KINETIC STUDY OF IN-PIPE WATER FLOW FOR POWER GENERATION

VENUTHAN A/L SURENDARAN

FACULTY OF ENGINEERING
UNIVERSITY OF MALAYA
KUALA LUMPUR

2020

KINETIC STUDY OF IN-PIPE WATER FLOW FOR POWER GENERATION

VENUTHAN A/L SURENDARAN

RESEARCH PROJECT SUBMITTED IN FULFILMENT OF THE REQUIREMENTS FOR THE MASTER OF MECHANICAL ENGINEERING

FACULTY OF ENGINEERING
UNIVERSITY OF MALAYA
KUALA LUMPUR

2020

UNIVERSITY OF MALAYA

ORIGINAL LITERARY WORK DECLARATION

Name of Candidate: Venuthan A/L Surendaran

Matric No: 17013727/1

Name of Degree: Masters in Mechanical Engineering

Title of Research Report: Kinetic Study of In-Pipe Water Flow for Power Generation

Field of Study: Fluids mechanics and CFD

I do solemnly and sincerely declare that:

(1) I am the sole author/writer of this Work;
(2) This Work is original;
(3) Any use of any work in which copyright exists was done by way of fair dealing and for permitted purposes and any excerpt or extract from, or reference to or reproduction of any copyright work has been disclosed expressly and sufficiently and the title of the Work and its authorship have been acknowledged in this Work;
(4) I do not have any actual knowledge nor do I ought reasonably to know that the making of this work constitutes an infringement of any copyright work;
(5) I hereby assign all and every rights in the copyright to this Work to the University of Malaya ("UM"), who henceforth shall be owner of the copyright in this Work and that any reproduction or use in any form or by any means whatsoever is prohibited without the written consent of UM having been first had and obtained;
(6) I am fully aware that if in the course of making this Work I have infringed any copyright whether intentionally or otherwise, I may be subject to legal action or any other action as may be determined by UM.

Candidate's Signature Date:

Subscribed and solemnly declared before,

Witness's Signature Date:

Name:

Designation:

UNIVERSITI MALAYA

PERAKUAN KEASLIAN PENULISAN

Nama: Venuthan A/L Surendaran

No. Matrik: 17013727/1

Nama Ijazah: Masters in Mechanical Engineering

Tajuk Kertas Laporan Penyelidikan: Penyelidikan Tenaga Kinetik Saliran Air Di Ruang Paip Untuk Penjanaan Kuasa

Bidang Penyelidikan: Makanik Bendalir dan CFD

Saya dengan sesungguhnya dan sebenarnya mengaku bahawa:

(1) Saya adalah satu-satunya pengarang/penulis Hasil Kerja ini;
(2) Hasil Kerja ini adalah asli;
(3) Apa-apa penggunaan mana-mana hasil kerja yang mengandungi hakcipta telah dilakukan secara urusan yang wajar dan bagi maksud yang dibenarkan dan apa-apa petikan, ekstrak, rujukan atau pengeluaran semula daripada atau kepada mana-mana hasil kerja yang mengandungi hakcipta telah dinyatakan dengan sejelasnya dan secukupnya dan satu pengiktirafan tajuk hasil kerja tersebut dan pengarang/penulisnya telah dilakukan di dalam Hasil Kerja ini;
(4) Saya tidak mempunyai apa-apa pengetahuan sebenar atau patut semunasabahnya tahu bahawa penghasilan Hasil Kerja ini melanggar suatu hakcipta hasil kerja yang lain;
(5) Saya dengan ini menyerahkan kesemua dan tiap-tiap hak yang terkandung di dalam hakcipta Hasil Kerja ini kepada Universiti Malaya ("UM") yang seterusnya mula dari sekarang adalah tuan punya kepada hakcipta di dalam Hasil Kerja ini dan apa-apa pengeluaran semula atau penggunaan dalam apa jua bentuk atau dengan apa juga cara sekalipun adalah dilarang tanpa terlebih dahulu mendapat kebenaran bertulis dari UM;
(6) Saya sedar sepenuhnya sekiranya dalam masa penghasilan Hasil Kerja ini saya telah melanggar suatu hakcipta hasil kerja yang lain sama ada dengan niat atau sebaliknya, saya boleh dikenakan tindakan undang-undang atau apa-apa tindakan lain sebagaimana yang diputuskan oleh UM.

Tandatangan Calon Tarikh:

Diperbuat dan sesungguhnya diakui di hadapan,

Tandatangan Saksi Tarikh:

Nama:

Jawatan:

KINETIC STUDY OF IN PIPE WATER FLOW FOR POWER GENERATION

ABSTRACT

In-pipe turbine generator has a potential to harvest energy from an existing water flow from distribution pipe which have natural and pumping capacity. That flow will be used to convert kinetic energy from the pipe to rotate the turbine. In turn, the turbine may be connected to a generator to produce electric power. The fact that the in-pipe turbine generator will be a useful energy harvesting tool with potential with industrial application is recommended by many scholars. However, there is a concern where there are many types and sizes of pipe that is readily used in the water works industry. Since that concern arises a kinetic study is required to analyze and observe the flow, velocity, and kinetic energy present in different pipes. A kinetic study of pipe water flow will further enhance and familiarize industry players with the in-pipe turbine generator. The reason is that a kinetic study will give more information to turbine manufacturer to size turbine accordingly based on flow data which is available. Observing the simulation results obtained from various pipe dimensions there is a trend that kinetic energy is observed maximum at a reduced diametrical point of the pipe rather than at the pipe's maximum diameter. When sizing turbine proper attention must be given details such as maximum velocity at various diameter to be able to maximize rate of energy harvesting. A turbine cannot be sized to be the same size as the pipe diameter as it will not be able to harvest energy at an optimum rate. This study focusses on geometry of different pipe sizes to evaluate the flow, velocity, and kinetic energy. The results obtained from this study is hoped to aid in proper sizing of turbine for industrial level application

PENYELIDIKAN TENAGA KINETIK SALIRAN AIR DI RUANG PAIP UNTUK PENJANAAN KUASA

ABSTRAK

Penjana turbin dalam paip berpotensi menuai tenaga dari aliran air yang ada dari paip pengedaran yang mempunyai kapasiti semula jadi dan mengepam. Aliran itu akan digunakan untuk menukar tenaga kinetik dari paip untuk memutar turbin. Sebaliknya, turbin boleh disambungkan ke penjana untuk menghasilkan tenaga elektrik. Fakta bahawa penjana turbin dalam paip akan menjadi alat penuaian tenaga berguna dengan potensi dengan aplikasi industri disarankan oleh banyak sarjana. Walau bagaimanapun, ada kebimbangan di mana terdapat banyak jenis dan ukuran paip yang mudah digunakan dalam industri penapisan air. Oleh kerana kebimbangan itu timbul, kajian kinetik diperlukan untuk menganalisis dan memerhatikan aliran, halaju, dan tenaga kinetik yang terdapat pada paip yang berlainan. Kajian kinetik aliran air paip akan meningkatkan dan membiasakan ahli industri dengan penjana turbin dalam paip. Sebabnya adalah bahawa kajian kinetik akan memberi lebih banyak maklumat kepada pengeluar turbin untuk mempiawai turbin sesuai berdasarkan data aliran yang tersedia. Memerhatikan hasil simulasi yang diperoleh dari pelbagai dimensi paip terdapat kecenderungan bahawa tenaga kinetik diperhatikan maksimum pada titik diameter paip yang dikurangkan dan bukannya pada diameter maksimum paip. Semasa mengukur turbin, perhatian yang betul mesti diberikan perincian seperti halaju maksimum pada pelbagai diameter agar dapat memaksimumkan kadar pengambilan tenaga. Kajian ini memfokuskan pada geometri ukuran paip yang berbeza untuk menilai aliran, halaju, dan tenaga kinetik. Hasil yang diperoleh dari kajian ini diharapkan dapat membantu dalam ukuran turbin yang tepat untuk aplikasi industri

ACKNOWLEDGEMENTS

I am a person who firmly believe in the saying that no man is an island. Hence, I am writing this column to convey my sincere gratitude to all who supported, guided, educated, and encouraged me to complete this research project successfully. Firstly, I would like to begin with recognizing Assoc Prof. Ir. Dr. Mohd Faizul Bin Mohd Sabri’s contribution to me for believing in my abilities to complete this study and accepting me as his research student. Appreciate your kindness, Sir!

Moving forward, I would like to convey my highest respect to my parents who did not show any emotions to me but was in every way proud of me in pursuing my master’s studies. Surely, I would like to complete my master’s studies with the best of my abilities to keep them proud. Furthermore, I would like to thank all my friend who are close and the well-acquainted ones from the same course. They may not have directly contributed to this written paper, but they were a positive influencing factor in all my coursework which were a pre-requisite for this research project.

Finally, I would like to invoke the blessing of the Lord to bless this work. It is well known that in this academic journey one must know right from wrong. Only then one will continue to improve his work. I dedicate this work to God and hope that He guides me to the path of right knowledge and correct all my wrongdoings along the way.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

LIST OF SYMBOLS AND ABBREVIATIONS

For examples:

Q	:	Volume flow rate (m^3/s)
v	:	Velocity (m/s)
Re	:	Reynolds number
E_k	:	Kinetic energy ($kg\ m^2/s^2$)
ρ	:	Density (kg/m^3)

LIST OF APPENDICES

CHAPTER 1: INTRODUCTION

This study is to find a suitable pipe size/diameter to harvest energy using In-pipe turbine generator (IPTG). Pressure, velocity, kinetic energy, and flow characteristics will be evaluated for pipe of different diameter. Evaluation will be done through Ansys Fluent simulation.

Modelling of geometry was done in three dimensional for five different pipe geometry. Details of modelling is discussed with specifics in the methodology section. Meshing and solving for results were performed in ANSYS FLUENT. Mesh details and simulation report is available in the Appendix section. Results of simulation is discussed along this report for all five pipe geometry.

This is a FLUENT based simulation study and proper consideration have been taken to choose the right solvers and input parameters that was used in this study. FLUENT based input is discussed in the methodology section with reasoning. A k-epsilon turbulence model with enhanced wall treatment function was chosen as the solving equation considering pre-simulation results which suggested flow is considered turbulent with high Reynolds number.

Velocity and turbulence kinetic energy profile at post-simulation was derived from the simulated geometry. Profile was extracted from 6 sections of the geometry to evaluate flow characteristics and maximum turbulence kinetic energy point. All profile extracted have been displayed in the Appendix section as reference.

Evaluation for this study will be done on the pipe flow to evaluate the velocity and kinetic energy present. This study will be conducted without the presence of an In-pipe turbine. In order for us to understand the potential of each pipe size used for energy harvesting.

