PARTICLE IMPACT PREDICTION OF AN ARCHIMEDES SCREW RUNNER BLADE FOR MICRO HYDRO TURBINE

NURUL SURAYA BT AZAHARI

UNIVERSITI MALAYSIA PAHANG

PARTICLE IMPACT PREDICTION ON AN ARCHIMEDES SCREW RUNNER BLADE FOR MICRO HYDRO TURBINE

NURUL SURAYA BINTI AZAHARI

Report submitted in partial fulfilment of the requirements

for the award of degree of

Bachelor of Mechanical Engineering

Faculty of Mechanical Engineering

UNIVERSITI MALAYSIA PAHANG

JUNE 2013

UNIVERSITI MALAYSIA PAHANG

BORANG PENGESAHAN STATUS TESIS♦

JUDUL: **PARTICLE IMPACT PREDICTION ON AN ARCHIMEDES SCREW RUNNER BLADE FOR MICRO**

SESI PENGAJIAN: 2012/2013

Saya **NURUL SURAYA BT AZAHARI (901104-06-5182)**
(HURUF BESAR)

mengaku membenarkan tesis (Sarjana Muda/~~Sarjana~~ /~~Doktor Falsafah~~)* ini disimpan di Perpustakaan dengan syarat-syarat kegunaan seperti berikut:

1. Tesis adalah hakmilik Universiti Malaysia Pahang (UMP).
2. Perpustakaan dibenarkan membuat salinan untuk tujuan pengajian sahaja.
3. Perpustakaan dibenarkan membuat salinan tesis ini sebagai bahan pertukaran antara institusi pengajian tinggi.
4. **Sila tandakan (√)

[ ]	**SULIT**	(Mengandungi maklumat yang berdarjah keselamatan atau kepentingan Malaysia seperti yang termaktub di dalam AKTA RAHSIA RASMI 1972)
[ ]	**TERHAD**	(Mengandungi maklumat TERHAD yang telah ditentukan oleh organisasi/badan di mana penyelidikan dijalankan)
[√]	**TIDAK TERHAD**	

Disahkan oleh:

______________________	______________________
(TANDATANGAN PENULIS)	(TANDATANGAN PENYELIA)

Alamat Tetap:

NO 25, LORONG 8,
TAMAN KEMPADANG MAKMUR,
26060, KUANTAN,PAHANG.

MUHAMMAD AMMAR NIK MU'TASIM
(Nama Penyelia)

Tarikh: **27 JUN 2013**

Tarikh: : **27 JUN 2013**

CATATAN:
* Potong yang tidak berkenaan.
** Jika tesis ini SULIT atau TERHAD, sila lampirkan surat daripada pihak berkuasa/organisasi berkenaan dengan menyatakan sekali tempoh tesis ini perlu dikelaskan sebagai atau TERHAD.
♦ Tesis dimaksudkan sebagai tesis bagi Ijazah doktor Falsafah dan Sarjana secara Penyelidikan, atau disertasi bagi pengajian secara kerja kursus dan penyelidikan, atau Laporan Projek Sarjana Muda (PSM).

UNIVERSITI MALAYSIA PAHANG

FACULTY OF MECHANICAL ENGINEERING

I certify that the project entitled "Particle Impact Prediction on an Archimedes Screw Runner Blade for Micro Hydro Turbine" is written by Nurul Suraya binti Azahari. I have examined the final copy of this report and in my opinion; it is fully adequate in terms of language standard, and report formatting requirement for the award of the degree of Bachelor of Engineering. I herewith recommend that it be accepted in fulfilment of the requirements for the degree of Bachelor of Mechanical Engineering.

MS AT-TASNEEM BINTI MOHD AMIN

Examiner

Signature

SUPERVISOR'S DECLARATION

I hereby declare that I have checked this project and in my opinion, this project is adequate in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering.

Signature :

Name of Supervisor : MR MUHAMMAD AMMAR BIN NIK MU'TASIM

Position : LECTURER

Date : 26 JUNE 2013

STUDENT'S DECLARATION

I hereby declare that the work in this report is my own except for quotations and summaries which have been duly acknowledged. The project has not been accepted for any degree and is not concurrently submitted for award of other degree.

Signature :

Name : NURUL SURAYA BINTI AZAHARI

ID Number : MC 09009

Date : 26 JUNE 2013

DEDICATION

I specially dedicated this project to my beloved parent, Mr Azahari bin Omar and Mrs Norhayati Binti Hamid, my dearest brothers and sister, Suhairie bin Azahari, Nurul Suhaina bt Azahari and Suhaidi Lukman Hafiz bin Azahari for your love, support and trust along my journey as a student, my supervisor and those who guided and motivated me for this project.

ACKNOWLEDGEMENTS

Appropriation and gratitude to my final year project supervisor, Mr Muhammad Ammar bin Nik Mu'tasim for working hard to instruct me, so I was able to finish my project. Besides, I am thankful for his guidance, encouragement, ideas and constant support in making this research possible. I am grateful for his support and always willing to share his knowledge with me. Besides, I am really grateful for his progressive vision about my training in science, his tolerance of my mistakes and his commitment to my future career.

My sincere thanks also go to all my group mates and members of the staff of Mechanical Engineering Department, UMP, who helped me in many ways such as willing to install the software that I need into my laptop and make my works easier. Many special thanks go to members of focus group for their support, co-operation and ideas sharing during this study.

Last but not least, I acknowledge my sincere indebtedness and gratitude to my lovely parents for their love, dream, support and sacrifice throughout my life. I just cannot find the appropriate words that could describe their kindness for the amazing support. Family's support could be the best catalyst in the world.

ABSTRACT

Energy is one of the important sources in the world and important for developing countries. In rural and remote areas, transmission and distribution of energy generated from fossil fuels can be difficult and expensive and producing renewable energy such as water turbine can locally offer a viable alternative. The subject study is conducted to investigate the flow behaviour of water inside the turbine and predict the impact of particle towards blade surface. For this reasons, computational fluid dynamics (CFD) methods are used. The three-dimensional flow of fluid is numerically analysed using the Navier-Stokes equation with standard k-ε turbulence model by applying some boundaries condition such as steady state flow condition, isentropic flow and isothermal temperature. The numerical results such as velocity streamlines, flow pattern and pressure contour for flow of water entering the blade are compared with the experimental results which obtained by other researches. This study shows that the prediction of particle impact occurs mostly on the entering surface blade and along the leading edge of the screw runner. Any modification on the design of the screw runner blade can be analyse for further study.

ABSTRAK

Tenaga merupakan salah satu sumber yang penting di seluruh dunia dan negara-negara yang sedang membangun. Bagi kawasan luar bandar dan pedalaman, penghantaran dan pembahagian tenaga yang dijana daripada bahan fosil adalah sukar dan penghasilan tenaga yang boleh diperbaharui merupakan salah satu alternatif. Kajian yang dijalankan adalah untuk mengkaji aliran air yang terbentuk di dalam turbin dan meramalkan kesan zarah ke atas permukaan bilah. Pengiraan bendalir dinamik (CFD) merupakan salah satu cara untuk mengkaji aliran air. Aliran cecair tiga dimensi (3D) dianalisis menggunakan persamaan Navier-Stokes dengan menggunakan model k-ε dan mengaplikasi keadaan aliran yang stabil, aplikasi aliran isentropi dan bersuhu malar. Halaju arus, corak aliran dan kontur tekanan untuk aliran air yang memasuki bilah dibandingkan dengan keputusan eksperimen yang dilakukan oleh pengkaji yang terdahulu. Keputusan kajian menunjukkan bahawa kesan zarah kebanyakannya berlaku pada permukaan bilah yang pertama dan sepanjang pinggir utama bilah turbin. Pengubahsuaian pada rekabentuk bilah boleh dianalisa untuk kajian di masa akan datang.

TABLE OF CONTENTS

	Page
TITLE	i
EXAMINER DECLERATION	ii
SUPERVISOR'S DECLARATION	iii
STUDENT'S DECLARATION	iv
DEDICATION	v
ACKNOWLEDGEMENTS	vi
ABSTRACT	vii
ABSTRAK	viii
TABLE OF CONTENTS	ix
LIST OF TABLES	xii
LIST OF FIGURES	xiii
LIST OF SYMBOLS	xv
LIST OF ABBREVIATIONS	xvi
LIST OF APPENDICES	xvii

CHAPTER 1 INTRODUCTION

1.1	Project Background	1
1.2	Problem Statement	2
1.3	Objectives	2
1.4	Scopes of the Project	3

CHAPTER 2 LITERATURE REVIEW

2.1	Micro Hydro Turbine	4
2.2	Design of the Archimedes Screw Blade	4
2.3	Governing Equations	6
2.4	Turbulence Flow Modelling	8
2.5	CFD Simulation	9
2.6	Turbine Blade Surface Erosion	11

CHAPTER 3 METHODOLOGY

3.1	Flow Chart	16
3.2	Design Concept	18
	3.2.1 Concept of Archimedes Screw	18
3.3	Design Phase using Solid works	20
3.4	Simulation Phase using ANSYS-CFX	23
	3.4.1 Setup	23
	3.4.2 Boundary Conditions	24
	3.4.3 Velocity Components	24
	3.4.4 Turbulence Characteristics	24
	3.4.5 Computational Fluid Dynamics Simulation	25
3.5	Validation of flow along the Z axis of the flow path	29

CHAPTER 4 RESULTS AND DISCUSSION

4.1	Introduction	30
4.2	Qualitative Analysis of Particle Flow Prediction	31
	4.2.1 Simulation Process	31
	4.2.2 Velocity Streamlines	32
	4.2.3 Flow Patterns	35
	4.2.4 Pressure Contour	37

CHAPTER 5 CONCLUSION AND RECOMMENDATIONS

5.1 Introduction 41
5.2 Conclusion 41
5.3 Recommendation 42

REFERENCES 43

APPENDICES 45

LIST OF TABLES

Table No.	Title	Page
Table 3.1	The diameter and dimension used	22
Table 3.2	Parameter Design	28

LIST OF FIGURES

Figure No.	Title	Page
2.1	The profile view of two-bladed Archimedes Screw	5
2.2	Side elevation of the Archimedes screw	6
2.3	The dimensionless radial-axial vector plots	10
2.4	The separation of flow in Pelton bucket	11
2.5	Volcanic ash deposition on turbine vanes	13
2.6	Erosion rate variation with impact angle	14
2.7	The streamlines and pressure contour of the fan blade	15
3.1	Flow Chart	17
3.2(a)	Side view of Archimedean screw	19
3.2(b)	Plan view of submerged blade	20
3.3	Design of the screw runner blade with the flow path	22
3.4	Blade design with flow in Solidworks	19
3.5	Schematic diagram of CFD code	23
3.6	Three-dimensional solid model of the helical blade	25
3.7	Method of meshing	26
3.8(a)	Zoom at the inlet velocity of the flow	27
3.8(b)	Zoom at wall of the water flow	27
3.9	Velocity profile of Pahang River	29
4.1	Vessel configuration with screw runner blade	32
4.2	Velocity streamline of water entering the blade	33
4.3	Velocity streamline at the inlet flow	34
4.4	Velocity streamline at the outlet flow	35
4.5	Velocity vector of the overall water flow	36
4.6	Velocity vector of the water at inlet flow	37
4.7	Pressure contour of the turbine blade	38

4.8	Front view pressure contour around the screw blade	39
4.9	Back view pressure contour around the screw blade	40

LIST OF SYMBOLS

ρ	Density
μ	Viscous velocity
Δ	Difference
α	Angle
$^{\circ}$	Degree
L	Length of blade
d	Blade diameter
D_h	Hub diameter
H	Head difference
P	Pitch
m	Number of turn
m/s	Meter per second
kW	Kilo watt
R	Radius
σ	Normal stress
Re	Reynolds number
F_D	Drag force
F_L	Lift force
F_G	Gravity force
F_{AM}	Added mass due to acceleration
k-ε	K-epsilon
k-ω	K-omega
%	Percentage
mm	milimeter
kg/m^3	Kilogram per meter cube
kg/ms	Kilogram per meter second

LIST OF ABBREVIATIONS

RANS	Reynolds Averaged-Navier Stokes
CAD	Computer Aided Design
CFD	Computational Fluid Dynamics
3D	Three dimensional
MHP	Micro hydro power
ANSYS	Analysis System

LIST OF APPENDICES

Appendix	Title	Page
A1	Gantt chart for Final Year Project 1	45
A2	Gantt chart for Final Year Project 2	46
B1	The velocity of Pahang River	47

CHAPTER 1

INTRODUCTION

1.1 PROJECT BACKGROUND

The Archimedes screw are created over 2000 years ago and used to raise the water. As the time goes by, it has been tested and modified for the other uses such as a blade to generate electricity as the diminishing of the energy resources. Historically, this screw was used in irrigation and drainage to lift the water to a higher level were generally powered by human or beast and only in small versions. On the other hand, when used as a hydro turbine, the principle of the Archimedes screw pump is still same, but acts in reverse. The screw pump function from the bottom of the channel and the blade will rotate while at the same time, flow the water through the screw blade and transfer the water to the upper channel.

On the other hand, for micro hydro turbine, the water will enter the screw at the top and the weight of the water will pushes on the helical blade and make the screw to rotate. The rotation can be used to produce the electricity. Basically, these screw runner turbine can work efficiently at head not less than 1 meter and not more than 5 meter, but practically people always used 3 meter maximum diameter. Besides, there are another impact that will cause the blade does not function maximally such as particle impact, acoustic impact, premature failure and etc. Based on (Brada, 1999) , the weight of the water that cover by the channel will make the screw blade rotate.

