COMPUTATIONAL FLUID DYNAMICS (CFD) SIMULATION OF SAND DEPOSITION IN PIPELINE

AZRIE BIN KINAN

MECHANICAL ENGINEERING

UNIVERSITI TEKNOLOGI PETRONAS

JANUARY 2017

COMPUTATIONAL FLUID DYNAMICS (CFD) SIMULATION OF SAND DEPOSITION IN PIPELINE

by

AZRIE BIN KINAN

17966

Dissertation submitted in partial fulfilment of

the requirements for the

Bachelor of Engineering (Hons)

(Mechanical)

JANUARY 2017

Universiti Teknologi PETRONAS
Bandar Seri Iskandar,
32610 Seri Iskandar,
Perak Darul Ridzuan,
Malaysia

CERTIFICATION OF APPROVAL

Computational Fluid Dynamics (CFD) Simulation of Sand Deposition in Pipeline

by

Azrie Bin Kinan
17966

A project dissertation submitted to the

Mechanical Engineering Programme

Universiti Teknologi PETRONAS

in partial fulfilment of the requirement for the

BACHELOR OF ENGINEERING (Hons)

(MECHANICAL)

Approved by,

____

(Dr. Tuan Mohammad Yusoff Tuan Ya)

UNIVERSITI TEKNOLOGI PETRONAS

BANDAR SERI ISKANDAR, PERAK

January 2017

CERTIFICATION OF ORIGINALITY

This is to certify that I am responsible for the work submitted in this project, that the original work is my own except as specified in the references and acknowledgements, and that the original work contained herein have not been undertake or done by unspecified sources or persons.

____

AZRIE BIN KINAN

ABSTRACT

Past reviews have demonstrated that transportation of reservoir fluid through pipeline is one of the most cost effective options for delivering the feed to the processing facility. However, most of the time sand particles are co-produced with the fluids. This will lead to sand deposition on the bottom of the pipeline whenever the transporting fluid velocity is below the critical velocity required. To prevent this from happening and ensure flow assurance, it is crucial to measure and identify the critical velocity.

This study presents the results obtained from computational fluid dynamics (CFD) simulation for identifying critical velocity where the formation of static sand bed occurs. The critical velocity is found to be fairly influenced by the sand volume fraction. It was observed that formation of sand dunes occur at the bottom of the pipe at low fluid velocity. The result from the simulations is compared with other studies for validation and analytical comparison.

ACKNOWLEDGMENT

I would like to express my deep thanks to my supervisor Dr Tuan Mohammad Yusoff who supported me during my Final Year Project I and II. I learned a lot from him not only in academic aspect but also personal things. CFD was a new area for me and I am pleased that I had an opportunity to work with such a professional person in this topic. His ideas and willingness to help impress me all the time.

My deep appreciation goes to Dr Feroz and Mr Calvin from PETRONAS GTS for the useful CFD discussions during the project work. Short but precise advice significantly helped me to perform my simulations in the most efficient and practical way.

Last but not least, I would like to thank my family for their endless support all the way through my degree study. Without their help and love, I would never come to UTP and would not make one of the most important steps in my life and career.

TABLE OF CONTENT

LIST OF FIGURES

LIST OF TABLES

ABBREVIATIONS

2D – Two Dimensional

3D – Three Dimensional

CFD – Computational Fluid Dynamics

DDPM – Dense Discrete Phase Model

DEM – Discrete Element Method

DPM – Discrete Phase Model

KTGF – Kinetic Theory of Glanular Flow

MTV – Minimum Transport Velocity

UDF – User-Defined Function

VOF – Volume of Fluid

NOMENCLATURE

C = sand volume fraction

d = particle diameter, m

D = pipe diameter, m

g = gravity, m/s^2

K = constant

R_e = Reynold’s number

s = particle to fluid density ratio

$\Delta\rho$ = density difference between particles and liquid, kg/m^3

μ_k = kinematic viscosity, m^2/s

μ_d = dynamic viscosity, $N.s/m^2$

V_m = minimum mixture flow velocity to avoid sand settling, m/s

V_{sl} / V_m = velocity ratio of supercial and mixture (1 for single phase)

ρ_f = liquid density, kg/m^3

μ_k = kinematic viscosity, m^2/s

V_c = critical velocity, m/s

CHAPTER 1

INTRODUCTION

1.1 Background

Sand problem is one of the common problems in petroleum industry. However only few studies had been covered in this particular area. This is due to the complexity of the model used for modelling the problem which includes several variables such as flow pattern, phase velocity and fluid properties. Not to mention the geometry features of the pipe such as diameter, roughness and leaning angle.

When the sand enters the pipeline system, it is important for the system to prevent the sand to settle. An experimental is set to investigate the critical velocity for the movement of the fluid where no to minimal sand deposition occurs.

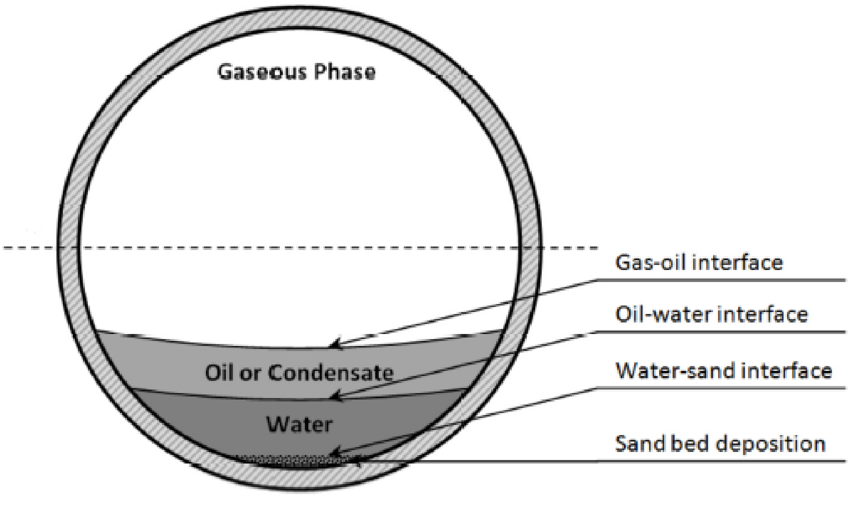


Figure 1.1: Deposition of sand in an oil pipeline

1.2 Problem Statement

"Flow Assurance" is the study of continuous fluid transportation between the reservoir to the processing facilities. The fluids from the reservoir such as black oil, dry gas, condensate gas and wet gas are mixed with water and sand during the transportation. The complexity of multiphase transport flow simulation is caused by the presence of the sand and it interacts with other transported fluids.

During the transportation of reservoir fluids to the processing plant, the rocks oil is often transported as a mixture with sand. The sand later may deposit on the walls due to pressure drop and causes other problems such as pipe blockage, corrosion, abrasion, reduction in flow area, pipe blockage and most importantly low output from the lines [1]. For that reason, it is crucial to predict the critical sand deposition velocity in order to maximize reservoir production.

1.3 Objective and Scope of Study

The objectives of the projects are to:

1. Develop a fluid simulation for the sand deposition in pipeline.
2. Find the critical velocity with respect to sand deposition.
3. Validate the result of the simulation with other published results from other studies.

The scope of study of this project will focus more on the deposition of sand particles in pipelines for oil and gas industries. This problem has costed millions of dollars in this particular industry due to the restricted rate of production.

CHAPTER 2

LITERATURE REVIEW

2.1 Computational Fluid Dynamics (CFD)

CFD is a study that involves numerical analysis, fluid mechanics and computer science. This technology has been developed since as early as 1955 but only limited to compressible flow and only accessible by large high-speed computer. As the computer hardware capabilities increase over time, CFD spreads to other industries such as aerospace, weapon simulation and many more. Nowadays, CFD is available to consumer level as a learning platform and engineering-related problem solver.

The 3 main steps for solving problem using CFD:

1. Data preparation (pre-processing)
    - Problem identification.
    - 3D modelling.
    - Identifying boundary conditions.
    - Mesh generation.
2. Problem solving
    - Solver such as ANSYS FLUENT® will do the calculation based on the conditions set earlier.
    - Time taken to solve the problem depends on the mesh size and model complexity.

3. Result gathering and analysis (post-processing)
    - The results can be obtained in graphical and numerical.
    - Data will be analysed and verified so that it will not contradict with engineering principles.

2.1.1 Multiphase Modelling

To solve a problem in CFD, a good understanding about the problem as well as the solver are needed since suitable approach is very important. Basically there are two types of multiphase models which can be found in ANSYS Fluent as shown in the figure below:

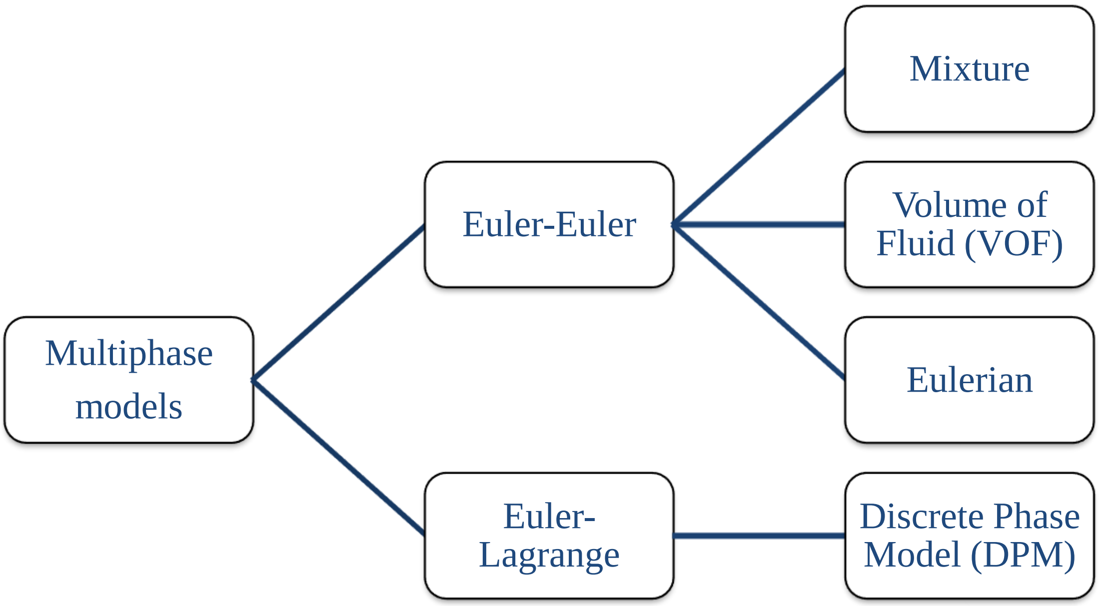


Figure 2.1: Multiphase modelling in ANSYS Fluent

2.1.2 Particulate Flows Modelling

To simulate a particulate system, ANSYS Fluent provides a wide range of configurations depending on the application.

