SUDAN UNIVERSITY OF SCIENCE &TECHNOLOGY

Faculty of Post Graduate Studies

Effect of Vortex in Kaplan Turbine-Using CFD

A Case Study: Rosseries Power Plant

أثرالدوامات المائية علي توربينة كابلن بإستخدام ديناميكا الموائع المحوسبة

تطبيق علي محطة كهرباء الروصيرص

A Thesis Submitted in Partial Fulfillment for the Requirement of the Degree of M.Sc in Mechanical Engineering

By:

El Mutaz Elsir Hassan Mohammed
B.Sc in Mechanical Engineering

Supervisor: Dr. Obai Younis Taha

June2016

Dedicate

To my

Parents'

Wife

My sons Mohammed

Munib

Daughter Alla

Acknowledgment:

I would like to thank Dr. Obai Yunis who guide me to conduct the theses and lead me to achieve all challenges.

I wish to extend my great thank to my colleges Mr. Yassir Elmahi for his participation and cooperation and helping in many aspects especially in the drawing of the turbine geometry.

Finally, I thank all those who assisted, encouraged me during this research.

Appendix

الملخص:

يهدف هذا البحث على التركيز في دراسة أثر الدوامات المائية علي عنفه كابلن وخاصة الدوامات عند أطراف الريشة. تم تنفيذ التحليل بواسطة (برنامج أنسيس) للتنبؤ بسلوك ومسار التيارات المائية داخل عنفه كابلن وذلك لمزيد من التشخيص والتحري حول المشاكل والتحديات السائدة بمحطات القدرة المائية بالسودان. وكمثال للمشاكل تأكل ريشة العنف،التكهف وفقدان في الطاقة الكهربائية نتيجة لذلك.

أتبع البحث في وصف أثر الدوامات طريقة التحليل العددي لديناميكا الموائع للتنبؤ بالعيوب التي يمكن تداركها .علما بإن معرفة شكل وتركيز الدوامات في أطراف العنف يساعد في حل المشاكل المذكورة أعلاه ووضع تصميم ملائم .

تم عرض أثر جودة التقسيم الشبكي لنموذج العنفة علي نتائج التحليل العددي لبرنامج ديناميكا الموائع المحوسب، كما تمت مقارنة النتائج مع الوضع الحقيقي لريش عنفه كابلن بمحطة كهرباء الروصيرص حيث يمثل الإنخفاض في الكفاءة نسبة (3.4%) نتيجة لأثر الدوامات المائية .

Abstract

The objective of this research is focused on the vortex flow in Kaplan turbine runner - a tip vortex. CFD analysis was performed to predict the water flow inside the turbine for more diagnosing and investigation of the hydro power problem.

On the basis of this analysis it is possible to estimate the existing and the intensity of vortex, danger of possible blade surface and runner chamber cavitations' pitting and power losses.

In the research, the ways how to predict vortex flow and the effect of the tip vortex are described. In order to prevent the blade surface against pitting and energy losses , the knowledge of the shape and intensity of the tip vortex helps to solve the main problem in the hydro turbine as well it help to design more sophistically as the change of geometry of the runner blade or other solution aspects. The affect of the mesh quality was presented.

After all, the results of the CFD analysis simulation and the actual status of some Kaplan runner in Rosseries hydro power are compared. There is significant decrease in efficiency (3.4%) due to vortex.

1.0 Introduction:

1.1Hydro power Background:

Now days, the awareness about the importance of a sustainable environment is increasing, it has been recognized that traditional dependence on fossil fuel extracts a heavy cost from the environment. The role of renewable energy has been recognized as great significance for the global environmental concerns. Hydro-power is a good example of renewable energy and its potential application to future power generation cannot be underestimated and it defines as is the force or energy of moving water, it is the leading source of renewable energy. It provides more than 97% of all electricity generated by renewable sources [1]. World-wide, about 20% of all electricity is generated by hydropower. Is the most efficient way to generate electricity? It has among the best conversion efficiencies of all known energy sources (about 90% efficiency, water to wire). It requires relatively high initial investment, but has a long lifespan with very low operation and maintenance costs; it is a proven, mature, predictable and typically price-competitive technology.

Leonardo da Vinci once said “The power of water has changed more in this world than emperors or kings”. It was very rightly stated by him as in present time Hydropower, the power generated from water, has a major contribution to the world’s total power production. This all was made possible by the development of hydraulic Turbines which can transfer the energy from flowing water to the shafts of dynamos producing electrical power [2].

1.2Problem Statement:

Many power plants, especially in Sudan, are getting old and needs to be upgraded. All of them are of Kaplan type. An upgrade often implies a new runner design and since there is a strong interaction between the flow in the runner and the draft tube in Kaplan turbines, there is a need to understand how water flows inside the turbine to design an efficient runner and assist to make a proper solution for the existing problem. To develop our capabilities it requires many steps and one is to find a model to make trust-worthy predictions of the pressure recovery and the existing of the vortices in the turbine through steady-state CFD-simulations.

1.3 Study Objective:

The research presents an investigation of CFD-simulations to predict the flow in a Kaplan turbine. The main purpose was to evaluate how well CFD simulations

Agree to actual situation in the turbine especially the influence of the vortex, secondly to build numerical model for researches and studies.

The simulations and the predicted pressure recovery of the turbine were made using the commercial software ANSYS CFX (14.5).

1.4 Approach

1. CFD simulations and research theory were made to understand the flow especially the influence of the vortices and any difficulties of Kaplan turbine and to find a general pattern for the velocity streamline and the pressure gradient.
2. Different meshes and boundary conditions performed to achieve good convergence.
3. CFD-simulations were performed and compared with the actual situation for Roessires hydro power plant turbine.
4. Many parameters were compared such as stream lines, pressure field, velocity and efficiency.

2.0 Theory of Hydro Power:

2.1 Waterpower Background:

Prior to the wide spread availability of commercial electric power, Humans was used the power of flowing water for thousands of years for irrigation and operation of various machines, such as watermills, textile machines and sawmills. By using water for power generation, people have worked with nature to achieve a better lifestyle.

The mechanical power of falling water is an old resource used for services and productive uses. It was used in the ancient times extensively by Egyptian and Chinese for irrigation and the Greeks to turn water wheels for grinding wheat into flour more than 2,000 years ago. In the 1700s, mechanical hydropower was used extensively for milling and pumping. During the 1700s and 1800s, water turbine development continued. The first hydroelectric power plant was installed in Cragside, Rothbury, England in 1870. Industrial use of hydropower started in 1880 in Grand Rapids, Michigan, when a dynamo driven by a water turbine was used to provide theatre and storefront lighting. In 1881, a brush dynamo connected to a turbine in a flour mill provided street lighting at Niagara Falls, New York.

The breakthrough came when the electric generator was coupled to the turbine and thus the world's first hydroelectric station (of 12.5 kW capacities) was commissioned on 30 September 1882 on Fox River at the Vulcan Street Plant, Appleton, Wisconsin, USA, lighting two paper mills and a residence. Early hydropower plants were much more reliable and efficient than the fossil fuel-fi red plants of the day (Baird, 2006). This resulted in a proliferation of small- to medium-sized hydropower stations distributed wherever there was an adequate supply of moving water and a need for electricity. As electricity demand grew, the number and size of fossil fuel, nuclear and hydropower plants increased. In parallel, concerns arose around environmental and social impacts. Hydropower plants (HPP) today span a very large range of scales, from a few watts to several GW. The largest projects, Itaipu in Brazil with14, 000MW and Three Gorges in China with 22,400 MW, both produce between 80 to 100TWh /yr (288 to 360 PJ/yr). Hydropower projects areal ways site-specific and thus designed according to the river system they inhabit. Hydropower plants do not consume the water that drives the turbines. The water, after power generation, is available for various other essential uses. In fact, a significant proportion of hydropower projects are designed for multiple purposes. In these instances, the dams help to prevent or mitigate floods and droughts, provide the possibility to irrigate agriculture, supply water for domestic, municipal and industrial use, and can improve conditions for navigation, fishing, and tourism activities [3] [4].

2.2Hydraulic Turbines:

2.2.1 General:

The industrial development throughout the 18.and 19. Century the need for more energy was increasing so much that the water wheels no longer got hold of sufficient energy supply. New energy sources like vapor power was adopted, but further development toke place also on utilization of water power. In 1750 the physicist J. A. Segner invented a reaction runner, which has got its name after him. This runner utilized the impulse force from a water jet and was the forerunner of the turbines. Just after this the mathematician Leonard Euler developed the turbine theory, which is valid today too [3] [4].

Turbine is a designation that was introduced in 1824 in a dissertation of the French engineer Burdin. Next step was made by engineer BenoitFourneyron when he designed and put to operation the first real turbine in 1827. At that time this machine was a kind of a revolution with the unusual great power output of 20 – 30 kW and a runner diameter of 500 mm and a speed of 2300 rev/min. Because the water flow has a radial direction through the turbine runner, it may be designated as radial turbine. However, two other turbine designers Henschel and Jonval, developed about 1840, independent of each other a turbine each with axial water flow through it. In addition they were the first ones to apply draft tube and in that way to utilize the water head between runner outlet and tail water level.

In the time interval until 1850 several designers improved the machines, but most serious was the work by the English engineer Francis. He developed and made a turbine in 1849, which has got its name after him. This turbine looked fairly similar to that one Euler had foreseen. An axial section through the guide vane cascade and the runner of this turbine is shown on Figure 2.1. The water flows radially through the guide vane cascade towards the runner and further out in the axial direction. In the year 1870 professor Fink introduced an important improvement by making the guide vanes turning on a pivot in order to regulate the flow discharge. The turbines mentioned above, were operating at relatively low heads compared with turbines nowadays. Additionally the water was flowing into the turbine runner with a certain pressure above the atmospheric through a guide vane cascade comprising the whole periphery of the runner. A turbine design principally different from these, was developed by the American engineer Pelton. He made his first runner in1890, and Figure 2.2 shows that the water in this case flows as a jet from a nozzle through atmospheric air into the runner buckets. The buckets are designed to split the jet in two halves, which are deflected almost 180° before they leave the bucket. In this way the impulse force from the deflection is transferred to the runner. By the invention of the Pelton turbine; a machine type was developed for utilization of the highest water heads in the nature.

In the region of low pressure turbines a great step forward was made when Professor Kaplan designed his propeller turbine, which was patented in 1913. This turbine had fixed runner blades, and was developed for utilizing the lowest water heads. Figure 2.3a shows an axial section through a Kaplan turbine. Not very long after making the propeller turbine, Kaplan further developed his turbine by making the runner blades revolving Figure 2.3b and Figure 2.3c show the mechanisms. This was an important improvement for an economic and efficient governing of the turbine output.

After the brief review above of the hydropower machines, one can summarize that the only machine types that have been developed to be dominant in modern times are Pelton, Francis and Kaplan turbines Figure. 2.4. The reason is that these three types supplement each other in an excellent way. As a basic rule one can say that Pelton turbines are used for relatively high heads and small water discharges, Kaplan turbines for the lowest heads and the largest water discharges and Francis turbines for the region between the other two turbine types [5].

Hydraulic Turbines transfer the energy from a flowing fluid (from higher to lower elevations) to a rotating shaft. Turbine itself means a thing which rotates or spins [6]. Flowing liquid, mostly water, when pass through the Hydraulic Turbine it strikes the blades of the turbine and makes the shaft rotate. While flowing through the Hydraulic Turbine the velocity and pressure of the liquid reduce, these result in the development of torque and rotation of the turbine shaft. There are different forms of Hydraulic Turbines in use depending on the operational requirements. For every specific use a particular type of Hydraulic Turbine provides the optimum output [7].