The usual problem in the screw runner blade for micro hydro turbine is the failure to achieve maximum performance of blade due to the particle impact. Particle impact may reduce the negative pressure spike and reduce energy extraction efficiency, which can affect the pressure on the blade surfaces. The review of this research, for example, was given by (Liu, 2010) mentioned in, the analysis between the screw propellers and horizontal axis turbines in terms of geometry and motion parameters, such as thrust coefficients, shaft/torque coefficients, blade surface distributions and downstream velocity profiles .CFD results verify the connection between the pressure fluctuations and more complex pattern has emerged comprising of the near wall fluid correlated with the actual inflow (Sheard, 2012). Besides, the computational methods is one of the best way to predict the particle trajectories, the areas that the erosion always occurred.

1.2 PROBLEM STATEMENT

Nowadays, the lacking of fuels and climate change issue become serious and one of the best solution to avoid from energy diminished is by using a renewable energy. The renewable energy is comes from natural resources such as sunlight, wind, rain, tides and geothermal heat. A development of strong renewable energy sector would ensure that the future energy security of the country and effects of climate change due the greenhouse gases emission. One of the renewable energy which is micro hydro turbine is always being used to generate energy, but there are such problems that affect the turbine performance, which is due to erosion. There are certain surfaces in the blade that always cause erosion due to particle impact during the inflow of river.

1.3 OBJECTIVES

The objectives of this study are as follow:-

1. To study the flow behaviour of water inside the turbine.
2. To predict the impact of particle towards the surfaces of the blade.

1.4 SCOPES OF THE PROJECT

The project scopes of this study to achieve the objectives are as listed below:-

1. Sketching two concept design of the screw runner blade of micro hydro turbine.
2. Choose the best design and draw the screw runner blade by using CAD software (Solid works).
3. The dimension of the screw runner blade are based on the previous research journal (Muller, 2009).
4. The length of the blade, L=5600mm, the screw angle,α= 23.58^0, the blade diameter d=1040mm,hub diameter D_h=390mm, casing blade diameter D= 1200mm, the head difference H=2240mm, pitch blade P=380mm and helix turns m=14.
5. The CFD-Ansys CFX software will be used to know the analysis result by applying the initial condition which is steady state condition (VA_{in}=VA_{out}) and water velocity of 2.47 m/s.
6. The vector velocity, the velocity streamline and pressure contour is found by set-up all the things in the Ansys software. The blade is assumed as stationary due to the complexity of the software set-up.

CHAPTER 2

LITERATURE REVIEW

2.1 MICRO HYDRO TURBINE

Micro hydro turbines is one of popular renewable energy that have been used nowadays. It may produce about 100 kW power and basically used by home owner or small business owner. The turbine extract energy from the fluid and change it into an energy such as mechanical energy output and usually in rotating form or shaft. The fluid from the outflow usually have energy loss, normally in pressure loss. In general, most turbine have a same problems which is noise, vibration, cavitation, cracking, the reducing of efficiency or performance and at the same time make the blade of the turbine damage. Other than that, the impact on the blade during the turbine's operation can cause the maintenance costs increase.

2.2 DESIGN OF THE ARCHIMEDES SCREW BLADE

Design is one of the important factor in creating a product because from the design, there are a lot of effect especially in the form of efficiency of the product. For example, the design of the screw blade is depends on several factor such as internal parameters and external parameters (Rorres, 2000). The example of internal parameters is including the outer radius of the turbine, the length and the slope while for the external parameters, it is consist of the inner radius, the number of blades, and the pitch of the blades. Figure 2.1 below shows the profile view of segment of two-bladed Archimedes screw.

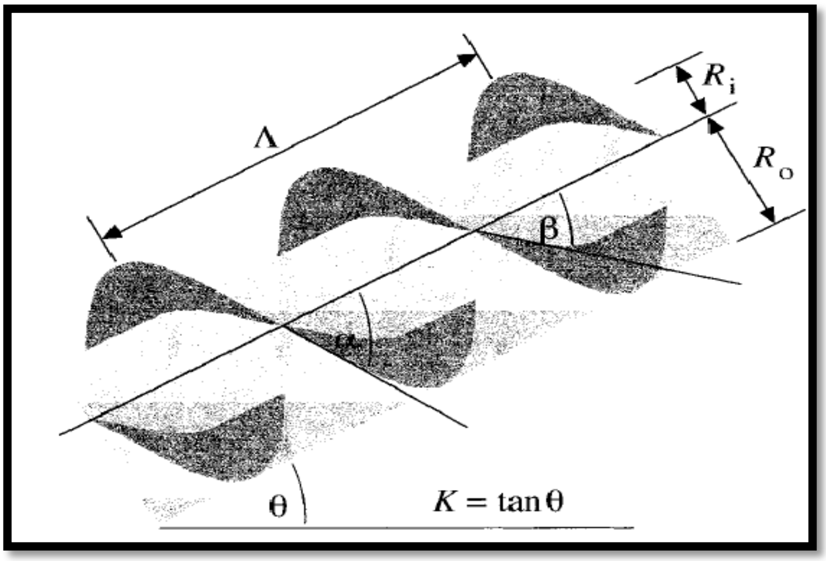


Figure 2.1: The profile view of two-bladed Archimedes Screw

Source: Rorres, 2000

The efficiency of the Archimedes screws is depends on the function of geometry and losses (Müller, 2009). The screw geometry for each blade can function as leakage losses and blade geometry. As the head difference between turns decrease, then the efficiency will increase. The downstream pressure which acting normally on the blade is smaller compared to the upstream pressure meanwhile the hydrostatic pressure and horizontal screw velocity can generate a power. The model of screw runner blade have a small diameter of hub, the submerged depth must smaller than radius R and the gap losses is assumed ignored. Figure 2.2 shows the side elevation of the Archimedes screw.

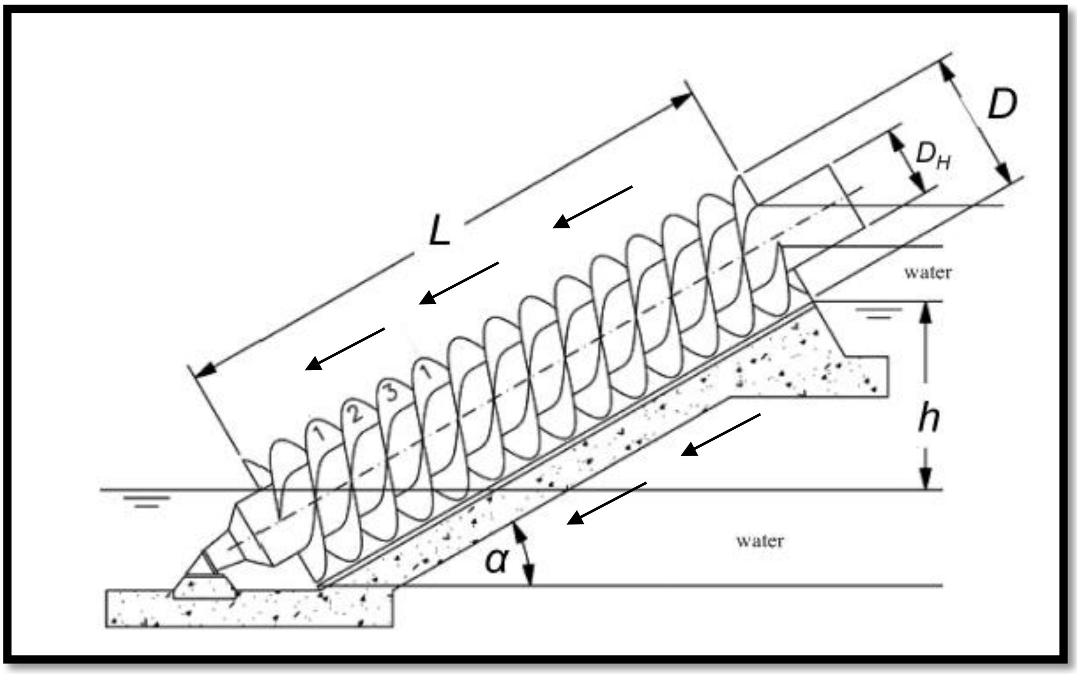


Figure 2.2: Side elevation of the Archimedes screw

Source: Möller, 2009

2.3 GOVERNING EQUATIONS

The fluid flow is being analysed by concern on conservation laws: conservation of mass, conservation of momentum and the equation of the particle impact.

The conservation of mass concept states that the increasing rate of mass inside the element is equal to the mass outside the element across its faces.

The integral equation for the conservation of mass is as followed:

$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}=0 \tag{2.1}$$

The conservation of momentum is based on the Newton's second law, which states that the fluid particle acceleration is related to the sum forces acting on the fluid particle. The flow field is defined in three components (u,v and w) or in generally x,y and z components.

The integral equation of momentum conservation for x,y and z direction are shown as in Eq. (2.2), Eq. (2.3) and Eq. (2.4) respectively :

Momentum for x-direction:

$$\rho\frac{Du}{Dt}=\frac{\partial\sigma_{xx}}{\partial x}+\frac{\partial\tau_{yx}}{\partial y}+\frac{\partial\tau_{zx}}{\partial z}+\sum F_x^{bodyforce} \tag{2.2}$$

Momentum for y-direction:

$$\rho\frac{Du}{Dt}=\frac{\partial\sigma_{xy}}{\partial x}+\frac{\partial\tau_{yy}}{\partial y}+\frac{\partial\tau_{zy}}{\partial z}+\sum F_y^{bodyforce} \tag{2.3}$$

Momentum for z-direction:

$$\rho\frac{Du}{Dt}=\frac{\partial\sigma_{xz}}{\partial x}+\frac{\partial\tau_{yx}}{\partial y}+\frac{\partial\tau_{zz}}{\partial z}+\sum F_z^{bodyforce} \tag{2.4}$$

The normal stresses σ_{xx}, σ_{yy} and σ_{zz} were due to the combination of pressure and normal viscous stress components τ_{xx}, τ_{yy} and τ_{zz} acting perpendicular to the control volume.

The particle move through the blade and follow the flow of the water. The forces that acting on the wall due to particles, have difference and own velocity. The particle usually have a sphere shape and mostly, the particle-particle interactions are ignored cause by low particle concentrations experienced.

The equation of motion is expressed in Eq. (2.5):

$$\rho_p V_p \frac{d\bar{v}_p}{dt} = \bar{F}_D + \bar{F}_L + \bar{F}_G + \bar{F}_{AM} \tag{2.5}$$

Based on Eq. (2.5) where F_D is the drag force, F_L is the lift force, F_G is gravity force and F_{AM} is added mass due to acceleration of neighbouring fluid.

2.4 TURBULENCE FLOW MODELLING

Turbulence flow is a most of a fluid flow that always occurred around us. For example, the flow around the bodies of the cars, aeroplanes and buildings are turbulent. The flow is considered turbulent when consists of irregular, diffusivity, large Reynolds Numbers and dissipative. There are several famous turbulence models that are always used by the previous researcher such as two-equation k-ω model of Wilcox, the two-equation of k-ε model of Launder and Sharma, the two-equation k-ω/k-ε SST model of Menter and one-equation model of Spalart and Allmaras (Bardina, 1997).

The choice of numerical methods are important to predict the turbulent kinetic energy. The k-ε and RNG models is used due to convergence difficulties related to Reynolds stress model and have a little effect on the mean flow and turbulent kinetic energy (Aubin, 2004). Every turbulence model have own specialities based on the complexity. Previous researcher (Gartmann, 2011) states that the characteristics of the channel flow properties is observed in portable wind tunnel with rainfall simulation by using Reynolds-averaged Navier-Stokes (RANS) k-ε turbulence model.

The standard k-ε turbulence model equation are expressed as in Eq (2.4) and (2.5) below:

$$k=\frac{(v_O)^2}{\sqrt{c_\mu}}(l_m\frac{dv_R}{dy})^2 \tag{2.4}$$

$$\varepsilon=\frac{(v_O)^4}{\sqrt{c_\mu}}(l_m\frac{dv_R}{dy})^3 \tag{2.5}$$

Based on Eq. (2.4) and Eq. (2.5) above where V_R is the velocity of neighbour to wall and y is the distance of the adjacent node to the solid wall.

2.5 CFD SIMULATION

Computational Fluid Dynamics is a solver that is used to illustrate the computational study. From the simulation, several factor can be determined and can be compared with the experiment data. The advantages of using the simulation is the researcher can study more on complex problem that is impossible to be done experimentally and other than that, the cost to do the product is also decrease as the researcher just need to set-up the simulation. The limitation of the CFD software is this simulation cannot be used to confirm the accuracy of the simulation is succeed. The results from simulation need to be validate with the suitable and relevant experimental data.

The simulation of the three dimensional distribution of solid particle in turbulent liquid flow with k-ε turbulence model have been used by (Micale, 2000) to study a simple settling velocity model and assume the particles are transported as a passive scalar. The increase in particle drag coefficient due to turbulent flow must be accounted to give a good agreement between simulation and experimental data.

The modelling approach, turbulence model and numerical scheme is the factor that influence the modelling turbulent flow in stirred tank (Aubin, 2004). The vector plot have been used to present the results as shown in Figure 2.3. The CFD modelling approaches predict the flow patterns which have good agreement, quantitatively and qualitatively with the experimental data.