Table 2.1: Particulate flow models available in ANSYS

Model	**Fluid**	**Particle**	**Interaction Between Particles**
DPM	Eulerian	Lagrangian	All particles are set as points
DDPM-KTGF	Eulerian	Lagrangian	Interactions of particles depend on the granular model
DDPM-DEM	Eulerian	Lagrangian	Interaction between particles are accurate
Euler-Granular	Eulerian	Lagrangian	Interaction between particles are modeled by the properties of the fluid

2.2 Boundary Layer and Fully Developed Flow

Boundary layer is the layers where shearing forces of a fluid acting on a wall and affect its velocity. A very simple example is a fluid flowing through a pipe as shown in the figure below:

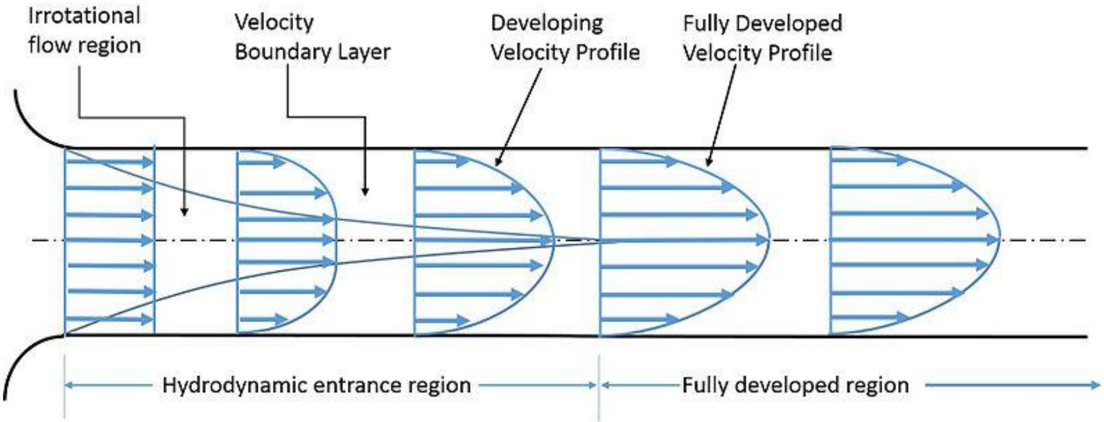


Figure 2.2: Transition of velocity profile

The length where the velocity starts to be fully developed is called entry length. The magnitude of entry length is influenced by the density and viscosity of the fluid, diameter of the pipe and the velocity when the fluid enters the pipe. The equations for finding entry length are given by:

$$L_{E,laminar} = 0.06R_eD \tag{1}$$

$$L_{E,turbulent} = 4.4{R_e}^{\frac{1}{6}}D \tag{2}$$

Where,

$R_e \ = \rho v D/\mu_d$, Reynold’s number

ρ = density of the fluid

v = velocity of the fluid at the entrance

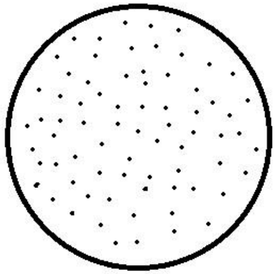
D = diameter of the pipe

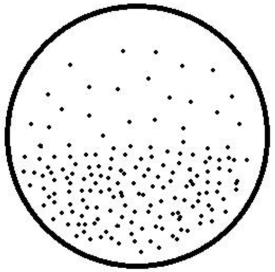
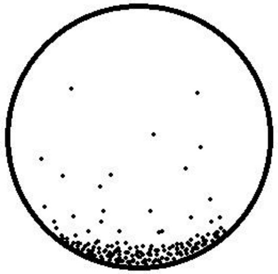
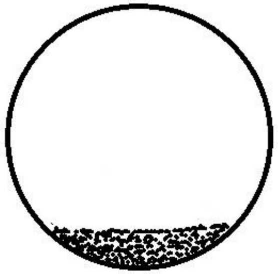
μ_d = dynamic viscosity of the fluid

2.3 Types of Flow Regimes in Slurry Transport

Turian and Yuan (1977) classified the flow regimes in slurry transport into four types. These four correlations were developed through extended pressure drop correlation scheme observed in slurry transport.

Table 2.2: Four main types of flow regimes in slurry transport

Types of Flow Regimes	Equation	Figure
1) Homogeneous Flow Regime Particles are transported together with the fluid and the distribution of the sand particles are equal at all sides.	$f - f_w = 0.8444\, C^{0.5024}\, {f_w}^{1.428}\, {C_D}^{0.1516} \left[\frac{v^2}{Dg(s-1)}\right]^{-0.3531}$	

2) Heterogenous Flow Regime Sand particles are still transported in suspension but densely populated near the low-side of the wall.	$f - f_w = 0.5513\, C^{0.8687}\, {f_w}^{1.200}\, {C_D}^{-0.1677} \left[\frac{v^2}{Dg(s-1)}\right]^{-0.6938}$	
3) Saltation Flow Regime A thin layer of sand bed is formed continually with the sand particle at the bottom side of the wall rolling/sliding slower compared to top.	$f - f_w = 0.9857\, C^{1.018}\, {f_w}^{1.046}\, {C_D}^{-0.4213} \left[\frac{v^2}{Dg(s-1)}\right]^{-1.354}$	
4) Stationary Bed Continuous sand bed formation at the low side of the pipeline wall while only the sand at the surface is rolling or sliding.	$f - f_w = 0.4036\, C^{0.7389}\, {f_w}^{0.7717}\, {C_D}^{-0.4054} \left[\frac{v^2}{Dg(s-1)}\right]^{-1.096}$	

2.4 Critical Velocity

The critical velocity v_c can be defined as the minimum velocity where the formation of solid particles bed occurs at the bottom of the pipe. K. Bello et al. used the term Minimum Transport Velocity (MTV) for their model and it was determined by measuring the flow rate at which the solid particles begin to drop out when the particles were initially in suspension.

2.4.1 Oroskar & Turian

Oroskar & Turian (1980) used various correlation to develop a new equation in finding critical velocity in his study. From these 7 correlations, Oroskar & Turian (1980) had developed an equation after various reasonable assumptions and conditions were made:

$$V_c = \sqrt{gd(s-1)} \left\{ \frac{5C(1-C)^{2n-1} \left(\frac{D}{d}\right) \left(\frac{D\rho_l \sqrt{gd(s-l)}}{\mu}\right)^{1/8}}{x} \right\}^{8/15} \qquad (3)$$

2.4.2 Salama Model

Salama then proposed an equation for predicting the critical velocity of solid particles bed formation in a horizontal pipe from other coorelation and relate it with the equation 3 which is presented below:

$$V_m = \left(\frac{V_{sl}}{V_m}\right)^{0.53} d^{0.17} \mu_k{}^{0.09} \left(\frac{\Delta\rho}{\rho_f}\right)^{0.55} D^{0.47} \tag{4}$$

where,

V_m = minimum mixture flow velocity to avoid sand settling, m/s

V_{sl} / V_m = velocity ratio of supercial and mixture (1 for single phase)

d = particle diameter, m

D = pipe diameter, m

$\Delta\rho$ = density difference between particles and liquid, kg/m^3

ρ_f = liquid density, kg/m^3

μ_k = kinematic viscosity, m^2/s

2.4.3 Danielson Model

Based on the sand transportation theory, critical velocity can be defined as the liquid velocity that is required to prevent stationary bed from forming. Danielson developed a liquid-sand modelling based on the analysis done by Wicks which is a single-phase flow but without considering the particle size. Danielson also refined this analysis because it was done with the coorelations of high sand-water ratio.

Danielson used the theory of turbulence and its eddies strength for the particles to go against the gravity. The equation can be written as the following expression:

$$V_c = K(\mu_k)^{-1/9} d^{-1/9} \big(gD(s-1)\big)^{5/9} \tag{5}$$

where,

V_c = critical velocity, m/s
K = constant
d = particle diameter, m
D = pipe diameter, m
μ_k = kinematic viscosity, m^2/s
g = gravity, m/s^2
s = particle to fluid density ratio

2.4.4 Oudeman Model

A horizontal pipeline study was conducted by Oudeman in 1993 stated that transition of the sand particles from static bed to moving or from moving bed to suspension is largely influenced by the superficial velocity of the liquid rather than gas. This is in my opinion true since water has higher density than air which of course carry more force to suspend the sand particle.

Even though gas is used in this experiment, but due to its weak effect to the sand particles flow transition, this equation will be used to be compared with the simulation result. The equation of Oudeman study is written as the following expression:

$$V_c = \left(\sqrt{0.25gd(s-1)} \Big/ 0.15\left(\frac{\mu_d}{\rho_l D}\right)^{1/8}\right)^{8/7} \tag{6}$$

where,

V_c = critical velocity, m/s
ρ_l = density of the liquid, kg/m^3
d = particle diameter, m
D = pipe diameter, m
μ_d = liquid dynamic viscosity, N.s/m^2
g = gravity, m/s^2
s = particle to fluid density ratio

CHAPTER 3

METHODOLOGY

3.1 Research Methodology

Various articles and studies are taken into account in doing this project. Most of the publications used as reference are from the studies done by doing expereimantal setup. The coorelations included in each of the papers need to be identified in developing a reliable CFD model.

