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

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

LIST OF TABLES

LIST OF FIGURES

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.

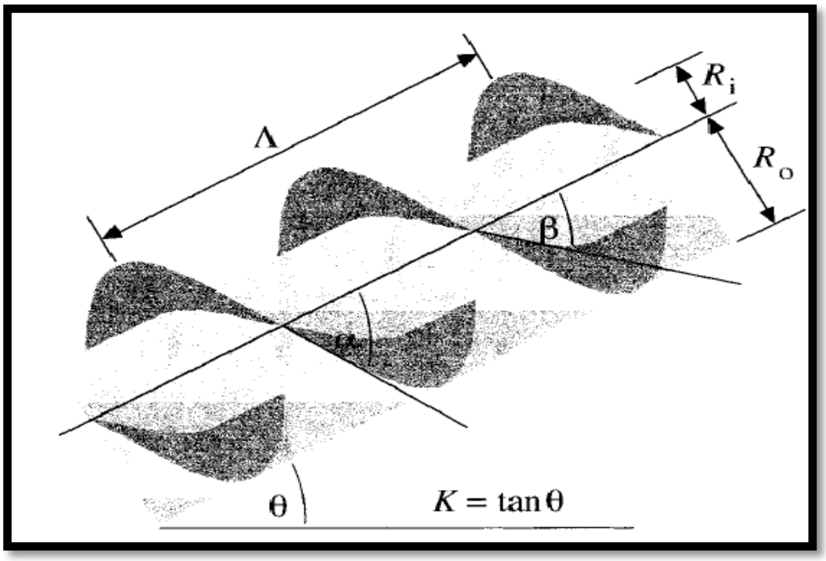

Figure 2.1: The profile view of two-bladed Archimedes Screw

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.

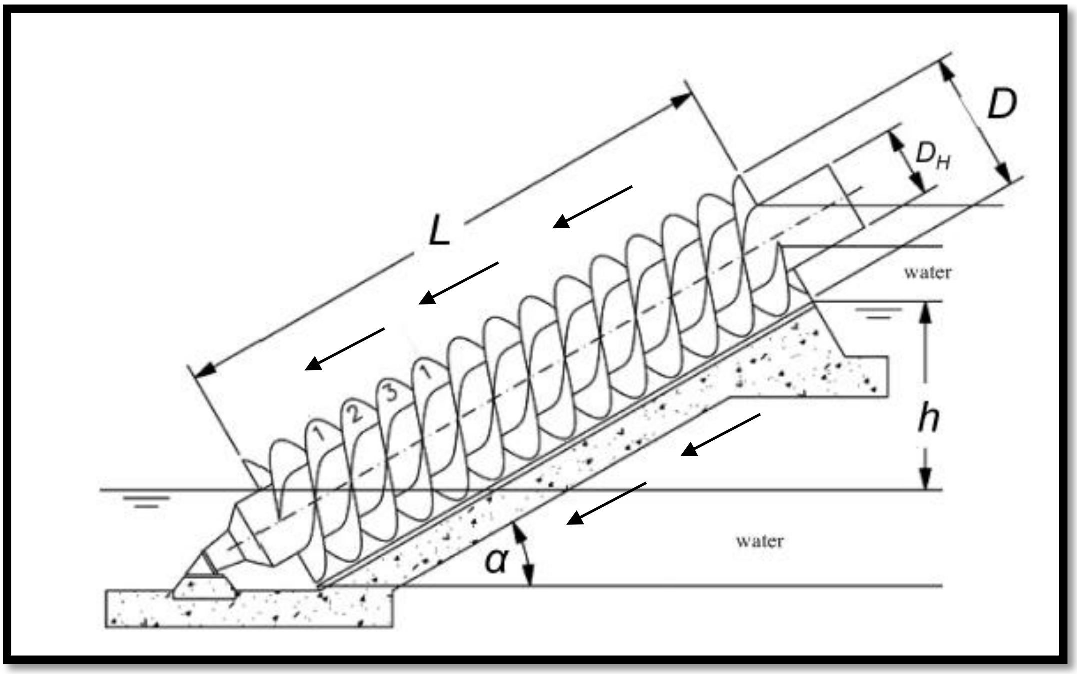

Figure 2.2: Side elevation of the Archimedes screw

Source: Mϋller, 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:

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

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.

