCUSSION

From the observation in the CFD simulation for the study grain size and concentration, it was noted that laminal flow was not sufficient to transport the sand from the inlet to the outlet. Although, at lower particle size the laminal flow would be able to transmit the sand particle.

For the particle size of 200 µm, the critical velocity was found to be 0.5m/s, and below this rate, the erosion of the was found to be more several that above it.

Below the critical velocity, majority of the erosions occurred at the bottom of the pipe wall and above it, the erosion distributes across the pipe, with minimized severity. Nonetheless, for particles transmitted by laminal flow, the erosion was found negligible.

Visual comparison was done between the simulation and the result obtained from the study done by Al-lababidi (2012) which displayed sand dune shape.

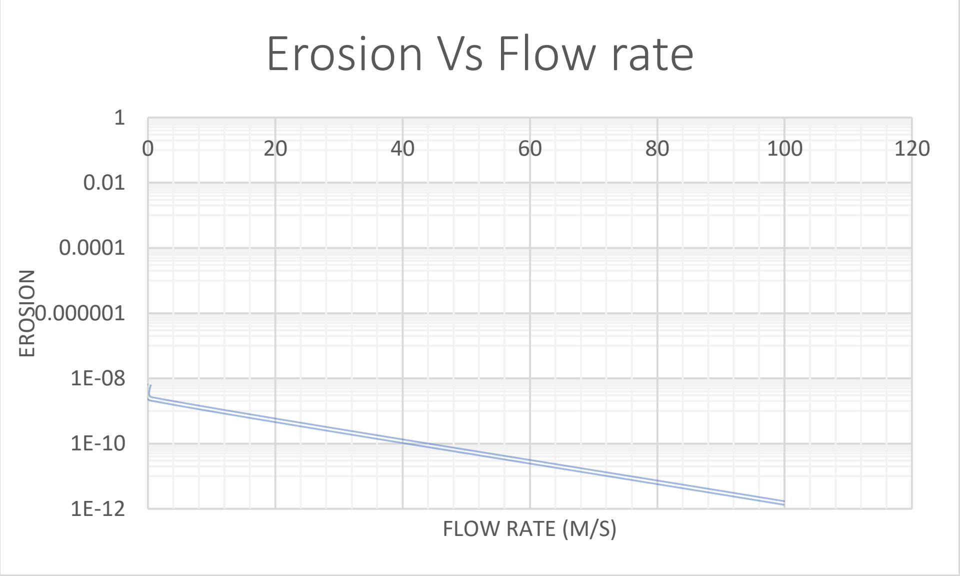


Fig.4. 18 Erosion Vs Flow rate

In addition, the cross-section meshing influenced the particle tracking, increasing in division leads to more particle tracking but the mesh size shouldn't be smaller than the particle size in order to allow the CFD model carryout proper tracking

CHAPTER 5: CONCLUSION AND RECOMMENDATION

5.1 CONCLUSION

After conduction the study and with reference to literature and pass experiment, it can be concluded that ANSYS Fluent can be used to model a multiphase flow with sand as the disperse phase. However, validating the CFD model will need further research because gas bubbles were neglected.

1. An appropriate CFD model for multiphase flow with sand particles as dispersed solid was developed using ANSYS Fluent CFD Software
2. The build model was valid for carrying-out studies in pipelines after proper validation with hand calculations and verifications with experimental data
3. While flow is laminal, transmission of sand particles by the continuous phase is organized and uniform but as the sand particle increases in size and downstream length increase, the particle tends to settle on the bottom of the pipe. Nonetheless, turbulent flow can be used to transport particles size transported and not transported by laminal flow and can cover a longer distance than laminal flow.
4. With reference to the CFD post-processing, while other conditions are kept constant, the sand particle size is the major factor that influences the critical velocity and they have a direct relationship. Another point to note, is, below the critical velocity erosion is expected to occur at the bottom of the pipe, at or slightly above critical velocity, erosion is found to be minimal and flowing at a velocity very high than the critical velocity, result to erosions on the pipe walls.

5. The major factor affecting sand transport in pipeline is the particle size, density and length of the transmission line.

5.2 RECOMMENDATION

For fluid flow through pipeline, laminal flow would be best because it gives the minimal value of erosion when it is successful. Nonetheless, when particle size is large, it might not be sufficient enough to transport the particle, flow should be flow at an optimum turbulent transmission rate that minimize the erosion and attain a long transmission distance.

After several validation of ANSYS, it can be used as a tool for determining this optimum rate.

REFERENCES

Al-lababidi, S., Yan, W., & Yeung, H. (2012). Sand Transportation and Deposition Characteristics in Multiphase Flows in Pipelines. Journal of Energy Resources Technology, 134. doi:10.1115/1/4006433

Anderson, T. Bo, and Roy Jackson. "Fluid mechanical description of fluidized beds. Equations of motion." Industrial & Engineering Chemistry Fundamentals 6, no. 4 (1967): 527-539.

ANSYS Fluent Lectures. (2019). Multiphase flow.

ANSYS Fluent Users' Guide. (2019). Release 19.1. ANSYS.

Azrie Bin Kinan (2017). Computational fluid dynamics (CFD) simulation of sand deposition in pipeline. Undergraduate Thesis, Univ. of Teknologi PETRONAS.

Bello, O.O., (2008). Modelling particle transport in gas-oil-sand multiphase flows and its applications to production operations. Ph.D. Thesis, Clausthal Univ. of Technology, Clausthal.

Bernard Massey (2005). Mechanics of fluids; revised by John Ward-Smith. –8th ed page 191

Bowen, R. M. "Theory of Mixtures in Continuum Physics, Vol. Ill, Eringen, A. C, ed." (1976).