1.1 Problem statement

The raw water processing industry pumps and supply water with potential of energy harvesting. Pumps works at full capacity to build pressure for the processed water to reach each household. The pressure and velocity available in the pipeline can be exploited to harness energy using an In-pipe turbine generator.

In addition to that, there are a lot of variation of pipe sizes used in transporting processed water depending on the demand available in the city. Pipes of different diameter have different rate of kinetic energy that can be made use of to harness energy. The pumping of processed water is a must to ensure that the processed water reaches the consumer of a distance. Running of pumps requires energy as well. A reverse method such as an IPTG can be employed into the pipelines to re-harness a small portion energy. This method may be able to be cost saving in the long run.

The is no recent study proposing proper potential of an In-pipe energy harvester with quantitative values. There is a study required to evaluate pipe flows only to understand the potential of energy harvesting. This is because the vast variety of pipe types and sizes which are available in the industry water works application.

1.2 Objective

This research project is aimed:

1. To model and mesh pipe of different dimensions using Ansys.

2. To perform simulation of the meshed pipe variations using ANSYS Fluent.

3. Evaluate velocity and flow behavior based on simulation results.

CHAPTER 2: LITERATURE REVIEW

2.1 In-pipe turbine generator

The study of an in-pipe turbine generator is generally a new study where there is patent filed in the title in-pipe hydroelectric power system and turbine in the year 2013 by a company named Lucid Energy Inc.

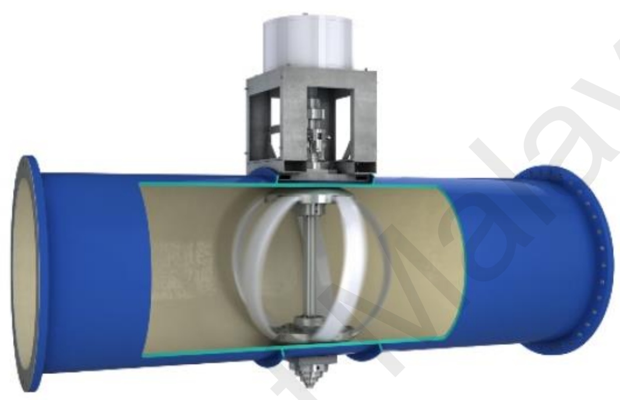

Figure 2.1: Cutout view of an In-pipe turbine generator

Figure 2.1 illustrates a cutout view of an in-pipe turbine. The turbine creates a motion by tapping the potential of water flow through the pipe. Water flow from the axis of the pipe will be used to create radial motion in the turbine and the motion is used to eventually power a generator with the aid of a shaft (United States of America Patent No. US 8,360,720 B2, 2013).

The turbine is sophisticated in its own way, whereby it is designed particularly for an in-pipe application. The idea of energy harvesting from this type of turbine may not be a complete solution in the sense whatever pumping power which is used will be re-harnessed back. In return, it may harvest a fraction of the energy back. Even though, the claim that it is not a complete solution it may still serve a purpose in water treatment plant.

In water treatment plant, there are other processes along the pipeline other water treating itself. Like, water sampling and flow monitoring equipment along the pipeline which will cover urban and remote areas. These tools are necessary in water treatment plants to provide good quality of water and undisrupted service of water to consumers. In remote areas access to electrical power to make use of these equipment can be a challenge. An in-pipe turbine generator may be the most suitable solution for cases as stated earlier (Langroudia, Afifia, Nobarib, & Najafia, 2019).

2.2 Types of fluid medium used for In-pipe application in urban planning

In an urban or sub-urban area, the supply and return of water is needed for consumption and sewerage purpose. In consumption, the method of supply will be via closed pipe from the dam or reservoir to the consumer. Whereas for sewerage application it is a combination of closed and open drainage system and in some cases closed pipe depending on the location and other urban factors.

In the context of water for consumption and sewerage the categories can be classified as raw water, treated water and wastewater. Utilities pipe serves its purpose for all three class of medium to channel its flow to the respected destination.

Raw water is generally sourced from river, lakes and sometimes from the ground water well. Then channeled or pumped to a dam for collection. Treated water comes when raw water is properly cleaned and tested to be safe for consumption. Wastewater is can be considered as water used for cleaning and washing. Where wastewater is usually channeled to a sewerage rain and will be treated to be toxic free before being left out to the environment again (Starkl, Brunner, Flogl, & Wimmer, 2008).

2.3 Pipe materials used for In-pipe flow application

Pipe materials for water works needs proper consideration prior to final selection. Even though, selection of material is not part of this projects objective but its worth stating from the literatures that will be useful in future studies.

The general pipe material used is mild steel, ductile iron, HDPE, ABS GRP & PVC-O. Relating to the material there is an order to select a pipe material. Whereby, for non-corrosive soil the order of preference is starting with mild steel and then to ductile iron, HDPE, ABS GRP & PVC-O. Whereas for corrosive soil it will start with HDPE and then to ABS, GRP, PVC - O, ductile iron and finally mild steel (Malaysia, 2018).

2.4 Pipe sizes used for in-pipe water application

In terms of pipe sizes there is no common size as size is fixed based on water demand and supply ability. There is a range specified which is 150mm till 1300mm. Based on the range there is 6 small range grouping based on specification. The first range is from 150mm – 400mm, next is 450mm – 550mm, subsequently 600mm – 750mm, 800mm – 1000mm, 1050mm – 1200mm and finally 1300mm and above. The classification is done in such a way complying to the local water management commission (Malaysia, 2018).

2.5 Fluid parameters

Formulas that are forecasted to be applied in this study is listed in this section. Also, conversion of units that are required to be performed is stated accordingly below.

Converting volume flow rate from m^3/hr to m^3/s.

$$\text{Volume flow rate, Q } (m^3/hr) = 1\frac{m^3}{hr}\ x\ \frac{1}{3600}\frac{hr}{s} = 0.000278\ \frac{m^3}{s} \tag{2-1}$$

Volume flow rate general formula and velocity formula after proper re-positioning of variable.

Volume flow rate, Q = A.v, A – Area (m^2), V – velocity (m/s) (2-2)

Velocity, $v = \frac{Q}{A}$ (2-3)

Area of circle formula as below to aid in Volume flow rate, Q, calculation.

Area of circle, $A = \pi r^2 = \frac{\pi D^2}{4}$ (2-4)

Reynolds number formula to understand if flow is laminar or turbulent.

Reynolds number, Re $= \frac{\rho v D}{\mu}$; Re < 2300 – laminar, 2300 < Re < 4000 – transitional, Re > 4000 – turbulent. (2-5)

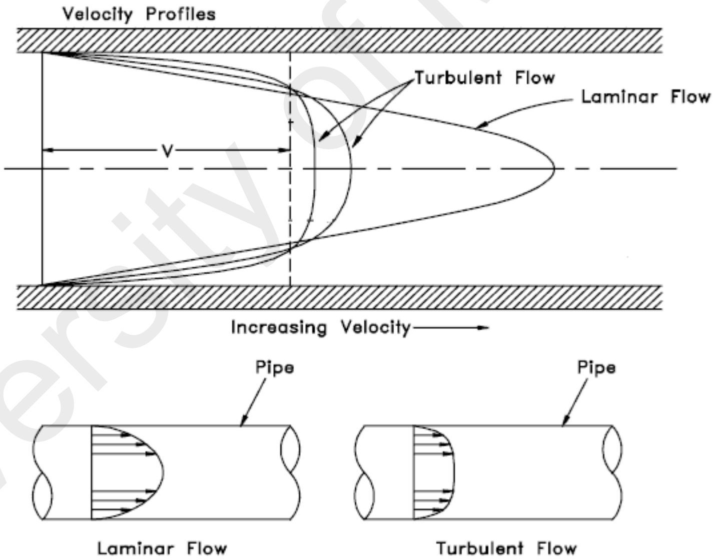

Figure 2.2: Image of velocity profile related to flow

Figure 2-1 illustrates velocity profile in a pipe. Two different type of flow can be analysed based on the image. Type of flow is laminar and turbulent. Another method to determine flow other than using the Reynolds formula is to analyse the velocity profile of the pipe.

Pressure in references are stated in terms of pressure head with SI unit of, m. Below states method to convert pressure head, m to bar then subsequently to Pascal (Cengel & Cimbala, 2006).

Pressure, meter head, 1m = 10.2 bar = 1,020,000 Pascal (2-6)

Kinetic energy formula as stated below. Based on Bernoulli's equation, kinetic energy is calculated per unit volume.

Kinetic energy, $\frac{E_K}{V} = \frac{1}{2}\rho v^2$ (2-7)

2.6 Mathematical modelling for pipe flow

ANSYS CFD is chosen as a simulation tool for this study. This simulation tool has a complexity with diverse solution method. In addition to that, proper validation is required based on experimental data. Basically, the process of simulation is about preparing, verifying, validating, and documenting the results obtained.

Most importantly, the right solution method must be chosen for the software to solve the case effectively with minimal error. Mass, momentum, and energy equation are considered governing equation that needs to be set up to solve a CFD simulation. To determine the specific equation the user must resolve whether the flow being used is laminar or turbulent. There will not be much discussions on laminar flow as in our case is related turbulent flow.

In turbulent flow, the discussion will be about a few solution methods and the best purpose that it will serve. Firstly, standard k–ε model is suitable for high Reynolds number turbulent flow. In cases of geometry with rotational effect the k–ε model may not solve with proper precision and stability. Secondly, Re-normalization Group (RNG) k–ε model is suitable low Reynolds number turbulent flow. The RNG k–ε method is suitable for rotating geometry but considering low Reynolds number. Thirdly, Realizable k−ε model is useful in planar and round jets application where prediction of speeding rate is more accurate. In cases with model with rotating and stationary flow, non-physical turbulent viscosity will be produced. Fourthly, standard k-ω model is good for measurements with problems of far wake, mixing layers and plane, round, and radial jets. It contains low Reynolds number effect, compressibility and shear flow spreading. Furthermore, Shear-stress transport (SST) k-ω model is used for a wide class flow and is precise for both near and far field region. Lastly, Reynolds stress model is more suitable in terms of accuracy than k−ε and k-ω model, but simulation may take too long with this solution (Cao, 2011).

CHAPTER 3: METHODOLOGY

3.1 Literature review

As per the context of this study, it will be carried out based on data obtained from various literatures and technical sheets available for public use. Main components of literature are ANSYS FLUENT related article, technical data from the water management commission, textbooks to relate theories and formula.

Based on objective of this study, there is not much study done on kinetic energy of a pipe. Maybe because the idea of harnessing kinetic energy from the flow medium of pipe is still new.

3.2 Geometry formation

Formation of geometry is done using AUTOCAD software and then exported into ANSYS geometry. Geometry is done based on dimensions as stated below.