3.2 Mathematical Modelling

2 models are selected in comparing the result from the simulation which are Salama and Danielson. 2 of the equations which are equation (4) and equation (5) are transferred into Microsoft Excel software. All variables that are needed in each equations are identified and will be used as the input data.

From all 4 equations, only Turian model include the variable of sand volume fraction. This will give constant critical velocity for all 3 volume fraction in other 3 equations when the result is tabulated in a graph. Other calculations that is done in Microsoft Excel are the sand volume fraction and the turbulence intensity of the pipe.

3.3 DPM Simulation

DPM is choosen because it is suitable for the problem with particle volume fraction that is less than 10%. All the parameters included in this simulation are carefully selected in order to give reliable results. Below are the assumptions made for this simulation:

- particles are injected in normal direction to the inlet surface which means all particles are initially suspended;
- initial velocity of the particles are zero so that they will settle faster and shorter pipe length can be used;
- water flow is steady;
- all particles have the same diameter and sphere in shape.

The result of this simulation is based on the visual observation only with the aid of the CFD post processing tool to filter them.

3.3.1 Modelling the Pipe

The diameter of the pipe is selected by referring the dimension used by past studies as well as considering the computational cost needed. The bigger and more complicated dimension of course will increase the simulation time and in return will slow down the progress of this project. After a few discussion with some of the experienced people in sand management for pipeline, the diameter of 0.07 m is selected for this project. The length of the pipe however is selected by considering the entry length of the liquid where the point of fully developed flow is achieved. This will be discuss further in the section **3.3.3** by relating equation (1) and (2).

The pipe is modelled by using the built-in modelling software in ANSYS Workbench which is DesignModeller. It is a good practice to use the built-in software since any alteration of the dimension can be done directly without needing to open other software which sometime can create problem due to the compatibility.

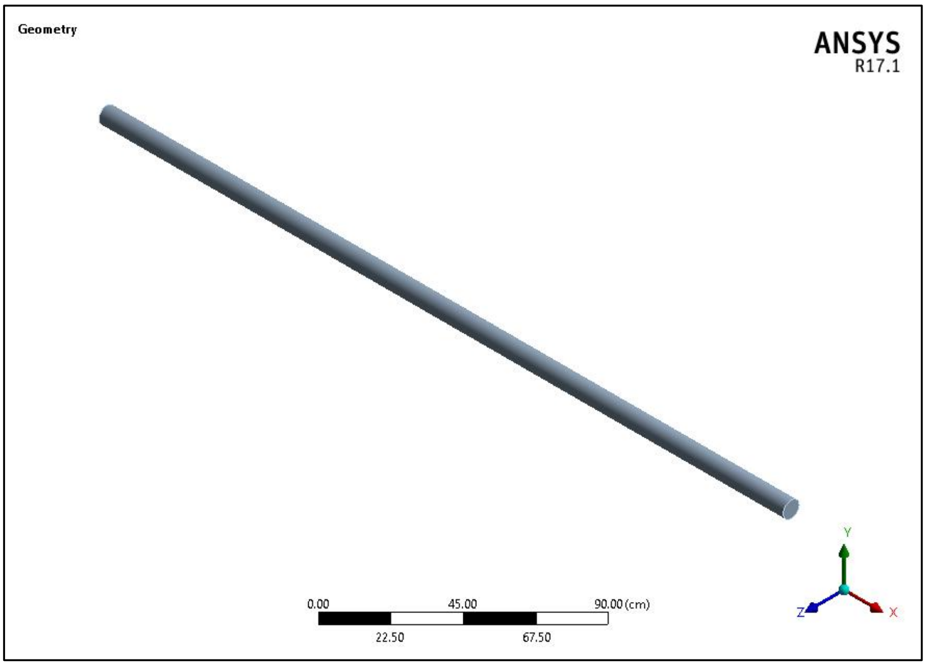


Figure 3.1: Modelling the pipe in ANSYS DesignModeller

3.3.2 Mesh

For generating the mesh, inflation method is used with multizone. With this approach, the thickness of the first layer can be controlled and at the same time avoiding poor mesh quality due to low orthogonal and high skewness.

It is observed that mesh cell size needs to be bigger than the particle size for obtaining a realistic result. Not only that, poor mesh quality will cause convergence problem in the simulation iteration later. Both ending of the pipe are set with inlet at the beginning of the x-axis and outlet at the other end.

At the axial direction of the pipe, the node is set to be 15 cm apart from each other using the multizone method as shown in the figure below.

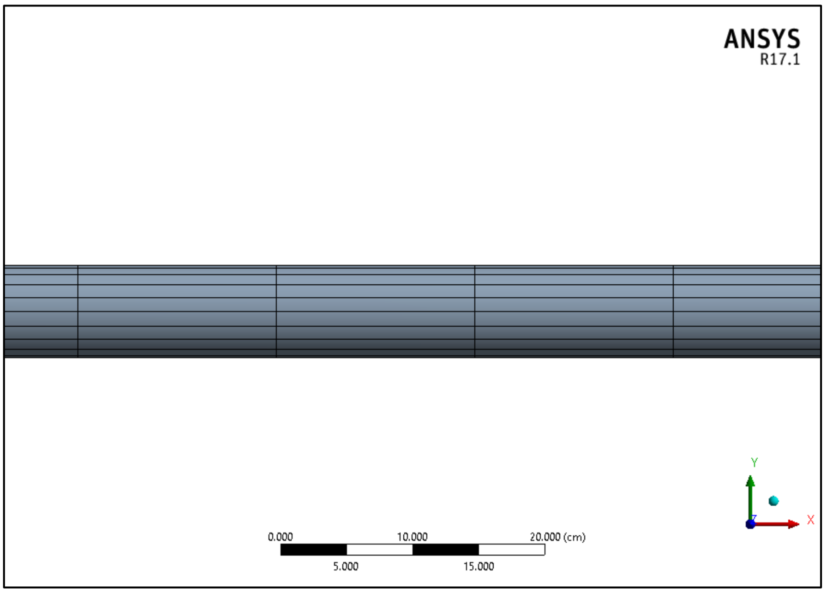


Figure 3.2: Node size of the mesh at the axial direction

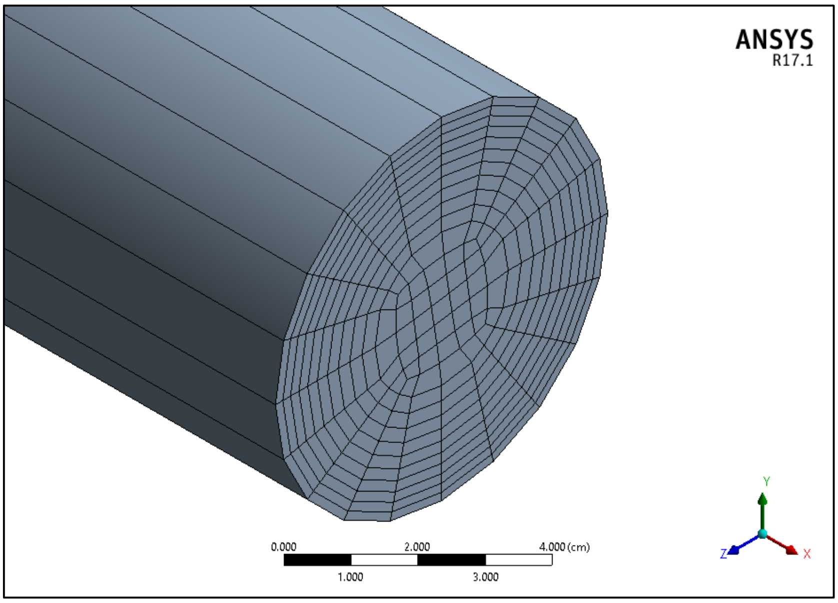


Figure 3.3: Mesh pattern at the cross sectional area of the pipe

3.3.3 Entry length

The entry length of the pipeline model needs to be calculated to ensure the velocity will be fully developed before it reaches the end. In other word, the entry length needs to be less than 3 m.

The parameters for the simulations are:

ρ = 998 kg/m^3
μ_d = 1.0002 x 10^{-3} N.s/m^2
D = 0.07 m
v = 0.1 m/s

Since the velocity of the fluid is inversely proportional to the entry length, the minimum velocity of 0.1 m/s is used for the calculation of entry length as it will give the longest entry length.

Finding the Reynold's Number,

$$Re = \rho vD/\mu_d = (998 \text{ kg/m}^3)(0.1 \text{ m/s})(0.07 \text{ m}) / 1.0002 \times 10^{-3} \text{ N.s/m}^2$$

$$= 6984.60 \text{ (> 4000, turbulent flow)}$$

Finding the entry length using equation (2),

$$Le = 4.4R_e^{1/6}D = 4.4(6984.60)^{1/6}(0.07\text{m})$$

$$= 1.347 \text{ m (< 3.00 m)}$$

This means with the length of pipe of 3 m, the flow will be developed and can be observed at about 1.3 m from the inlet. The calculation is then verified by using a simple CFD simulation.