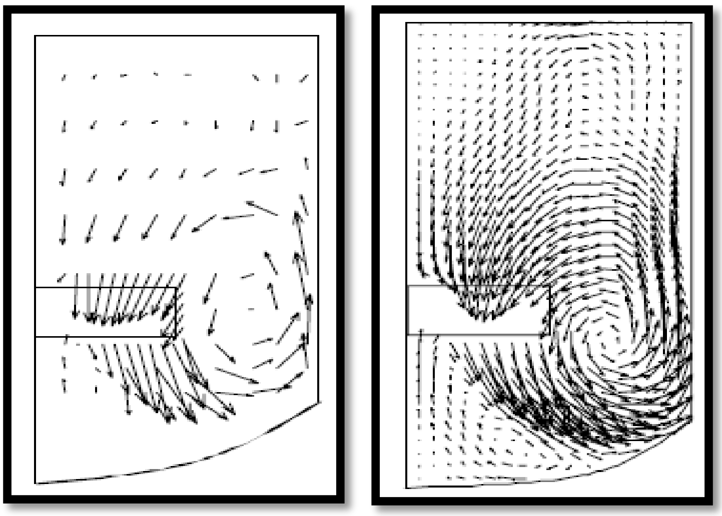


Figure 2.3: The dimensionless radial-axial vector plots

Source: Aubin, 2004

2.6 TURBINE BLADE SURFACE EROSION

The normal particle acceleration will separate the particle from the direction of the flow. Most of the accelerating particle will strikes the first surface of the turbine. The size of particle may affect the surface of the blade. As stated by (Thapa, 2004), the erosion will occurred most on the needle if the particle are fine, more erosion in bucket if the particle is course and for the medium particle, then both bucket and needle are equally have eroded area. Figure 2.4 indicates the separation of particle in a Pelton bucket. The red-dot line is shows the affect due to the larger moving particle, while the black-dot line indicate the smaller particle impact to the blade. The blue line illustrate the original of water surface.

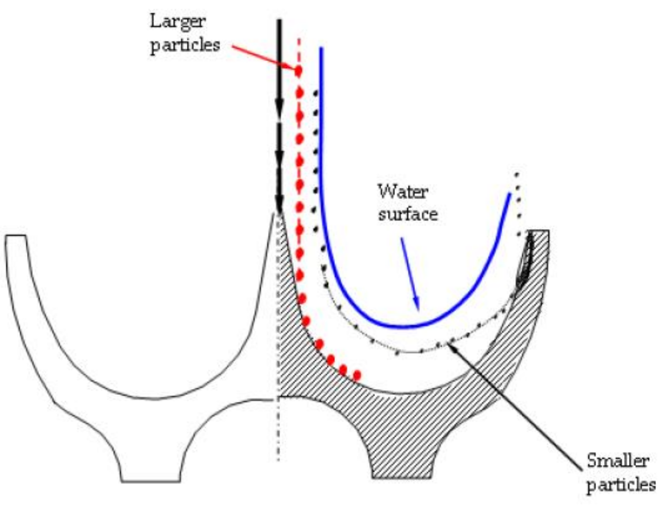


Figure 2.4: The separation of flow in Pelton bucket

Source: Thapa, 2004

The dynamic action of sediment of flowing water can cause the sediment erosion to the turbine. The particle dynamics simulation indicates that many particles hit the vane pressure surface which the larger particles across over and impact the vane suction surface towards trailing edge (Tabakoff, 2005). Besides, the trajectories shows that the impact of particle always occurred at the pressure surface and part of the suction surfaces. Besides, the blade surface that was hit by some particles will diverge from flow due to high inertia.

The parameter effect the losses of surface material as consequences of erosion (Hamed, 2006). For example, at the engine inlet engine, the aircrafts standing or moving can blow sand, ash, dust and other particles that enter into the engine and erosion can occurred. Test that have been done at University of Cincinnati showed that the quartz sand is less erosive than volcanic ash. Figure 2.5 below shows the volcanic ash deposition on turbine vanes.

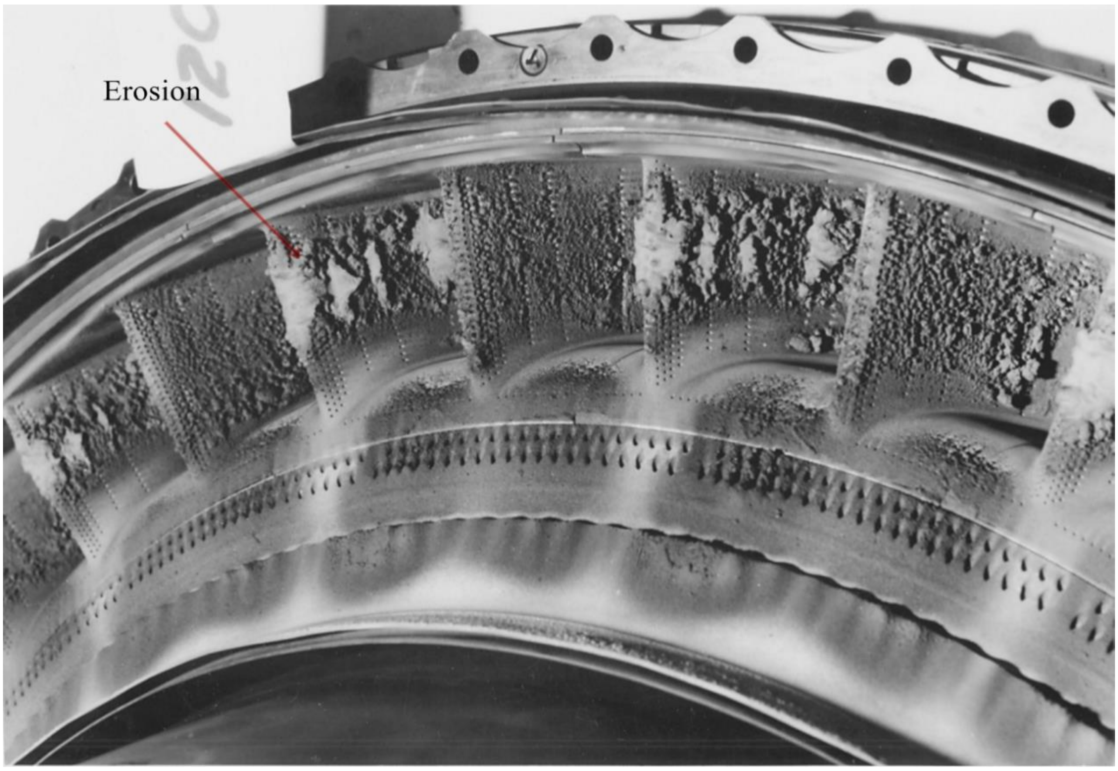


Figure 2.5: Volcanic ash deposition on turbine vanes

Source: Hamed, 2006

The ductile and brittle material have a different erosion rate with impact angle due to the predominantly different mechanism of cutting and brittle material. For ductile case as shown in Figure 2.6, the impacting particle cause the material surface become rough but for brittle case, although there are impacting material on the surface material, the surface is not too rough.

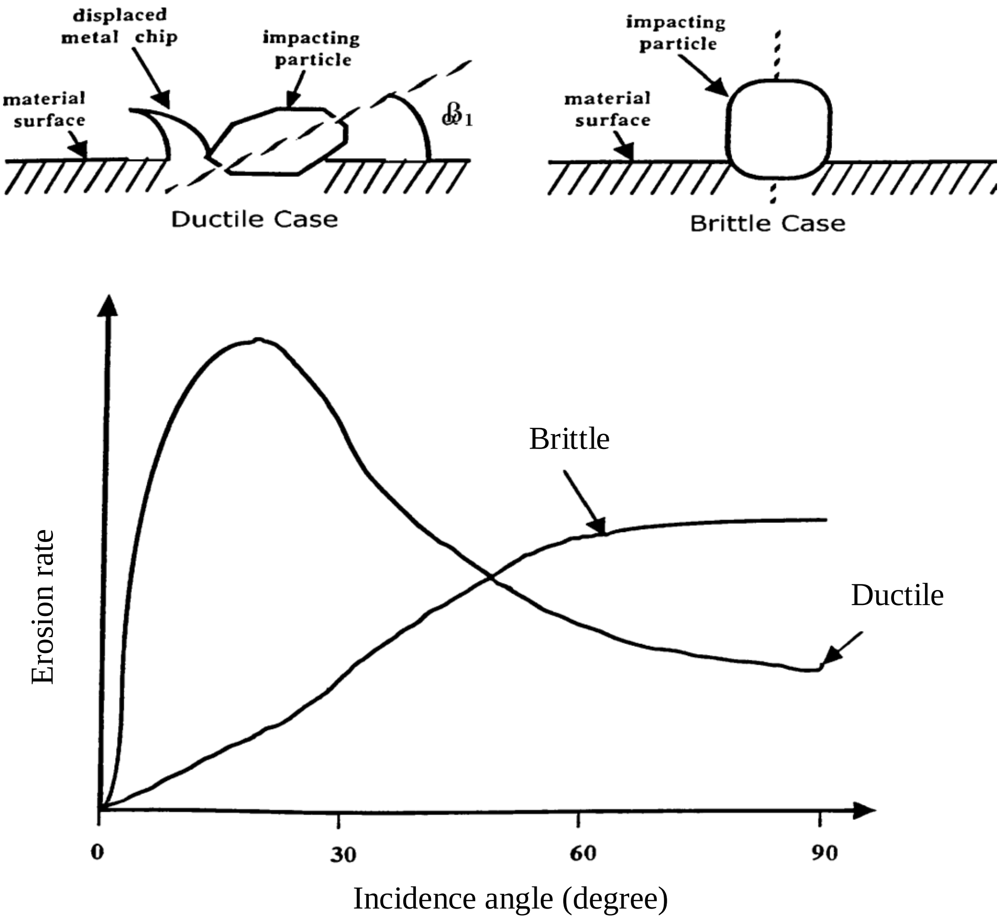


Figure 2.6: Erosion rate variation with impact angle

Source: Hamed, 2006

The erosion pattern is depends on particle impact velocity and also drop in fan efficiency and pressure rise coefficient due to blade leading edge and pressure side at the corner blade (Sheard, 2012). Figure 2.7 indicates the streamlines and pressure contour of the fan blade. The streamline indicates the velocity at the wall with forward and backward from a seeding location while the pressure contour specify the line of the pressure for the detailed view.

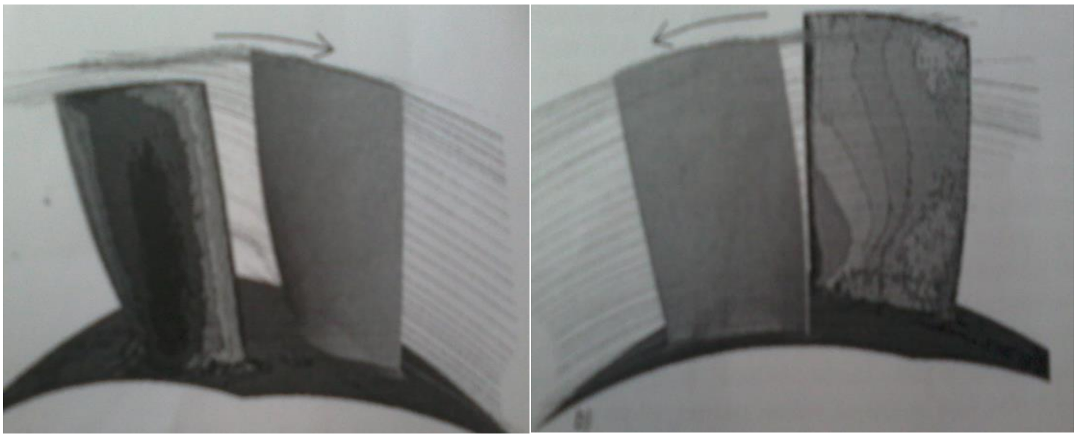


Figure 2.7: The streamlines and pressure contour of the fan blade

Source: Sheard, 2012

Particle transport through the rotor also influenced the streamlines and pressure contour of the blade. The complex pattern is seen near the wall fluid caused by the actual inflows incidence angle. Both figure above shows the existing of leading edges separation that radially outward. The blunting of the leading edge thickness on the pressure and suction sides results an erosion and the pressure side surface is larger than the suction side.

CHAPTER 3

METHODOLOGY

3.1 FLOW CHART

The flow chart of methodology is designed to show the step by step of the processes in a systematic and proper way and used in analysing, designing, documenting and handling a process in a many fields. The flow chart is important in a way to achieve a project's objectives as it visualizes what is going on and at the same time find the flaws. Figure 3.1 shows the flow chart of this project from the initial starting to the end of the processes. The project is started with finding all the information regarding to the impact on screw runner blade, then the problem that always occurred to the blade surface and the objectives on how to solve the problem happened. The project starts with the literature review and some research from the previous researcher as a research study. From the literature review, the author make a decision on a methods that are suitable to be applied on the project. Firstly, the author sketch the design of the screw runner blade and draw the design by using one of the Computational Aided Design (CAD) software, SolidWorks. Secondly, the author run the simulation by applying some boundary conditions. The results from the simulation is analyse according towards objectives. If the results is converged, then the future and detailed analysis is made and otherwise, the design of the screw runner blade should be considered again until the results is converged. During these work in progress, the thesis writing is written parallels to the work done.