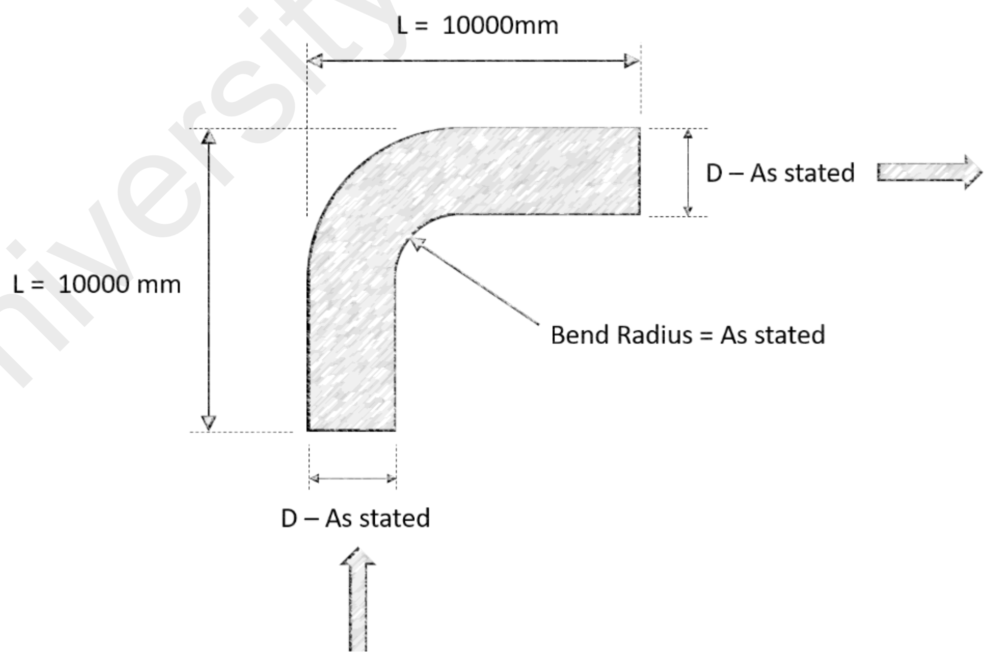

Figure 3.1: Design of pipe constructed with 90 deg elbow

Figure 3.1 gives a rough layout of how the pipe design will be constructed. The length of the pipe before the elbow and after the elbow is set at 10m length for all simulation cases. Bend radius is set at 0.4m for all geometry constructed. Diameter of pipe will vary from 200 mm to 400 mm with step difference of 50 mm for every geometry. Therefore, in total there are 5 geometry constructed.

3.3 Meshing of geometry

All geometry is later meshed as per the requirement of ANSYS to run simulation. The whole pipe geometry will be considered 1 body. Named selection was given to each section of the pipe for convenience of data acquisition and analysis. The respective parts to be properly named inlet, outlet, pipe before elbow, elbow, and pipe after elbow. Maximum number of layers was defined as default from ANSYS which is 5 layers. The choice of 5 layers were chosen to avoid long simulation time and considering the size of the geometry.

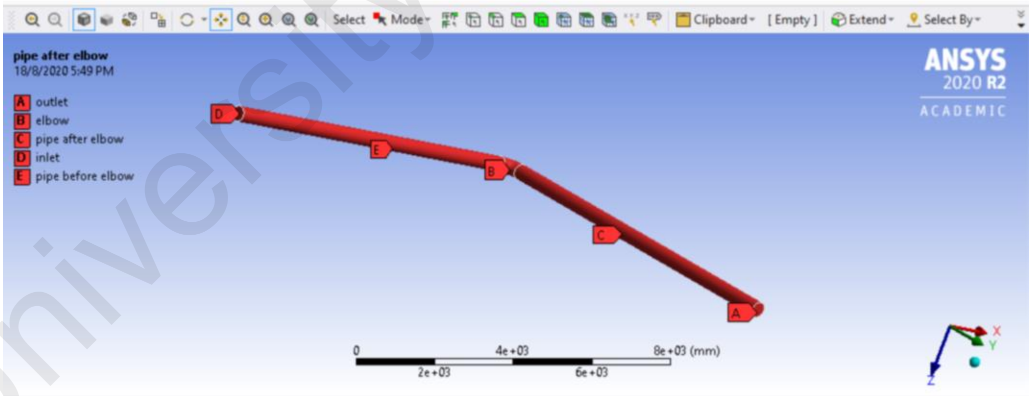

Figure 3.2: Named selection of part of pipe

Figure 3.2 illustrates named selection give to each section of the pipe. Evaluation of data will be done based on each section as portrayed in the image.

3.4 Simulation of meshed geometry

Meshed geometry will be simulated with conditions preset in the FLUENT software. The proposed conditions will be based on general solvers selection, solution models, materials, cell zone conditions, boundary conditions, method of solution and number of iterations.

In the general selection, the conditions set is pressure based, absolute velocity and steady time method. Next, in models the energy equation was put to ON and standard k-epsilon enhanced wall treatment method was chosen considering turbulent flow condition. Moving on, for materials the medium is water. Other than that, cell zone condition will be fluid flowing through (ANSYS Inc, 2011). t

Boundary conditions will vary according to geometry prepared. Thus, a table is prepared to state details of settings.

Table 3.1: Pressure, Velocity and Reynolds number calculation for each geometry simulated

No.	Diameter, D (m)	Pressure, Pa	Velocity, V (m/s)	Reynolds number, Re
1	0.2	294124	0.111	27567.4
2	0.25	294124	0.071	22053.9
3	0.3	294124	0.049	18378.2
4	0.35	294124	0.036	15752.8
5	0.4	294124	0.028	13783.7

Pressure and velocity stated in Table 3.1 will be considered boundary conditions used for inlet of the pipe. Outlet conditions will be checked with Prevent reverse flow with compliance with original cases where most treated water pipes are equipped with check valves to prevent reverse flow.

Before calculating solution, method for calculating turbulent kinetic energy and turbulent dissipation rate will be done in second order upwind. Subsequently the meshed geometry will be initialized with standard initialization and set to compute from the inlet of the pipe. Solution of the meshed geometry will be calculated with 2000 iterations.

3.5 Type of data and data acquisition

Data which can be computed from ANSYS FLUENT based on the simulation condition discussed are as follows. Related to this study, the required data will be velocity, pressure, and turbulence kinetic energy. All data acquisition will be from FLUENT software.

Data displayed is suggested to be velocity and turbulence kinetic energy contour of all 3D pipe dimensions for understanding the flow. For flow analysis, velocity profile will be extracted from 6 sections of 1 pipe geometry. A conclusion will be given for whether the flow is laminar or turbulent. Furthermore, as this study is about harnessing of energy the turbulence kinetic energy profile of pipe at 6 sections will be analysed too.

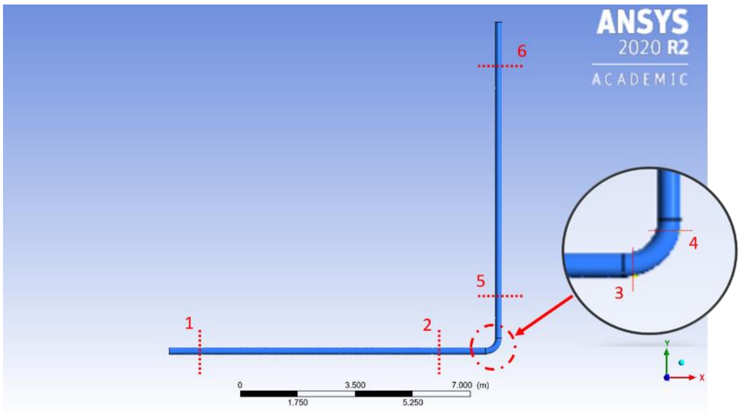

Figure 3.3: Section on pipe for dynamic profile analysis

Figure 3.3 illustrates a total of 6 sections that will be set to analyse the dynamic quantity such as velocity and turbulence kinetic energy.

3.6 Analysis of Data in theory and simulation

In theory, the calculation of velocity and Reynolds number will be done using the respective formula. A constant preset value of pressure will be determined for the system. Pressure will be considered a constant variable. From flow simulation, velocity profile will be obtained from 5 points of the constructed geometry to analyze flow. In addition to that, turbulent kinetic energy profile will be obtained and analyzed comparing with the objective of this study.

3.7 Findings & Discussions

All findings will be well documented and illustrated according to necessity based on the given objective. Findings of five different pipe dimensions will be discussed in detail in line with this study's objective. A specific feedback will be given based on the results obtained. In addition to that, suggestion for further improvement or if another study required will be proposed accordingly.

3.8 Conclusions

A conclusion will be drawn based on the findings and discussions that were obtained from this study. Conclusion drawn will not compare much between different geometry as this study is a steppingstone to go further and analyze the suitable in-pipe turbine generator. In this case the suitably will mean the right sizing based on the kinetic energy of the pipe and the right radial location to install the in-pipe turbine generator in the pipeline to maximize the energy harnessed.

3.9 Expected outcomes based on previous literature

There are previous literatures which have test run an in-pipe turbine generator and obtained experiment data based on its blade profile. Most of which have focused on the turbine blade design but not on kinetic study of the pipe. This study will focus on the kinetic study of the pipe. Where the initial flow is calculated to be turbulent. A velocity profile of 6 sections along the simulated pipe geometry will be obtained to compare the flow characteristics. Flow is expected to be turbulent after simulation like theoretical calculation. ANSYS FLUENT have the capability to solve for turbulence kinetic energy. Turbulence kinetic energy profile for 6 sections along the simulated pipe geometry will be observed. This observation will be used to determine the optimized diameter of an in-pipe turbine. Based on theory, the near wall velocity will be zero. Velocity and kinetic energy are closely related through a general formula which have been stated in the literature review section. Thus, an in-pipe turbine may not be suitable to be sized at the same maximum diameter of the pipe. It may need to be sized to be smaller by a few percent. An ideal size can be proposed based on the observation done on the turbulence kinetic energy profile.