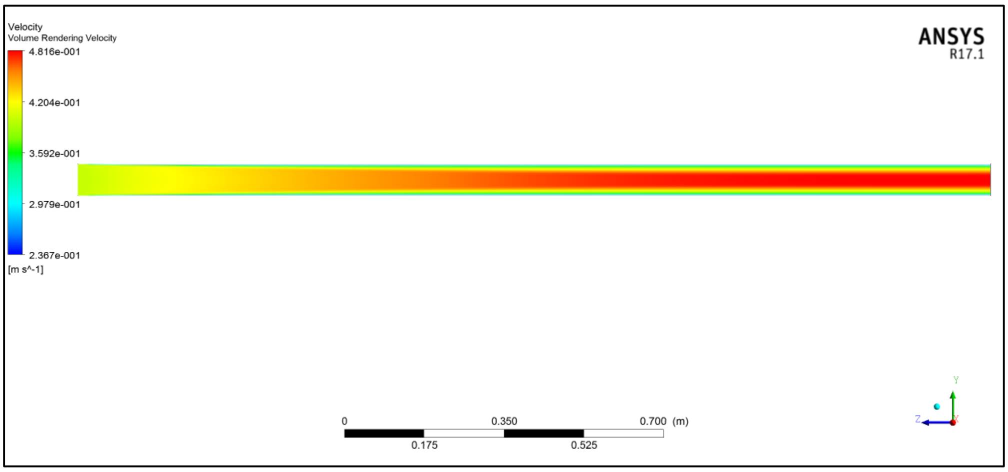


Figure 3.4: Cross-section of the pipe showing velocity contour

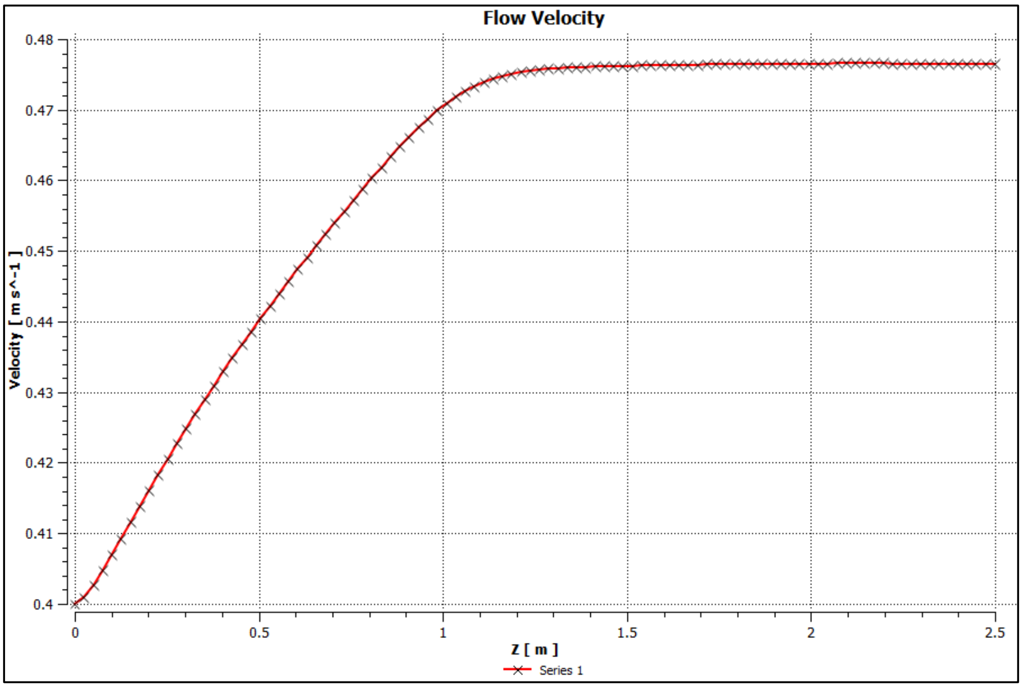


Figure 3.5: Length of pipe in x-axis direction versus the velocity magnitude

From the figure and chart above, it can be concluded that with the length of 3 m, a fully developed flow can occur with the velocity of 0.1 m/s.

3.3.4 DPM setting

Different setting needs to be put for the discrete and the continuous phases of the DPM model. Some parameters are quite straight forward while for more complicated parameters, some calculations are needed. For complicated parameters, it will be discussed in details while the rest will only be explained briefly.

Water is selected as the fluid medium of this simulation and its properties are taken directly from the ANSYS default material library. Steady flow is selected as the experiments conducted for comparing the simulations are in steady condition as well.

For the discrete phase, the density of the sand is set to be 2650 kg/m^3 with the constant diameter of 200 µm. The interaction of the discrete phase and the continuous phase is also enabled in order to observe its effect to the suspension and the deposition of the sand particles. Continuous phase iteration is set to 20 for each 1 discrete phase iteration after considering the convergence and accuracy since there is no specific number required for this parameter.

Table 3.1: Phase properties for DPM simulation

Phase properties	
Discrete phase (sand)	
Density, kg/m^3	2650
Diameter, µm	200
Continous phase (water)	
Density, kg/m^3	998
Viscosity, kg/m.s	1.003×10^{-3}

The particles flow propargation is tracked by using steady tracking. As mentioned earlier, this simulation is a steady state simulation and there is no need to use unsteady particle tracking function as it is not the point of interent of this study. Steady tracking function will track the particles until it reaches the outlet.

For turbulent dispersion, stochastic model is selected as it will contribute to the effect of particles lifting, significantly at the boundary layer. Virtual mass force is enabled as it is possible for the particles to move faster than the water flow especially when the particles are suspended. To make the effect of force more realistic, Shaffman lift force function is also enabled because the lifting effect also can be caused due to shear. These parameters are very crucial in determining the critical velocity of this simulation. The summary of the parameters can be observed in the table below.

Table 3.2: DPM simulation setting

Parameters	**Remark**
Time	Steady
Viscous model	Realizable k-ε with Enhanced Wall Treatment
Gravity	9.81 m/s^2
Continous phase interaction and iteration per DPM iteration	On, 20
Particle tracking mode	Steady
Stochastic model	On
Virtual mass factor	On
Shaffman lift force	On
Virtual mass force	On

The volume fraction of the sand needs to be specified at the inlet as it is one of the parameters needed for the discrete phase. The mass flowrate needs to be calculated separately for each of the simulation since different velocity will give different sand mass flowrate when the sand volume fraction is different. The sand mass flowrate can be calculated by using the following equation:

$$\rho_s \times C \times V_m \times A = G_s \tag{7}$$

where,

ρ_s = sand density, kg/m^3
C = sand volume fraction
V_m = mixture velocity, m/s
A = pipe cross sectional area, m^2
G_s = Sand mass flow rate, kg/s

Table 3.3: Sand mass flowrates based on the given volume fraction

Sand mass flowrate, kg/s			
Water velocity, m/s	Sand volume fraction		
	1.61×10^{-5}	1.08×10^{-4}	5.38×10^{-4}
0.1	1.64×10^{-5}	1.10×10^{-4}	5.49×10^{-4}
0.2	3.28×10^{-5}	2.20×10^{-4}	1.10×10^{-3}
0.3	4.93×10^{-5}	3.30×10^{-4}	1.65×10^{-3}
0.4	6.57×10^{-5}	4.41×10^{-4}	2.19×10^{-3}
0.5	8.21×10^{-5}	5.51×10^{-4}	2.74×10^{-3}
0.6	9.85×10^{-5}	6.61×10^{-4}	3.29×10^{-3}
0.7	1.15×10^{-4}	7.71×10^{-4}	3.84×10^{-3}
0.8	1.31×10^{-4}	8.81×10^{-4}	4.39×10^{-3}
0.9	1.48×10^{-4}	9.91×10^{-4}	4.94×10^{-3}
1.0	1.64×10^{-4}	1.10×10^{-3}	5.49×10^{-3}

Table 3.4: Boundary condition for the DPM simulation

Boundary conditions	
Inlet	
Water velocity, m/s	0.1 – 1.0
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