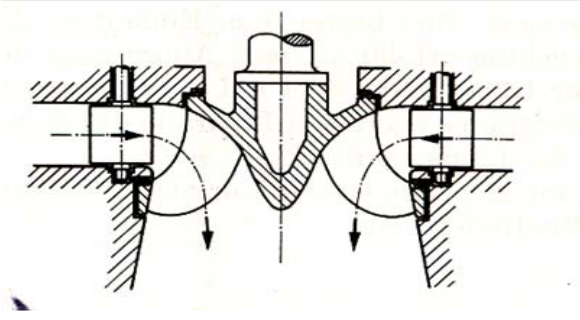

Figure 2.1a Francis Turbine

Figure 2.1b 3 Georges Francis Turbine runner

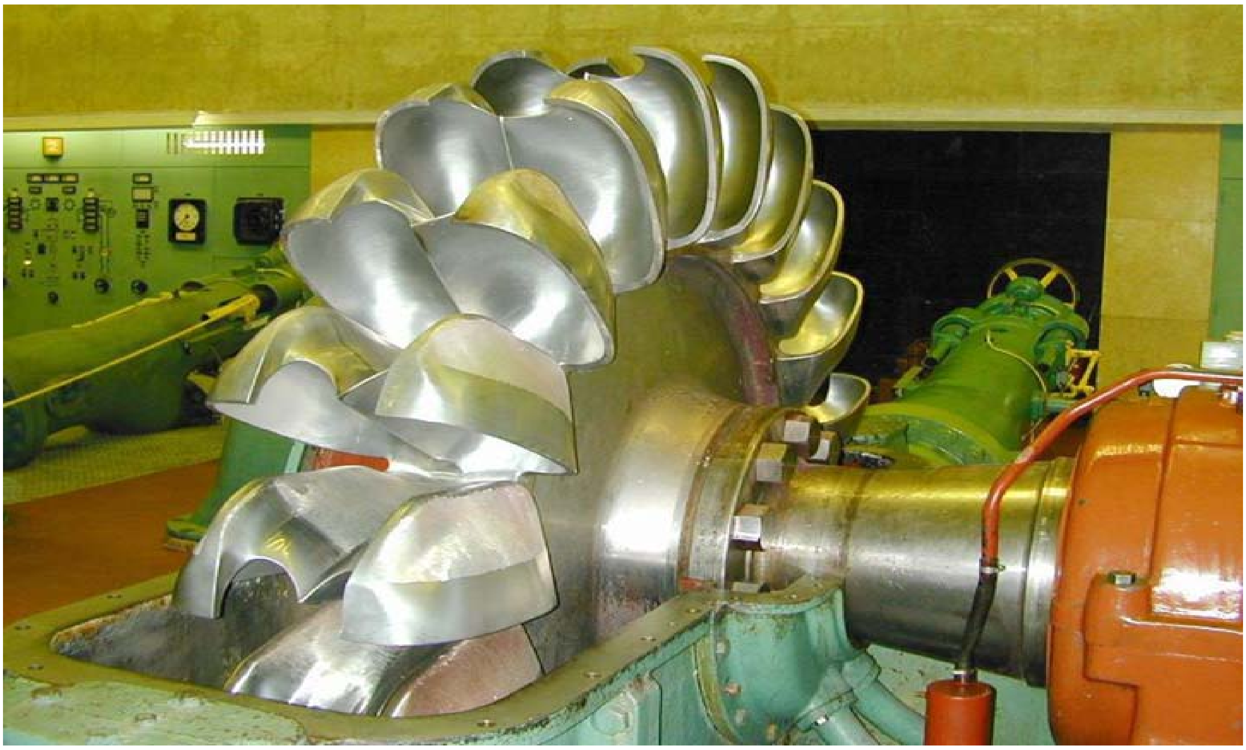

Figure 2.2 Pelton Turbine

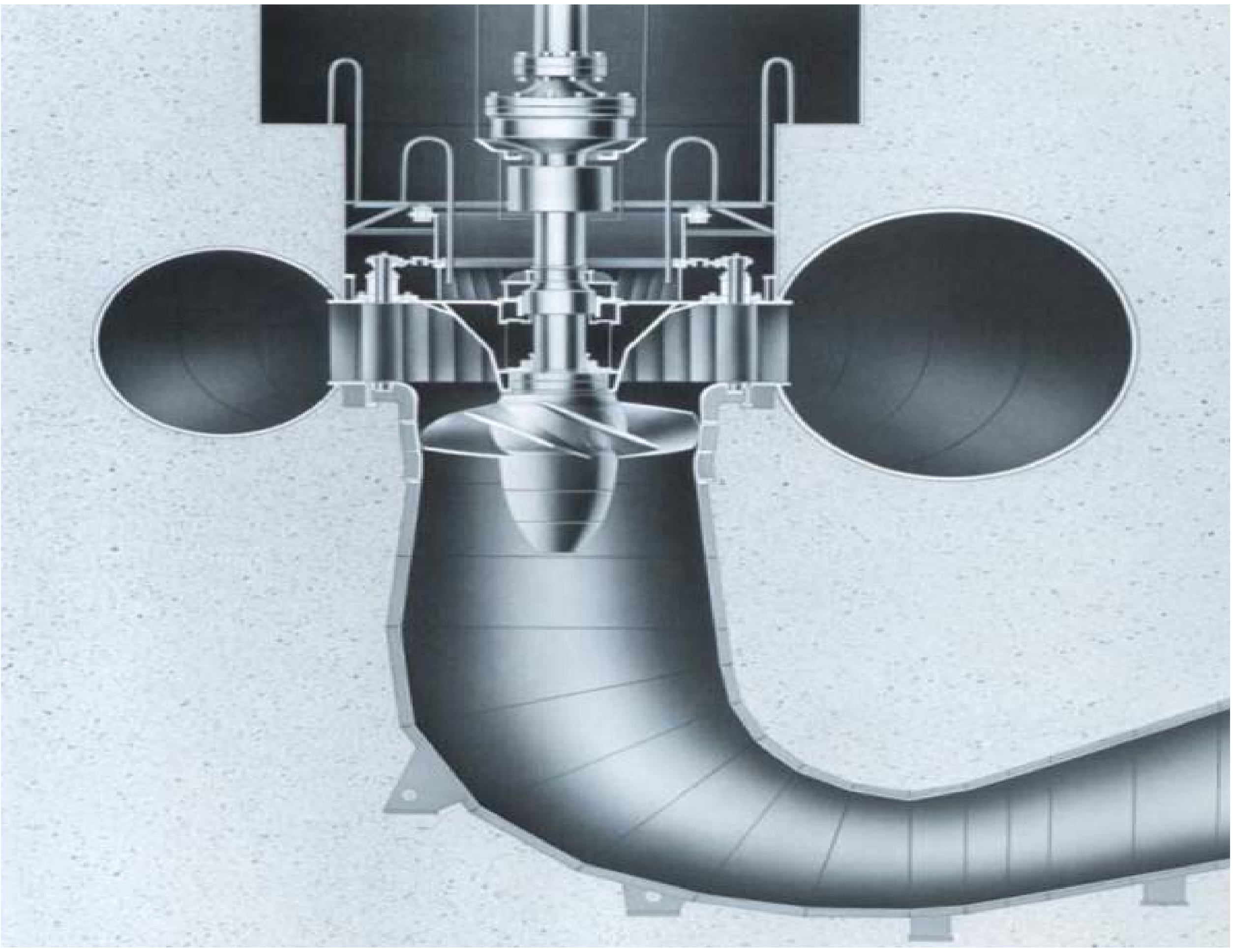

Figure. 2.3a illustrate the axial section

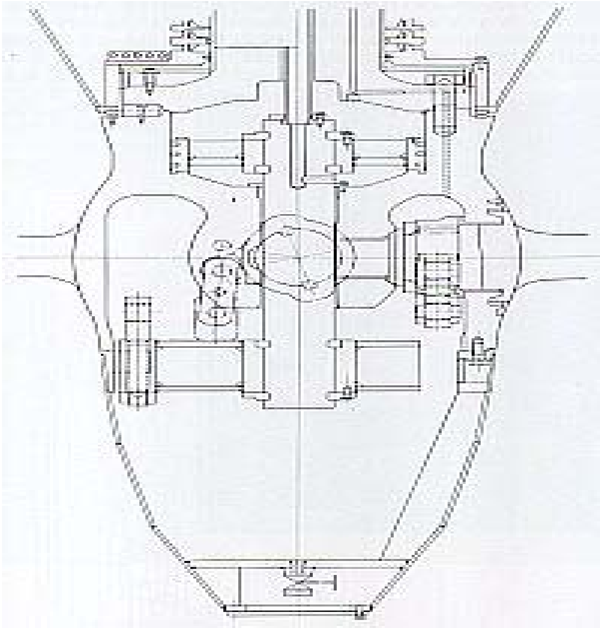

Figure. 2.3b Kaplan turbine revolving blade

Figure. 2.3c Revolving blade mechanism

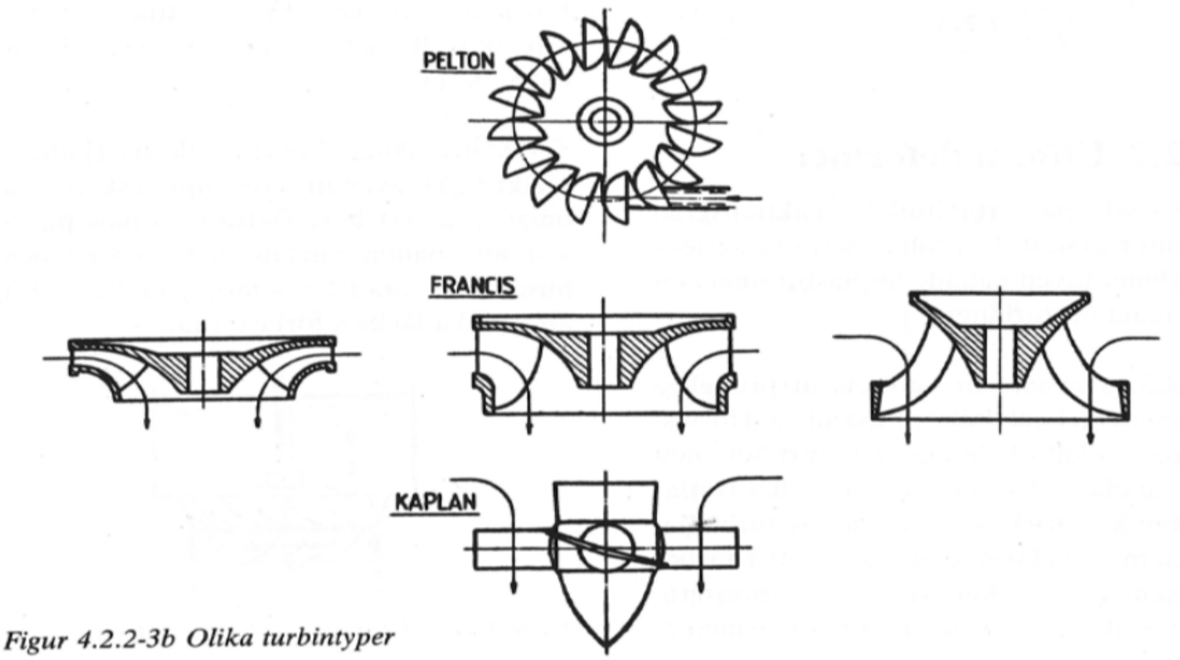

Fig. 2.4 illustrates the main types of Turbine

2.2.2 Classification of Hydraulic Turbines:

The hydraulic turbines were classified into different categories based on:

2.2.2 .1-Based on flow path:

Water can pass through the Hydraulic Turbines in different flow paths. Based on the flow path of the liquid Hydraulic Turbines can be categorized into three types.

(i) Axial Flow Hydraulic Turbines: This category of Hydraulic Turbines has the flow path of the liquid mainly parallel to the axis of rotation. Kaplan Turbines has liquid flow mainly in axial direction.

(ii) Radial Flow Hydraulic Turbines: Such Hydraulic Turbines has the liquid flowing mainly in a plane perpendicular to the axis of rotation.

(iii) Mixed Flow Hydraulic Turbines: For most of the Hydraulic Turbines used there is a significant component of both axial and radial flows. Such types of Hydraulic Turbines are called as Mixed Flow Turbines. Francis Turbine is an example of mixed flow type, in Francis Turbine water enters in radial direction and exits in axial direction.

None of the Hydraulic Turbines are purely axial flow or purely radial flow. There is always a component of radial flow in axial flow turbines and of axial flow in radial flow turbines.

2.2.2 .2- Based on pressure change:

One more important criterion for classification of Hydraulic Turbines is whether the pressure of liquid changes or not while it flows through the rotor of the hydraulic turbines. Based on the pressure change hydraulic turbines can be classified as of two types.

(i) Impulse Turbine: The pressure of liquid does not change while flowing through the rotor of the machine. In Impulse Turbines pressure change occur only in the nozzles of the machine. One such example of impulse turbine is Pelton Wheel.

(ii) Reaction Turbine: The pressure of liquid changes while it flows through the rotor of the machine. The change in fluid velocity and reduction in its pressure causes a reaction on the turbine blades; this is where from the name Reaction Turbine may have been derived. Francis and Kaplan Turbines fall in the category of Reaction Turbines.

2.3 Hydro power in Sudan:

The potential hydro power in Sudan is more than 3000 MW concentrate in Nile river and its branches .Recently the total hydro generation about1500 MW, most of the turbine are Kaplan turbine according to the available head. So in this study the work will focused on Kaplan turbine type.

2.4 Kaplan Turbine:

Most of the turbines developed earlier were suitable for large heads of water. With increasing demand of power need was felt to harness power from sources of low head water, such as, rivers flowing at low heights. For such low head applications Viktor Kaplan (1876-1934) Figure 2.5 designed a turbine similar to the propellers of ships. Its working is just reverse to that of propellers. The Kaplan Turbine is also called as Propeller Turbine. Instead of displacing the water axially using shaft power and creating axial thrust, the axial force of water acts on the blades of Kaplan Turbine and generating shaft power, it is necessary to have large flow rates through the turbine. It is designed to accommodate the required large flow rates. Except the alignment of the blades the construction of the Kaplan Turbine is very much similar to that of the Francis Turbine. It has a runner with three to six blades in which water impinges continuously at a constant rate. The pitch of the blades may be fixed or adjustable and it is cast as integral parts of the runner or may be welded to the hub; the major components besides the runner are a scroll case, wicket gates, and a draft tube sees Figure 2.6 [8]; the overall path of flow of water through the Kaplan Turbine is from radial at the entrance to axial at the exit. Similar to the Francis Turbine, Kaplan Turbine also has a ring of fixed guide vanes at the inlet to the turbine. Unlike the Francis Turbine which has guide vanes at the periphery of the turbine rotor (called as runner in the case of Francis Turbine), there is a passage between the guide vanes and the rotor of the Kaplan Turbine.

The shape of the passage is such that the flow which enters the passage in the radial direction is forced to flow in axial direction with the initial swirl imparted by the inlet guide vanes which is now in the form of free vortex Figure2.7, the tangential (whirl) velocity is inversely proportional to radius. The runner blades must be long in order to accommodate the large flow rate and, consequently, considerations of strength required to transmit the tremendous torques involved impose the necessity for large blade chords. Thus, pitch chord ratios of 1.0to 1.5 are used by manufacturers and, consequently, the number of blades is small [7] [10].

The axial flow of water with a component of swirl applies force on the blades of the rotor and loses its momentum, both linear and angular, producing torque and rotation (their product is power) in the shaft, the scheme for production of hydroelectricity by Kaplan Turbine is same as that for Francis Turbine. The rotor of the Kaplan Turbine is similar to the propeller of a ship. The rotor blades are attached to the central shaft of the turbine. The blades are connected to the shaft with moveable joints such that the blades can be swiveled according to the flow rate and water head available.The blades are not planer as any other axial flow turbine; instead they are designed with twist along the length so as to allow swirling flow at entry and axial flow at exit. The velocity of the blades is directly proportional to the radius whereas, the fluid whirl velocity is inversely proportional to it. To cater for this difference, the runner blades are twisted so that the angle they make with the axis is greater at the tip than at the hub [7].

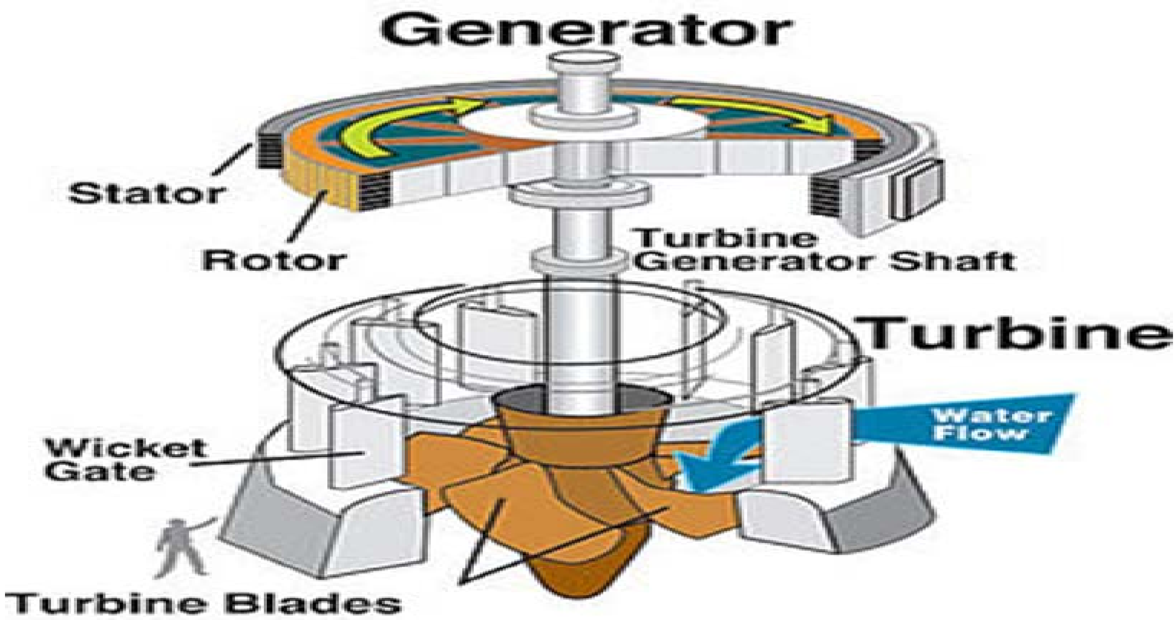

Figure 2.6 illustrate Kaplan turbine

2.4.1: Basic fluid mechanics:

2.4.1.1Governing equations:

The equation for describing the motion of fluids is named the Navier-Stokes Equation from the great physicists Claude-Louise Navier and George Gabriel Stokes.

The governing equations behind it are derived from elementary mechanics and assumptions concerning fluid motions and are based on the idea that mass, Momentum and energy are conserved [8].

2.4.1.2velocity triangles:

The velocity triangles shown in Figure.2.8, the velocity of flow is axial at inlet and outlet and, of course, remains the same. The whirl velocity is tangential. The blade velocity at inlet and outlet is the same, but varies along the blades with radius from hub to tip.

If the angular velocity of the runner is ω the blade velocity at radius r is given by

u = ωr **(2.1)**

Since at maximum efficiency = 0 and v2 = vf, it follows that

$$E = uvw1/g \qquad (2.2)$$

In which = vf cot θ. Since E should be the same at the blade tip and at the hub, but u is greater at the tip, it follows that must be reduced. Similarly, the velocity of flow vf should remain constant along the blade and, therefore, cot θ must be reduced towards the tip of the blade. Thus, θ has to be reduced and, consequently, the blade must be twisted so that it makes a greater angle with the axis at the tip than it does at the hub.