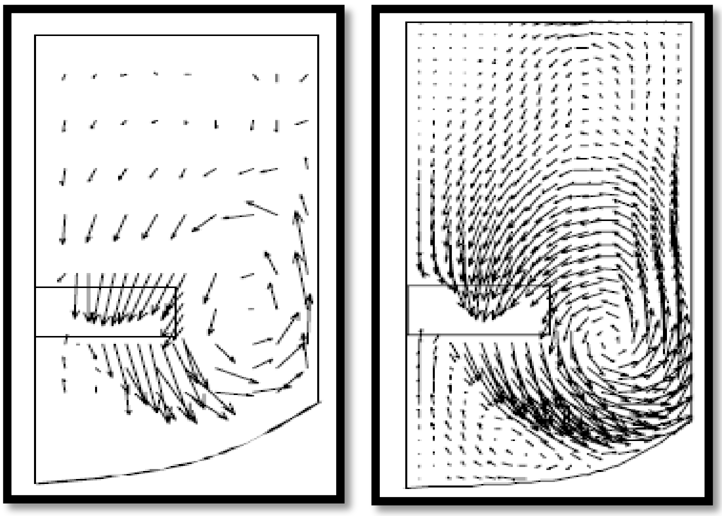

Figure 2.3: The dimensionless radial-axial vector plots

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.

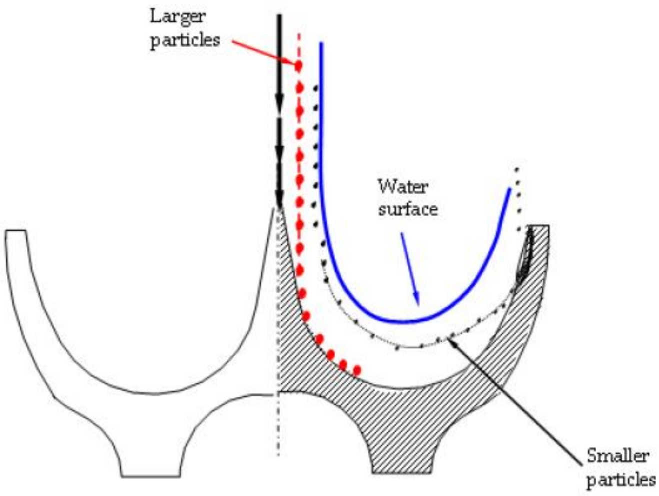

Figure 2.4: The separation of flow in Pelton bucket

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.

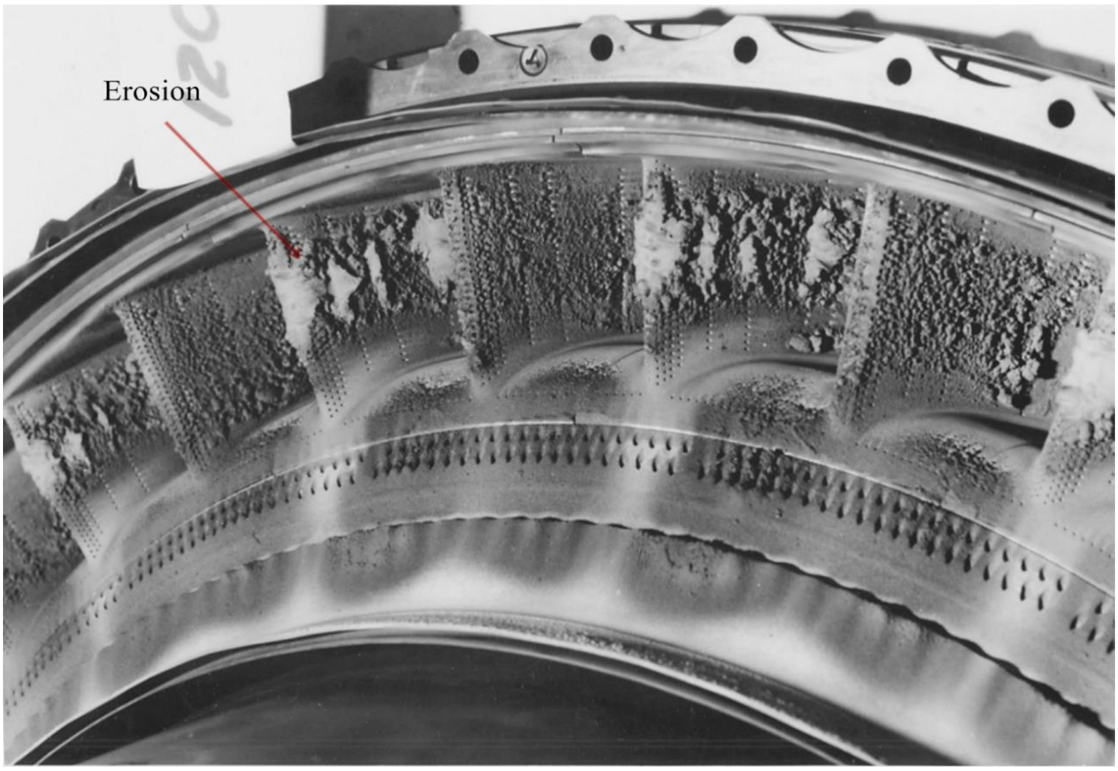

Figure 2.5: Volcanic ash deposition on turbine vanes

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.