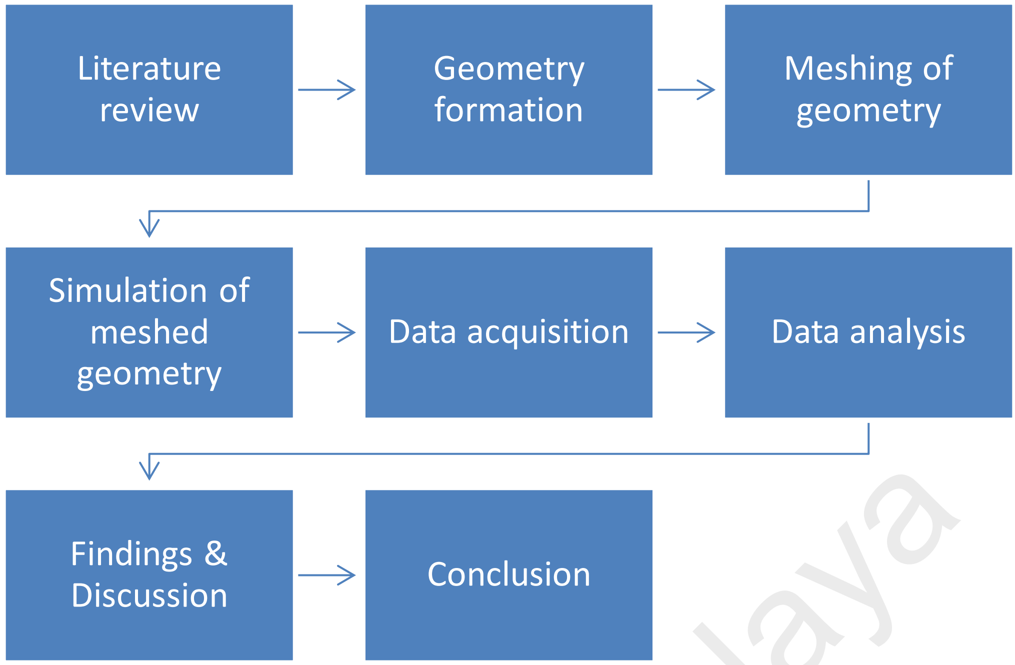

Figure 3.4: General flow of research project

Figure 3.4 is the general flow of how the research project will be conducted. Details in each section stated may vary according to the findings of this research along the way. All necessary details will be updated in the methodology if any changes were made.

CHAPTER 4: RESULTS AND DISCUSSIONS

4.1 Results

Results for this study have been calculated theoretically initially and computed using CFD ANSYS. The results will be based on velocity and turbulence kinetic energy profile. 3D contour will be displayed for proper understanding of geometry.

4.1.1 Theoretical calculation

Below is theoretical calculation done with available data prior to simulation.

Table 4.1: Tabulated data of theoretical calculation

No	Diameter, D (m)	Bend radius, R (m)	Pressure, Pa	Area, A (m2)	Velocity, V (m/s)	Reynolds number, Re	Kinetic Energy, Ek
1	0.2	0.4	294124	0.126	0.111	27567.4	6.095
2	0.25	0.4	294124	0.196	0.071	22053.9	2.497
3	0.3	0.4	294124	0.283	0.049	18378.2	1.204
4	0.35	0.4	294124	0.385	0.036	15752.8	0.650
5	0.4	0.4	294124	0.503	0.028	13783.7	0.381

Table 4.1 shows fixed data considered in this experiment. Data presented is pipe diameter, pipe elbow bend radius, pressure, pipe cross sectional area, velocity, Reynolds number and Kinetic energy. All values presented are obtained theoretically.

A general assumption was made in this study for the theoretical calculation. The pressure as sated is fixed to 294124 Pascal. As this is the maximum allowable pressure in distribution pipes of water treatment plant. The flow rate is fixed at 50 m^3/hr. or 0.0139 m^3/s for ease of calculation.

4.1.2 Velocity contour

Velocity contour for all 5 variations of pipe from 200 mm to 400 mm diameter will be illustrated as below. Contour range is set from 0 m/s to 0.2 m/s. Maximum achieved velocity is 0.111 m/s. Contour range is set uniform for all pipe diameter for convenience of illustration.