3.4 Project Flowchart

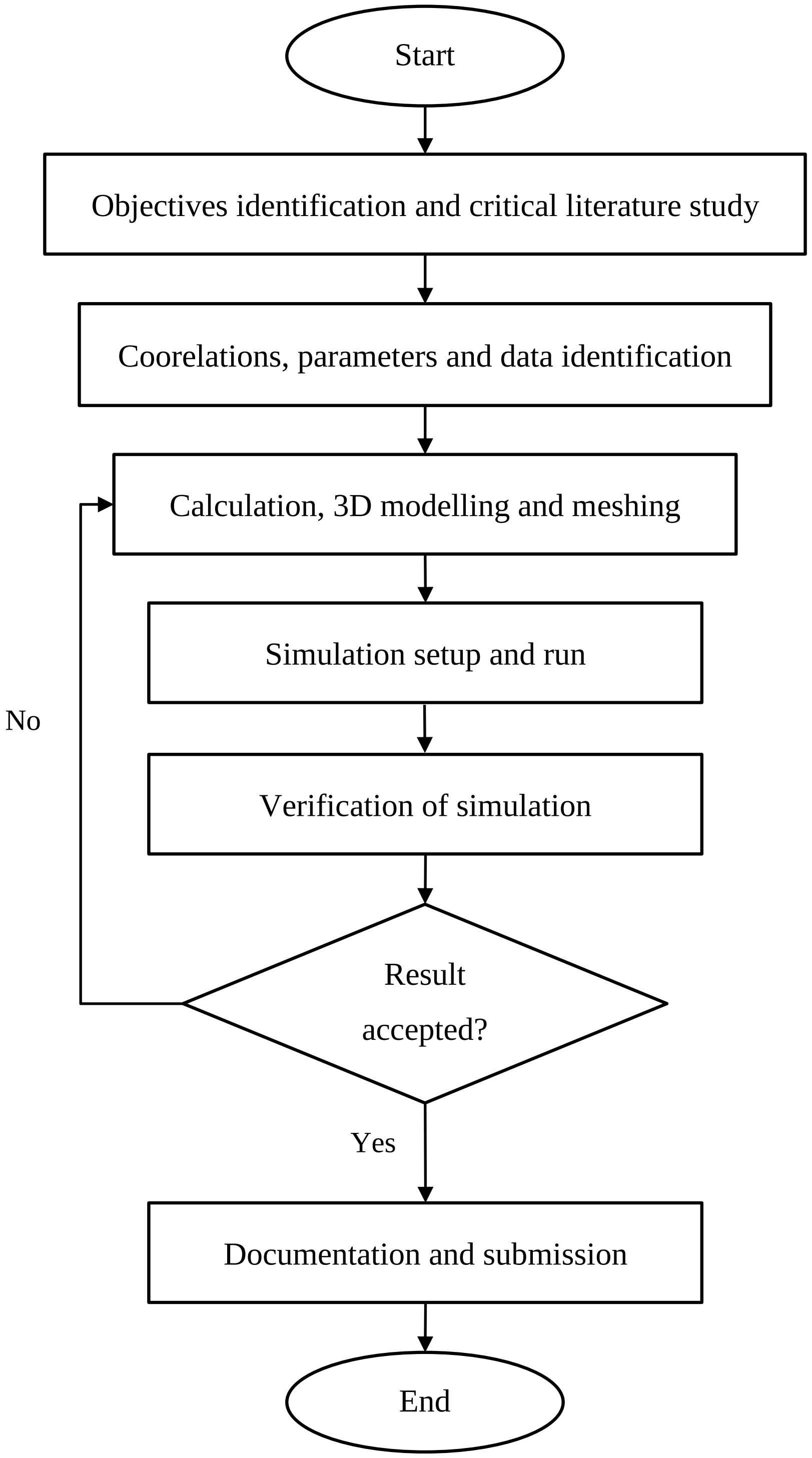


Figure 3.6: Research methodology of this project

3.5 Project gantt Chart and Key Milstone

Table 3.5: Project Gantt Chart

KEY MILESTONES AND PROJECT ACTIVITIES	**DURATION**																											
	SEPTEMBER 2016 – JANUARY 2017														**JANUARY 2017 – MAY 2017**													
	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			■	■	■	■	■	■	■	■	■	■	■	■														
Data gathering from outsource			■	■	■	■	■	■	■	■	■	■	■	■	■	■	■											
Boundary conditions and parameters identification based on the data gathered																		■	■									
Modelling and Simulation																			■	■	■	■						
End of parameter study																				■	■	■	■					
Data analysis																							■	■	■	■		

CHAPTER 4

RESULT AND DISCUSSION

This section will discuss about the results of simulations for all 3 volume fraction with 10 different liquid velocities. The simulations are done with the conditions and parameters mentioned in the previous chapter. The data in Table 4.1 will summarize the observation obtained from the simulation.

The pictures of particle distribution for 1.61×10^{-5} sand volume fraction will be attached together in the Appendix **A**. The result from this simulation is also will be compared with other publication and research for the comparison.

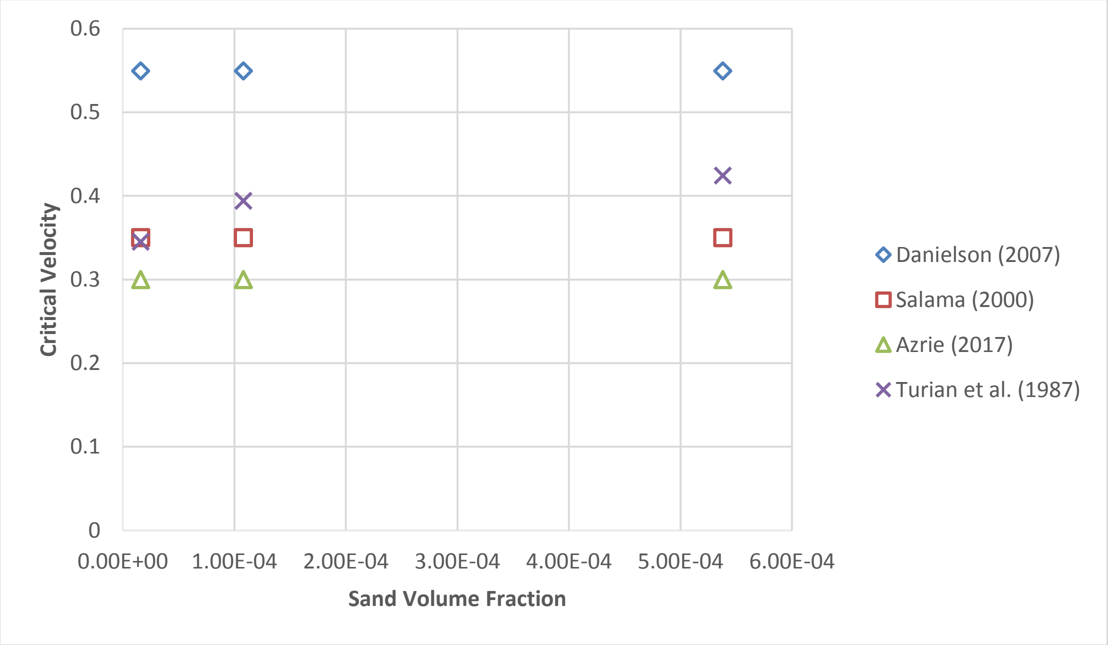


Figure 4.1: Comparison of the results

Table 4.1: Results from the DPM simulations

Water velocity, m/s	Sand volume fraction		
	1.61×10^{-5}	1.08×10^{-4}	5.38×10^{-4}
0.1	Slowly moving dunes developed	Slowly moving dunes developed	Slowly moving dunes developed
0.2	Sand dunes formation	Sand dunes formation	Sand dunes formation
0.3	Highest streaks concentration	Highest streaks concentration	Highest streaks concentration
0.4	Highest streaks concentration	Highest streaks concentration	Highest streaks concentration
0.5	Highest streaks concentration	Highest streaks concentration	Highest streaks concentration
0.6	Sand streaks mostly at the bottom	Sand streaks mostly at the bottom	Sand streaks mostly at the bottom
0.7	Sand streaks mostly at the bottom	Sand streaks mostly at the bottom	Sand streaks mostly at the bottom
0.8	Sand streaks mostly at the bottom	Sand streaks mostly at the bottom	Sand streaks mostly at the bottom
0.9	Few sand streaks at the bottom	Few sand streaks at the bottom	Few sand streaks at the bottom
1.0	Few sand streaks at the bottom	Few sand streaks at the bottom	Few sand streaks at the bottom

From the observation in DPM simulation, it is found that the critical velocity sits between 0.2 m/s to 0.4 m/s where the transition of the sand flow occurred. Visual comparison has been done between DPM simulation and the result obtained from study done by Al-lababidi (2012) and it can be found that formation of sand dunes started at the velocity of 0.2 m/s and 0.3 m/s respectively (refer Appendix B and Appendix C). Besides that, it can be proved that the critical velocity is influenced by the sand volume fraction.

The critical velocity value obtained from CFD simulation is below from the published results. The reason behind this is due to the mesh dependent simulation as well as other models that are neglected such as particle-particle interaction and the diameter of the particle which can be found in DEM model. Since, the length of this study is only limited to 8 months and only a few source materials available, it is very difficult to use that approach.

Another reason is the models used for this comparison are mostly involving more than one phase which include gas and oil while the simulation only used water as the transporting fluid. This is true because since oil is more viscous than water, the boundary layer of oil is thicker and requires higher velocity for transporting the sand particles.

However, the result from Salama (2000) shows small difference. From the equation, Salama (2000) predicted that oil-wetted sand will require lower velocity than water-wetted sand. This required further study to explore the physics behind it and the model can be included in simulation for more accurate result.

CHAPTER 5

CONCLUSION

After conducting the study in this topic, it can be concluded that ANSYS DPM model is able to simulate slurry flow regimes where the formation of sand bed as well as sand suspension can be successfully predicted. However, the results from this study needs more in-depth research since particle-particle interaction is neglected and the particle diameter does not give a significant impact on the continuous flow.

Another thing to point out is, this simulation is mesh dependent where coarser mesh gives more logical result than fine mesh. In depth mesh study need to be done to give more understanding on how the mesh can affect the result, especially for further study related to this topic. Besides that, the result from this experiment shows that the critical velocity does depend on the sand volume fraction. However, most of the experiment and research discussed in the literature review chapter did not include sand volume fraction except for the study conducted by Oroskar and Turian.

In general, DPM model in ANSYS Fluent really shows reasonable potential in predicting sand behavior in pipeline. However, if more models are included in the simulation where missing physics were not discussed such as DDPM, DEM and KTGF, it might give more reliable result despite its computational cost is high. Another function that was not utilized is User-Defined Function (UDF) where users can manually override the physics in the model for more specific set of simulation environment.

Overall, with the duration of 2 semesters for Final Year Project which is 8 months in total, only the surface of this study could be covered due to time constrain. However, it really gave a good insight on how research is done regarding sand production management especially in oil and gas industry.

REFERENCES

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

ANSYS Fluent Lectures. (2016). *Multiphase flow.*

ANSYS Fluent Users's Guide. (2016). *Release 17.1.* ANSYS.

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.

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.*

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.