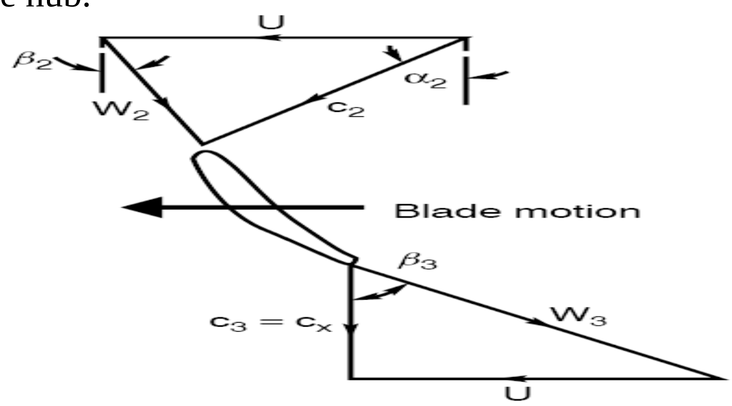

Figure. 2.7 illustrate the Swirling flow

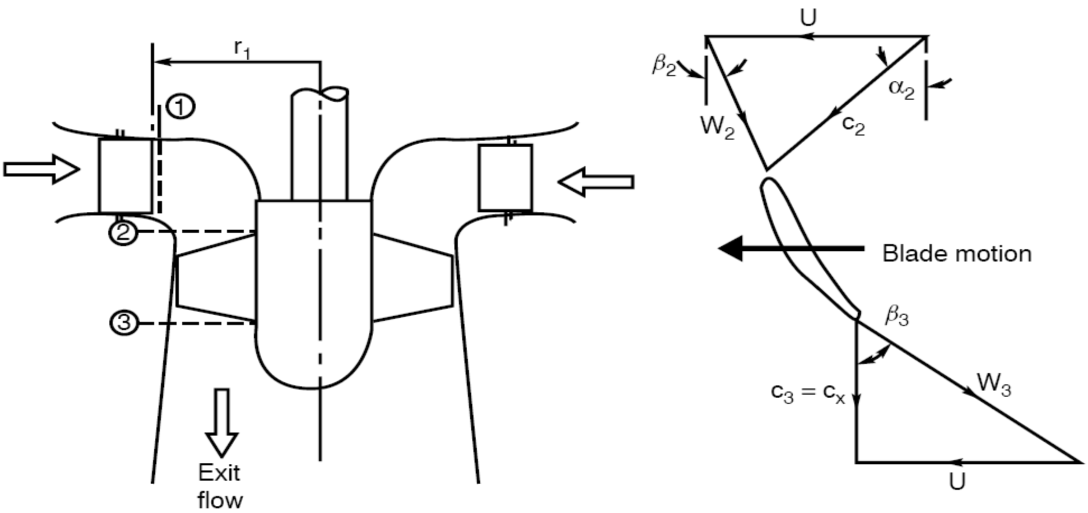

Figure 2.8 illustrate Section of a Kaplan

2.4.2 Kaplan turbine Details:

Kaplan turbine consists of the following parts:

1. Spiral or scroll air tight casing; the whole mechanism of the Kaplan turbine is closed in a casing called scroll casing. Scroll casing take water from the resource and direct it towards the blades with the help of the guide vanes.
2. Guide mechanism; Function of the wicket gate is to direct the flowing water toward the blades of the turbine.
3. Runner: consist of Blades of the Kaplan turbine are like the propeller. Blades of other axial turbine are planer but that of Kaplan are not planers instead they are of twist shape along the length to allow the swirling flow of water at entrance and exist.
4. Draft tube Waster after passing through the blades is take put of the turbine by the means of the draft tube. Draft tube decreases the velocity of the water by increasing its area**.**
5. Governor: Governing mechanism control the movement of the guide vanes. When power requirement is high it opens the guide vane and when power requirement is low is close the guide vanes.
6. Hub is the part on which blades of the turbine are attached which is attached to the central shaft of the turbine.
7. Shaft is Rotation motion of the blades is transfer to the generator by means of shaft which is a long solid part of the turbine.

2.4.2.1 Basic equations:

In the Euler for work done or energy transfer, in case of a series of radial curved vanes was derived as

work done per unit mass per second

W/unit mass/s =VwiUwi ±VwoUo **(2.3)**

orEnergy transfer,

E/unit mass/s =VwiUi ± VwoUo. **(2.4)**

This is the fundamental equation of hydraulic machines, turbines and pumps and is known as Euler's equation. The equation expresses the energy conversion in a runner. The equation in its present form indicates the energy transfer to the runner by fluid which gives motion to the runner, this principle of the turbine motion .And the fluid velocity is **(V)**, runner speed is **(U)** and the resultant is **(W).**

W= U-VE **(2.5)**

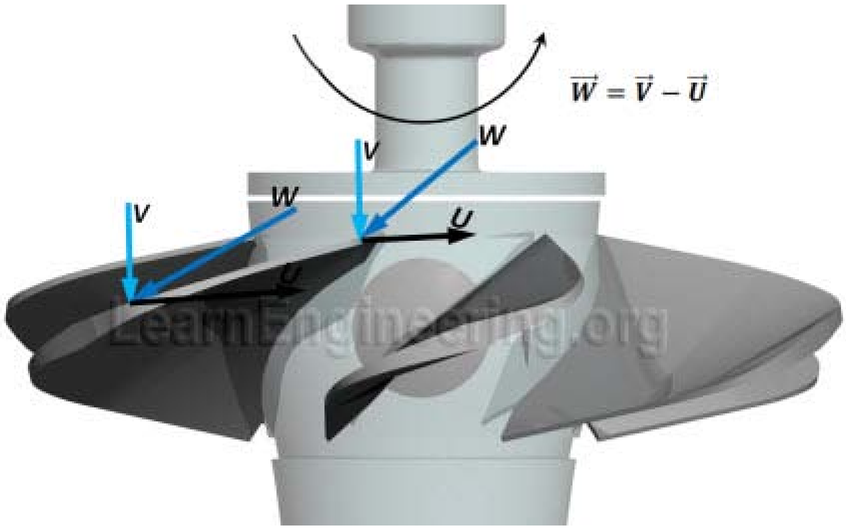

Figure 2.9 illustrate the velocities

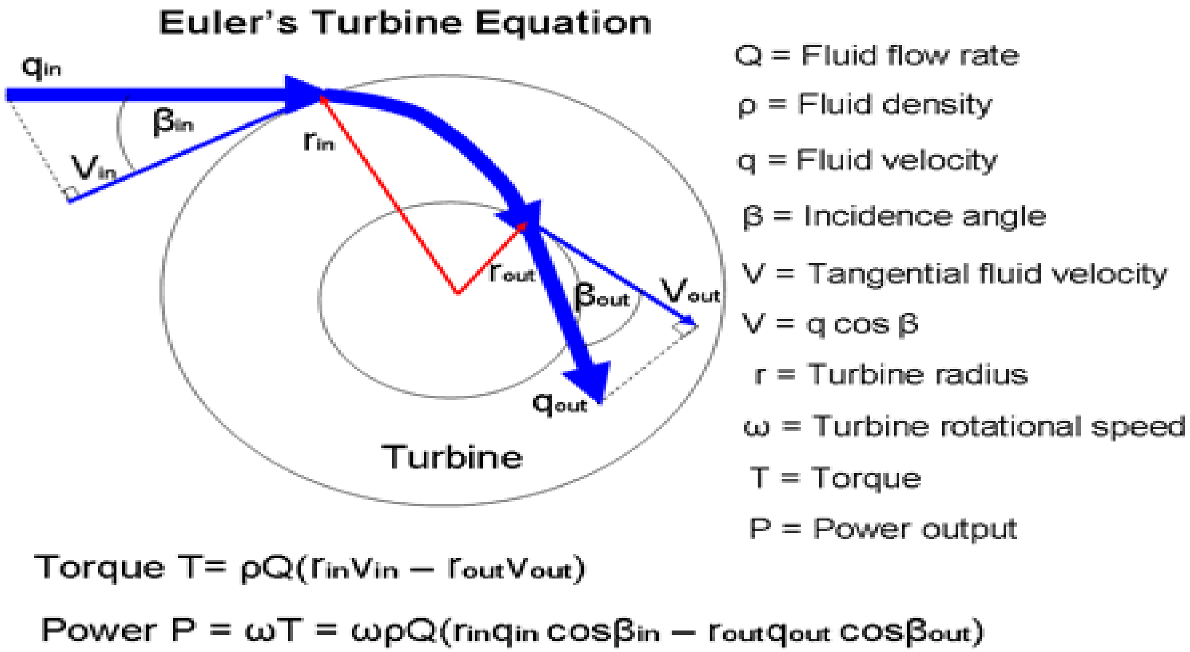

Figure 2.10 illustrate the Euler's equation

The principle of hydropower is extraction of potential energy from nature to convert it into mechanical energy and then to electrical energy by utilizing the head available and the discharge. Equation 1.1 shows the calculation of magnitude of the power developed.

Hydro Power = Efficiency x Pressure drop x Flow rate

$P = \eta \rho g Q H$ **(2.6)**

Here,

P = Power developed (watts)
= Efficiency of the turbine

ρ= density of the water (kg/m3)

g = Acceleration due to gravity (m/s2)

Q = discharge (m3/s)

H = available head (m)

The turbine's hydraulic capacity is determined by the specific speed of the turbine which is given by equation 1.2. It is the speed of a turbine of specific size which generates a power of 1 hp at 1 m of head.

$$\mathbf{Ns} = \frac{\mathbf{N} * \sqrt{\mathbf{Q}}}{(\mathbf{g} * \mathbf{H})\text{^}¾}$$

(2.7)

An axial turbine being low head and high discharge turbine will have high Specific speed. High specific speed of a turbine can be subjected to cavitations as well.
The cavitations' coefficient depends upon the value of specific speed. It can be calculated from the equation

$$\boldsymbol{\sigma} = \frac{(\mathbf{Ns} + \mathbf{30})\text{^}\mathbf{1.8}}{\mathbf{200,000}}$$

(2.8

Where, σ is cavitation index and Ns is specific speed of a turbine. This equation clearly explains that cavitation is directly proportional to the specific speed. Therefore high head plant demands low specific speed turbine to make it free from cavitations' (GrigoriKrivchenko, 1994). [15]

The efficiency of the hydro turbine is determined by the loss coefficients. There are mainly two types of losses: mechanical loss which is due to the friction imposed by the rotation of the turbine and impact loss.

2.4.2.3 Advantages of Kaplan Turbine:

- In Kaplan Turbine Runner vanes are adjustable.
- In Kaplan Turbine very low head is required.
- In Kaplan Turbine has very small number of blades 3 to 8.
- In Kaplan Turbine very less resistances have to be overcome.

2.4.2.4Disadvantages of Kaplan Turbine:

- In Kaplan Turbine disposition of shaft is only in vertical direction.
- In Kaplan Turbine very large flow rate is required.
- In Kaplan Turbine specific speed is 250-850.
- In Kaplan Turbine heavy duty generator is required.

The operation conditions by large flow rate may lead to many problems such as vibration, efficiency dropping, cavitation, and blade cracks caused by unstable flow in each of the flow passage components of the turbine seriously affect the safe operation of the unit and because of these problems, many power plants are forced to undergo downtime for repairs or renovation [9].some vortex formulated inside the turbine especially in the lower part of the hub and the tip clearance of the runner blade with the shroud. Pressure fluctuations are a serious problem in hydraulic machinery and are usually the result of a strong vortex created in a center of a flow at the outlet of a runner. The draft tube vortex appears at partial load operating regimes usually in radial turbines and also at single regulated axial turbines. The consequences of the vortex developed in the draft tube are very unpleasant pressure pulsation, axial and radial forces and torque fluctuation as well as turbine structure vibration. [11].

3.0 Literature Review

3.1 Vortex:

In fluid dynamics, a vortex is a region, in a fluid medium, in which the flow is mostly rotating around an axis line; Vortices are a special existence form of fluid motion with origin in the rotation of fluid elements. The vortices flow that occurs either on a straight-axis or a curved-axis. [12][13] Vortices are a major component of turbulent flow. In the absence of external forces, viscous friction within the fluid tends to organize the flow into a collection of irrotational vortices, possibly superimposed to larger-scale flows, including larger-scale vortices. In each vortex, the fluid's flow velocity is greatest next to its axis, and decreases, in inverse proportion, to the distance from the axis. The vortices (the curl of the flow velocity) are very high in a core region surrounding the axis, and nearly nil in the greater vortex; and the pressure drops with proximity to the axis of the vortex. Once formed, vortices can move, stretch, twist, and interact in complex ways. A moving vortex carries with it some angular and linear momentum, energy, and mass .In a stationary vortex, the streamlines and path lines are closed. In a moving or evolving vortex the streamlines and path lines are stretched by the overall flow into loopy but open curves. [14]

Fig 3.1 Water Vortex

A key concept in the dynamics of vortices is the vorticity, which defined as a vector that describes the local rotary motion at a point in the fluid, as would be perceived by an observer that moves along with it.

In theory, the speed of the particles (and, therefore, the vorticity) in a vortex may vary with the distance from the axis in many ways. There are two important special cases, however:

If the fluid rotates like a rigid body – that is, if the angular rotational velocity (Ω) is uniform, so that the speed of the particles (***u***) increases proportionally to the distance ***r*** from the axis –the flow would also rotate about its center as if it were part of that rigid body. In such a flow, the vortices are the same everywhere: its direction is parallel to the rotation axis, and its magnitude is equal to twice the uniform angular velocity (Ω) of the fluid around the center of rotation, the local rotation measured by the vorticity $\vec{\omega}$.

$$\begin{aligned} \vec{\Omega} &= (0,0,\Omega), \quad \vec{r} = (x,y,0), \\ \vec{u} &= \vec{\Omega} \times \vec{r} = (-\Omega y, \Omega x, 0), \\ \vec{\omega} &= \nabla \times \vec{u} = (0,0,2\Omega) = 2\vec{\Omega}. \end{aligned} \tag{3.1}$$

If the particle speed u is inversely proportional to the distance r from the axis, then the imaginary test ball would not rotate over itself; it would maintain the same orientation while moving in a circle around the vortex axis. In this case the vorticity is $\vec{\omega}$ zero at any point not on that axis, and the flow is said to be irrotational.

$$\begin{aligned} \vec{\Omega} &= (0,0,\alpha r^{-2}), \quad \vec{r} = (x,y,0), \\ \vec{u} &= \vec{\Omega} \times \vec{r} = (-\alpha y r^{-2}, \alpha x r^{-2}, 0), \\ \vec{\omega} &= \nabla \times \vec{u} = 0. \end{aligned} \tag{3.2}$$

3.1.1 Irrotational vortices:

In the absence of external forces, a vortex usually evolves fairly quickly toward the irrotational flow pattern [citation needed], where the flow velocity u is inversely proportional to the distance r. For that reason, irrotational vortices are also called free vortices.