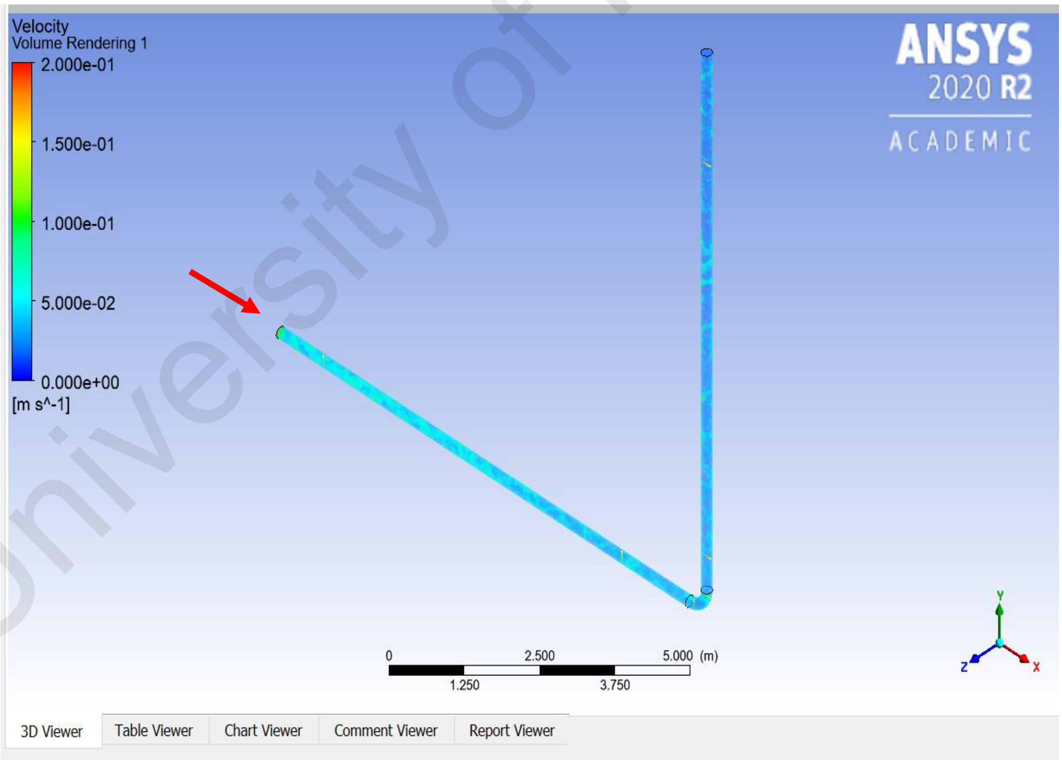

Figure 4.1: Velocity contour of 200 mm diameter pipe

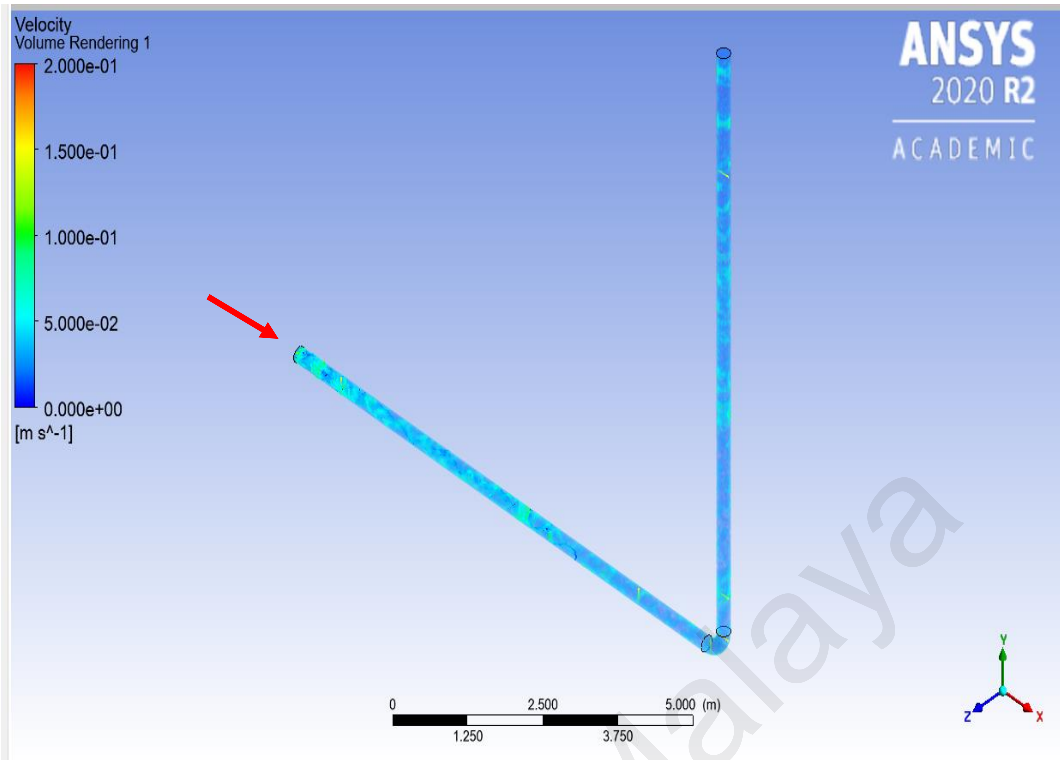

Figure 4.2: Velocity contour of 250 mm diameter pipe

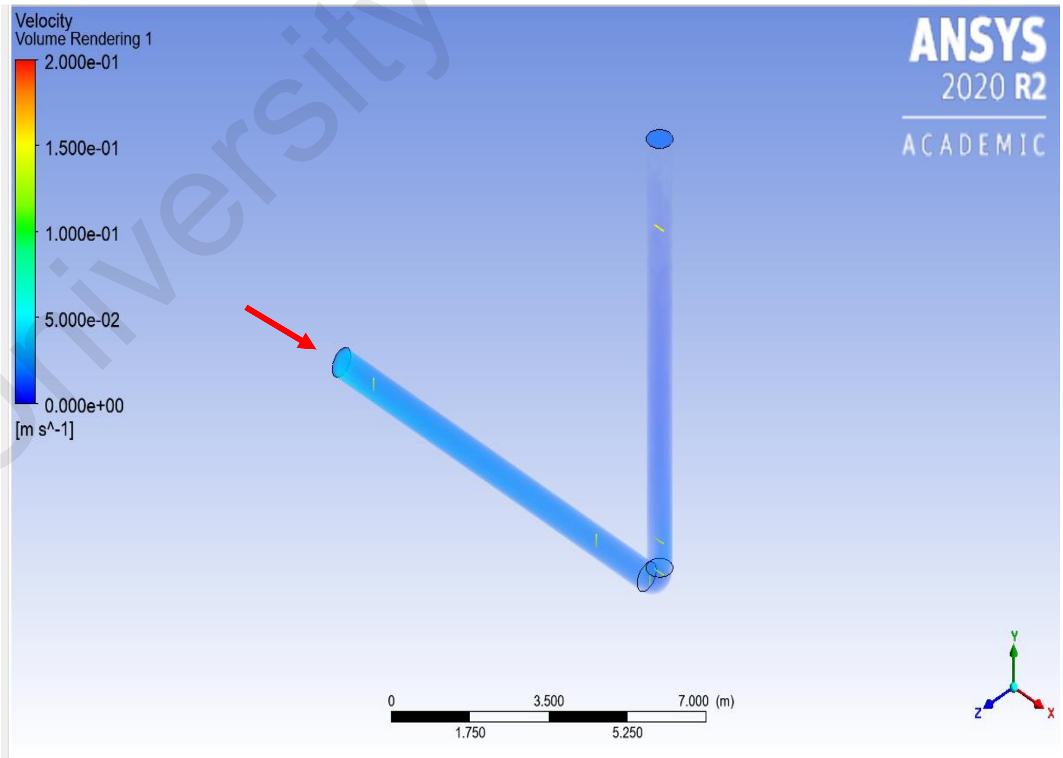

Figure 4.3: Velocity contour of 300 mm diameter pipe

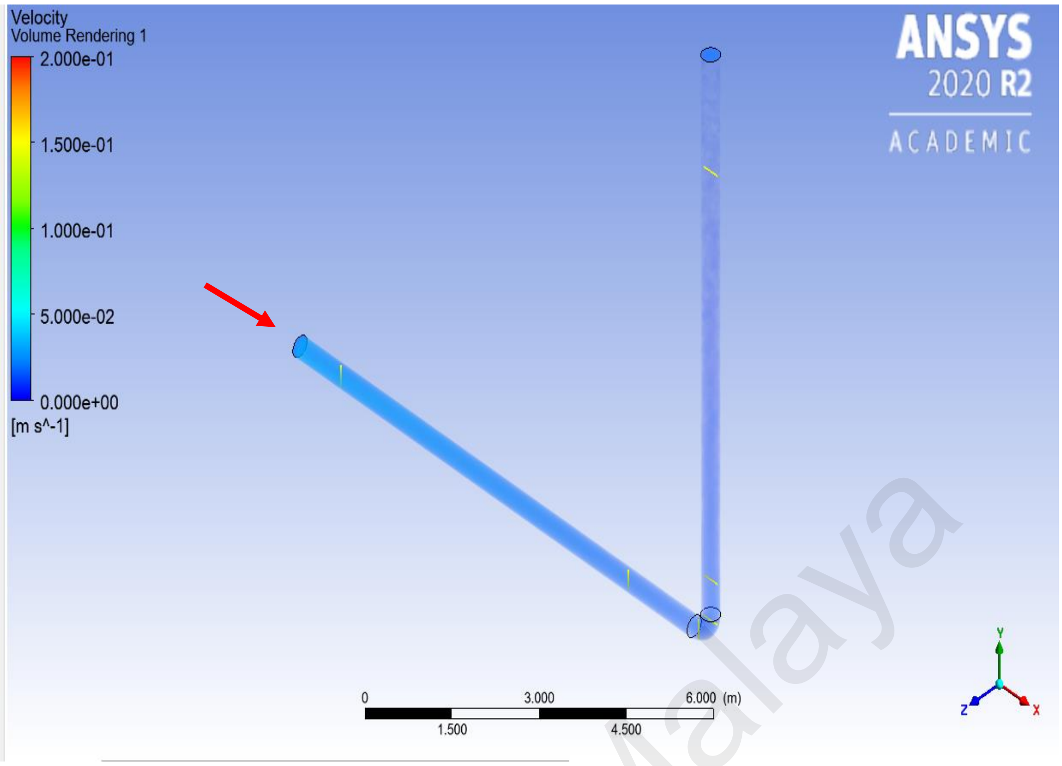

Figure 4.4: Velocity contour of 350 mm diameter pipe

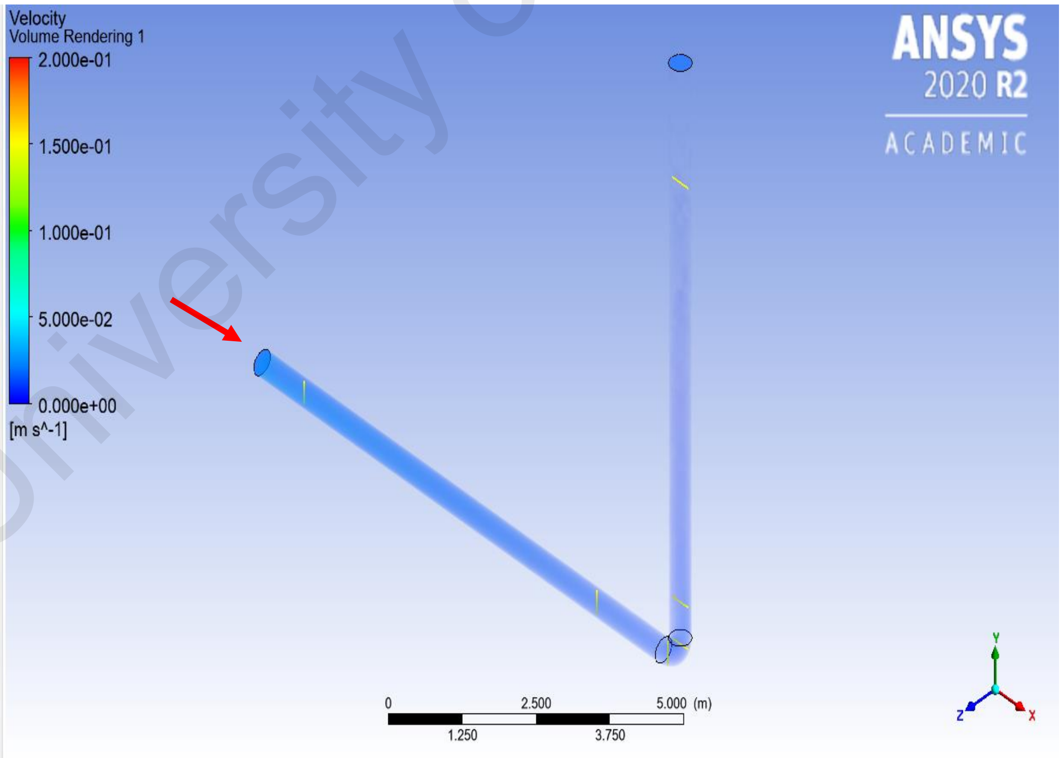

Figure 4.5: Velocity contour of 400 mm diameter pipe

Figure 4.1, 4.2, 4.3, 4.4 & 4.5 shows the velocity contour of all five variation of pipe as described below the image. The inlet of the flow is from positive x – axis direction and its outlet are towards the positive y – axis direction. As illustrated, there is an observation where velocity is decreasing along the pipe. Towards the outlet of the pipe the velocity may be observed to be almost 0 m/s.

4.1.3 Turbulence kinetic energy contour

Turbulence is defined to be caused by high energy in kinetic form. Turbulence kinetic energy is considered the average kinetic energy for one-unit mass.