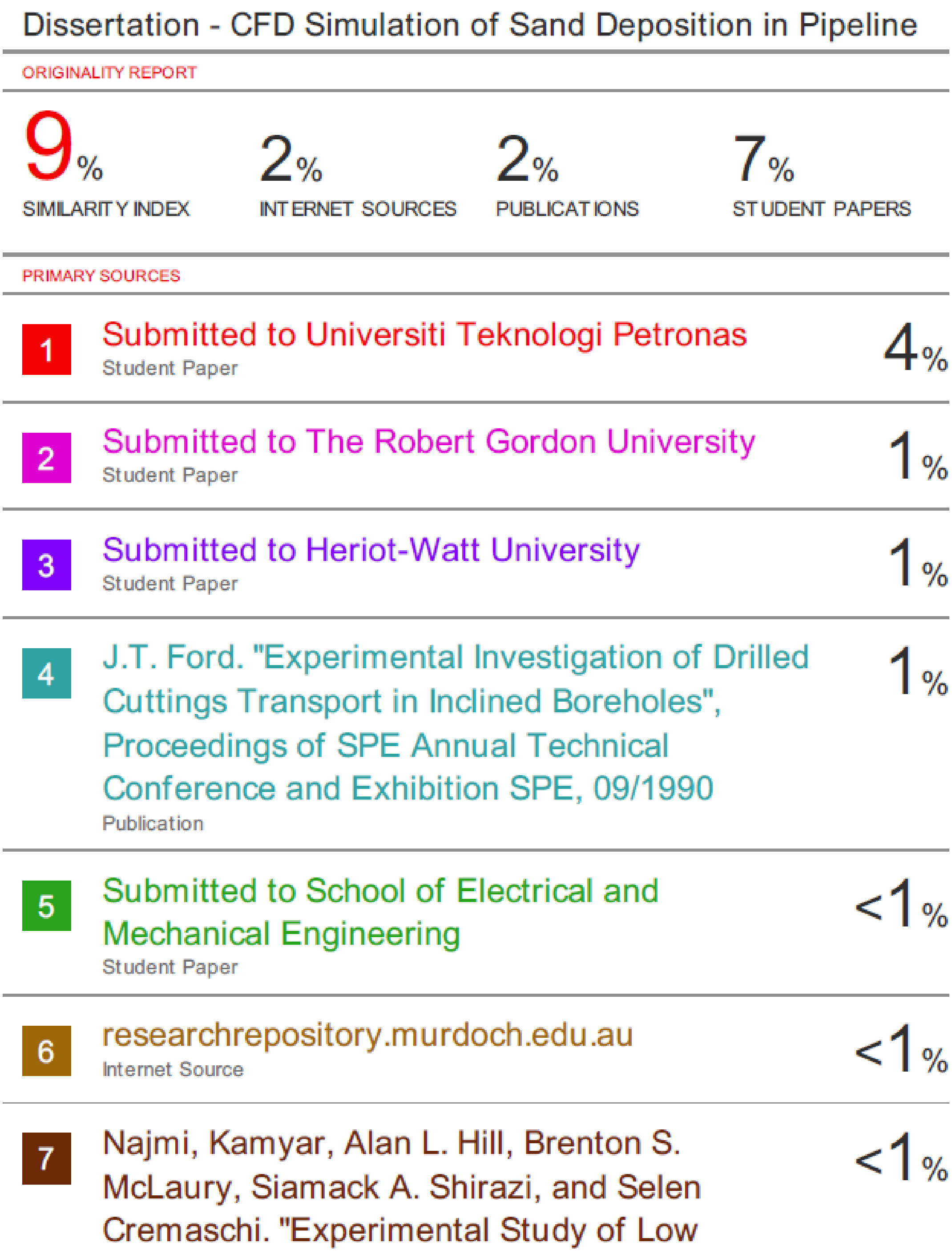
Salama, M. M. (2000). Sand Production Management. Energy Resour. Technol, 122, 29–33.

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.

APPENDIX A

TURNITIN SIMILARITY



Dissertation - CFD Simulation of Sand Deposition in Pipeline

ORIGINALITY REPORT

9%	2%	2%	7%
SIMILARITY INDEX	INTERNET SOURCES	PUBLICATIONS	STUDENT PAPERS

PRIMARY SOURCES

1	Submitted to Universiti Teknologi Petronas Student Paper	4%
2	Submitted to The Robert Gordon University Student Paper	1%
3	Submitted to Heriot-Watt University Student Paper	1%
4	J.T. Ford. "Experimental Investigation of Drilled Cuttings Transport in Inclined Boreholes", Proceedings of SPE Annual Technical Conference and Exhibition SPE, 09/1990 Publication	1%
5	Submitted to School of Electrical and Mechanical Engineering Student Paper	<1%
6	researchrepository.murdoch.edu.au Internet Source	<1%
7	Najmi, Kamyar, Alan L. Hill, Brenton S. McLaury, Siamack A. Shirazi, and Selen Cremaschi. "Experimental Study of Low	<1%

	Concentration Sand Transport in Multiphase Air–Water Horizontal Pipelines", Journal of Energy Resources Technology, 2015. Publication	
8	research.wsulibs.wsu.edu:8080 Internet Source	<1%
9	Submitted to Universiti Teknikal Malaysia Melaka Student Paper	<1%
10	www.scribd.com Internet Source	<1%
11	www.jbstockimages.com Internet Source	<1%