For an irrotational vortex, the circulation is zero along any closed contour that does not enclose the vortex axis and has a fixed value, Γ, for any contour that does enclose the axis once.[15]The tangential component of the particle velocity is then:

$$u_\theta = \Gamma/(2\pi r). \tag{3.3}$$

The angular momentum per unit mass relative to the vortex axis is therefore constant,

$$r u_\theta = \Gamma/(2\pi). \tag{3.4}$$

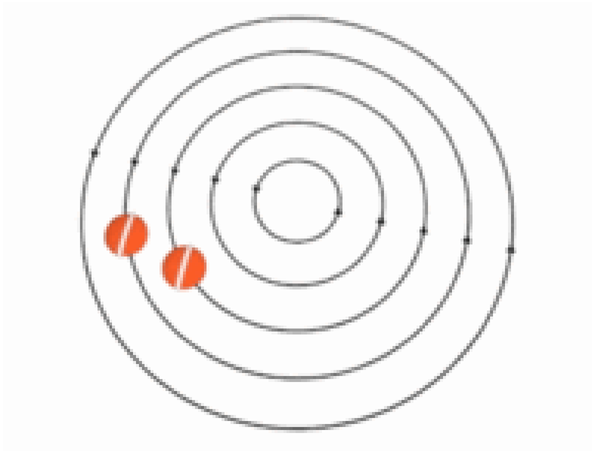

Figure: 3.2(a) Irrotational vortex flow

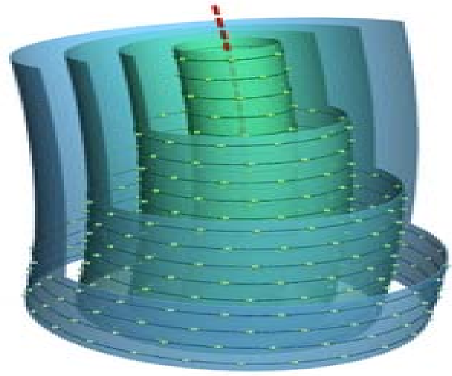

Figure: 3.2(b)Path lines of fluid particles

However, the ideal irrotational vortex flow is not physically realizable, since it would imply that the particle speed (and hence the force needed to keep particles in their circular paths) would grow without bound as one approaches the vortex axis. Indeed, in real vortices there is always a core region surrounding the axis where the particle velocity stops increasing and then decreases to zero as r goes to zero. Within that region, the flow is no longer irrotational: the vorticity $\vec{\omega}$ becomes non-zero, with direction roughly parallel to the vortex axis. The Rankin vortex is a model that assumes a rigid-body rotational flow where r is less than a fixed distance r0, and irrotational flow outside that core region. The Lamb-O seen vortex model is an exact solution of the *Navier–Stokes equations governing fluid flows and assumes cylindrical symmetry, for which*

$$u_\theta = (1 - e^{-r^2/(4\nu t)})\Gamma/(2\pi r). \qquad \textbf{(3.5)}$$

In an irrotational vortex, fluid moves at different speed in adjacent streamlines, so there is friction and therefore energy loss throughout the vortex, especially near the core.

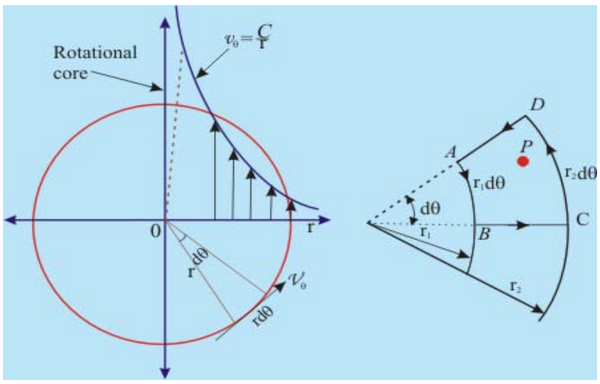

Figure 3.2(c) free vortex flow

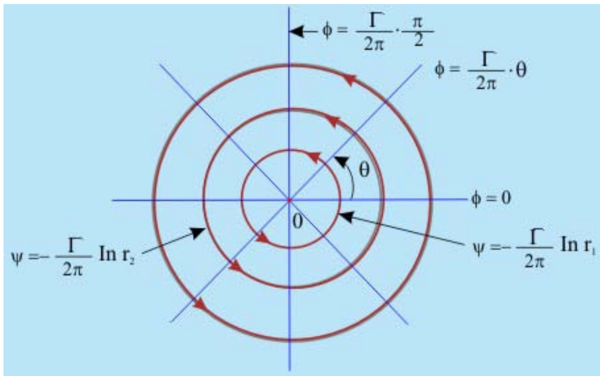

Figure 3.2(d) Flow net for a vortex

3.1.2 Rotational vortices:

A rotational vortex – one which has non-zero vorticity away from the core – can be maintained indefinitely in that state only through the application of some extra force, that is not generated by the fluid motion itself.

For example, if a water bucket is spun at constant angular speed ***w*** about its vertical axis, the water will eventually rotate in rigid-body fashion. The particles will then move along circles, with velocity ***u*** equal to ***wr***.[14] In that case, the free surface of the water will assume a parabolic shape.

In this situation, the rigid rotating enclosure provides an extra force, namely an extra pressure gradient in the water, directed inwards, that prevents evolution of the rigid-body flow to the irrotational state.

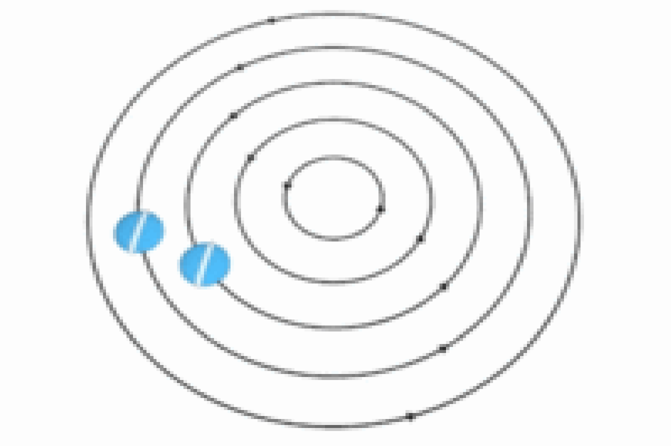

Figure 3.3 Rotational vortexes (A rigid-body)

In a stationary vortex, the typical streamline (a line that is everywhere tangent to the flow velocity vector) is a closed loop surrounding the axis; and each vortex line (a line that is everywhere tangent to the vorticity vector) is roughly parallel to the axis. A surface that is everywhere tangent to both flow velocity and vorticity is called a vortex tube. In general, vortex tubes are nested around the axis of rotation. The axis itself is one of the vortex lines, a limiting case of a vortex tube with zero diameters.

The fluid motion in a vortex creates a dynamic pressure (in addition to any hydrostatic pressure) that is lowest in the core region, closest to the axis, and increases as one moves away from it, in accordance with Bernoulli's Principle. One can say that it is the gradient of this pressure that forces the fluid to follow a curved path around the axis. In a rigid-body vortex flow of a fluid with constant density, the dynamic pressure is proportional to the square of the distance r from the axis. In a constant gravity field, the free surface of the liquid, if present, is a concave parabolic. In an irrotational vortex flow with constant fluid density and cylindrical symmetry, the dynamic pressure varies as $\mathbf{P\infty - K/r^2}$, where $P\infty$ is the limiting pressure infinitely far from the axis. This formula provides another constraint for the extent of the core, since the pressure cannot be negative. The free surface (if present) dips sharply near the axis line, with depth inversely proportional to $\mathbf{r^2}$.

3.2 Vortex in Kaplan Turbine:

Mainly the Kaplan turbine is used to create the power from the low head turbines to satisfy large power demands, very large volume flow rates need to be accommodated in the Kaplan turbine, i.e., the product (QHE) is large, so large flow rate is required from the Kaplan turbine. The water flow in the Kaplan turbine is in the radial direction, the flow is entered and exists axially. In the inlet of the turbine guide vanes are fixed. We can see the passage in between the rotor and guide vane which the flow is in the radial direction. Initially the flow must be in radial direction but the radial direction is forced to move in the axial direction. We can observe the similarity in between the rotor and propeller of a ship. To the central shaft of the turbine rotor blades are attached. With the help of moveable joints blades are connected to the shaft. The blades are rotated according to the water flow rate and the water head available. Compare to the other axial flow turbines the blades of the Kaplan turbine are not planer. So they are designed with a twist along the total length so it allows rotation of the water flow at the inlet and leaves at the axial flow. The overall flow configuration is from radial to axial; in which the flow enters from a volute into the inlet guide vanes, which impart a degree of swirl to the flow determined by the needs of the runner. The flow leaving the guide vanes is forced by the shape of the passage into an axial direction and the swirl becomes essentially a free vortex, i.e.

$$\mathbf{rc_{\varphi} = \text{a constant}} \qquad \mathbf{(3.6)}$$

The water from the scroll casing flows over the guide vanes. It is then deflected through 90° and enters the adjustable runner vanes,. The water enters with maximum potential energy and with little kinetic energy. It flows through the; blades in the axial direction .the force exerted on the vanes causes the shaft to rotate. In this turbine only 3to6 blades are used and they are fixed at equidistance.

The runner is in the form of boss having a bigger diameter. As the blades of the runner as well as guide blades can be adjusted during operation, the governing of the turbine is easy. The water after doing work passes on the tail race through a draft tube. The specific speed of this turbine is between 300 to 1000 rpm.

Due to the low water heads it allows the water flow at larger in the Kaplan turbine. With help of the guide vane the water enters. So the guide vanes are aligned to give the flow a suitable degree of swirl. The swirl is determined according to the rotor of the turbine. The water flow from the guide vanes are passes through the curved structure which forces the radial flow to direction of axial. The swirl is imparted by the inlet guide vanes and they are not in the form of free vortex as shown in Figure (3.4) . With a component of the swirl in the form of axial flow are applies forces on the blades of the rotor. Due to the force it loses both angular and linear momentum.

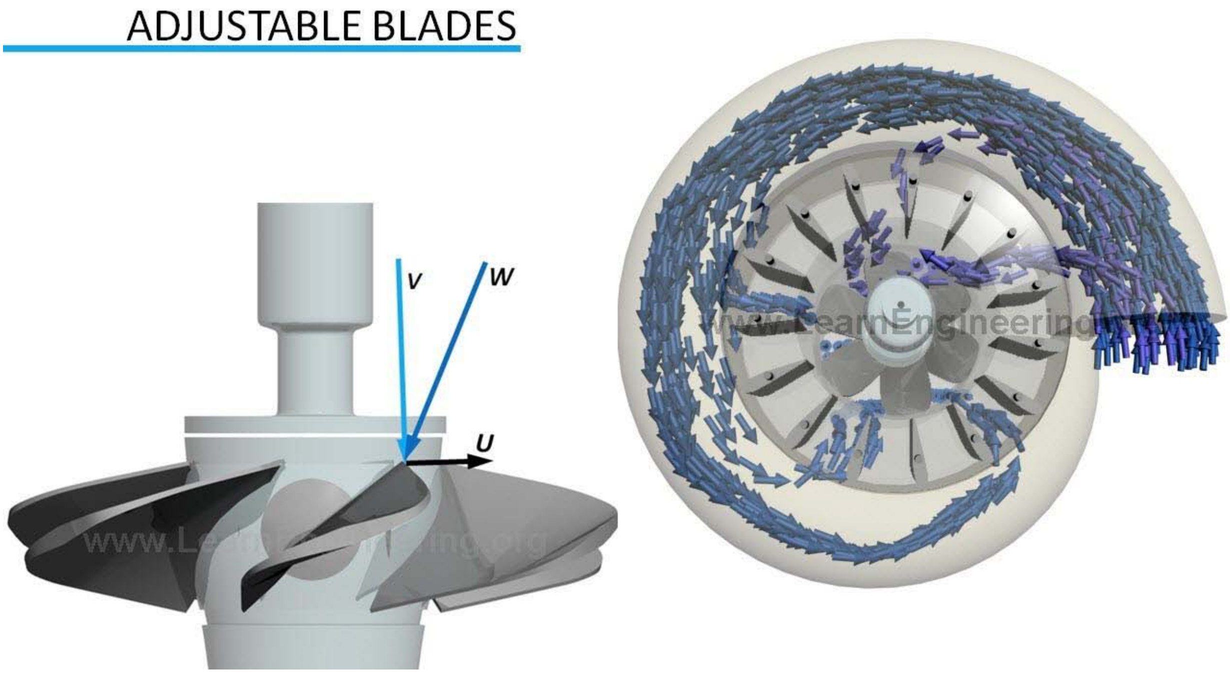

Figure (3.4) illustrate the water flow enter

It is more useful to study the water flow inside the turbine for more analysis and investigation to the chronic problem case which Lead to decrease the equipment efficiency.

3.3 Previous study:

There are several studies were done to predict the flow inside Kaplan turbine, most of them by numerical models, in the following paragraph the research will explain and introduce some of previous studies concerned about the vortex in blade tip and draft tube, such as follows:

1. Mikael Grekula and G¨oran Bark from Chalmers University of Technology, G¨oteborg, Sweden (CAV2001:sessionB9.004):

They conclude their study after the identification of the cavitation process on the suction side of the blades, it is suggested that there are re-entrant jets in the attached sheet cavities at the blade leading edge observed, and that they are of major importance to the shedding of cavitating vortices and cloud cavitation. Although the cavities at the blade root seem to be mainly of sheet type or travelling bubble type, depending on running condition, several facts indicate the contribution of vortex motion as well. However it was not possible to draw any final conclusion as to the type of these cavities. Some flow disturbances that most likely influence the root cavitations were also found. The cavitations tip vortex behaved strangely, possibly due to influence of wakes or vortices originating from the guide vanes. After its initial formation, the cavitations tip vortex leaves the blade surface, whereupon it turns back and approaches the blade surface again at a place where erosion damage is commonly reported to appear. Moreover, the tip vortex cavitation is characterized by fine -scaled cloud cavitation at the points where it approaches the blade surface, which indicates an erosion risk. Although some minor in homogeneity did modulate the root cavitation, no major flow in homogeneities were observed to influence the cavitation processes specifically.

2.VasileCojocaru, Daniel Balint, Viorel Constant in Campian, Dorian Nedelcu and Camelia Jianu show in their study which named Numerical Investigations of Flow on the Kaplan Turbine Runner Blade Anti cavitation Lip with Modified Cross Section (ISBN: 978-1-61804-020-6) that :

The results of numerical simulations on the Kaplan turbine runner blade with original and modified anti cavitation lip show a decrease of the pressure variation at the tips of the lip. This decrease is shown by the limit values of the pressure coefficient and by the gradient of the variation; also, for the lip with modified profile, the tip vortex detaches from the blade and the runner chamber, the tip vortex is separated in two components (a secondary vortex appears in the trailing edge area).Due to these aspects a decrease of the cavitation erosion is expected.

3. BjörnJedvik clarify in his Master's thesis which titled Evaluation of CFD Model for Kaplan Draft Tube, from Luleå University of Technology; that

(i) Using CFD for estimating pressure recovery in a low head draft tube is difficult. The simulations are difficult to converge and are very sensitive to input parameters.
(ii) When using an appropriate mesh, unstructured with y+-values not exceeding 515 and physically reasonable turbulence intensity as function of the radius, the simulations converge and give results that agree quite well experiments.
(iii) A steady-state simulation work as well as a transient and they predict nearly the same pressure recovery.

(iv) There is a strong interaction between the runner and the draft tube. A full stage simulation yields different velocity profile and turbulence intensity than split stage simulation (simulating the flow in the runner alone and use the output flow as input for a draft tube simulation).
(v) When validating simulations with experiments, the k-ε turbulence model performs better than the SST model, which seems to over-predict separation. The RSM model converged poorly.
(vi) The boundary layer thickness, shape or even presence does not influence the flow particularly. Anyhow, when validating to experiments, the thinner the boundary layers, the more accurate solutions.

4 **.**Bernd NENNEMANN and Thi C. VU were showed in their paper which titled (KAPLAN TURBINE BLADE AND DISCHARGE RINGCAVITATION PREDICTION USING UNSTEADY CFD) in 2nd IAHR International Meeting of the Workgroup on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems, Timisoara, Romania, October 24 - 26, 2007

The effect of the tip vortex in Kaplan turbine in the research concludes as follows:

Cavitation is a significant aspect in the design and operation of Kaplan turbines. Important types of cavitation in Kaplan turbines are leading edge, blade surface and discharge ring cavitation. In the present investigation we have primarily studied discharge ring cavitation. Blade surface cavitation is also looked at. Our primary method of investigation is Computational Fluid Dynamics (CFD). Experimental data that would allow us to quantitatively validate our method with respect to the specific physics is still unavailable. We have some experimental data that satisfactorily verifies the validity of the CFD calculations on a qualitative basis. High-speed camera movies show similar shapes and volumes of vapour as predicted by CFD. Unsteady pressure measurements on a model scale turbine show similar pressure distributions as predicted. Most importantly, the CFD results of this study permit us to explain at least one commonly found type of cavitation damage on prototype Kaplan turbines: discharge ring cavitation. The CFD results have significantly contributed to understanding the underlying physics. As such, CFD has proven to be a useful tool of exploring previously vaguely understood phenomena. Now that the CFD method is developed we are using it to improve future designs. We have performed extensive parameter studies to investigate all the significant influence factors. Areas that need to be addressed are:

• Scaling of cavitation intensity, vortex intensity and pressure fluctuations from model to prototype;

• Quantification of cavitation damage to be expected on prototype based on CFD calculations.