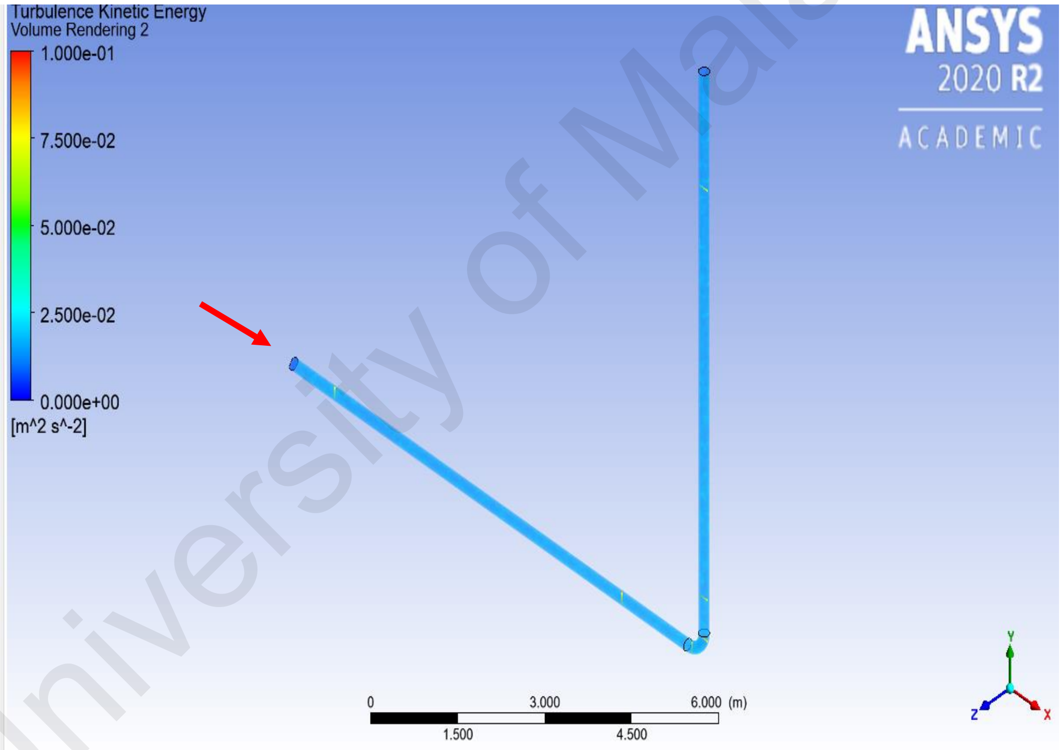

Figure 4.6: Turbulence kinetic energy contour of 200 mm diameter pipe

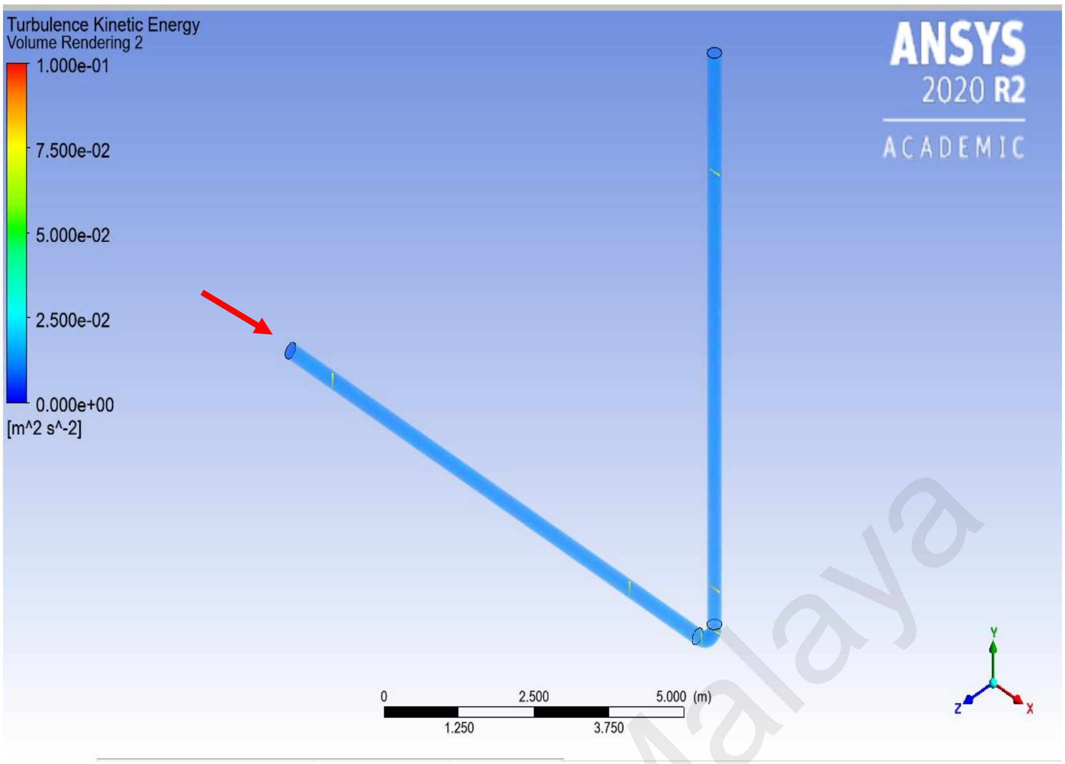

Figure 4.7: Turbulence kinetic energy contour of 250 mm diameter pipe

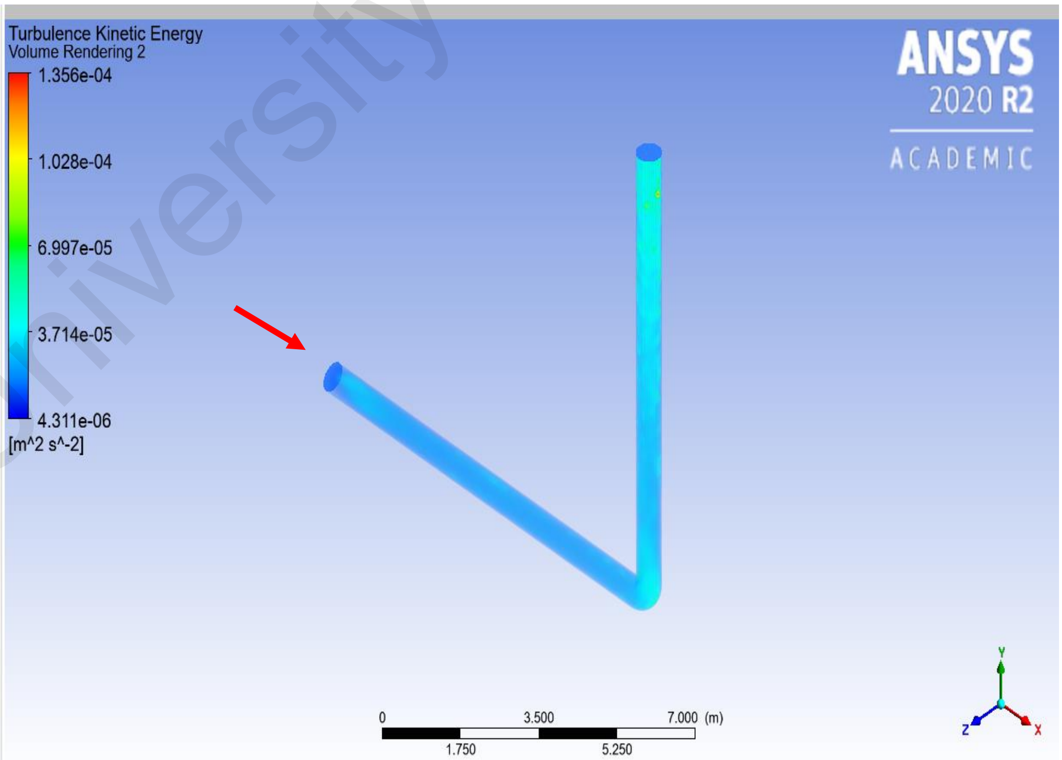

Figure 4.8: Turbulence kinetic energy contour of 300 mm diameter pipe

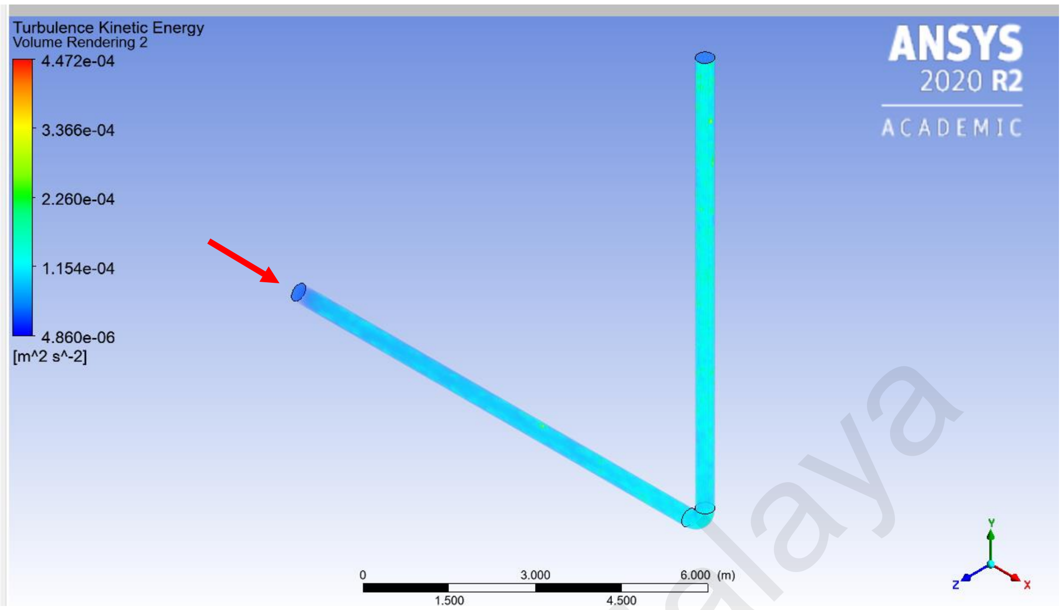

Figure 4.9: Turbulence kinetic energy contour of 350 mm diameter pipe

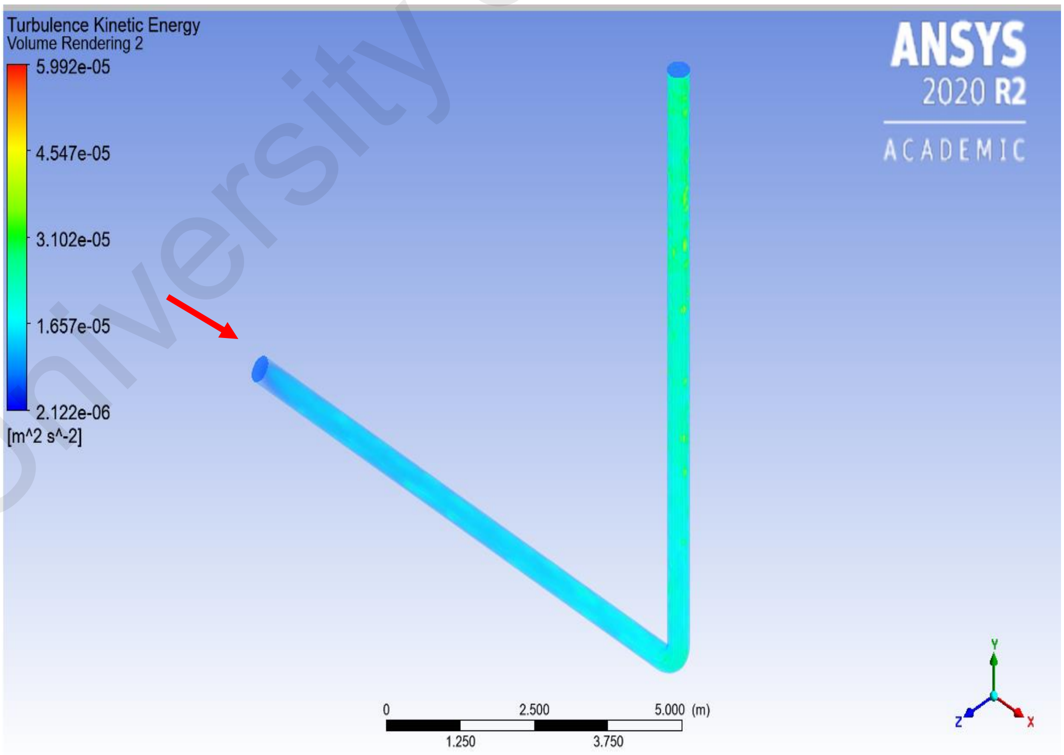

Figure 4.10: Turbulence kinetic energy contour of 400 mm diameter pipe

Figure 4.6, 4.7, 4.8, 4.9 & 4.10 shows the contour of turbulence kinetic energy. A smaller diameter of pipe achieved a higher turbulence kinetic energy. The rate of turbulence kinetic energy decreases as the pipe diameter get bigger. A decrease in turbulence kinetic energy mean that the pipe will not be useful for energy harvesting.

4.1.4 Velocity profile

Velocity profile is obtained from 6 sections of the pipe in all 5 different geometry constructed. A simple flow evaluation by comparing Figure 2.1 can be done to analyse type of flow. This is to compare theoretical data with experimental data.