EXCLUDE QUOTES	ON	EXCLUDE MATCHES	< 3 WORDS
EXCLUDE BIBLIOGRAPHY	ON		

APPENDIX B

DPM SIMULATION

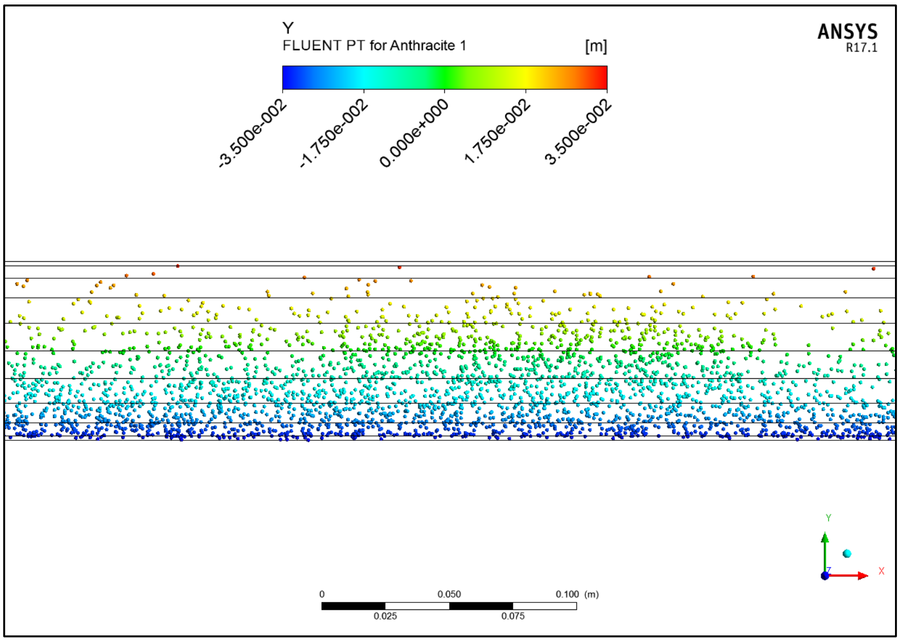


Figure B-1: 1.0 m/s – 1.61e-5 – side view*

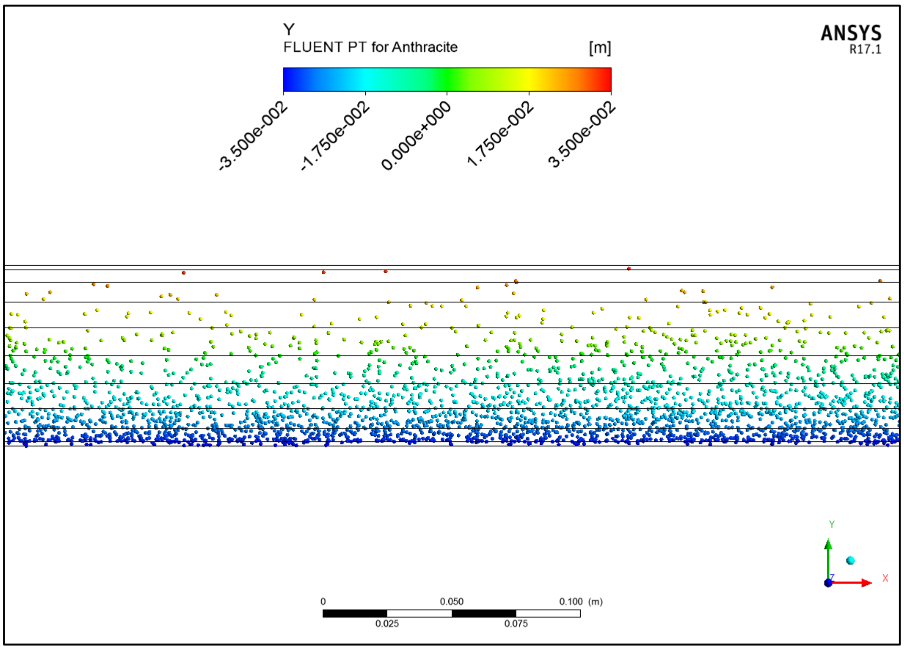


Figure B-2: 0.9 m/s – 1.61e-5 – side view

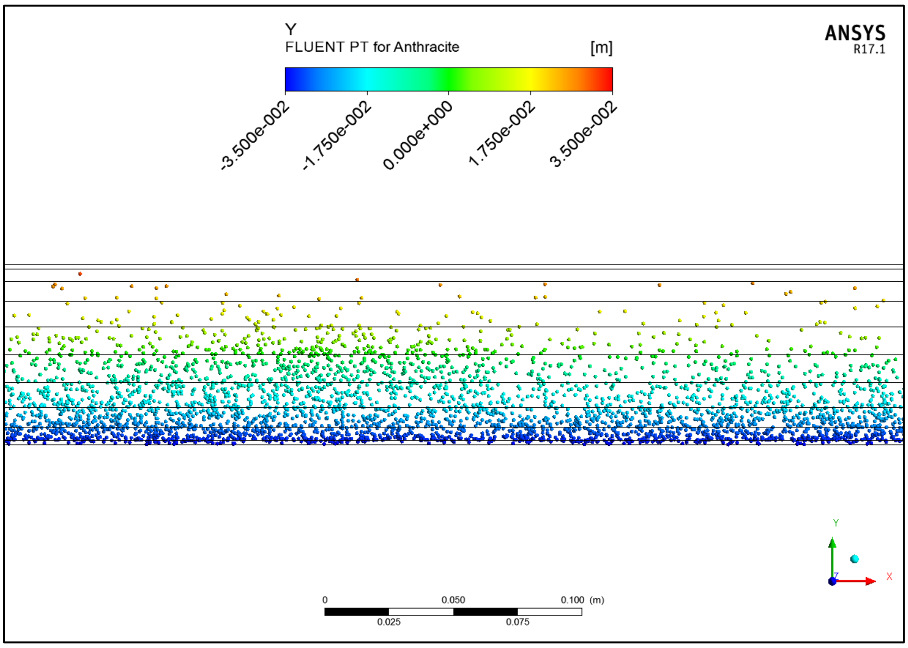


Figure B-3: 0.8 m/s – 1.61e-5 – side view

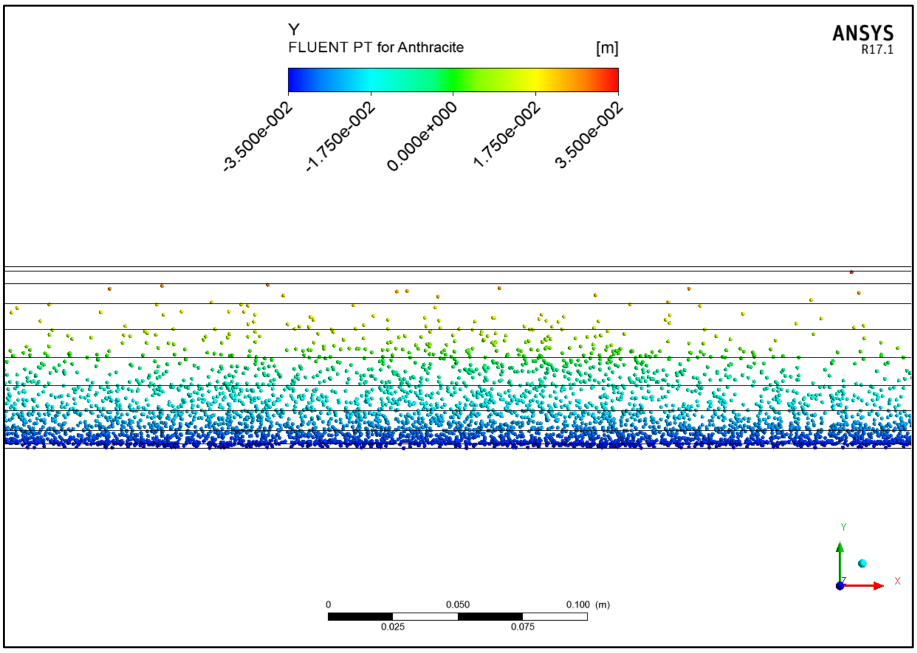


Figure B-4: 0.7 m/s – 1.61e-5 – side view

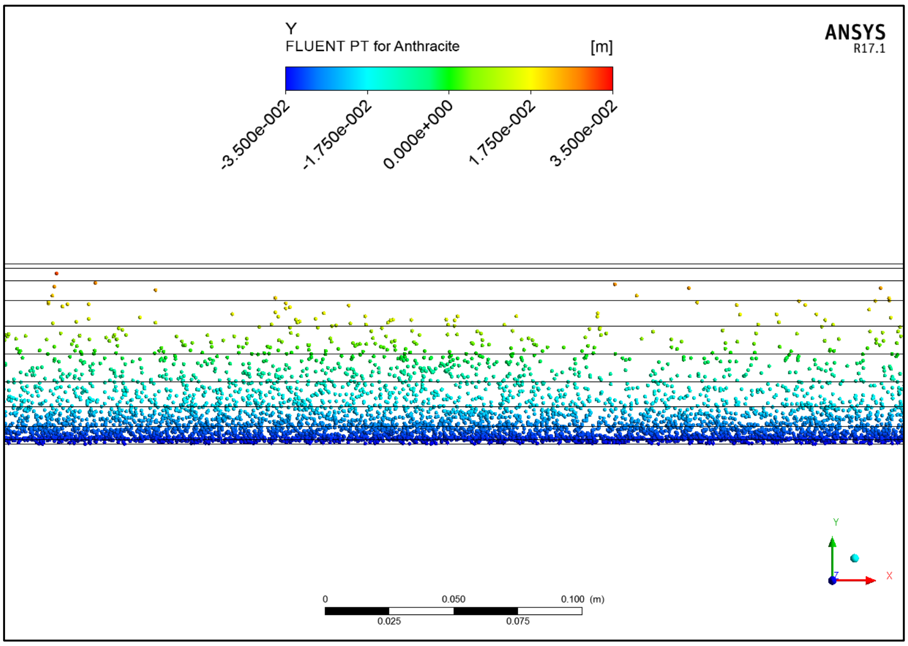


Figure B-5: 0.6 m/s – 1.61e-5 – side view

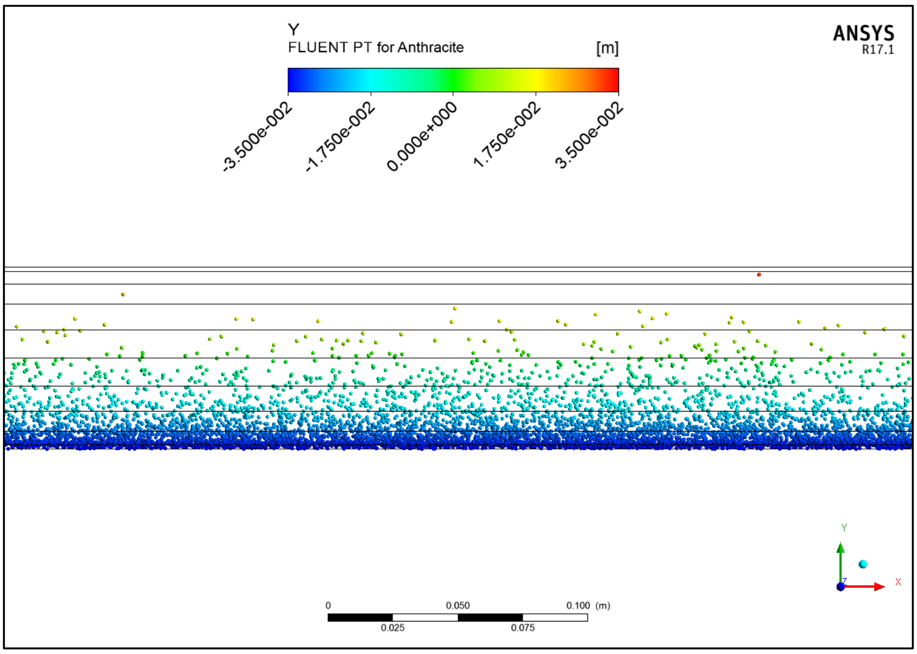


Figure B-6: 0.5 m/s – 1.61e-5 – side view

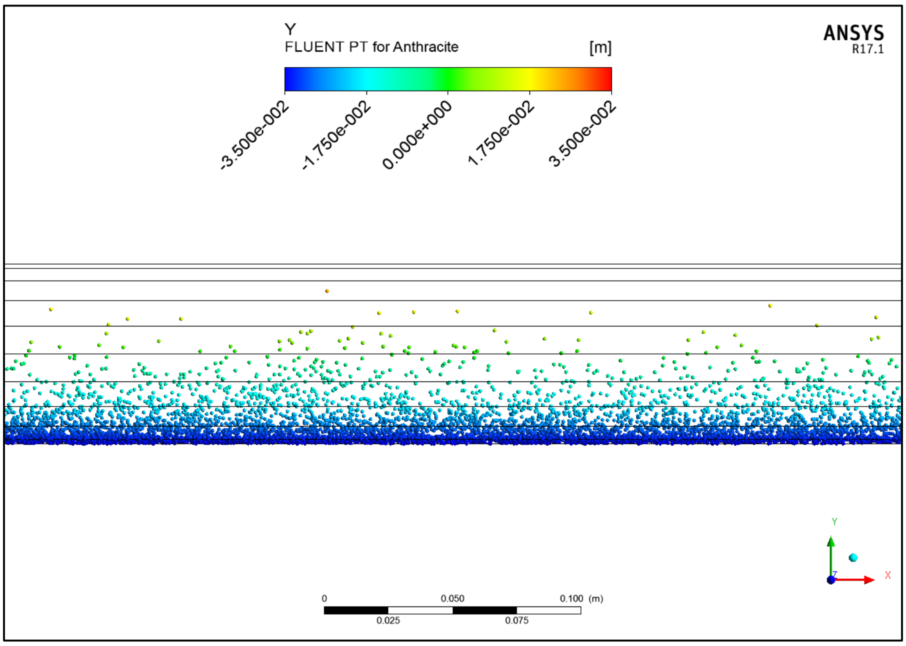


Figure B-7: 0.4 m/s – 1.61e-5 – side view

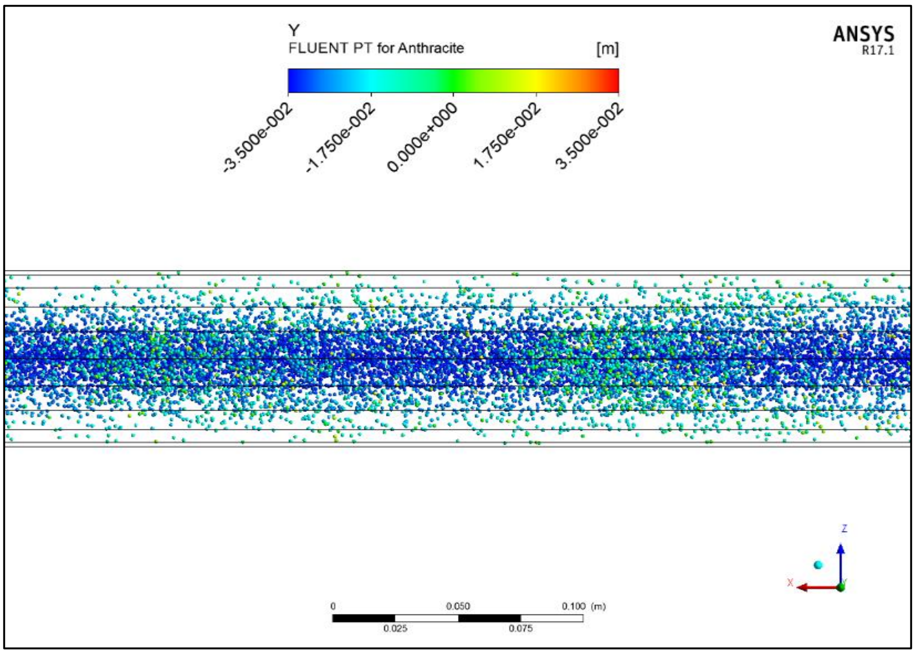