5.J.Decaix,M. Dreyer,C. Münch-Alligné and M. Farhat were showed presentation of their study regarding to the Experimental and numerical study of the tip vortex (Application to Kaplan Turbine) in SHF Conference on Cavitation, Hydraulic Machines. Air in Water Pipes. On June 5-6th, 2013, Grenoble, France.

They were compared the experimental and numerical results in tip vortex case which create severe erosion in axial turbines and lead to increase the gap at the tip of the blades and it would made a negative influence to the efficiency.

6. L motycak, A Skotak and R Kupcik clarify in their research which named Kaplan turbine tip vortex cavitation - analysis and prevention and they conclude that:

The flow within the gap has mainly negative effects. It decreases the efficiency of the runner because the water flowing through the gap does not generate any torque on the runner blades. Another negative effect is the cavitation pitting which results from above mentioned cavitation phenomena associated with the tip clearance.

a. The positive effect of the tip clearance could be found as well. In the case of the power plant rehabilitation, when the old runner is changed for the new one, the velocity profile downstream the runner is very important for proper draft tube flow or draft tube efficiency, the tip clearance causes the local increase of the axial and tangential velocity near the hub and chamber. This phenomenon usually suppresses the flow separation and stabilizes flow in a draft tube.

b. In Kaplan turbines, the tip clearance cavitation usually occurs as the first one from all types of cavitation. Depending on the head, the limited amount of the tip clearance cavitation and tip vortex cavitation is allowable in standard operating points.

4.0 Numerical Methods

4.1 CFD:

CFD is a branch of fluid mechanics that uses numerical analysis and algorithms to solve and analyze problems that involve fluid flow. It is an abbreviation for Computational Fluid Dynamics, and includes all Ways to solve fluid dynamics problems numerically, Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. With high-speed supercomputer, better solutions can be achieved. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial experimental validation of such software is performed using a wind tunnel with the final validation coming in full-scale testing, e.g. flight tests. [17]

The result of CFD analyses is relevant engineering data:

- Conceptual studies of new designs.
- Detailed product development.
- Troubleshooting.
- Redesign.

4.2 Background:

The fundamental basis of almost all CFD problems are the Navier–Stokes equations, which define any single-phase (gas or liquid, but not both) fluid flow. These equations can be simplified by removing terms describing viscous actions to yield the Euler equations. Further simplification, by removing terms describing vorticity yields the full potential equations. Finally, for small perturbations in subsonic and supersonic flows (not transonic or hypersonic) these equations can be linearized to yield the linearized potential equations.

Historically, methods were first developed to solve the linearized potential equations. Two-dimensional (2D) methods, using conformal transformations of the flow about a cylinder to the flow about an airfoil were developed in the 1930s.[17]

One of the earliest types of calculations resembling modern CFD are those by Lewis Fry Richardson, in the sense that these calculations used finite differences and divided the physical space in cells. Although they failed dramatically, these calculations, together with Richardson's book "Weather prediction by numerical process", set the basis for modern CFD and numerical meteorology. In fact, early CFD calculations during the 1940s using ENIAC used methods close to those in Richardson's 1922 book.[17]

The computer power available paced development of three-dimensional methods. Probably the first work using computers to model fluid flow, as governed by the Navier-Stokes equations, was performed at Los Alamos National Lab, in the T3 group. This group was led by Francis H. Harlow, who is widely considered as one of the pioneers of CFD. From 1957 to late 1960s, this group developed a variety of numerical methods to simulate transient two-dimensional fluid flows, such as Particle-in-cell method (Harlow, 1957), Fluid-in-cell method (Gentry, Martin and Daly, 1966), Vorticity stream function method (Jake Fromm, 1963), and Marker-and-cell method (Harlow and Welch, 1965). Fromm's vorticity-stream-function method for 2D, transient, incompressible flow was the first treatment of strongly contorting incompressible flows in the world. [17]

The first paper with three-dimensional model was published by John Hess and A.M.O. Smith of Douglas Aircraft in 1967. This method discretized the surface of the geometry with panels, giving rise to this class of programs being called Panel Methods. Their method itself was simplified, in that it did not include lifting flows and hence was mainly applied to ship hulls and aircraft fuselages. The first lifting Panel Code (A230) was described in a paper written by Paul Rubbert and Gary Saaris of Boeing Aircraft in 1968.[17]

The Navier–Stokes equations were the ultimate target of development. Two-dimensional codes, such as NASA Ames' ARC2D code first emerged. A number of three-dimensional codes were developed (ARC3D, OVERFLOW, CFL3D are three successful NASA contributions), leading to numerous commercial packages.[17]

4.3 Methodology:

In all of these approaches the same basic procedure is followed,
during preprocessing

- The geometry (physical bounds) of the problem is defined.
- The volume occupied by the fluid is divided into discrete cells (the mesh). The mesh may be uniform or non-uniform.
- The physical modeling is defined – for example, the equations of motion + enthalpy + radiation + species conservation
- Boundary conditions are defined. This involves specifying the fluid behavior and properties at the boundaries of the problem. For transient problems, the initial conditions are also defined.

- The simulation is started and the equations are solved iteratively as a steady-state or transient.
- Finally a postprocessor is used for the analysis and visualization of the resulting solution.

4.4 Discretization methods:

The stability of the selected discretization is generally established numerically rather than analytically as with simple linear problems. Special care must also be taken to ensure that the discretization handles discontinuous solutions gracefully. The Euler equations and Navier–Stokes equations both admit shocks, and contact surfaces, some of the discretization methods being used are:

4.4.1Finite volume method (FVM):

Is a common approach used in CFD codes, as it has an advantage in memory usage and solution speed, especially for large problems, high Reynolds number turbulent flows, and source term dominated flows."Finite volume" refers to the small volume surrounding each node point on a mesh. In the finite volume method, volume integrals in a partial differential equation that contain a divergence term are converted to surface integrals, using the divergence theorem. These terms are then evaluated as fluxes at the surfaces of each finite volume. Because the flux entering a given volume is identical to that leaving the adjacent volume, these methods are conservative. Another advantage of the finite volume method is that it is easily formulated to allow for unstructured meshes. The method is used in many computational fluid dynamics packages.

In the finite volume method, the governing partial differential equations (typically the Navier-Stokes equations, the mass and energy conservation equations, and the turbulence equations) are recast in a conservative form, and then solved over discrete control volumes. This discretization guarantees the conservation of fluxes through a particular control volume. The finite volume equation yields governing equations in the form,

$$\frac{\partial}{\partial t}\iiint Q\,dV + \iint F\,d\mathbf{A} = 0, \tag{4.1}$$

Where Q is the vector of conserved variables.

F is the vector of fluxes (see Euler equations or Navier–Stokes equations), V is the volume of the control volume element

$\mathbf{A}$ is the surface area of the control volume element

The finite volume method is used in ANSYS CFX to solve the Navier-Stokes Equation. It is based on the idea of dividing a continuum into finite control

Volumes. The idea is then to approximate the net flux through the CV boundaries as theSum of integrals over all surfaces.

4.4.2-Finite element method(FEM):used in structural analysis of solids, but is also applicable to fluids, the (FEM) formulation requires special care to ensure a conservative solution, and it has been adapted for use with fluid dynamics governing equations.(FEM) can require more memory and has slower solution times than the (FVM).

4.4.3- Finite difference method: the finite difference method (FDM) has historical importance and is simple to program. It is currently only used in few specialized codes, which handle complex geometry with high accuracy and efficiency by using embedded boundaries or overlapping grids (with the solution interpolated across each grid).

4.4.4- Spectral element method: Spectral element method is a finite element type method. It requires the mathematical problem (the partial differential equation) to be cast in a weak formulation. This is typically done by multiplying the differential equation by an arbitrary test function and integrating over the whole domain. Purely mathematically, the test functions are completely arbitrary - they belong to an infinite-dimensional function space. Clearly an infinite-dimensional function space cannot be represented on a discrete spectral element mesh; this is where the spectral element discretization begins. The most crucial thing is the choice of interpolating and testing functions. In a standard, low order (FEM) in 2D, for quadrilateral elements the most typical choice is the bilinear test or interpolating function of the form$v(x,y) = ax + by + cxy + d$. In a spectral element method however, the interpolating and test functions are chosen to be polynomials of a very high order (typically e.g. of the 10th order in CFD applications). This guarantees the rapid convergence of the method. Furthermore, very efficient integration procedures must be used, since the number of integrations to be performed in numerical codes is big. Thus, high order Gauss integration quadratures are employed, since they achieve the highest accuracy with the smallest number of computations to be carried out. At the time there are some academic CFD codes based on the spectral element method and some more are currently under development, since the new time-stepping schemes arise in the scientific world.

4.4.5- Boundary element method: in the boundary element method, the boundary occupied by the fluid is divided into a surface mesh.

4.4.6-High-resolution discretization schemes: High-resolution schemes are used where shocks or discontinuities are present. Capturing sharp changes in the solution requires the use of second or higher-order numerical schemes that do not introduce spurious oscillations. This usually necessitates the application of flux limiters to ensure that the solution is total variation diminishing.

4.5 Turbulence models:

A turbulence model is a procedure to close the system of mean flow equations, for most engineering applications it is unnecessary to resolve the details of the turbulent fluctuations, turbulence models allow the calculation of the mean flow without first calculating the full time-dependent flow field. We only need to know how turbulence affected the mean flow.

There are several numerical models for resolving turbulent flow, in computational modeling of turbulent flows; one common objective is to obtain a model that can predict quantities of interest, such as fluid velocity, for use in engineering designs of the system being modeled. For turbulent flows, the range of length scales and complexity of phenomena involved in turbulence make most modeling approaches prohibitively expensive; the resolution required to resolve all scales involved in turbulence is beyond what is computationally possible. The primary approach in such cases is to create numerical models to approximate unresolved phenomena. This section lists some commonly-used computational models for turbulent flows.

Turbulence models can be classified based on computational expense, which corresponds to the range of scales that are modeled versus, resolved (the more turbulent scales that are resolved, the finer the resolution of the simulation, and therefore the higher the computational cost). If a majority or all of the turbulent scales are not modeled, the computational cost is very low, but the tradeoff comes in the form of decreased accuracy. [17]

In addition to the wide range of length and time scales and the associated computational cost, the governing equations of fluid dynamics contain a non-linear convection term and a non-linear and non-local pressure gradient term. These nonlinear equations must be solved numerically with the appropriate boundary and initial conditions. [17]

The most accurate numerical models for resolving turbulent flow is called Direct Numerical Simulation (DNS). It resolves even the smallest eddies which are in the Kolmogorov scale, the theoretically smallest length scale in a flow. DNS is far too expensive in terms of computational time for practical use.

4.6 The DNS Approach

As we know, turbulent flow is 3-D in nature and the fluctuations are very rapid in space and time. Hence, it is required to keep the time step low as well as equally small grid spacing is required. But the problem comes into picture when, we have to simulate it. Low time steps and finer grid leads to exponentially increased computational time. However, there are attempts where, finer grids with small time steps are simulated without any numerical approximations. This approach is called DNS.

DNS stands for Direct Numerical Simulation. In DNS the N-S equations are directly solved, without further approximations, except the one, the fluid is Newtonian in nature. As we discussed earlier, this is computationally expensive and most of the time we do not need much-detailed results. Hence, there is a requirement of sufficiently less detailed results with lesser calculation. Lesser calculations mean lesser time and lower computational power requirements. Reynolds Averaging is one of the alternatives for this. Because the large scale motions are usually what carries the most energy and are the once that are the most important, a way of approximating DNS is to resolve only the large eddies in a flow up to the largest theoretical length scale, the Taylor scale. The method is called Large Eddy Simulation (LES) and assume the large eddies to dissipate energy to the small eddies in a predestined manner depending on the eddy viscosity.

4.7 RANS

Another simulation technique evolved from Reynolds's averaging, known as RANS. RANS stands for Reynolds Averaged Navier-Stokes equation. RANS forms the basis of simulation of turbulent flow in CFD. All the equations are time-averaged.[18]

The idea is to consider the instantaneous velocity of a particle, U ,as the sum of the ($\bar{u}$)mean velocity, , and the deviation, $\H{U}$

$$U = \bar{u} + \H{U} \quad \textbf{(4.2)}$$

The mean fluctuation over time is regarded to be zero.

Other models evolved are basically described as one-equation model and two-equation model. Examples being Spallart-Allmaras, K-Epsilon, K-Omega, K-Omega SST etc.

4.8 K-epsilon models:

The K-epsilon model is one of the most common used turbulence models in CFD, It is described in detail by Rodi (1980).one of the main strengths of the model is its universality, as it has been applied successfully to large number of different flow situations. Its empirical constants are relatively universal. Anthor characteristic is that it may over predict the turbulent eddy-viscosity for some situations. This increase the stability of the CFD model, but it may also lead to inaccurate results for some cases. Although it just doesn't perform well in cases of large adverse pressure gradients.[19] It is a model, which means it includes two extra transport equations to represent the turbulent properties of the flow. This allows a two equation model to account for history effects like convection and diffusion of turbulent energy.

The first transported variable is turbulent kinetic energy, k. The second transported variable in this case is the turbulent dissipation, ϵ. It is the variable that determines the scale of the turbulence, whereas the first variable, k, determines the energy in the turbulence.

There are two major formulations of K-epsilon models. [17]That of Launder and Sharma is typically called the "Standard" K-epsilon Model. The original impetus for the K-epsilon model was to improve the mixing-length model, as well as to find an alternative to algebraically prescribing turbulent length scales in moderate to high complexity flows. The K-epsilon model has been shown to be useful for free-shear layer flows with relatively small pressure gradients. Similarly, for wall-bounded and internal flows, the model gives good results only in cases where mean pressure gradients are small; accuracy has been shown experimentally to be reduced for flows containing large adverse pressure gradients. One might infer then, that the K-epsilon model would be an inappropriate choice for problems such as inlets and compressors.

4.9 Transport equations for standard k-epsilon model:

For turbulent kinetic energy k

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + P_k + P_b - \rho\epsilon - Y_M + S_k \tag{4.3}$$

For dissipation ϵ

$$\frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial}{\partial x_i}(\rho \epsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial \epsilon}{\partial x_j}\right] + C_{1\epsilon}\frac{\epsilon}{k}\left(P_k + C_{3\epsilon}P_b\right) - C_{2\epsilon}\rho\frac{\epsilon^2}{k} + S_\epsilon \tag{4.4}$$

4.10 Modeling turbulent viscosity:

Turbulent viscosity is modeled as:

$$\mu_t = \rho C_\mu \frac{k^2}{\epsilon} \tag{4.5}$$

Production of k:

$$P_k = -\rho \overline{u'_i u'_j} \frac{\partial u_j}{\partial x_i} \quad \textbf{(4.6)}$$

$$P_k = \mu_t S^2 \quad \textbf{(4.7)}$$

Where S is the modulus of the mean rate-of-strain tensor, defined as:

$$S \equiv \sqrt{2 S_{ij} S_{ij}} \quad \textbf{(4.8)}$$

Effect of buoyancy:

$$P_b = \beta g_i \frac{\mu_t}{\mathrm{Pr}_t} \frac{\partial T}{\partial x_i} \quad \textbf{(4.9)}$$

Where Prt is the turbulent Prandtl number for energy and gi is the component of the gravitational vector in the ith direction. For the standard and realizable - models, the default value of Prt is 0.85.