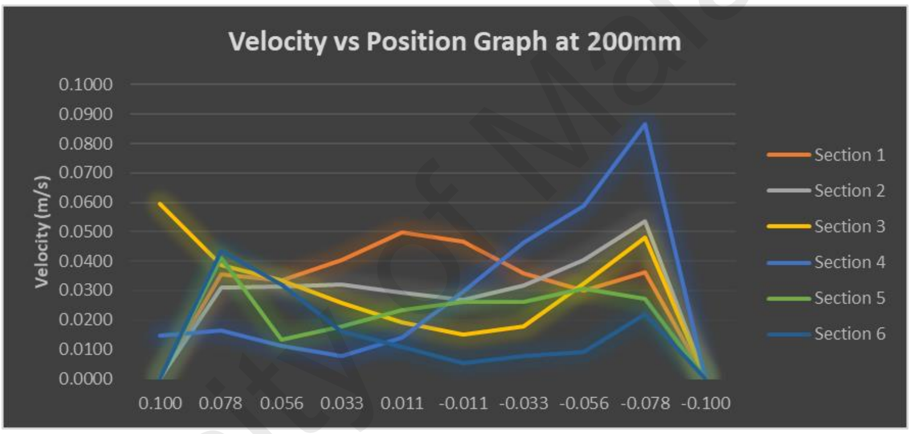

Figure 4.11 Velocity vs Position Graph at 200mm of 6 sections of pipe

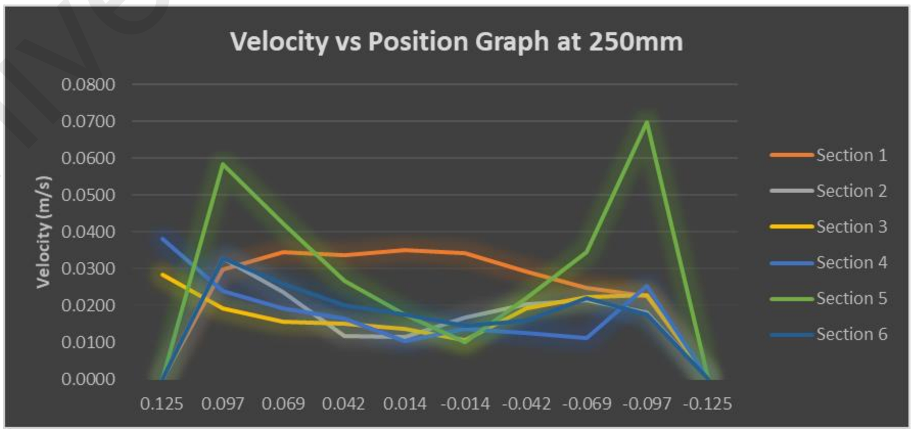

Figure 4.12 Velocity vs Position Graph at 250mm of 6 sections of pipe

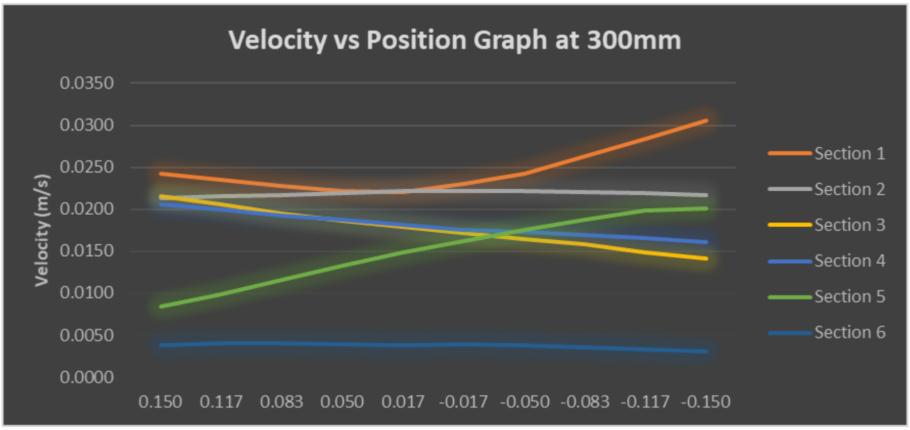

Figure 4.13 Velocity vs Position Graph at 300mm of 6 sections of pipe

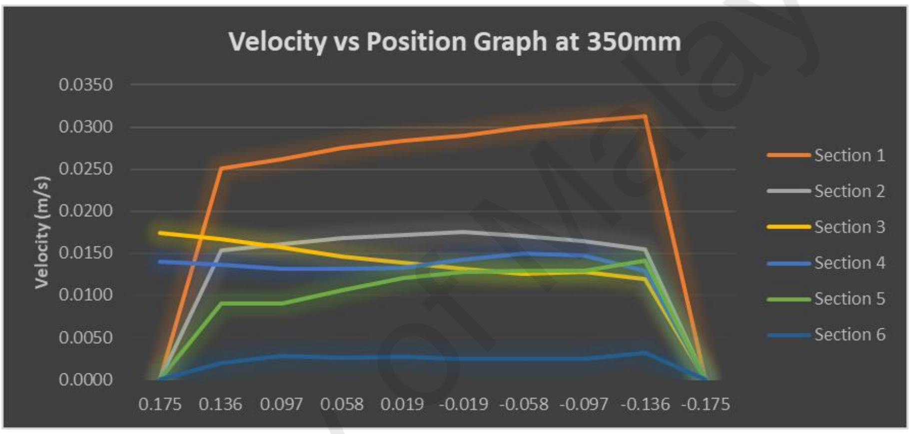

Figure 4.14 Velocity vs Position Graph at 350mm of 6 sections of pipe

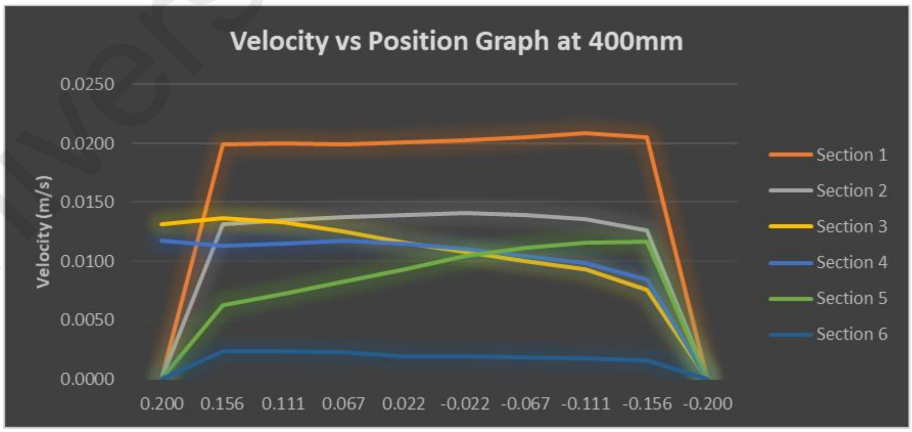

Figure 4.15 Velocity vs Position Graph at 400mm of 6 sections of pipe

Figure 4.11, 4.12, 4.13, 4.14 & 4.15 illustrates a graph that plots velocity profile of all 6 sections of a pipe. This graph will be able to show properties of velocity along the pipe.

Table 4.2: Type of flow evaluated at section 1 of each pipe

Section 1		
No	Pipe size (mm)	Flow
1	200	Turbulent
2	250	Turbulent
3	300	Turbulent
4	350	Turbulent
5	400	Turbulent

Table 4.3: Type of flow evaluated at section 2 of each pipe

Section 2		
No	Pipe size (mm)	Flow
1	200	Turbulent
2	250	Turbulent
3	300	Turbulent
4	350	Turbulent
5	400	Turbulent

Table 4.4: Type of flow evaluated at section 3 of each pipe

Section 3		
No	Pipe size (mm)	Flow
1	200	Turbulent
2	250	Turbulent
3	300	Turbulent
4	350	Turbulent
5	400	Turbulent

Table 4.5: Type of flow evaluated at section 4 of each pipe

Section 4		
No	Pipe size (mm)	Flow
1	200	Turbulent
2	250	Turbulent
3	300	Turbulent
4	350	Turbulent
5	400	Turbulent

Table 4.6: Type of flow evaluated at section 5 of each pipe

Section 5		
No	Pipe size (mm)	Flow
1	200	Turbulent
2	250	Turbulent
3	300	Turbulent
4	350	Turbulent
5	400	Turbulent

Table 4.7: Type of flow evaluated at section 6 of each pipe

Section 6		
No	Pipe size (mm)	Flow
1	200	Turbulent
2	250	Turbulent
3	300	Turbulent
4	350	Turbulent
5	400	Turbulent

Table 4.2, 4.3, 4.4, 4.5, 4.6, & 4.7 gives a simplified evaluation of flow on the pipe section. It must be noted that all velocity profile is not of the same shape. Suitability of installing an energy harvester must be decided upon doing further study specific to different velocity profile.

4.1.5 Turbulence kinetic energy profile

APPENDIX A, B, C, D & E is furnished with turbulence kinetic energy profile of the simulated geometry. Profile was obtained from 6 sections of the pipe geometry similar to the position of the velocity profile. In addition to that, all five different pipe geometry have been evaluated. A specific relationship could not be decided due to very less journal available on this subject. Further studies may be required to properly relate to an in-pipe turbine generator. Nevertheless, the presence turbulence kinetic energy with a specific value is convincing that an in-pipe turbine generator is possible.

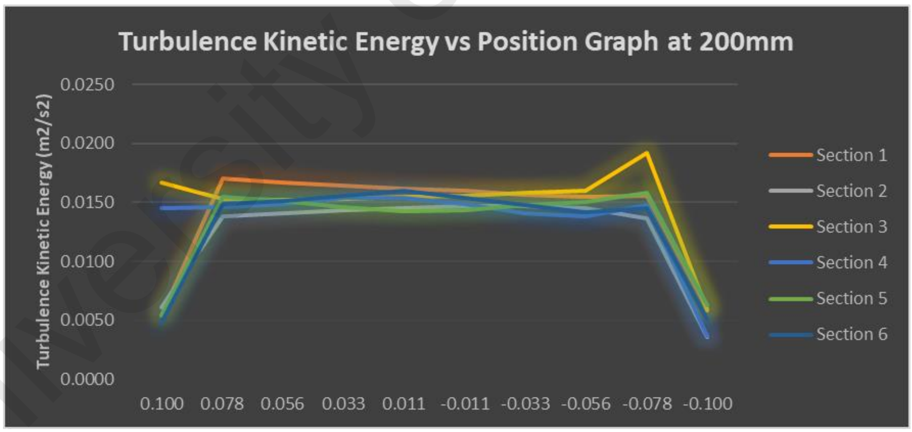

Figure 4.16 Turbulence Kinetic Energy vs Position Graph at 200mm of 6 sections of pipe

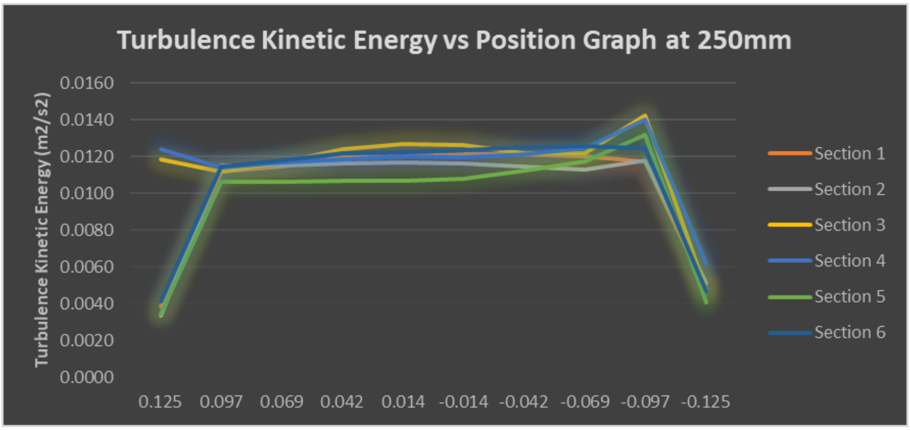

Figure 4.17 Turbulence Kinetic Energy vs Position Graph at 250mm of 6 sections of pipe