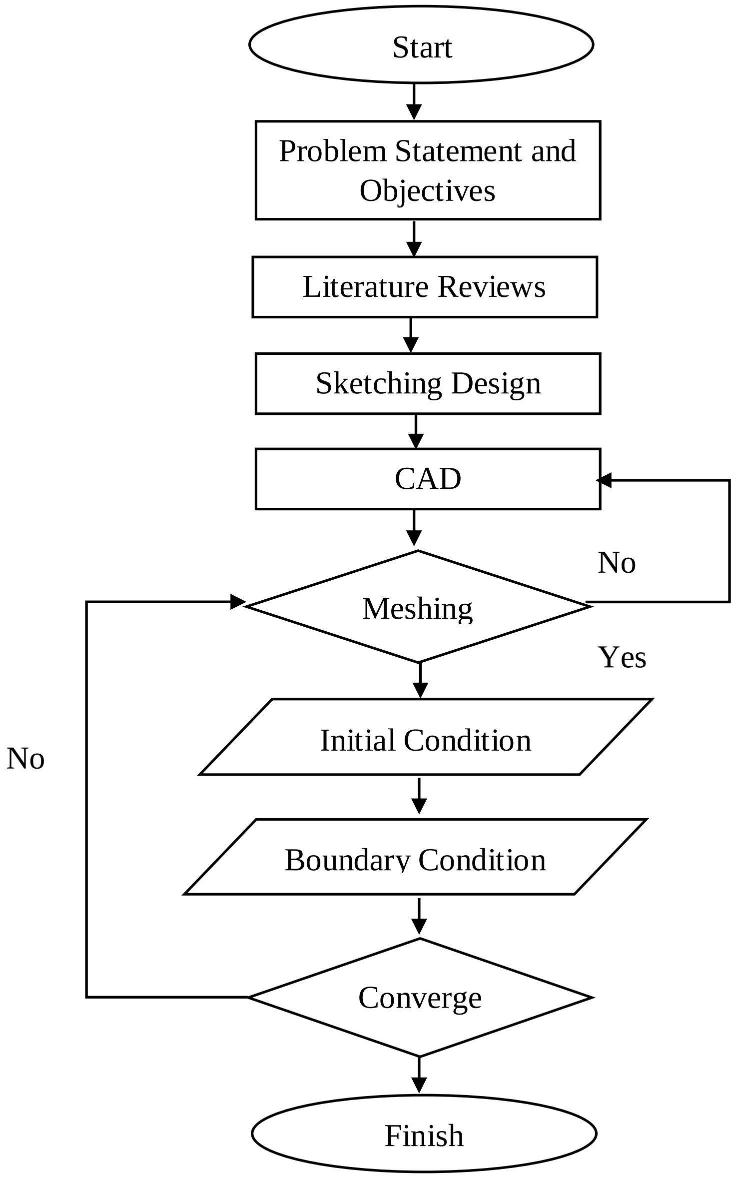


Figure 3.1: Flow Chart

3.2 DESIGN CONCEPT

Design concept is the basic step before the design can be created. It will become a direction of a design and not out of the scope. The concept will lead and the design is created. There are two types of design concept such as verbal and visual. Verbal means the focus on the design to communicate with people. While, the visual is the specific one.

3.2.1 Concept of Archimedean screw

The design of the screw runner blade is similar with the concept design of Archimedean screw. The Archimedean screw pump is one of the oldest hydraulic machines but nowadays, it have been develop as a screw runner turbine which function as energy converter. The water weight enclosed is assume to drive the screw blades (Brada, 1999). If there are no losses, the theoretical efficiency can achieved the maximum efficiency of 100%. The Figure 3.2 (a) and Figure 3.2 (b) below shows the idealised Archimedean screw from the side view and plan view of submerged blade respectively. Figure 3.2 (a) shows the level of water inside the screw blade and horizontal hydrostatics force that developed due to the head difference.

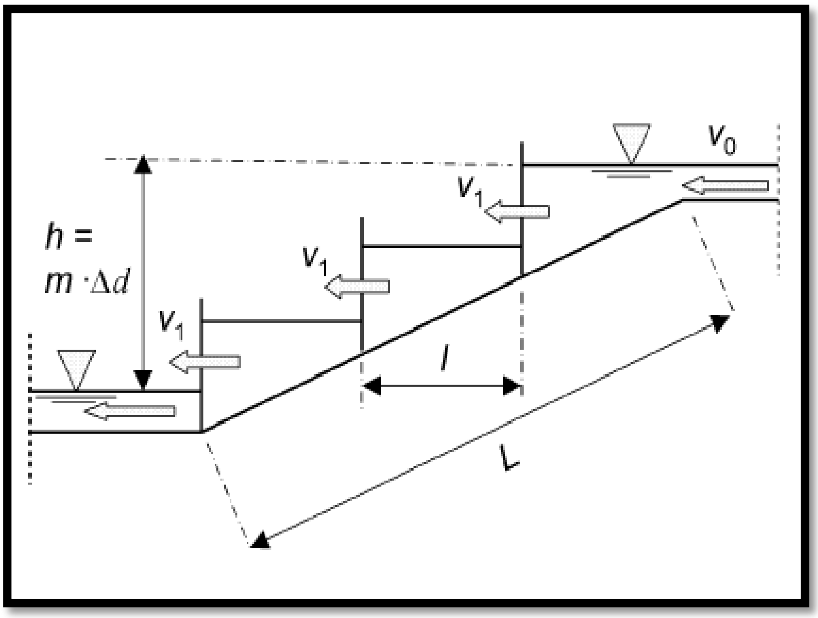


Figure 3.2 (a): Side view of Archimedean screw

Source: Mϋller, 2010

Figure 3.2 (b) illustrate the hydrostatics pressures on the incline blade that have upstream pressure larger than the downstream pressure acting normally on the blades.

The formula of flow velocity, v_1 to the screw, which is smaller than the entry velocity,v_0 are:-

$$v_1 = \frac{d_0}{d_0 + \Delta d} v_0 \tag{3.1}$$

Based on Eq. (3.1) where v_1 is the flow velocity, d_0 is the inflow depth, Δd is the increase of water depth and v_0 is the entry velocity.

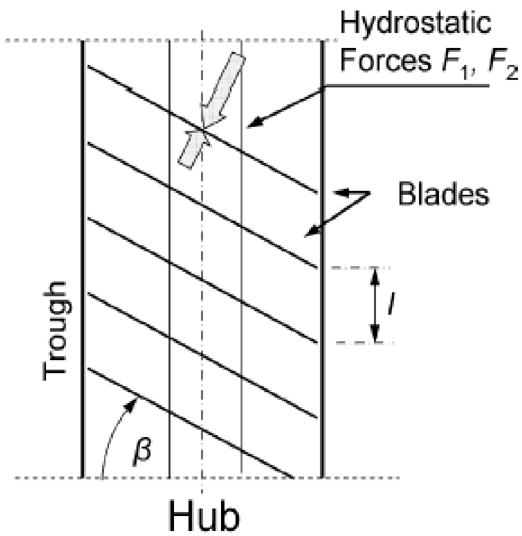


Figure 3.2 (b): Plan view of submerged blade

Source: Möller, 2010

3.3 DESIGN PHASE USING SOLIDWORKS

The design or screw runner blade is drawn by using Computational Aided Design (CAD). CAD is one of the best software to design a model as have many advantages such as increase the productivity of the designer, improve the quality design, creating database for manufacturing and better communication as it have standard criteria and widely used. In this study, the author use SolidWorks 2012 version to draw the screw runner blade with the suitable dimension. The author make a previous researcher blade dimension as the guidance. Figure 3.3 shows the blade design by using a SolidWorks software.

From Figure 3.3 below, the design is drawn by using SolidWorks. The first step is the author create a new sheet, then choose front plane. The other instruction are by using the boss-extrude to draw the path flow, the cut-sweep to draw the helical blade, the sweep and cut loft to create a tapering at the end of the blade. The drawing and the dimension of the blade is listed in Figure 3.4 and Table 3.1 respectively.

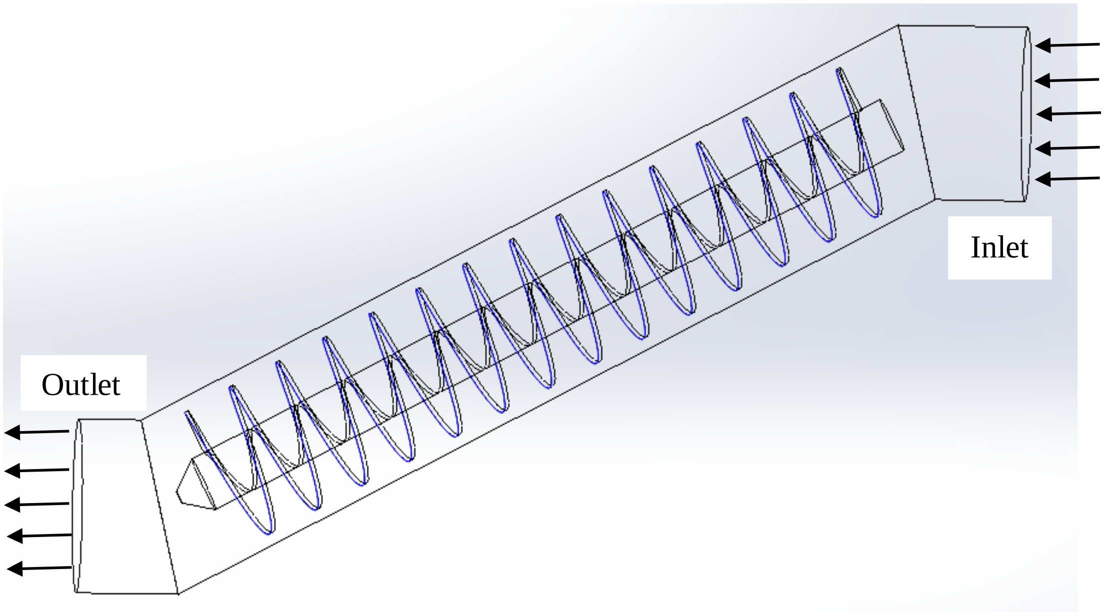


Figure 3.3: Design of the screw runner blade with the flow path

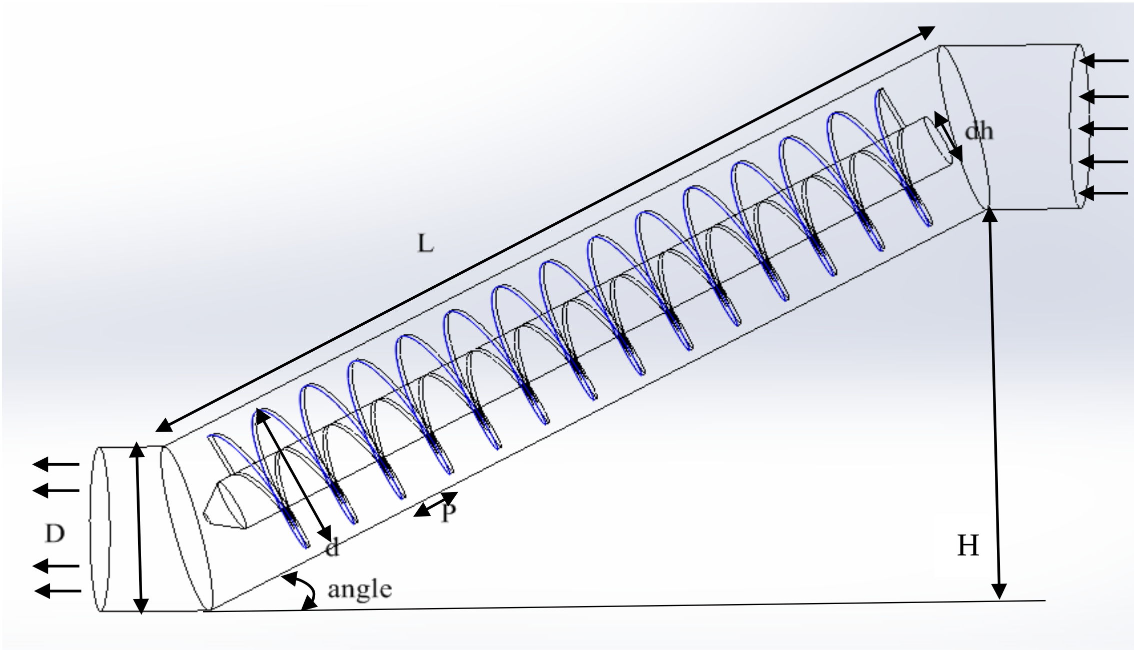


Figure 3.4: Blade design with flow in Solidworks

Table 3.1 below indicates the parameter and dimension used by the author. The author makes the dimension referring from the previous researcher.

Table 3.1: The diameter and dimension used

Parameters	**Dimension**
Length, L	5600 mm
Angle, α	23.58^0
Casing diameter, D	1200 mm
Blade diameter, d	1040 mm
Hub diameter, D_h	390 mm
Head difference, H	2240 mm
Helix turns, m	14 turns
Pitch, P	380 mm

3.4 SIMULATION PHASE USING ANSYS-CFX

3.4.1 Setup

The physical characteristics of fluid motion can be described through fundamental of mathematical equations or called governing equation in Computational Fluid Dynamics (CFD). The computer programs executed a high speed digital computers to get the numerical solutions. Besides, the CFD is one of the method for the industries to solve problems in fluid dynamics and heat transfer. From the simulation, there are several factor can be determined and compared with the experimental data and at the same time, can reduce the experimental costs. The CFD code have three main elements such as pre-processor, processor and post-processor (Figure 3.5).