The coefficient of thermal expansion, β , is defined as

$$\beta = -\frac{1}{\rho} \left(\frac{\partial \rho}{\partial T} \right)_p \quad \textbf{(4.10)}$$

Model constants:

$$C_{1\epsilon} = 1.44, \quad C_{2\epsilon} = 1.92, \quad C_\mu = 0.09, \quad \sigma_k = 1.0, \quad \sigma_\epsilon = 1.3$$

4.11 Solver:

An extensive set of (CFD) solvers has evolved (and is forever growing) that are available to users. Some solver is direct and other is iterative, direct will establish a matrix for the system of equation, and invert it to obtain the solution. This is extremely time consuming from computational point of view and requires very large computer memory, so some type of iterative has been used. Commercial solver (ANSYS14.5) CFX has been used for this study.

5.0 Implementation:

Implementations of the model represent the methodology of the project, and we should execute the same basic procedure as follows

5.1 Geometry:

The geometry for Kaplan turbine build in cad format (Auto Cad and Solid works), the dimensions of the domain is scaled to Roessierse turbine. And imported as a Para solid. Recomposed using ANSYS (14.5).

The turbine data and dimension were presented in the following table.

NO.	Items	Description
1	Type of Turbine	Kaplan
2	Type of The runner	6 Blade -Vertical
3	Runner Diameter	4.5m
4	Spiral case Centerline	440.00m
5	Runner Centerline	438.165m
6	Rated Net head	33.0m
7	Discharge at rated Load	145.10m^3/h
8	Rated Out put	44 MW
9	Maximum Out put	48 MW
10	Synchronize Speed	136.4 rpm
11	Tip clearance	4(mm)
12	Spiral Case Pressure	4.2 (bar)
13	Draft Tube Pressure	0.225 (bar)
14	Net Head	42.1(m)
15	Upstream Level	486.85 (m)
16	Downstream Level	443.00 (m)

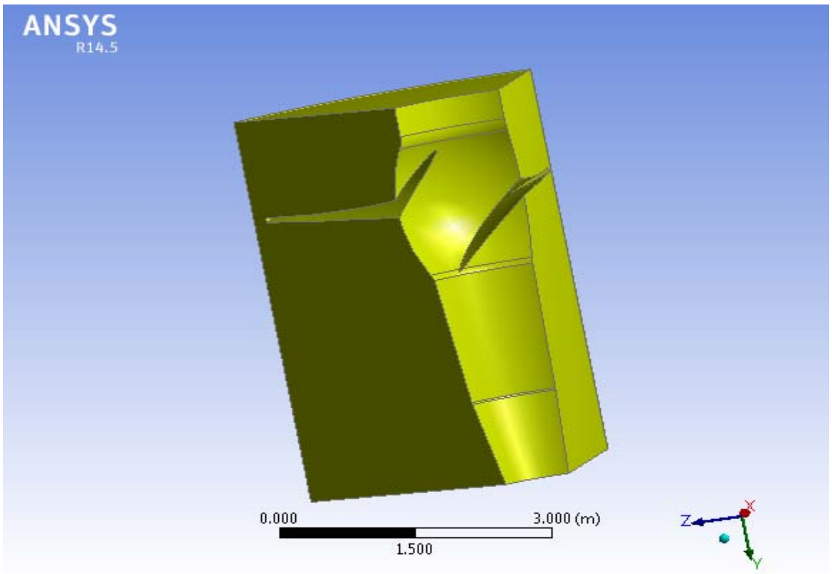

Figure 5.1 illustrate the domain in ANSYS

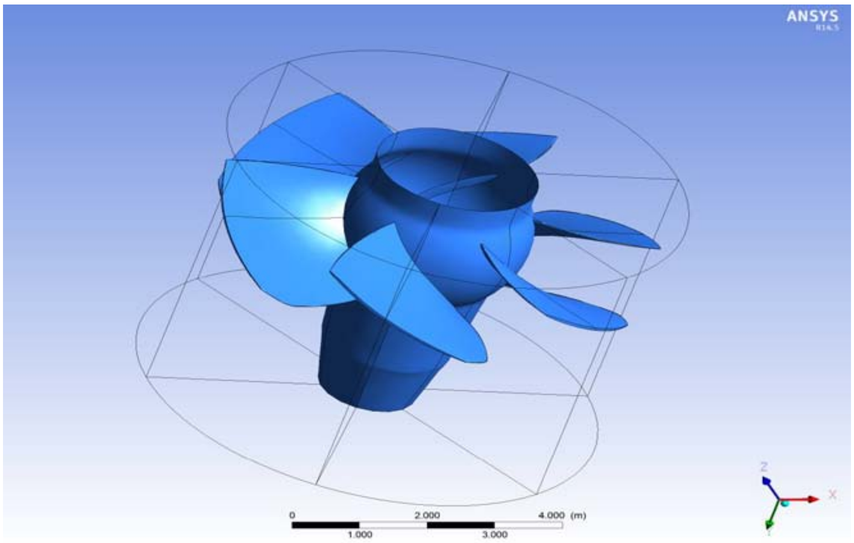

Figure 5.2 illustrate the geometry of Kaplan

5.2 mesh:

An unstructured mesh was used to discretize the domain by ANSYS (14.5) work bench meshing automatic feature. For the main part, the CFX mode was used and it made tetrahedrons and the part was divided into three parts with different elements sizes according to the relevance center of the element (fine, medium and coarse),and two smoothing level (high and low).

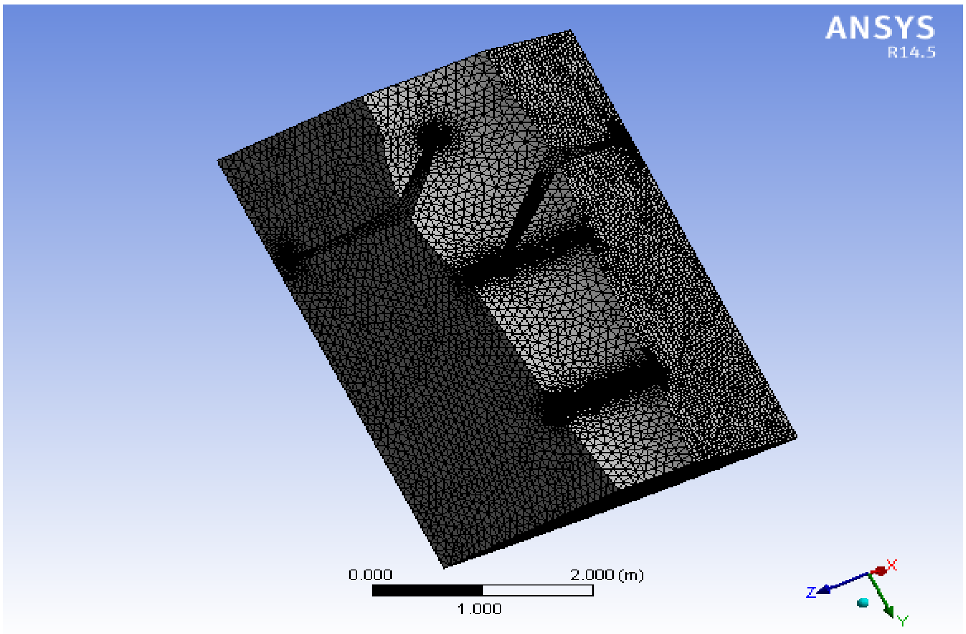

Figure 5.3 illustrate Fine unstructured mesh

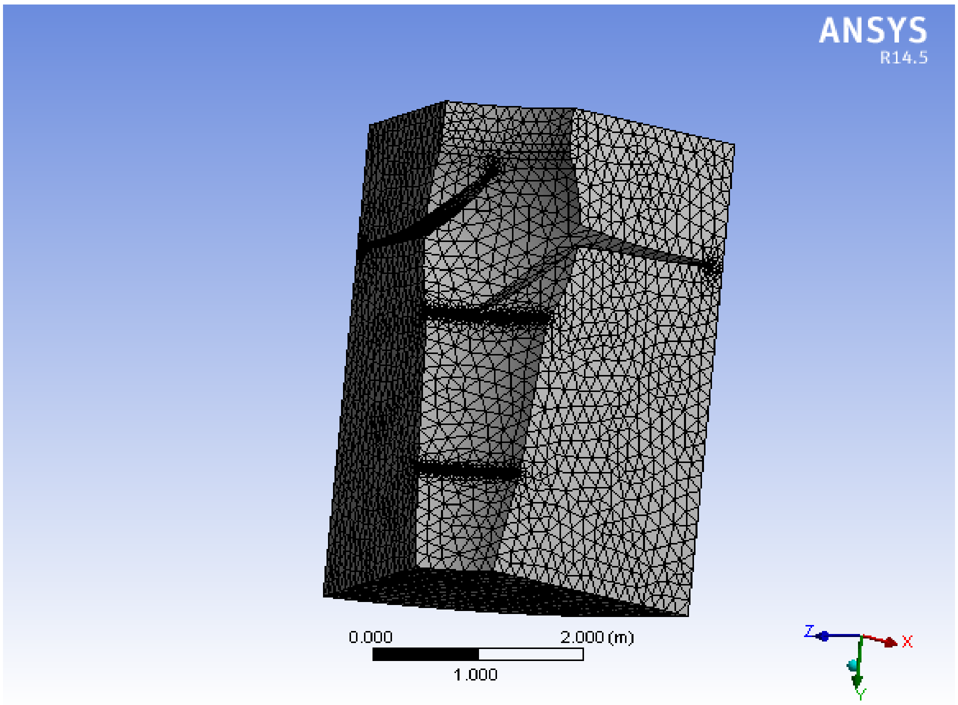

Figure 5.4 illustrate medium unstructured

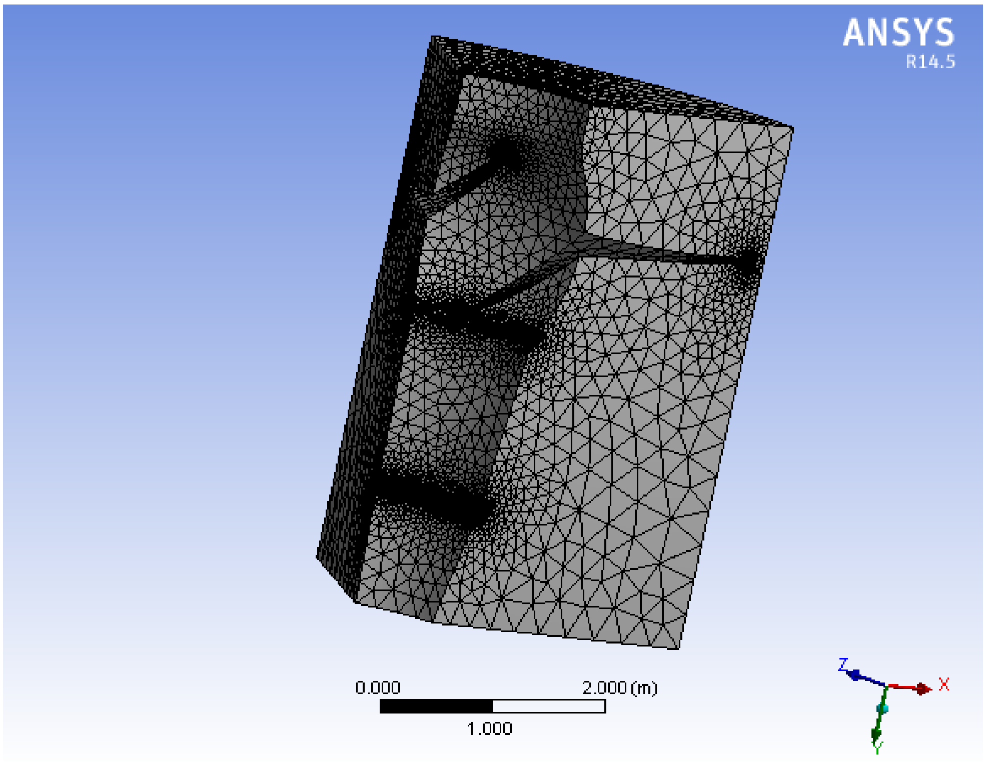

Figure 5.5 illustrate coarse unstructured

5.3 Pre-processing in ANSYS CFX

The pre-processing was made using (ANSYS CFX) and k-ε turbulence models was used; a constant turbulence intensity of 1% and high resolution turbulent numeric was used, minimum iteration one and maximum is 300. For the convergence criteria, the residual target is 0.000001.

A steady-state simulation was done, the hub velocity of 134 rev/min. The medium was set to water and the temperature to 25°C isothermal with zero value for reference pressure.

For the inlet flow the flow regime was subsonic and the boundary condition for the mass and momentum is total pressure with value varied from 4.5 to 1.5 bar and stationary frame type.

Outflow boundary was static pressure about 0.5 bar, subsonic flow regime, stationary frame type.

The hub and shroud were called wall as the boundary type, smooth roughness and with no slip wall, and it was rotated for the hub and stationary for shroud as the frame type.

5.4 Stage simulation:

A stage simulation on a turbine domain was performed from the guide vanes (inlet) until a small distance down the hub and consider as the top of the draft tube (Outlet).

The velocity profile, turbulence profile and pressure gradient were then exported and used as inlet profiles for runner simulations.

Another stage simulation on the same solution or simulation stage was performed from the power and efficiency as the result of the work done by the simulated flow.

5.5Postprocessing:

In order to make velocity profile and pressure gradient for the geometry, symmetric imitations for stream line, velocity vector and pressure contour were made for the Kaplan turbine default domain in the CFX model.

Some chart and tables were obtained from the post processing.

6.0 Results and discussion:

6.1 Mesh Investigation:

Three different unstructured meshes were made, the first one was fine relevance center and fine span center, used advanced size function on proximity and curvature with high smoothing, and normal curvature angle is 18.0°,minuimm size is 9.7437e^-4 m, and maximum size is 9.7437e^-2 m,second mesh arrangement was medium relevance center and medium span center, used advanced size function on proximity and curvature with medium smoothing, and normal curvature angle is 18.0°,minuimm size is 1.6634e^-3 m, and maximum size is 0.33268m .And the last one was coarse relevance center and coarse span center, used advanced size function on proximity and curvature with low smoothing, and normal curvature angle is 18.0°,minuimm size is 3.3268e^-3 m, and maximum size is 0.66536 m.

The results from the above meshes were obtained and compared with their efficiencies to distinguish them, as shown in the following table.

NO	Mesh Type	Node	Element	Iterations	Efficiency
1	Fine Mesh	229278	1287314	220	89.6
2	Medium Mesh	103530	556701	255	89.3
3	Coarse Mesh	58529	280248	290	88.8

It noted that the mesh quality affects the convergence, and the iteration decreases in the fine mesh but the time was increase due to huge for big matrix calculation, number of nudes and element were increase and the efficiency result was improved .in the medium mesh it was observed the number of nodes and elements less than fine mesh and the efficiency less than the result in fine mesh, number of iteration also was increased, but the calculation time was less than the fine mesh calculation time. And it was similar criteria for coarse mesh.

Other factor were affected the mesh quality such mesh refinement and mesh inflation like growth rate ,inflation algorithm, number of maximum layers ,transition ratio and etc…

The figure below show the mesh quality of the fine mesh in mid span of the runner blade (Default Domain).