Camçı, Gülden. "Application of isokinetic sampling technique for local solid densities in upward liquid-solid flows through an annulus." PhD diss., METU, 2003.

Choong, K.W., Wen, L.P., Tiong, L.L, Anosike, F., Shoushtari, M.A., & Saaid, I.M. (2013). A comparative study on Sand Transport Modelling for Horizontal Multiphase Pipeline. Research Journal of Applied Sciences, Engineering and Technology, 7(6): 1017-1024

Danielson, T. J. (2007). Sand Transport Modeling in Multiphase Pipelines. Offshore Technology Conference.

Franz Durst. Fluid Mechanics - An Introduction to the Theory of Fluid Flows 524, 525

Fukuda, T., & Shoji, Y. (1986). Pressure drop and heat transfer for tree phase flow: 1st report, flow in horizontal pipes. Bulletin of JSME, 29(256), 3421-3426.

Kago, Tokihiro, Tetsuya Saruwatari, Makoto Kashima, Shigeharu Morooka, and Yasuo Kato. "Heat transfer in horizontal plug and slug flow for gas-liquid and gas-slurry systems." Journal of chemical engineering of Japan 19, no. 2 (1986): 125-131.

Kaushal, D. R., A. Kumar, Yuji Tomita, Shigeru Kuchii, and Hiroshi Tsukamoto. "Flow of monodispersed particles through horizontal bend." International Journal of Multiphase Flow 52 (2013): 71-91.

Kaushal, D. R., Kimihiko Sato, Takeshi Toyota, Katsuya Funatsu, and Yuji Tomita. "Effect of particle size distribution on pressure drop and concentration profile in pipeline flow of highly concentrated slurry." International Journal of Multiphase Flow 31, no. 7 (2005): 809-823.

Kelessidis, V. C., G. E. Bandelis, and J. Li. "Flow of dilute solid-liquid mixtures in horizontal concentric and eccentric annuli." Journal of Canadian Petroleum Technology 46, no. 05 (2007).

Kelessidis, Vassilios C., Panagiotis Dalamarinis, and Roberto Maglione. "Experimental study and predictions of pressure losses of fluids modeled as Herschel–Bulkley in concentric and eccentric annuli in laminar, transitional and turbulent flows." Journal of Petroleum Science and Engineering 77, no. 3 (2011): 305-312.

Kocamustafaogullari, G., and Z. Wang. "An experimental study on local interfacial parameters in a horizontal bubbly two-phase flow." International journal of multiphase flow 17, no. 5 (1991): 553-572.

Krieger, Irvin M. "Rheology of monodisperse latices." Advances in Colloid and Interface Science 3, no. 2 (1972): 111-136.

Kumar Gopaliya, Manoj, and D. R. Kaushal. "Modeling of sand-water slurry flow through horizontal pipe using CFD." Journal of Hydrology and Hydromechanics 64, no. 3 (2016): 261-272

Liu_Henry_-_Pipeline_Eng O'Brien, Morrough P. "Review of the theory of turbulent flow and its relation to sediment-transportation." Eos, Transactions American Geophysical Union 14, no. 1 (1933): 487-491.

Mariella Leporini [a, *], Barbara Marchetti [b], Francesco Corvaro [a], Giuseppe di Giovine [a], Fabio Polonara [a, c],

Alessandro Terenzi; Sand transport in multiphase flow mixtures in a horizontal pipeline: An experimental investigation

Orell, Aluf. "The effect of gas injection on the hydraulic transport of slurries in horizontal pipes." Chemical Engineering Science 62, no. 23 (2007): 6659-6676.

Özbelge, T. A., and A. Beyaz. "Dilute solid–liquid upward flows through a vertical annulus in a closed loop system." International journal of multiphase flow 27, no. 4 (2001): 737-752.

Özbelge, Tülay A., and Ahmet N. Eraslan. "A computational hydrodynamic and heat transfer study in turbulent up-flows of dilute slurries through a concentric annulus." Turkish Journal of Engineering and Environmental Sciences 30, no. 1 (2006): 1-13.

Özbelge, Tülay A., and Sema H. Köker. "Heat transfer enhancement in water—feldspar upflows through vertical annuli." International journal of heat and mass transfer 39, no. 1 (1996):

135-147.ineering page 19

Oroskar, A.R. and R.M. Turian. (1980) "The Critical Velocity in Pipeline Flow of Slurries". AlChE J., 26(4): 550-558.

Oudeman, P. (1993). Sand transport and deposition in horizontal multiphase trunklines of subsea satellite developments. SPE Prod. Facil., 8(4): 237-241.

Reynolds number (2021 May 27). In *Wikipedia*. https://en.wikipedia.org/w/index.php?title=Reynolds_number&oldid=1025423200

Rasel A Sultan (2018). A comprehensive study on multiphase flow through pipeline and annuli using CFD Approach. MSC. Thesis. Univ. of Newfoundland

Salama, M. M. (2000). Sand Production Management. Energy Resour. Technol, 122, 29–33.

Sanni, S. E., et al. (2015). Modeling of Sand and Crude Oil Flow in Horizontal Pipes during Crude Oil Transportation. Journal of Engineering. Retrieved from https://www.hindawi.com/journals/je/2015/457860/

Turian, R.M., F.L. Hsu and T.W. Ma, (1987). Estimation of the critical velocity in pipeline flow of slurries. Powder Technol., 51(1): 35-47

Turbulence (2021 June 18). In *Wikipedia*.

https://en.wikipedia.org/w/index.php?title=Turbulence&oldid=1029129386