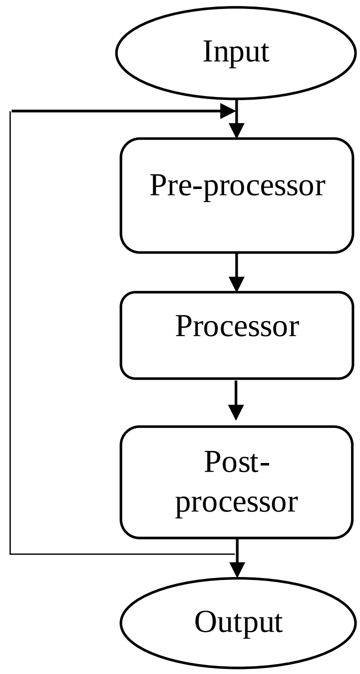


Figure 3.5: Schematic diagram of CFD code

3.4.2 Boundary Conditions

The initial boundary condition have been used in this simulation. The condition of the flow is a steady state flow which have velocity inlet is equal to velocity outlet. 2.47 m/s constant inlet velocity was used as a boundary condition and no-slip wall is applied due to the fluid velocity at the wall point is zero because the existence of the viscosity.

3.4.3 Velocity Components

The condition of no-slip condition is suggested on all rigid wall. The velocity components mesh nodes which intersect with turbine, is assume equal to zero. The condition are as below:-

U (I, J, K) = 0

V (I, J, K) = 0

W (I, J, K) = 0

3.4.4 Turbulence Characteristics

The value of k and ε was connected when the flow is near to the wall. The equation that is related to the k-ε turbulence model are as follow:-

$$k = \frac{(v_o)^2}{\sqrt{C_\mu}}(l_m \frac{d v_R}{dy})^2 \tag{3.2}$$

$$\varepsilon = \frac{(v_o)^4}{\sqrt{c_\mu}}(l_m)(\frac{d\,v_R}{dy})^3 \tag{3.3}$$

Based on Eq. (3.2) and Eq. (3.3) above where V_R is the velocity of neighbour to wall and y is the distance of the adjacent node to the solid wall.

3.4.5 Computational Fluid Dynamics simulation

The pre-processor stage involve the geometry of the design blade, the mesh generation and the specification of the boundary conditions. The nodes is described to separate the solid domain from the liquid domain. The mesh in the flow domain is automatically generated for the three-dimensional simulations as shown in Figure 3.6.

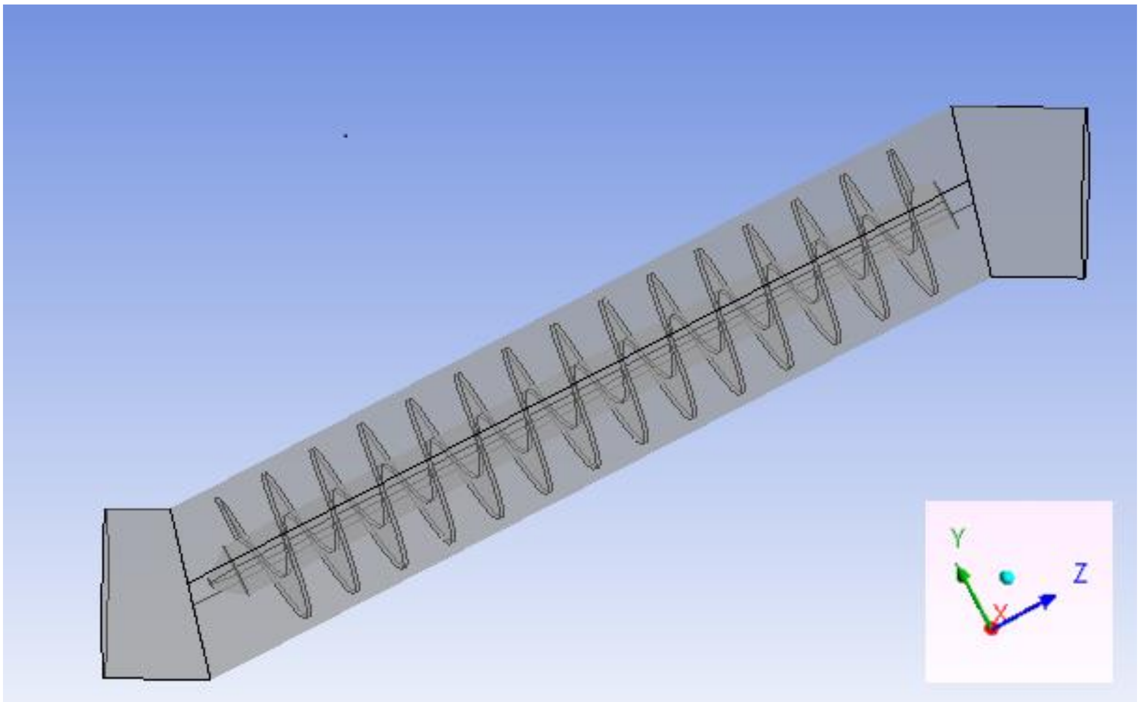


Figure 3.6: Three-dimensional solid model of the helical blade

Unstructured mesh with tetragonal elements is used with minimum edge length of 0.003m. Grid is shown in Figure 3.7.

The mesh consists about 945048 linear tetrahedral elements with 179690 nodes. The mesh size influence the accuracy of the results and smaller meshing is selected.

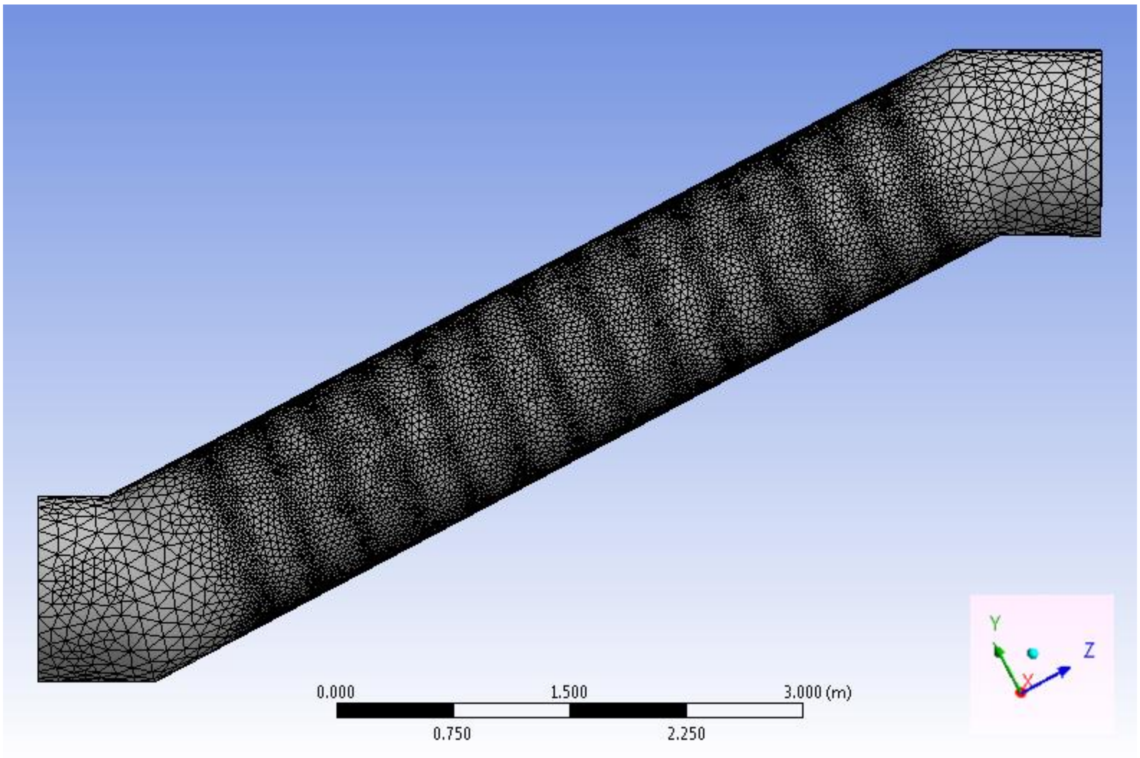


Figure 3.7: Method of meshing

The Figure 3.8 (a) and 3.8 (b) indicates the meshing of the fluid flow. The Figure 3.8 (a) is the meshing of flow at the inlet. The size of meshing at the inlet boundary without the existent of blade is bigger. Meanwhile, for Figure 3.8 (b) shows the meshing size of the fluid that located at the wall with the existing of the blade is quite tiny.

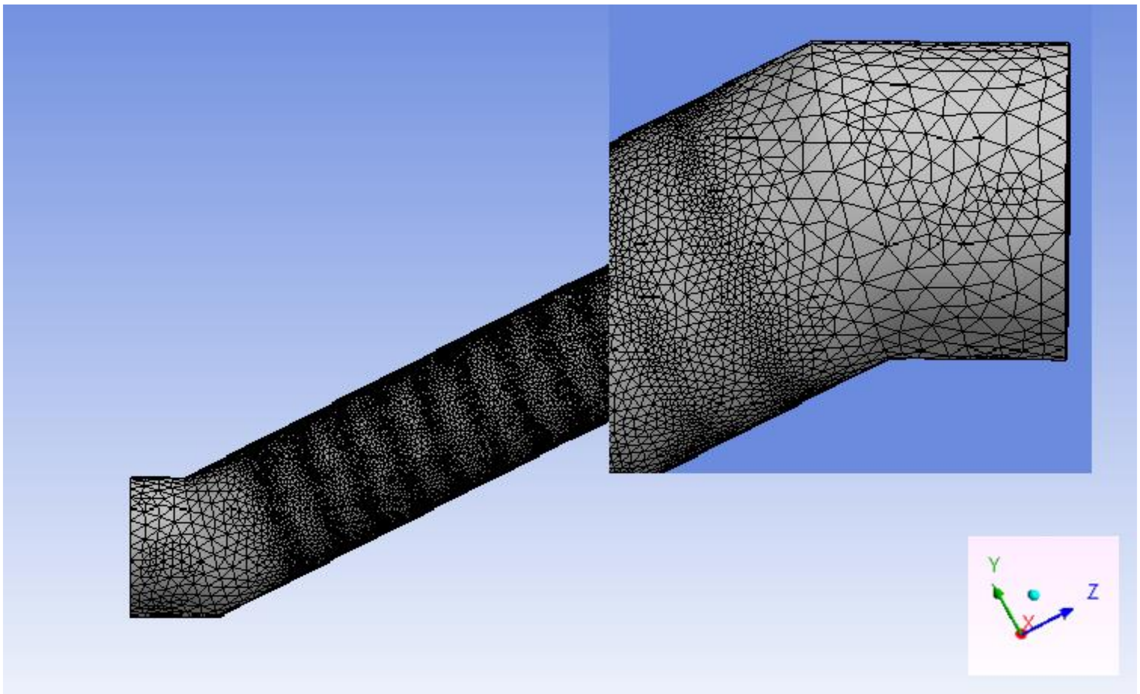


Figure 3.8 (a): Zoom at the inlet velocity of the flow

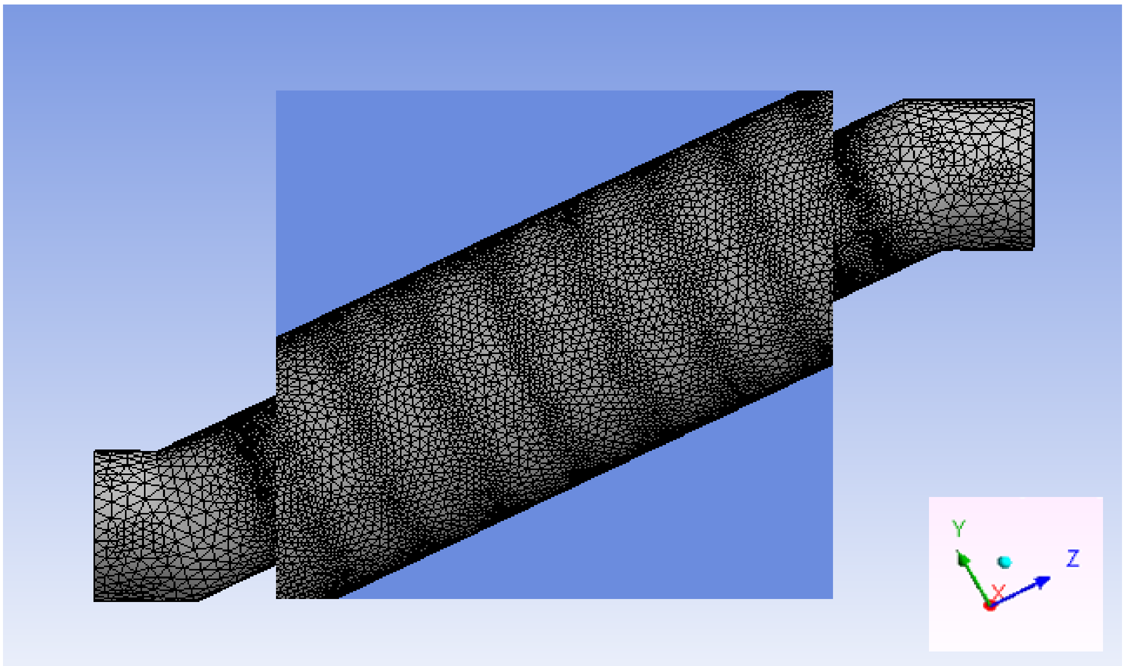


Figure 3.8 (b): Zoom at wall of the water flow

The solution of Navier-Stokes equation in conjunction with standard k-ε turbulence model are developed using a control volume discretization method. The standard k-ε turbulence model is used to expect the flow because of its robustness, but this model also have the weakness which is not good to predict the complex flow.

The post-processor is used to display the results of the simulation, which contain geometry domain, grid display, vector plots, velocity streamline, contour pressure, plane pressure and etc. All the display can be used to manipulate and analysed. Table 3.2 below shows the parameter design.