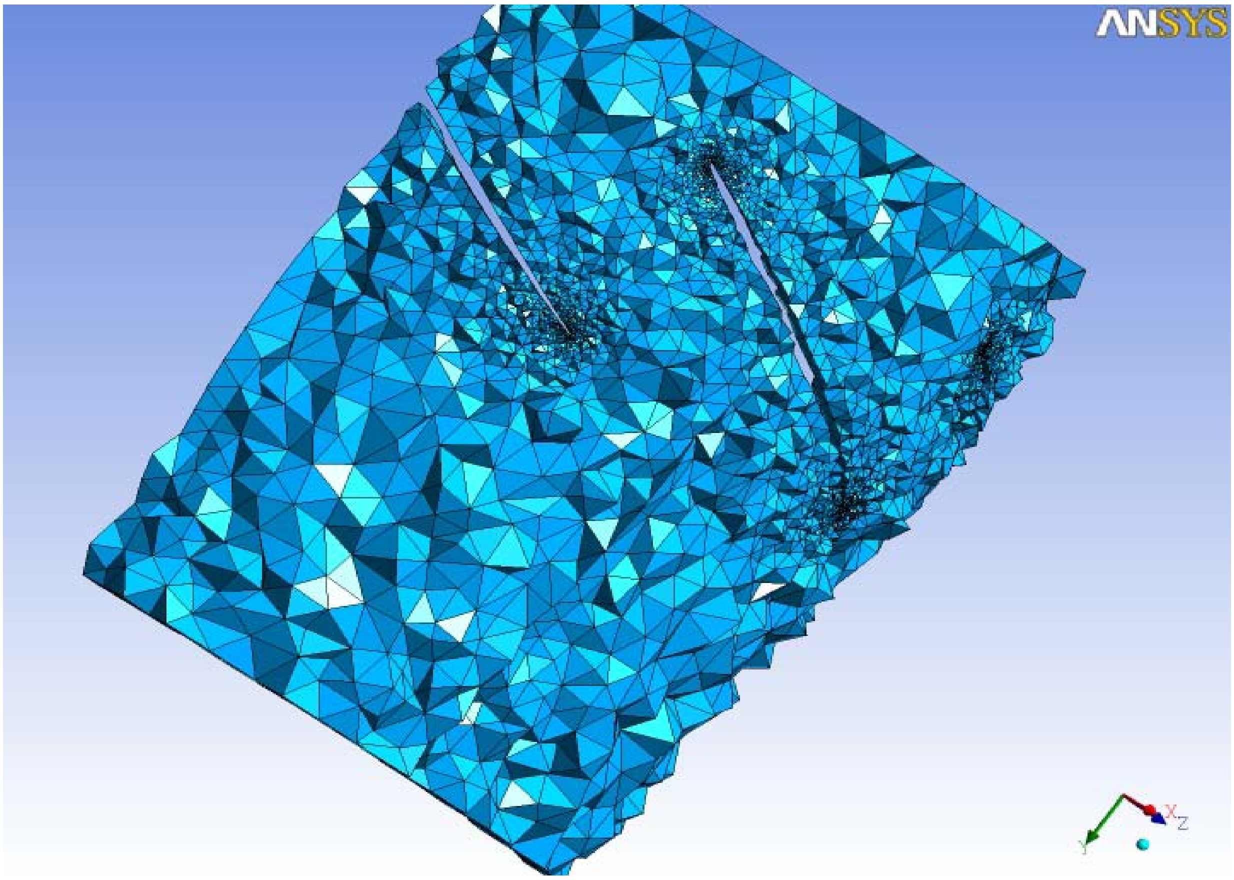

Fig 6.1 Mesh in domain

6.2Simulation Results:

The simulations were run by commercial solver (ANSYS CFX14.5), with different unstructured mesh, k-ε turbulence model, with inlet total pressure and outlet static pressure. In order to understand how Kaplan turbines affected by the vortex.velocity and pressure profiles were conduct, the convergences were accomplished with a good accuracy. The model is getting almost good results, The stream line profile gave a positive sign for the existing of the tip vortex, the pressure gradient from the hub into the runner chamber shows the lowest value at the bottom blade tip(suction side) which matching with the actual situation in Reoserisse turbine .

The figure below illustrated the simulation result for the Kaplan turbine runner blades and hub in 3D view.

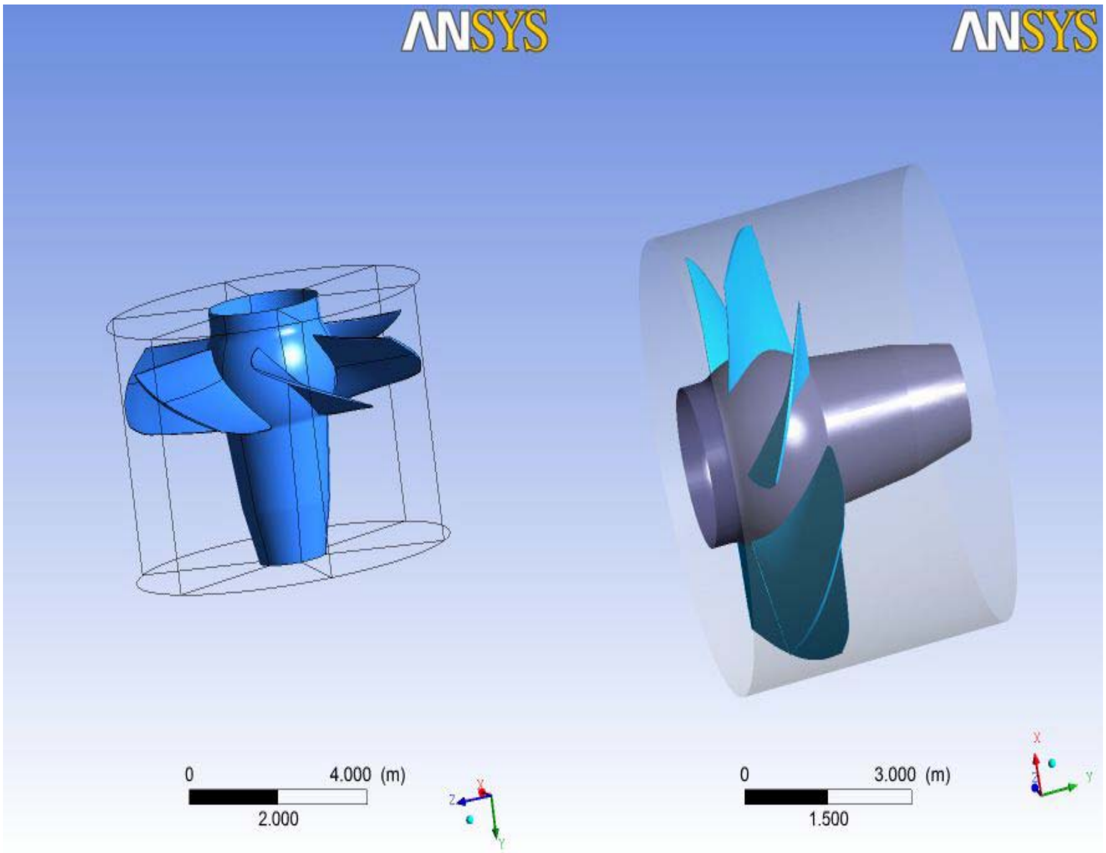

Figure 6.2 illustrate 3D view for Kaplan

The figure below show the stream line in Kaplan turbine simulation performed by ANSYS 14.5.

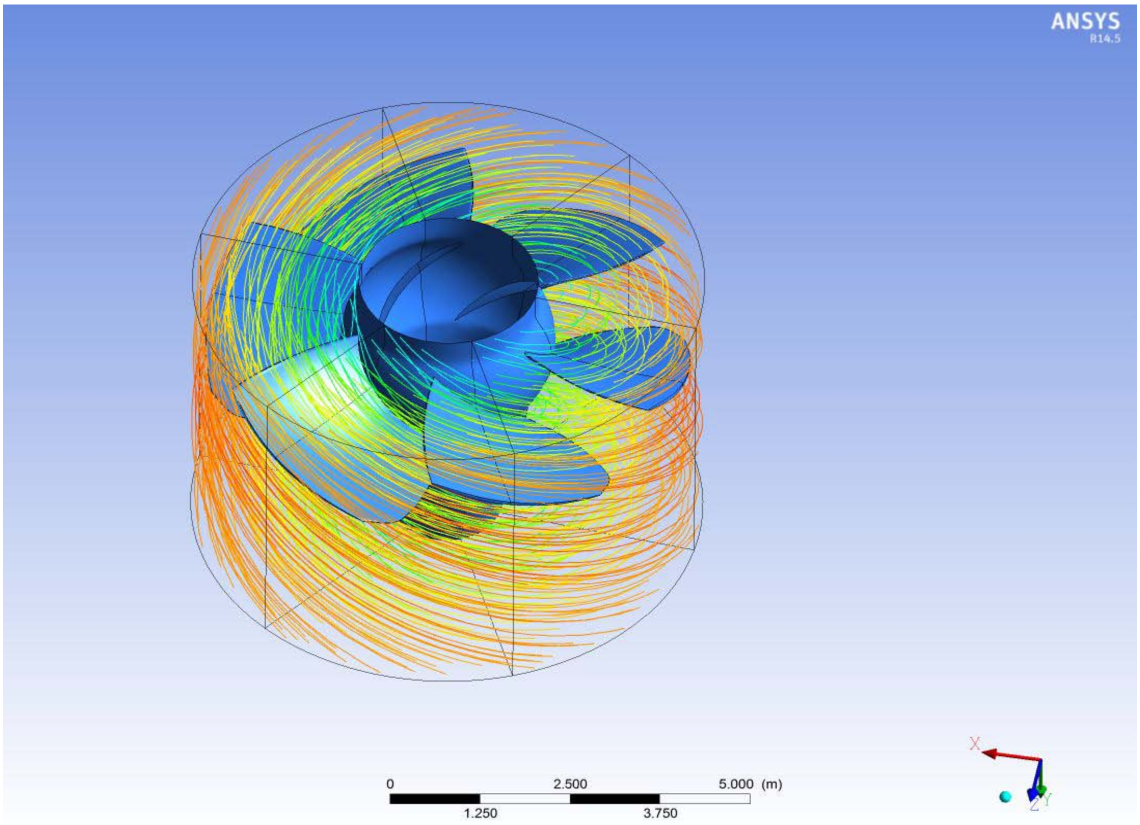

Fig 6.3 show the stream line in Kaplan

The streamline shows vortices develop in the runner chamber as shown in Fig 6.3 and Figure 6.4 shows the velocity vector which indicates the potential and expected erosion area and it proofed the vortex intensity.

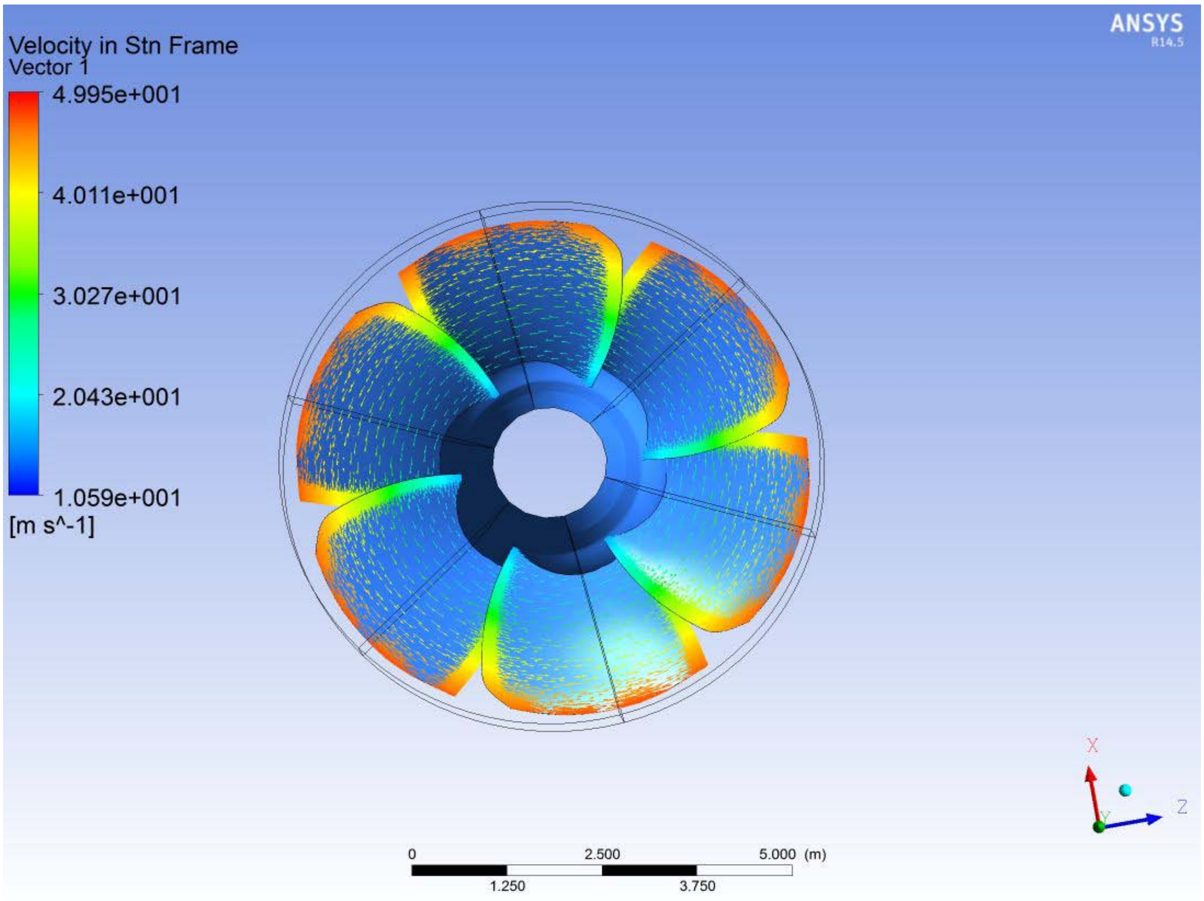

Fig 6.4 illustrate the stream line vector

The input variables for the CFD analysis are used from the actual data of Ressories turbine in order to compare with the simulation output. The objective of the study is to plot the pressure, velocity and performance curve of the turbine for different head and compare the result obtained from the operation data. The boundary condition for the simulation is chosen as total pressure inlet and static pressure outlet.

Total pressure = static pressure +dynamic pressure

The simulations were conducted for 45m of head which is the net head of the turbine. Six cases for different head were run in CFX with boundary conditions as total pressure at inlet and static pressure at outlet. The data were extracted from the cases shows the velocity stream line in CFX.. The power developed by the turbine is directly proportional to the net head developed between inlet and outlet of the turbine which can be expressed by the following equation. As

$$P = \int gHQ$$

Performance curve was plotted for different speed to get the optimum speed as shown in Figure 6.5.

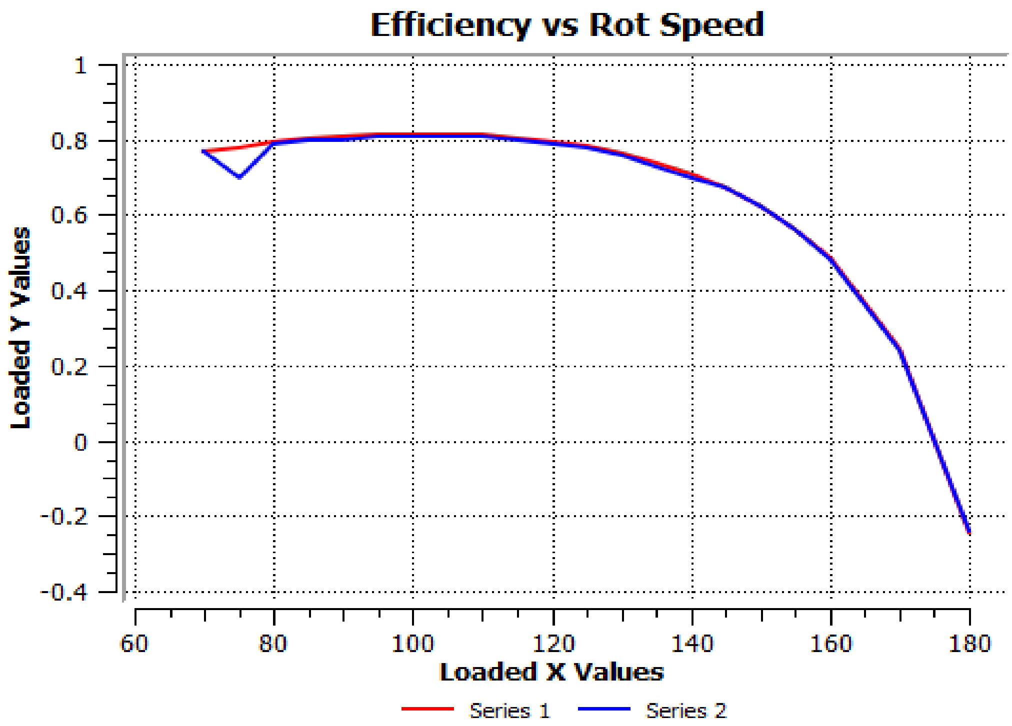

Figure 6.6 show the efficiency vs the speed

The figure below show the turbine efficiency. The differences in the efficiencies between actual and simulation might be due to tip clearance between the runner and runner chamber.