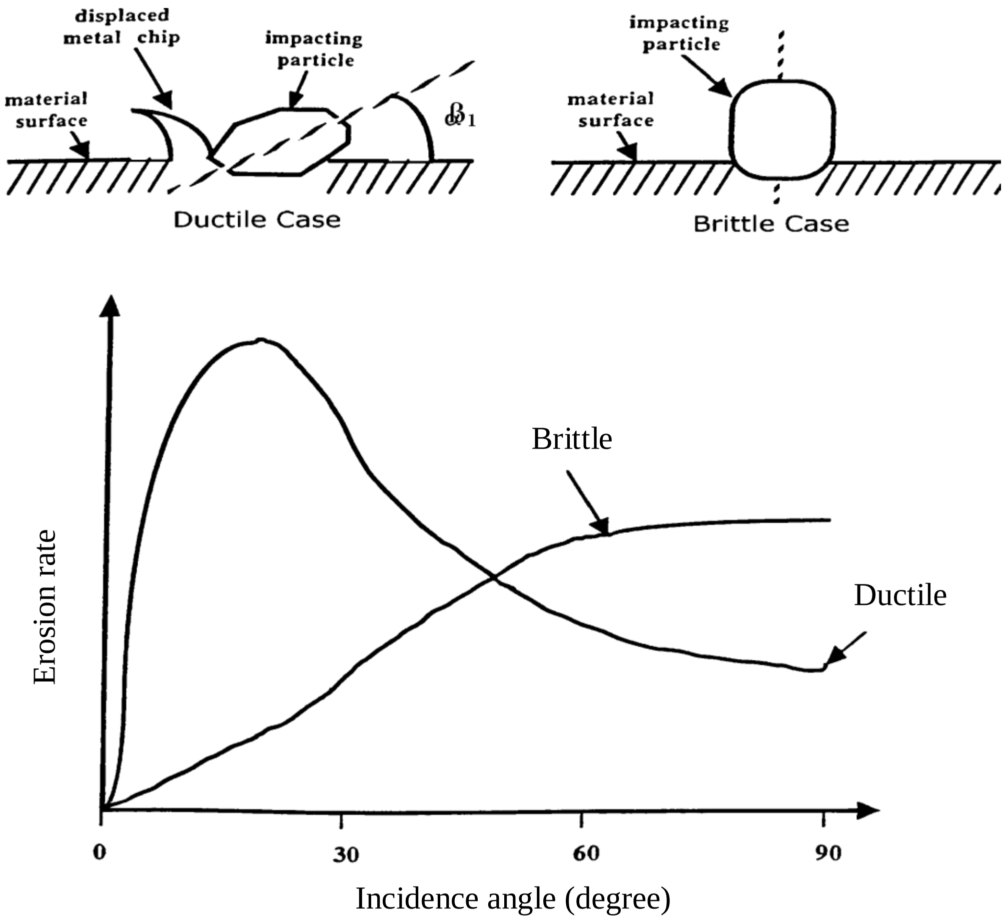

Figure 2.6: Erosion rate variation with impact angle

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.

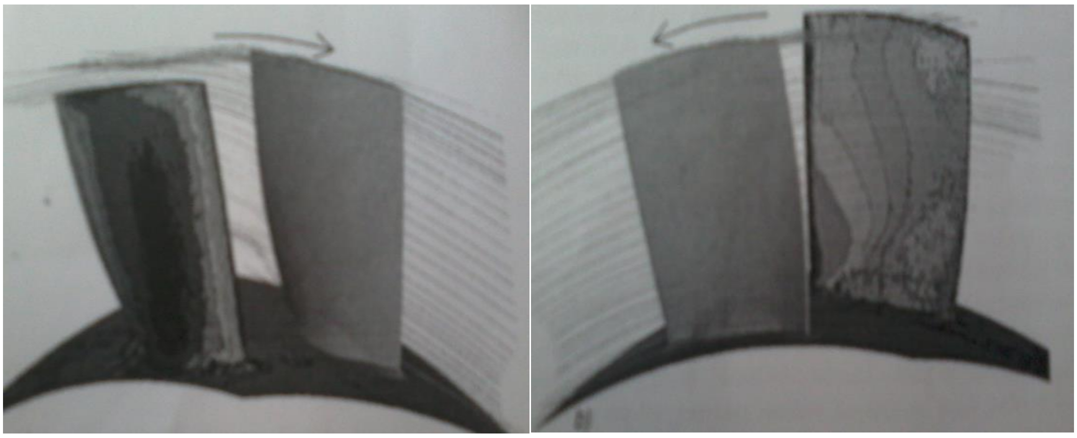

Figure 2.7: The streamlines and pressure contour of the fan blade

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.