Table 3.2: Parameter Design

Variables	**Specifications**
Material	Water
Pressure	0 Pa
Domain motion	Stationary
Heat transfer	Isothermal
Fluid temperature	25^0C
Turbulence model	Standard k-ε model
Wall function	Scalable
Turbulence intensity	5%
Inlet velocity	2.47 m/s
Density	997 kg/m^3
Dynamic viscosity	8.8990e-04 kg/m.s

The screw blade consists of 14 helical blade. The water is supplied to the inlet and will flowing inside the casing of the blade by using a velocity of 2.47 m/s. The weight of the water will make the helical screw blade to rotate, but in this case, the author setup the blade as a stationary blade due to the complexity of the software set up. The subject study is conducted to investigate the flow behaviour of water when the water is entering the blade and predict the blade pressure surface.

3.5 VALIDATION OF FLOW ALONG THE Z AXIS OF THE FLOW PATH

Figure 3.9 show the velocity and distance from the z axis of turbine flow with inlet velocity of 2.47 m/s which is based on the velocity of Pahang River. The velocity increasing as the distance at z axis increase due to the complexity of the flow throughout the water. The existing of the blade also effect the flow path of the water velocity. The flow will be messier with the present of the blade. The graph shows that the velocity is constantly increase but drop at distance of 4.75m. This is caused maybe due to some leakage.

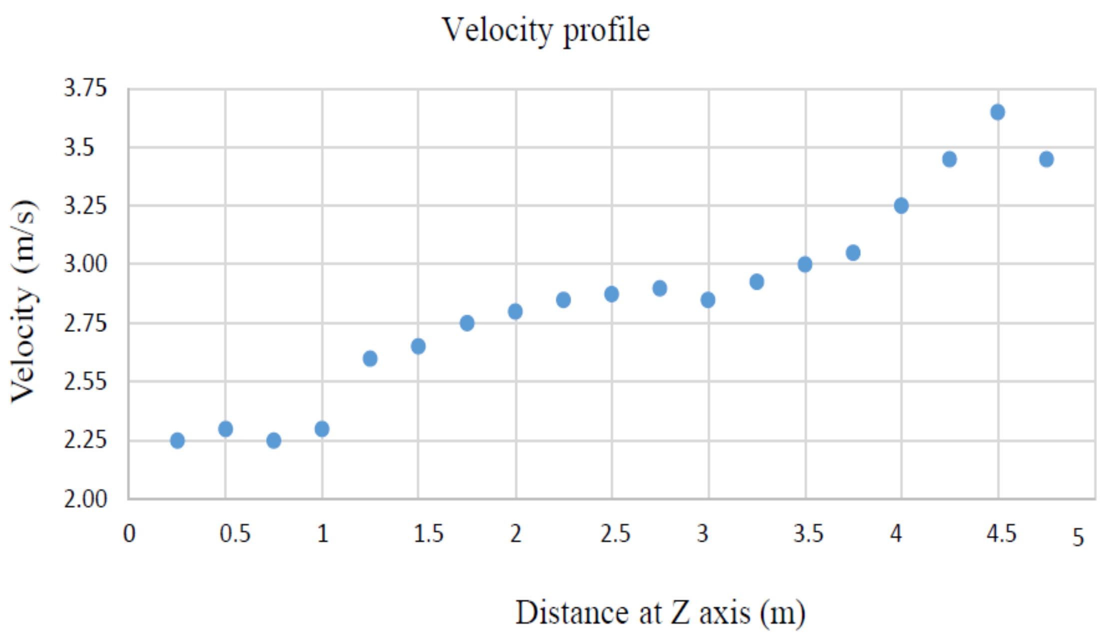


Figure 3.9: Velocity profile at Pahang River

CHAPTER 4

RESULTS AND DISCUSSIONS

4.1 INTRODUCTION

The application of computational flow modelling tools to simulate the flow created by screw runner blade turbine is discussed in details. The three dimensional flow of fluid is numerically analysed using some equation continuity equation and incompressible Navier-Stokes equation. The turbulence model used is standard k-ε model because of its economy, robustness, and reasonable accuracy of wide range of flows. The computer results give a several information regarding to the turbulent flow in a tank. From the simulation, the results that the author get are the velocity streamline pattern, the viscous dissipation rate, the turbulent viscosity rate and the pressure contour of the blade. All of these results give a better understanding and concept of hydrodynamics mechanism. The pressure contour of the blade have been compared with the result of previous researcher.

4.2 QUALITATIVE ANALYSIS OF PARTICLE FLOW PREDICTION

Simulation on the turbulent flow is already investigated by the previous researcher. For example, the main aim of Micale (2000) are three dimensional distribution of solid particles in the turbulent liquid flow of an impeller blade and have shown that the increase in particle drag coefficient due to turbulent flow must be considered to give a good simulation of experimental data. The grid type and turbulent flow also affect the numerical prediction of the flow around the marine propellers as hexa-structured meshes give a more accurate results compared to hybrid-unstructured meshes, Morgut (2012).

4.2.1 Simulation Process

The turbine in this study is a tank with a 14 helical screw blade. The tank is being modelled with angle of 23.58^0 from horizontal axis and have head difference of 2240 mm. Figure 4.1 below shows the vessel configuration with screw blade turbine.

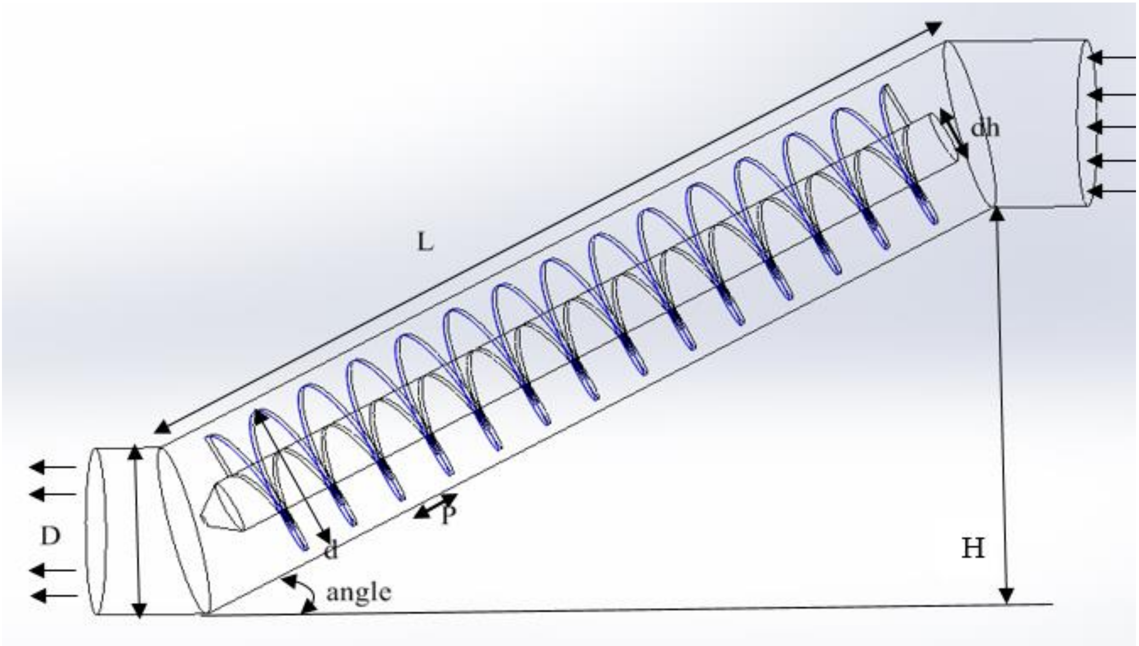


Figure 4.1: Vessel configuration with screw runner blade

4.2.2 Velocity streamlines

In this study, the author considered the flow behaviour with the pressure surface. The simulation is done using a standard design of helical blade that consists of 14 helical blade which have about 5600 mm length and pitch of 380 mm. The water is inflowing the inlet turbine with velocity 2.47 m/s with a steady state flow. The streamlines are field line that are instantaneous tangent to the velocity vector of the flow. In this case, the streamline indicate the flow behaviour of water which go through the turbine and follow the path of the blade. The Reynolds number based on the diameter is 4.3971×10^5 and verify that the author ran simulation for normal condition. The results from the modelling of water with 25 points of streamline are shown as velocity vectors in Figure 4.2 below. There are 25 streamlines that are being analysed for 14 helical blade. The streamlines is traced by a moving particle, at every point along the path and the mass does not cross the streamlines. The pattern of flow is seen smooth according to there are no disturbance but become rapidly mixed as the flow of water go through the blade.

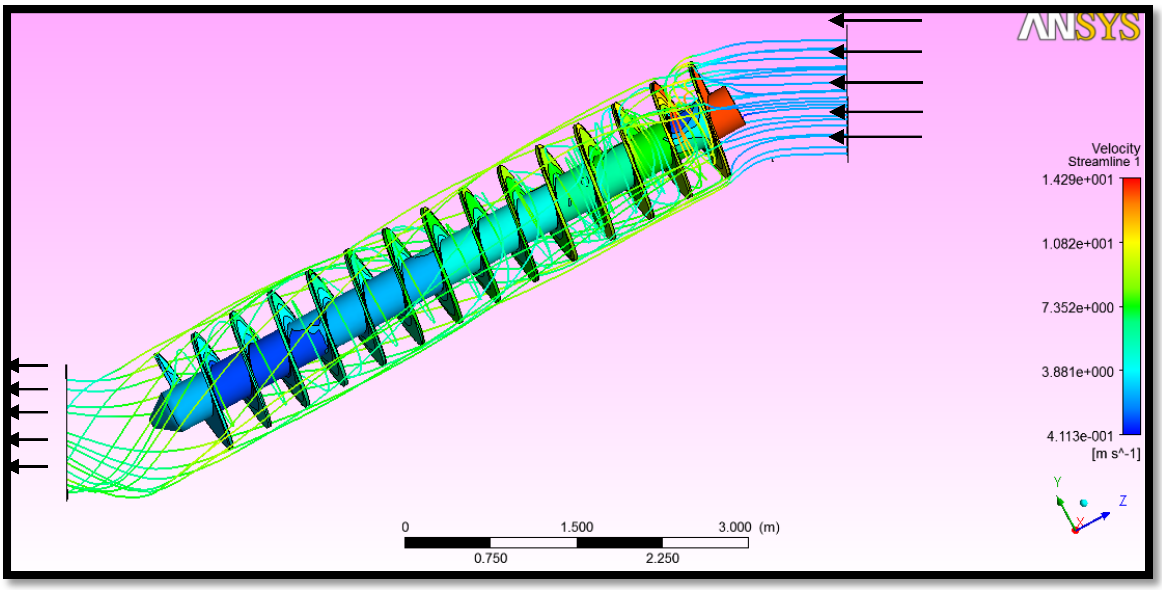


Figure 4.2: Velocity streamline of water entering the blade

Figure 4.3 below indicates the velocity streamlines at the inlet of the turbine. As seen, the velocity streamlines is lowest at the inlet and smooth because the flow is not being disturbed yet by the surface of blade and the kinetic forces tend to maintain the flow in the inlet direction. The situation is based on the continuity equation which have the volume flow of the inlet is equal to the volume of outlet if there are no disturbed flow of the surrounding field. The highest velocity streamlines occurred at the backside of the first blade. This is due to the instability of the flow and from the Reynolds, the instability of the flow is predicted in term of relative magnitudes of velocity and the viscous force acts.

The equation of the Reynolds number:

$$\mathrm{Re} = \frac{\rho v D}{\mu} \tag{4.1}$$

Based Eq (4.1) where Re is the Reynolds number, ρ is the density, v is the velocity, D is the diameter and μ is the dynamic viscosity.

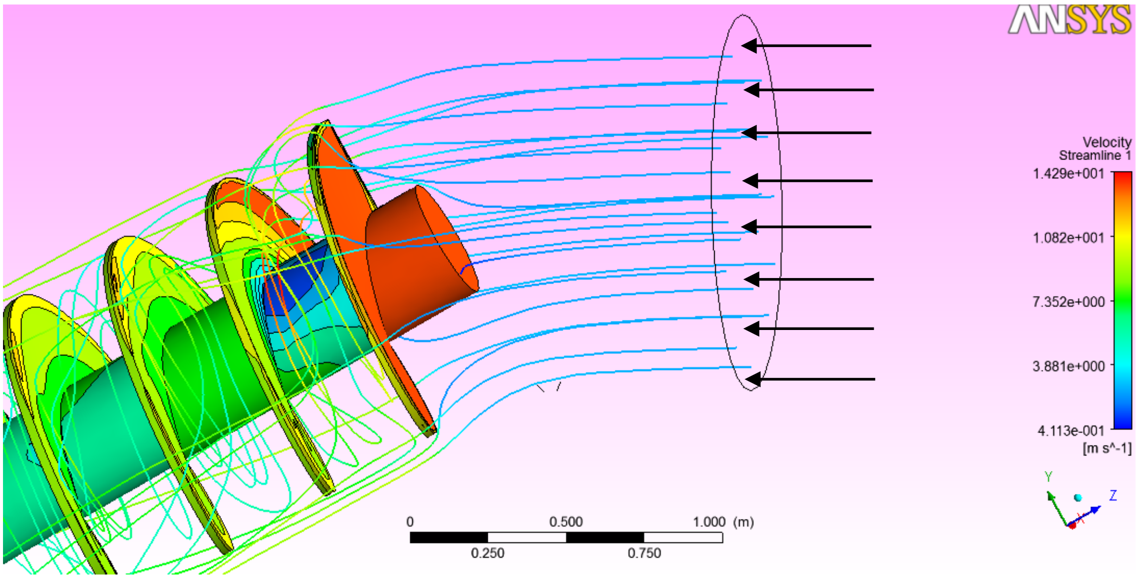


Figure 4.3: Velocity streamline at the inlet flow

Figure 4.4 shows the streamlines of the velocity at the outlet of the turbine. The velocity streamlines is quite low after the flow go through the screw runner blades. The loss of the blades as the disturbance make the flow smooth again. However, the flow pattern is seen quite messy along the 2^{nd} to 14^{th} blade due to the turbulent flow.