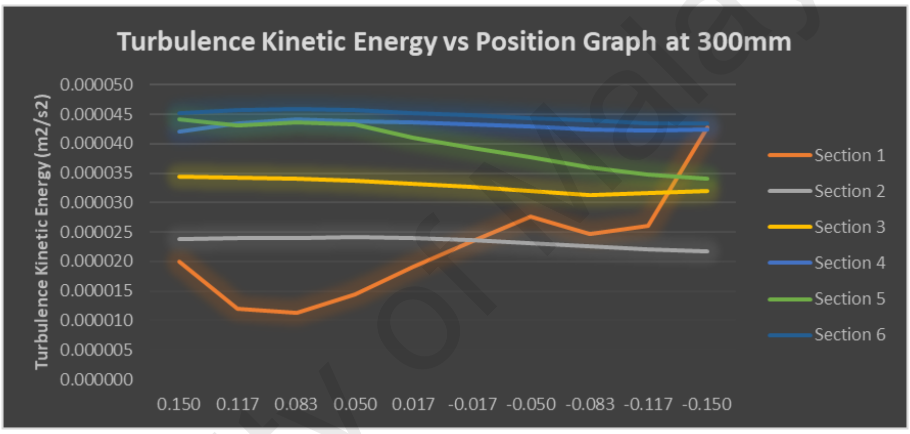

Figure 4.18 Turbulence Kinetic Energy vs Position Graph at 300mm of 6 sections of pipe

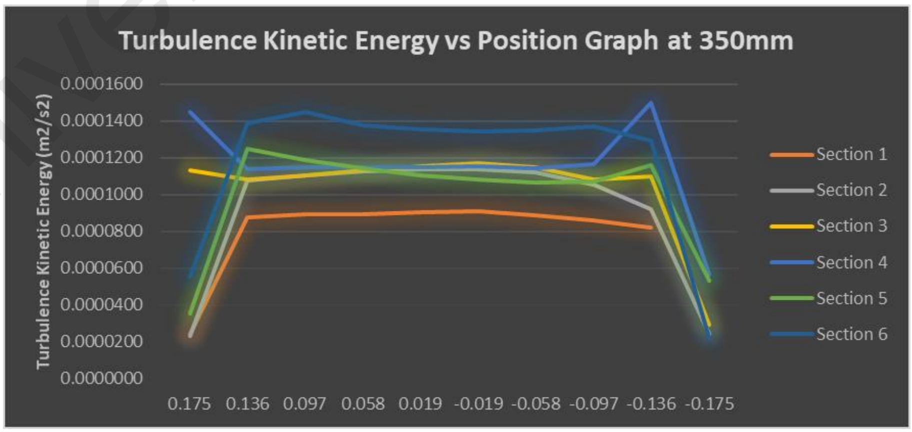

Figure 4.19 Turbulence Kinetic Energy vs Position Graph at 350mm of 6 sections of pipe

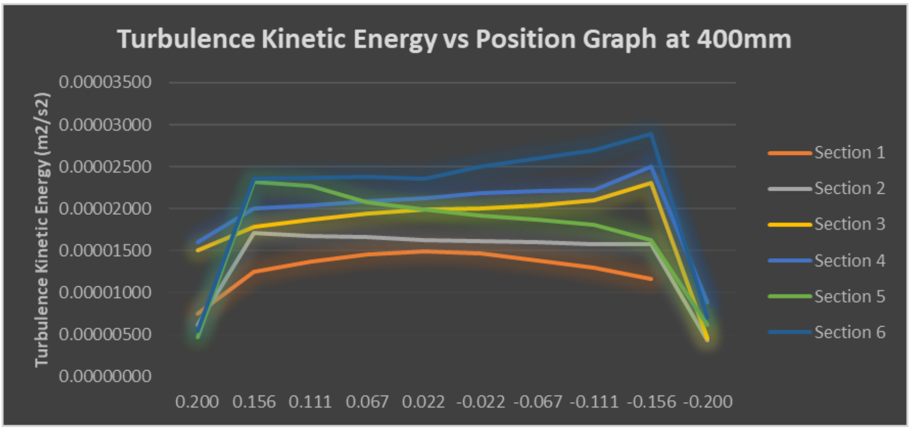

Figure 4.20 Turbulence Kinetic Energy vs Position Graph at 400mm of 6 sections of pipe

Figure 4.16, 4.17, 4.18, 4.19 & 4.20 illustrates the Turbulence kinetic energy of a pipe at all 6 sections. These graphs show energy value plotted in a unit of mass.

4.2 Discussions

The velocity contour of 5 different pipes have shown a pictorial description of each pipe's velocity along the body. In a fixed pressure and flow rate pumping system it calculates that a pipe with bigger diameter will have a lower velocity. While analysing along the pipe geometry it is found that velocity at the inlet of the pipe is highest and as proves the same results. Where all 5 variation of pipe dimensions have calculated value of more than 4000. Reynolds number which is calculated more than 4000 is turbulent.

Turbulence kinetic energy profile is analysed using ANSYS CFD. Turbulence kinetic energy for fluid is considered average or mean kinetic energy per unit mass whereas kinetic energy consideration based on Bernoulli's equation is kinetic energy is calculated per unit volume. In general, the calculation of kinetic energy is needed to see the potential area to best fit an energy harvester or an in-pipe turbine generator. Potential area in the authors opinion is if a pipe has a specific diameter, sizing the energy harvester to the pipe's maximum diameter may not be suitable to maximize the potential harvest of energy that is available.

While observing the turbulence kinetic energy profile of each pipe at all 6 sections as illustrated in Figure 3-3. The highest reading where turbulence kinetic energy is observed is not at the maximum diametrical point of the pipe. The maximum reading is observed at about 70% - 80% reduction in the pipe diameter. That is the point where it is observed as the best diameter to size an energy harvester.

Collectively, simulation was done on pipe with diameter of 200mm, 250mm, 300mm, 350mm and 400mm. The length of the pipe 10m by 10m with 90-degree angle bend. The bend section is equipped with an elbow with a general bend section of 400 mm for all five pipe diameter difference.

Discussing about different pipe dimensions, it must be understood that this study is not about choosing the most suitable pipe size. The reason for that is when simulation was performed or even when literature review was done all pipe of different sizes will produce some amount of velocity as there is a pump present at the initial stage of the system. What matters in this study is at what rate the kinetic energy is for different sizes of pipe. In relation to that, there may be different sizing of energy harvester or and in-pipe turbine generator.

Comparing values obtained through CFD simulation and theoretical calculation. In fixed flow rate system, pipe with smaller dimension will have a higher velocity and as the diameter of the pipe get bigger the velocity will decrease relating to the flow rate, Q formula. When observing turbulence kinetic energy which was obtained from ANSYS FLUENT, the same relation can be made where smaller diameter pipe has a higher turbulence kinetic energy and as the pipe diameter increases the turbulence computed kinetic energy value decrease.

Relating to power generation, there was no simulation done with an in-pipe turbine generator installed together in the pipe. However, there can be seen a potential for an installation of in-pipe turbine. The right size of an in-pipe can be considered based on velocity and turbulence kinetic energy profile. Based on the point where energy is plotted to be high, a proper diametrical sizing must be done to harness maximum energy from the available pipe flow. Power generation is possible if an in-pipe turbine possibly rotates with the flow provided. Assuming rotating is possible, a suitable rotating motion power generator may be applied to make use of the turbine rotation produce power. Calculation may be required based on minimum and maximum RPM that an in-pipe turbine may produce. Calculation is suggested to size a proper generator as generator are not universal and designed specific to case of potential power that can be harnessed.

This study is done with very specific objectives and there is room for further improvement. It is worth discussing a few areas where this study can be improved and recommendation for any further studies. Firstly, since this study was fully done by ANSYS FLUENT there is insufficient data present for validation of simulation data. It is suggested an experimental study is done to validate simulation data. Secondly, sizing matters when it comes to applying the energy harvester into an in-pipe system. Thus, a proper study must be initiated to run an in-pipe energy harvester of different sizes in one pipe system. This also can be used validate findings of simulation data where it is found that towards the far wall region higher energy can be expected compared to the near wall region. Thirdly, is vibration study. Although this part is not related to energy harvesting, but it is worth to consider a vibration study. Turbulence is defined to occur when there is excess kinetic energy. Also, with presence of turbulence flow there will be vibration present in the pipeline. Vibration may not affect the pipe construction or its reinforcement and mounting significantly but the concern is will it affect an in-pipe energy harvester which is installed into the pipeline where high vibration is present. Excess vibration is well known to be an enemy to any mechanical system. A relationship of rate of vibration and it affect to an in-pipe energy harvester in operation is a worthy study.

CHAPTER 5: CONCLUSION

This study aimed at designing, meshing, simulating, and evaluating fluid and flow characteristics in a pipe. A total of 5 different pipe dimension geometry was used. Data obtained was compared and analysed accordingly with respect to the objective of this study. Test report of the simulation is compiled in the appendix section for reference purpose.

Energy harvesting is possible at anywhere energy is present. In the case of in pipe fluid the simple requirement is motion of fluid which will exhibit certain velocity depending its own factors. Kinetic energy will be present at cases where velocity is present as velocity and kinetic energy theoretically related.

Pipe with 5 different diametrical sizes were simulated. All 5 sizes show the potential where energy harvesting is possible. Although rate of energy present will differ according to sizes as pumping capacity is kept constant in this study. Furthermore, there is a need to decide proper sizing of an energy harvester. Reason being the kinetic energy at the maximum pipe diameter is computed to be lower in ANSYS FLUENT. A reduction of 70% - 80% in diameter of energy harvester compared to the dimension of the pipe. This is to maximise the rate of energy harvest to be converted into useful purpose.

The main purpose to harvest energy is to be converted into other forms of energy. Specifically, the harvested energy will be converted into electric and may be able to reduce cost of electricity in the long run if applied effectively. The raw water, treated water and wastewater treatment plant will benefit from this method of energy harvesting. A lot of in-pipe application are being used in all three types of water treatment plant and this study will be useful to promote in pipe energy harvesting in those plants.

REFERENCES

ANSYS Inc. (2011). *ANYSY FLUENT User's guide.* ANSYS.

Cao, H. (2011). Aerodynamics Analysis of Small Horizontal Axis Wind Turbine Blades by Using 2D and 3D CFD Modelling. 13-16.

Cengel, Y., & Cimbala, J. (2006). *Fluid mechanics: Fundamentals and Applications.* Mc Graw Hill.

Langroudia, A. T., Afifia, Z., Nobarib, A. H., & Najafia, A. (2019). Modeling and numerical investigation on multi-objective design improvement of a novel cross-flow lift-based turbine for in-pipe hydro energy harvesting applications. *Energy conversion and management*, 1 - 7.

Malaysia, S. P. (2018). Uniform technical guidelines water reticulation and plumbing. 67-129.

Schlabach, R. A., Cosby, M. R., Kurth, E. J., Palley, I., & Smith, G. (2013). *United States of America Patent No. US 8,360,720 B2.*

Starkl, M., Brunner, N., Flogl, W., & Wimmer, J. (2008). Design of an institutional decision-making process: The case of urban water management. *Environment management*, 1 - 8.