Figure B-8: 0.3 m/s – 1.61e-5 – top view

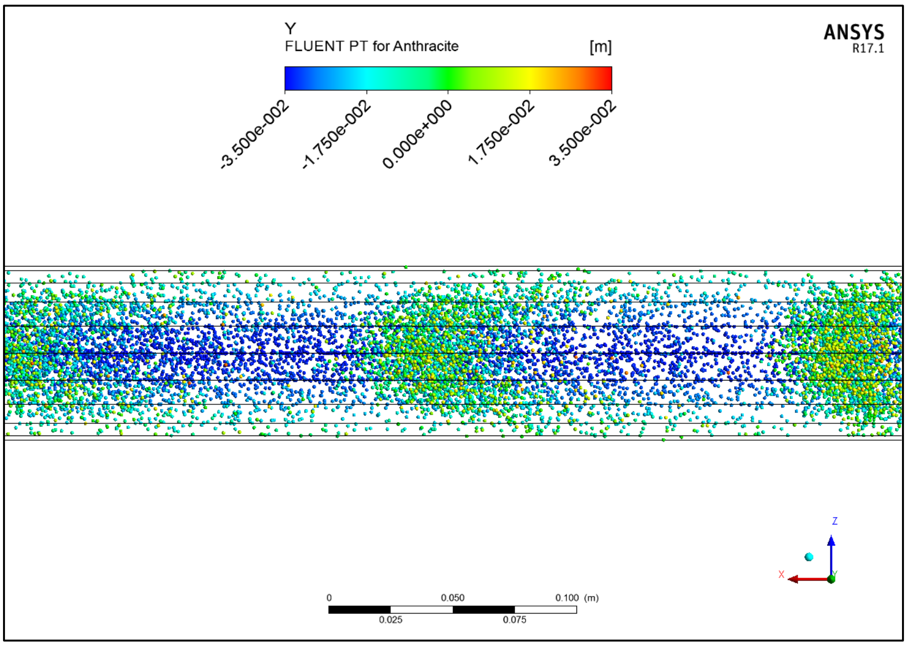


Figure B-9: 0.2 m/s – 1.61e-5 – top view**

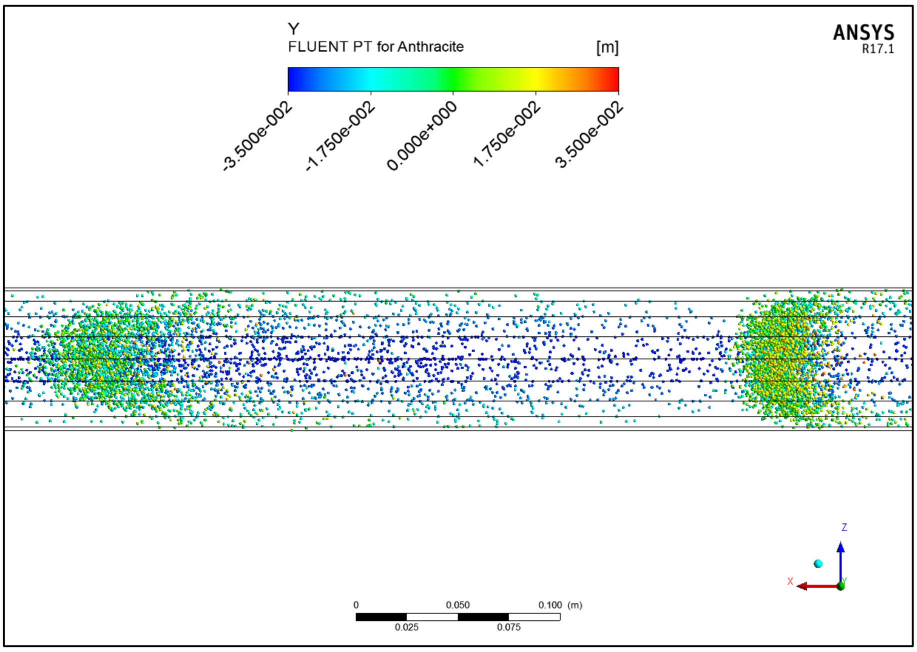


Figure B-10: 0.1 m/s – 1.61e-5 – top view

*Side view – flow according to x-axis (left to right)

**Top view – flow according to x-axis (right to left)

APPENDIX C

VISUAL COMPARISON

Figure C-1: water velocity at 0.5 m/s, Al-lababidi (2012)

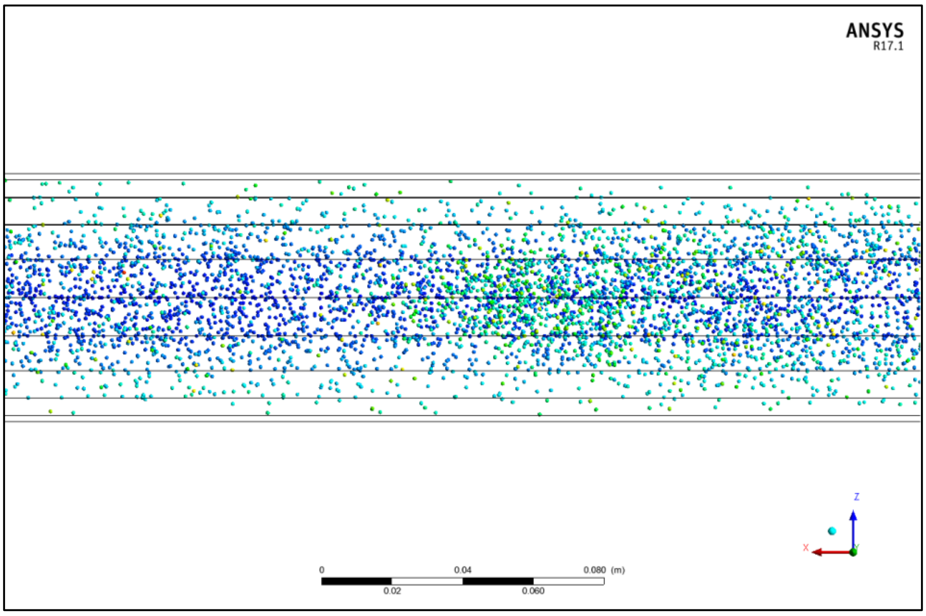


Figure C-2: water velocity at 0.5 m/s, DPM simulation

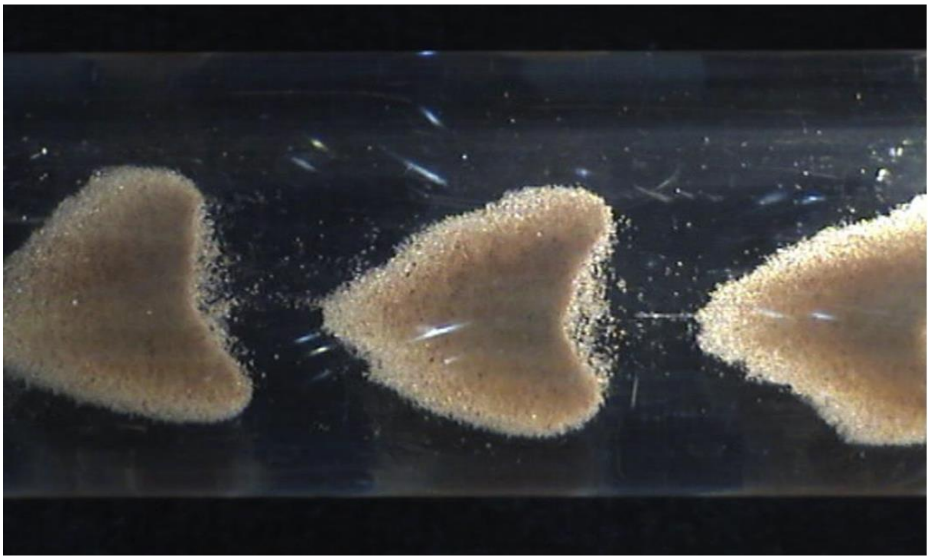


Figure C-3: water velocity at 0.3 m/s, Al-lababidi (2012)

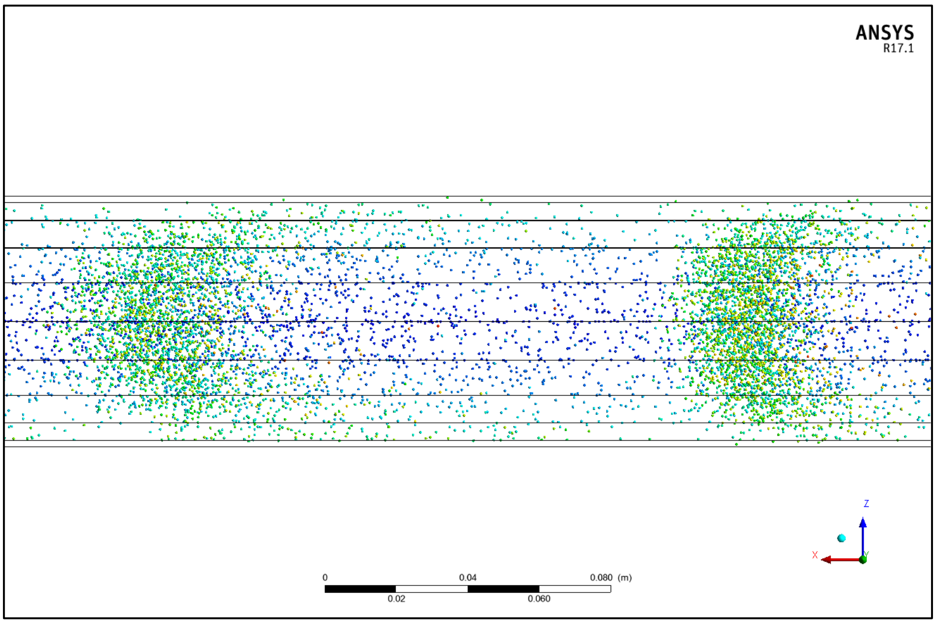


Figure C-4: water velocity at 0.2 m/s, DPM simulation