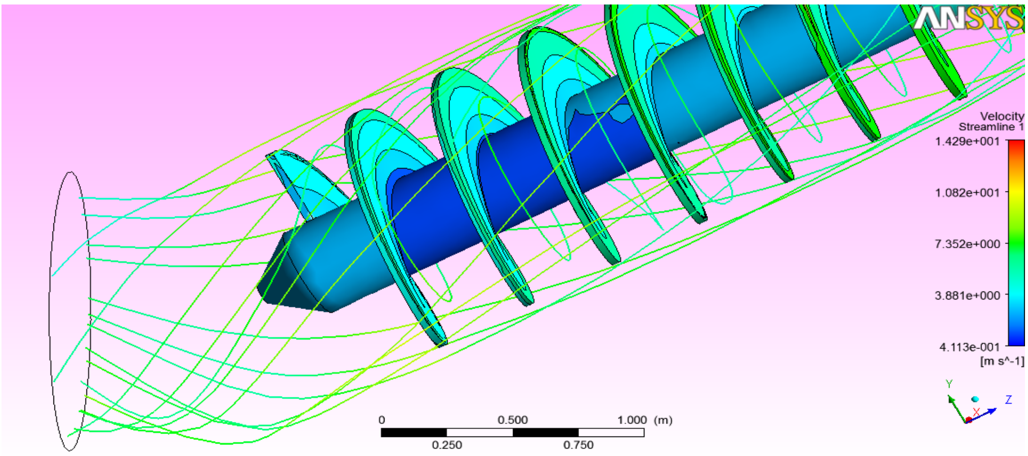


Figure 4.4: Velocity streamline at the outlet flow

4.2.3 Flow patterns

The velocity vector indicates the flow moving from the inlet of the turbine. The velocity vector shows that in Figure 4.5 and at the first blade, the velocity is not too high. Then, at the back of the first blade, the velocity vector move unstable and have high velocity vector compared to the second blade.

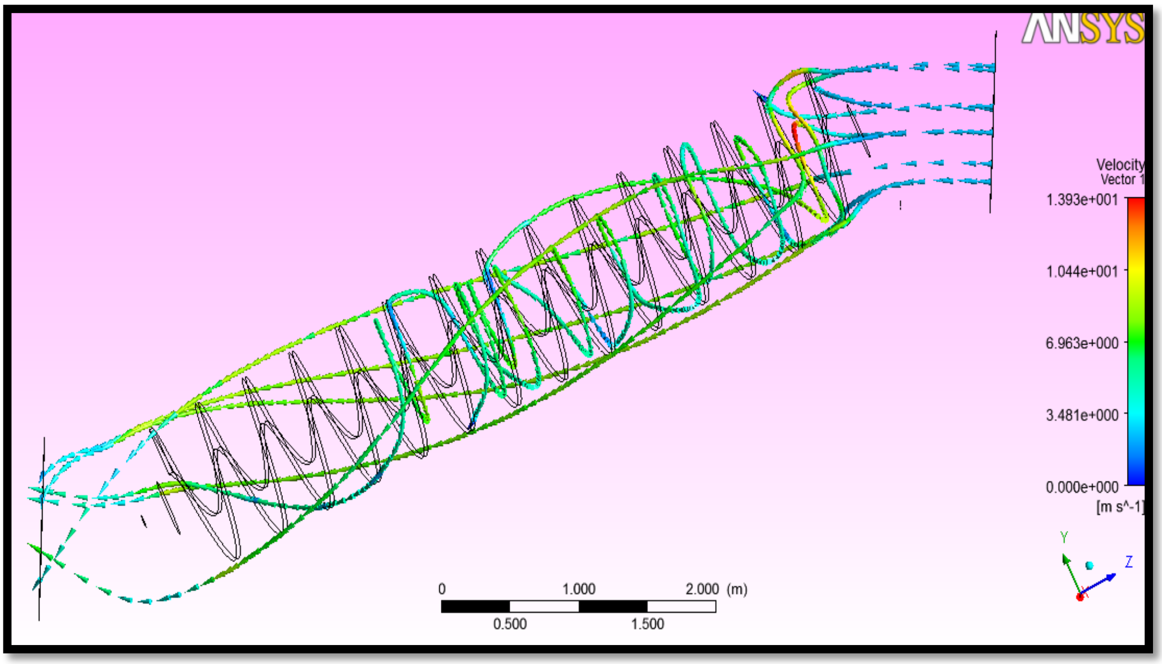


Figure 4.5: Velocity vector of the overall water flow

The velocity vector plot flow in streamlines. It seemed that the flow is dominated by the tangential component. The downstream of the blade streamline is shown in this Figure 4.6. Besides, the velocity vector was affected by the turbine blade as the stream of the turbine hit the wall and at the same time the velocity vector diverged in two direction which is upper and lower vertices. The minimum velocity occurred at the inlet of the turbine and shows that the velocity is inactive.

The maximum velocity vector was at back of the first blade of turbine. This is due to the increasing of the movement in flow behind the blade. The first blade have an active velocity fields because the first blade acts as the main disturbance to the flow. The flow becomes weak when reaching the side surface.

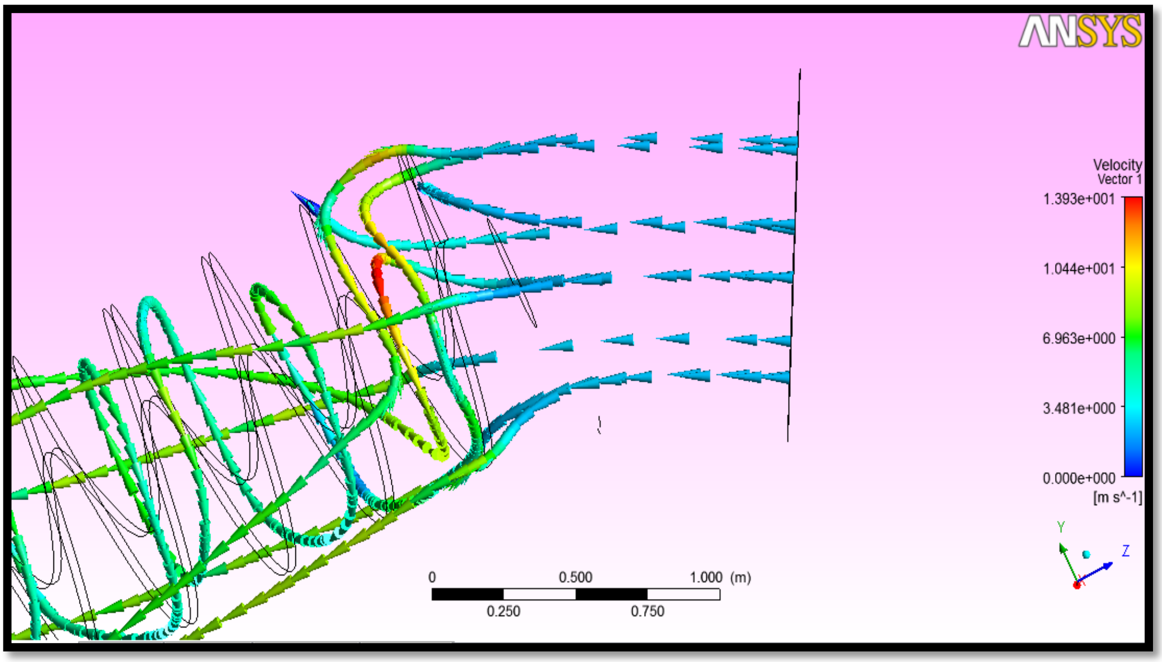


Figure 4.6: Velocity vector of the water at inlet flow

4.2.4 Pressure Contour

The highest pressure contour occurred at the first and second blade while the minimum one is located at the bottom of the turbine as indicates in Figure 4.7. The change of magnitude and direction of the impacts velocities is due to the turbine surface blades. The pressure contour also show that the first blade that got hit by the flow of water will experience a greater impact compared to the other blade and the static pressure shows the location of the stagnation region on the leading edge and blade tip. The blade surface at the first blade have high pressure because the separation of the fluid is more exposed to the wall. In particular, it can be directly seen that the maximum pressure contour with 8.174e+2 occurred at the first and second blade from inlet flow of turbine. The velocity at inlet also affect the number and location of the maximum pressure. Meanwhile, the minimum pressure is situated near the outlet of turbine and is due by the blade profile.

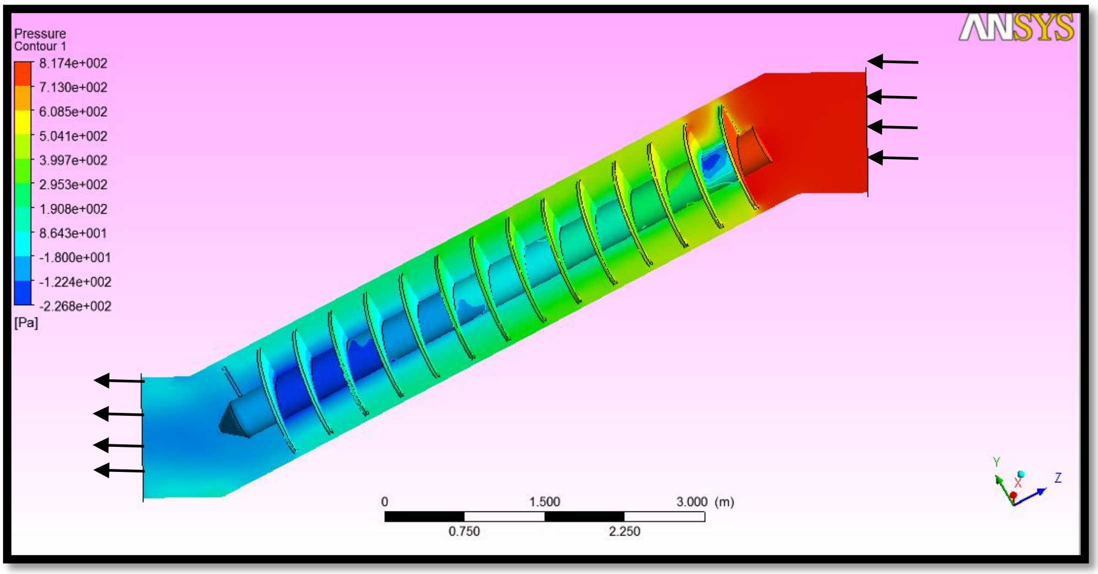


Figure 4.7: Pressure contour of the turbine blade

The front view pressure contour around the screw blade as shown in Figure 4.8 shows that the layer flow of the blade boundary is develops without a large separated region. Besides, the section that located close the wall fluid linked with the actual inflow of blade angle have more complex pattern and the leading edge separation bubble that affect the pressure contour which is radially outward. From the pressure contour, the prediction of particle impact can be recognised at the certain points. The particle impact will always occurred at the blade surfaces, at the blade tip and at the leading edge as shown in Figure 4.8.

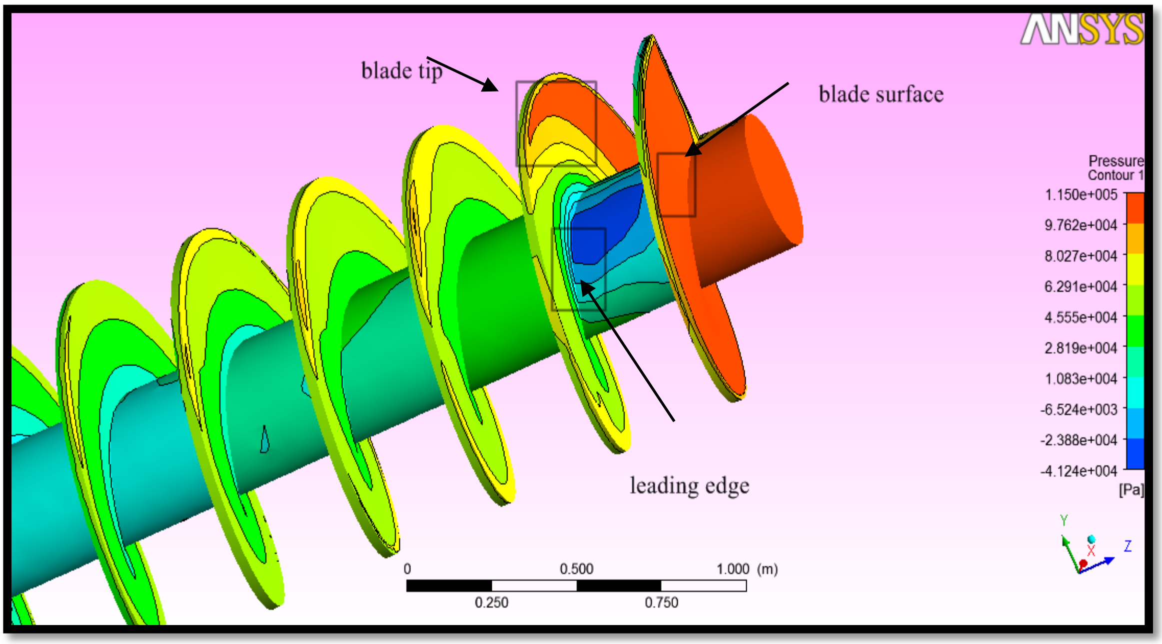


Figure 4.8: Front view pressure contour around the screw blade

On the blade leading edge from the Figure 4.9 above shown that the various pressure is resulting from the blade boundary layer flow, and from that, the prediction that can be made is the impact may occurred at the leading edge's erosion is extended over the entire blade and as the said by previous researcher, the areas sensitive to the erosion were more concern on the tip corner as per high peripheral velocity. Since the high inertia particle produced from the flow, the force is greater at the blade surfaces, blade tips and leading edges. So, the predictions of particle impact on a blade surfaces, blade tips and leading edges will lower the particles absolute velocities.

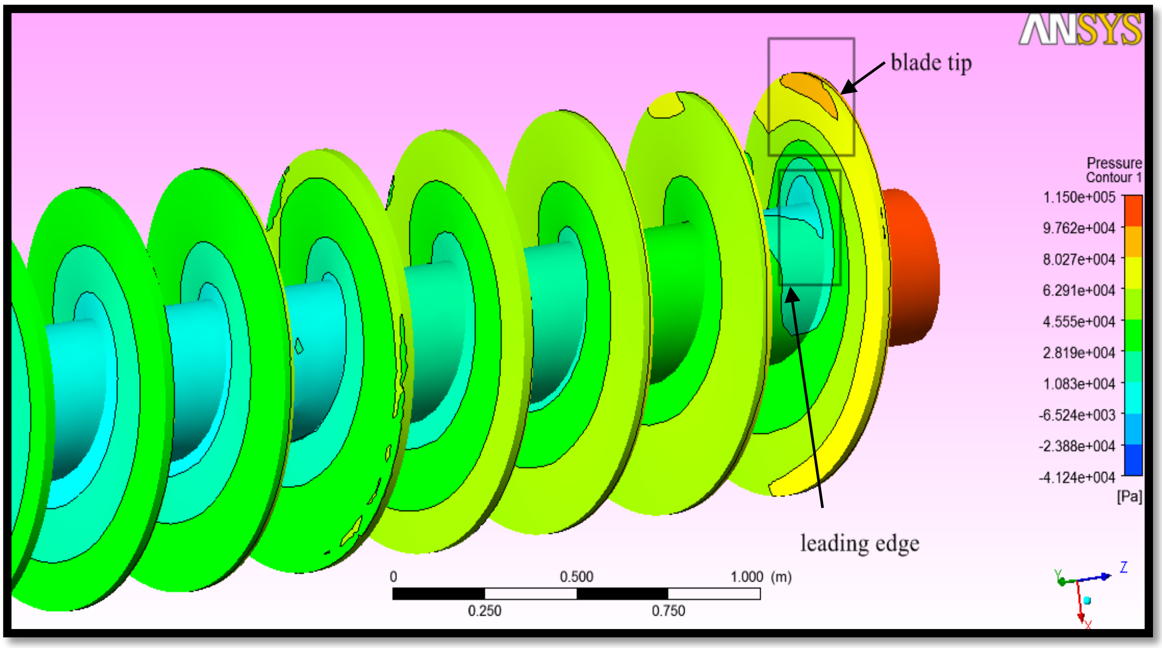


Figure 4.9: Back view pressure contour around the screw blade

Based on the results from the CFD simulation, this study shows that the prediction of particle impact occurs mostly on the entering surface blade and along the leading edge of the screw runner. Any modification on the design of the screw runner blade can be analyse for further study.

CHAPTER 5

CONCLUSIONS

5.1 INTRODUCTION

This chapter consists of a conclusion to summarize the overall project findings in the prediction of the blades and recommendation for the improvement for the future research.

5.2 CONCLUSION

This project outlines is to predict the particle impact on a Archimedes screw runner blade for micro hydro turbine. A 3-dimensional flow of a water is numerically analysed using the governing equation and k-ε turbulence model. The project was carried out to study the behaviour of the flow pattern inside the blade and predict the impact acted on the surface blade. The computational method was used to analyse a complex geometry and reduce the costs to conduct an experiment. The computational results of flow pattern indicates that streamline velocity is highest at the helical blade suction surface compared with discharged surface. Prediction based on computational simulation and the experimental indicates a slender impact occurred more at the leading edge and pressure surface impact decrease towards the entire helical edge.

5.3 RECOMMENDATION

Based on the results that was observed from the numerical simulation, there are some recommendation that is suitable to be apply in the next research. The Archimedes screw runner blade should be rotating because the real function of the turbine is to convert the rotating mechanical device into some useful energy. Moreover, the material selection of the blade should be concern as the type of material play an important role in every mechanical design.

REFERENCE

Aubin,J. Fletcher,D.F. and Xuereb.C. 2004. Modelling turbulent flow in stirred tanks with CFD: The influence of the modelling approach, turbulence model and numerical scheme. *Experimental Thermal and Fluid Science*. **28**: 431-445.

Bardina,J.E. Huang.P.G. and Coakley.T.J. 1997. Turbulence modelling validation, testing and development. National Aeronautics and Space Administration. 18-23.

Brada,K. 1999. Hydraulic screw generates electricity from micro hydropower stations. **14**: 52-56.

Georgescu,A.M. Georgescu, S.C. Cosoiu, C.I. Alboiou,N and Petre. A.M. 2010. Experimental versus numerical results on the velocity field in the wake of a hydropower farm equipped with three achard turbines. *Hydaulics and Environmental Protection Department, Technical University of Civil Engineering Bucharest, Romania*. **72**(1): 133-140.

Gartmann,A. Fister,W. Schwanghart,W and Möller, M.D. 2011. CFD modelling and validation of measured wind field data in a portable wind tunnel. *Aeolian Research*. **3**: 315-325.

Hamed, A.A. and Tabakoff,W. 2005. Turbine blade surface deterioration by erosion. *Journal of Turbomachinery ASME*. **127**: 445-452.

Hamed,A. Tabakoff,W. and Wenglarz,R. 2006. Erosion and deposition in turbomachinery. *Journal of Propulsion and Power*. **22**(2): 350-360.

Liu,P. 2010. A computational hydrodynamics method for horizontal axis turbine-Panel method modelling migration from propulsion to turbine energy. *Energy*. **35**: 2843-2851.

Möller,G. and Senior,J. 2009. Simplified theory of Archimedean screws. *Journal of Hydraulic Research*. **47**(5): 666-669.

Micale,G. Montante,G. Grisafi,F. Brucato,A. and Godfrey.J. 2000. CFD simulation of particle distribution in stirred vessels. *Trans IChemE*. **78**(A): 435-444.

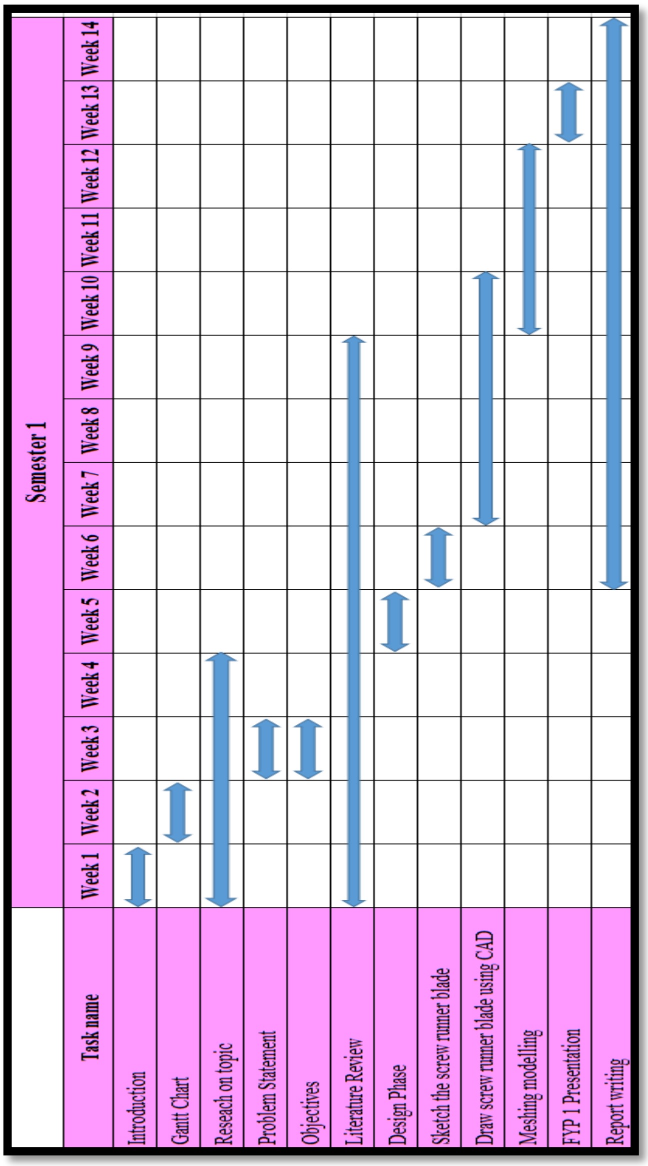
Rorres,C. 2000. The turn of the screw: Optimal design of an Archimedes screw. *Journal of Hydraulic Engineering ASCE*. **126**(1): 72-80.

Sheard,A.G. 2012. Predicting blade leading edge erosion in an axial induced draft fan. *Journal of Engineering for Gas Turbine and Power ASME*. **134**: 1-9.

Thapa,B. and Brekke,H. 2004. Effect of sand particle size and surface curvature in erosion of hydraulic turbine. Proc. Of 22nd IAHR Symposium on Hydraulic Machinery and Systems, Stockholm.

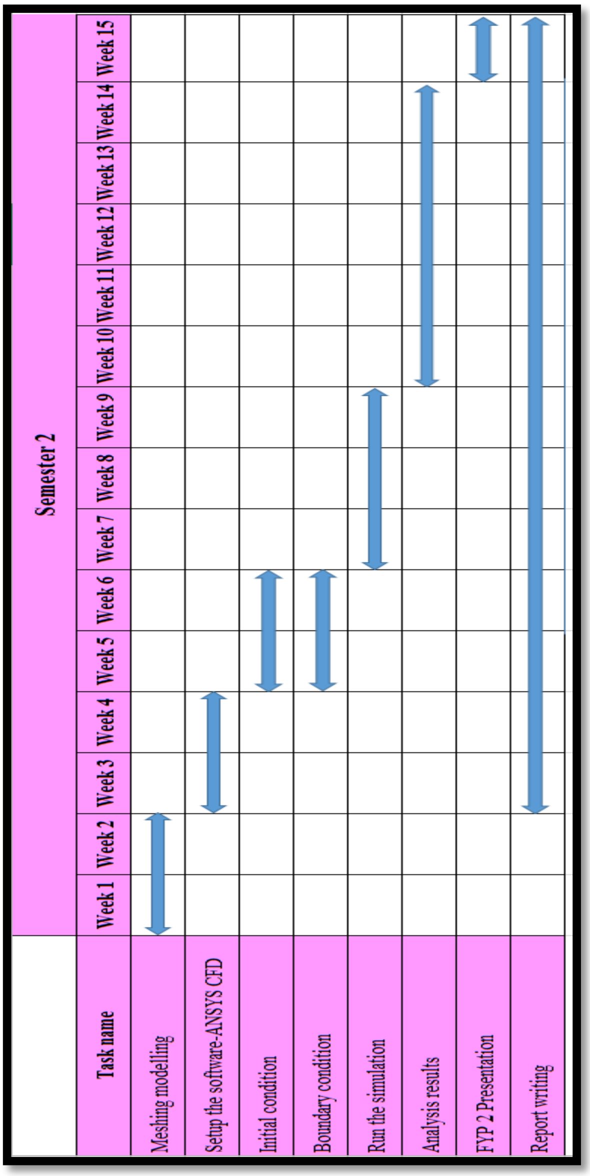
APPENDIX A1

Gantt chart for Final Year Project 1



APPENDIX A2

Gantt chart for Final Year Project 2



APPENDIX B1

The velocity of Pahang River

		Velocity of Pahang River (m/s)								
River part		Jan	Feb	March	April	May	June	July	Aug	Mean
Pahang river (Temerloh)	Minimum	2.1	2.3	2.6	2.3	2.1	2.3	2.6	2.1	2.3
	Mean	2.3	2.5	2.5	2.5	2.3	2.5	2.5	2.3	2.425
	Maximum	2.5	2.7	2.4	2.7	2.5	2.7	2.4	2.5	2.55
Jelai river (Kuala Medang)	Minimum	2.3	2.3	2.2	2.3	2.3	2.6	2.1	2.3	2.3
	Mean	2.5	2.5	2.4	2.5	2.5	2.5	2.3	2.5	2.4625
	Maximum	2.7	2.7	2.6	2.7	2.7	2.4	2.5	2.7	2.625
Pahang river (Pekan)	Minimum	2.3	2.6	2.3	2.1	2.6	2.3	-	-	2.36667
	Mean	2.5	2.5	2.5	2.3	2.5	2.5	-	-	**2.46667**
	Maximum	2.7	2.4	2.7	2.5	2.4	2.7	-	-	2.56667

(Source: Department of Irrigation and Drainage Malaysia)

The mean velocity of Pahang River specifically at Pekan area was 2.46667 m/s. This velocity value was used for the simulation of the river with MHP turbines.