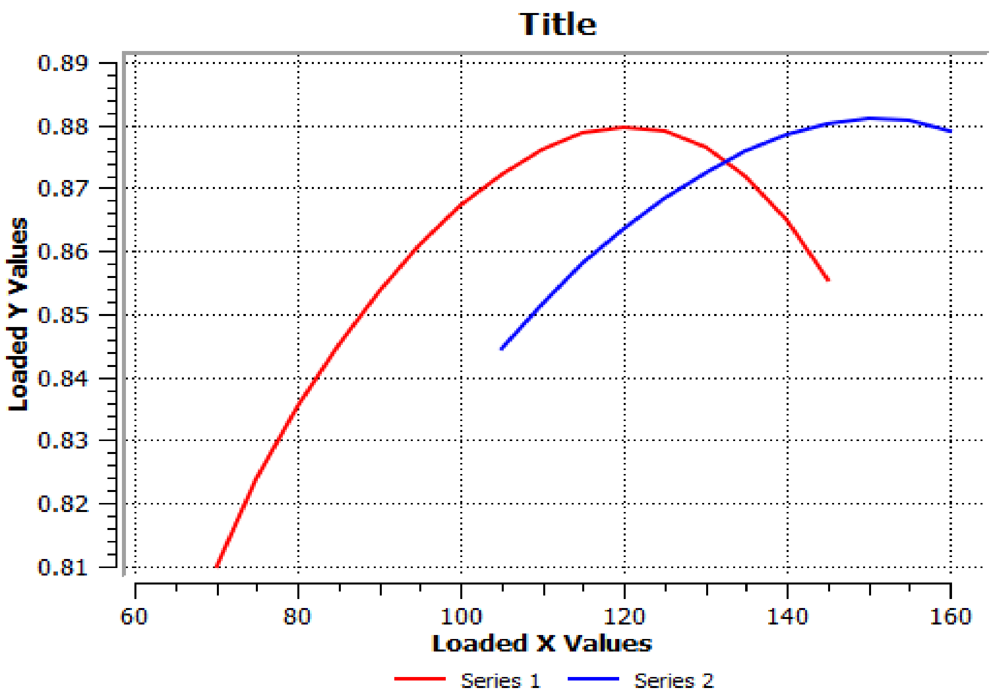

Figure 6.7 show the Turbine efficiency

Figure 6.8 shows the pressure along the pressure side and suction side of the runner blade. It is seen clearly that the negative pressure zone is observed on the suction side of the blade. And Fig 6.9 presented the pressure gradient along the blade in the area extended from the hub to the runner chamber.

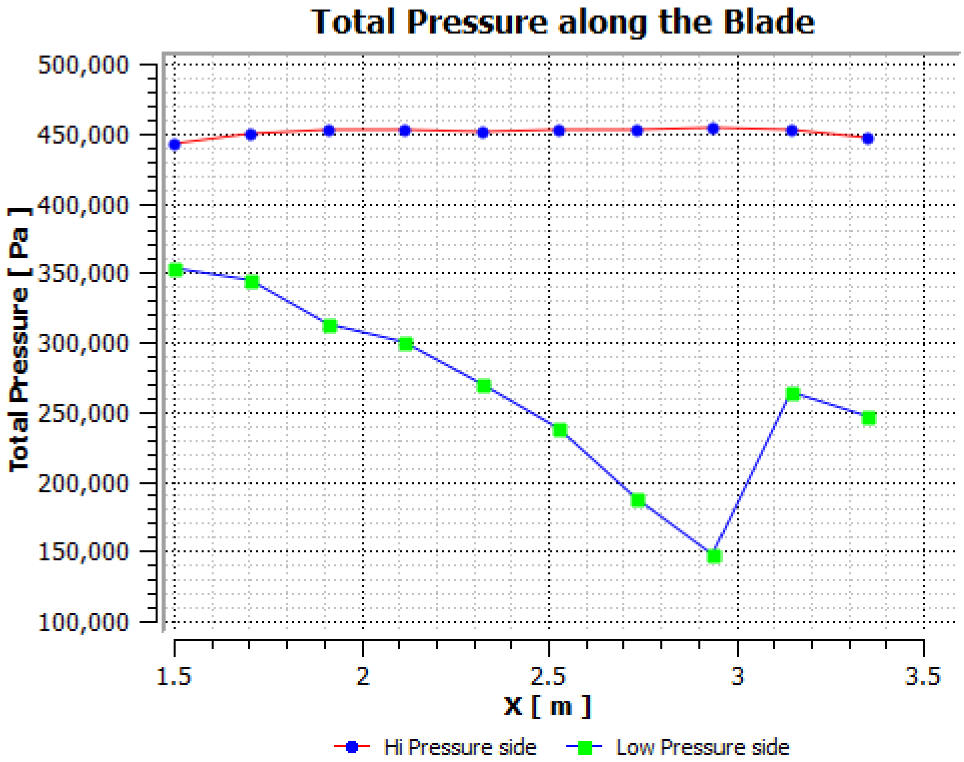

Fig 6.8 show the pressure along the blade

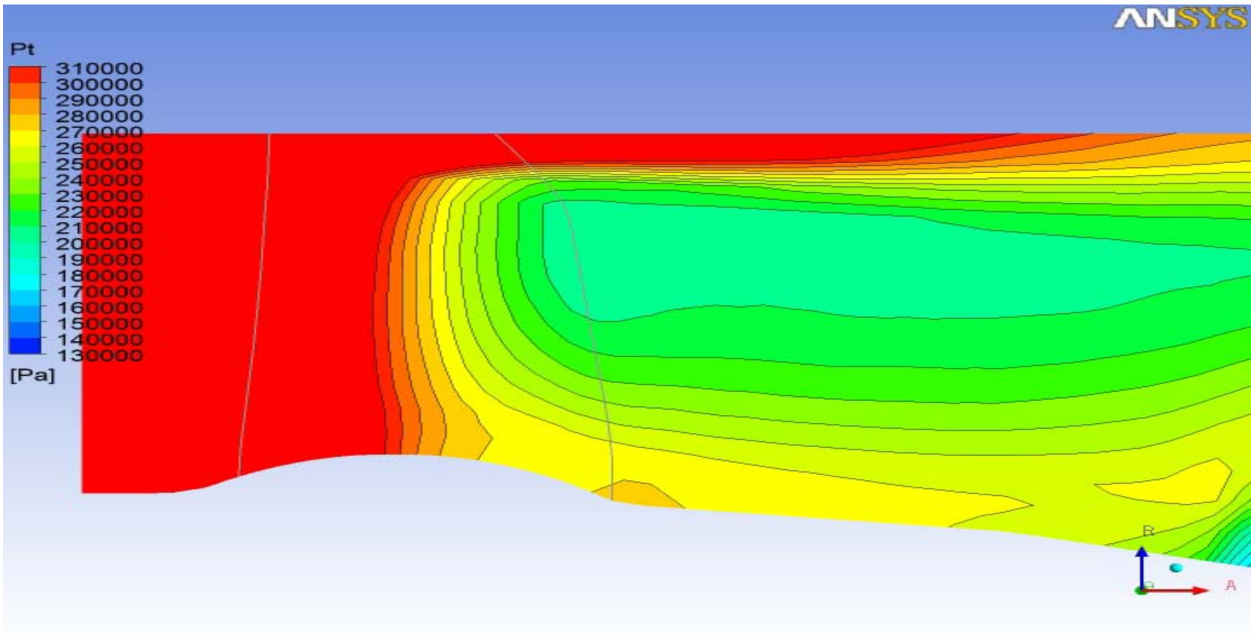

Fig 6.9 show the pressure contour

Figure 6.10 shows the pressure contours on the blade and hub, and the pressure gradient was noted along the blade.

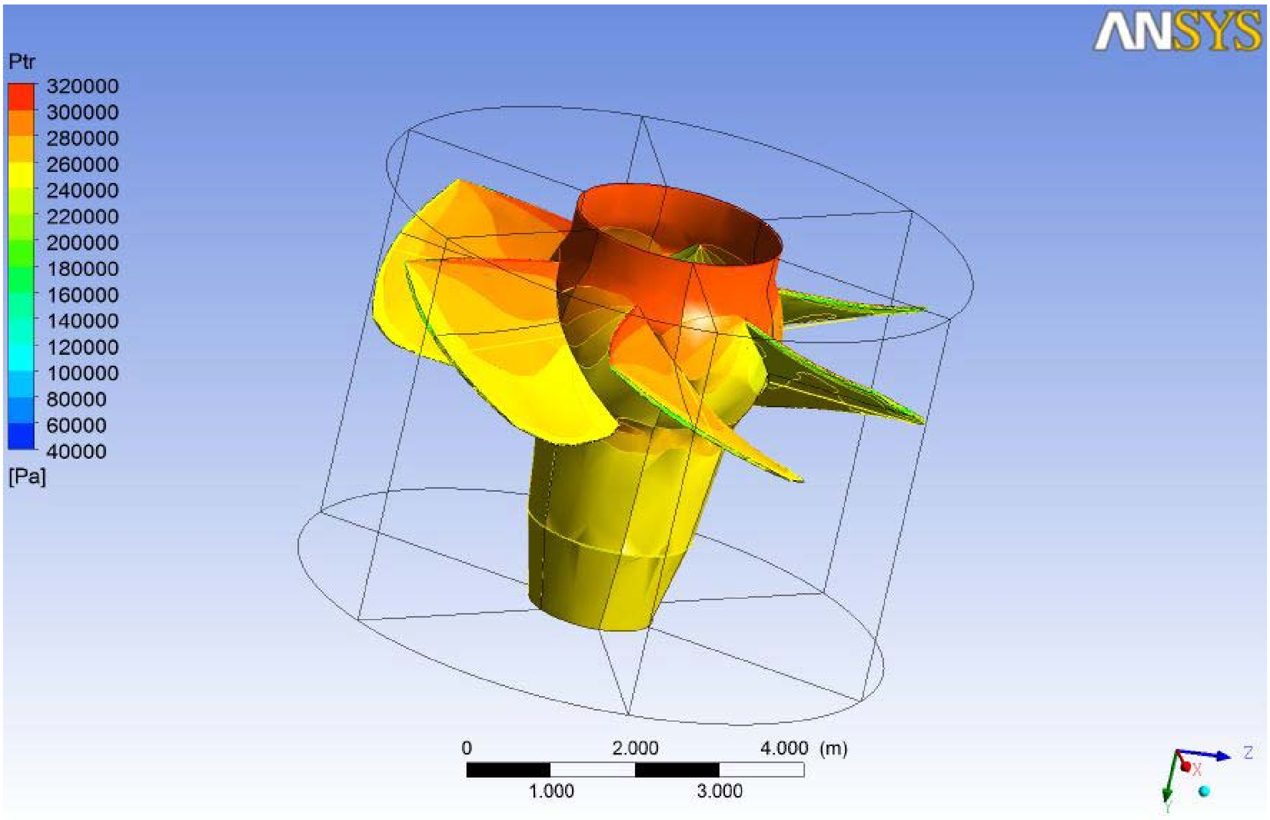

Figure 6.12 show pressure contours

Figure 6.11and 6.12 were show the pressure contours on mid span of the runner blade and indicates negative pressure formation at the suction side of the leading edge of the blade. Negative pressure distribution on the suction side of the blade is mainly due to the dynamic action of the blade. High velocity of the water on this location creates a low pressure area. When local static pressure becomes less than the vapor pressure, cavitations will take place. It is evident from the contours plot that there is an occurrence of cavitations due to vortices and figure 6.13 a and figure 6.13 b were show the pressure gradient which indicate the existing of vortex. As RPM increases with increasing discharge, negative pressure is developing on the leading edge as well as on the trailing edge of the blade as shown in figure 6.14.

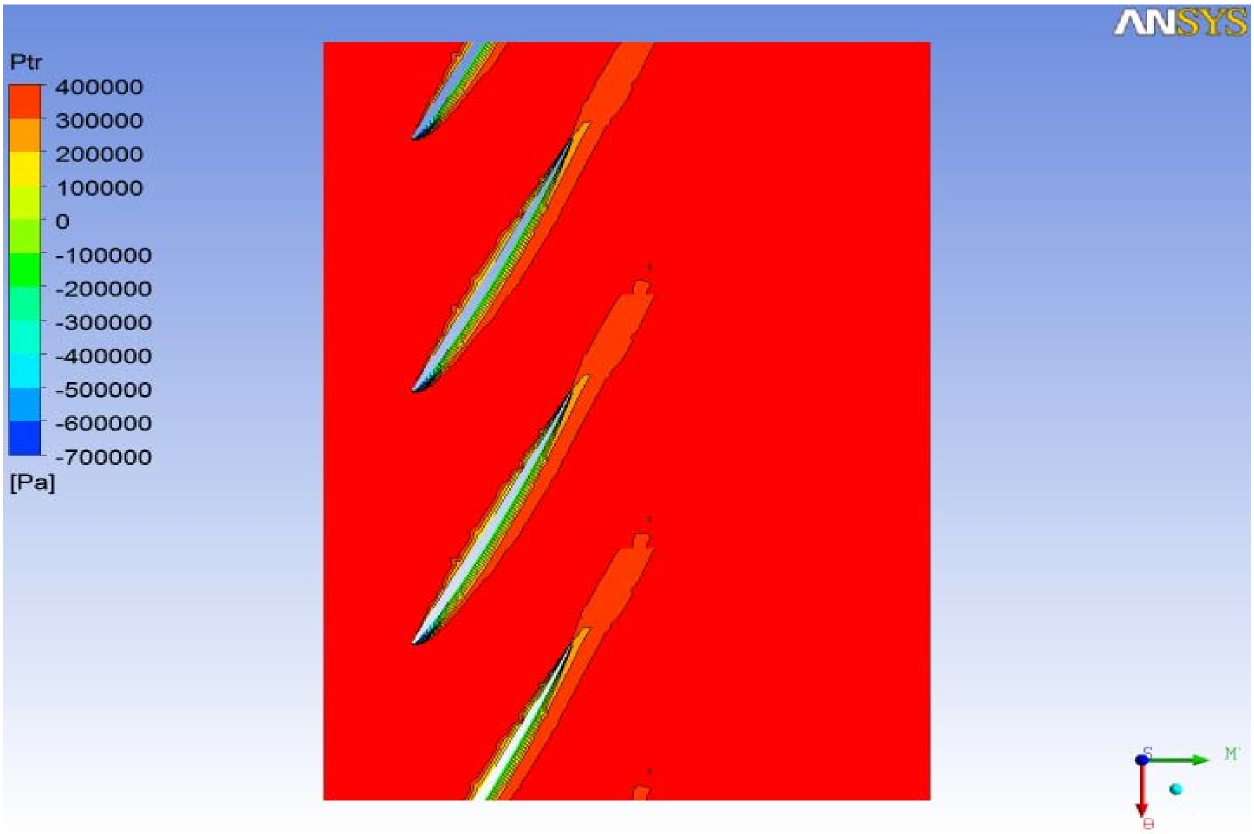

Figure 6.11 shows the pressure contours

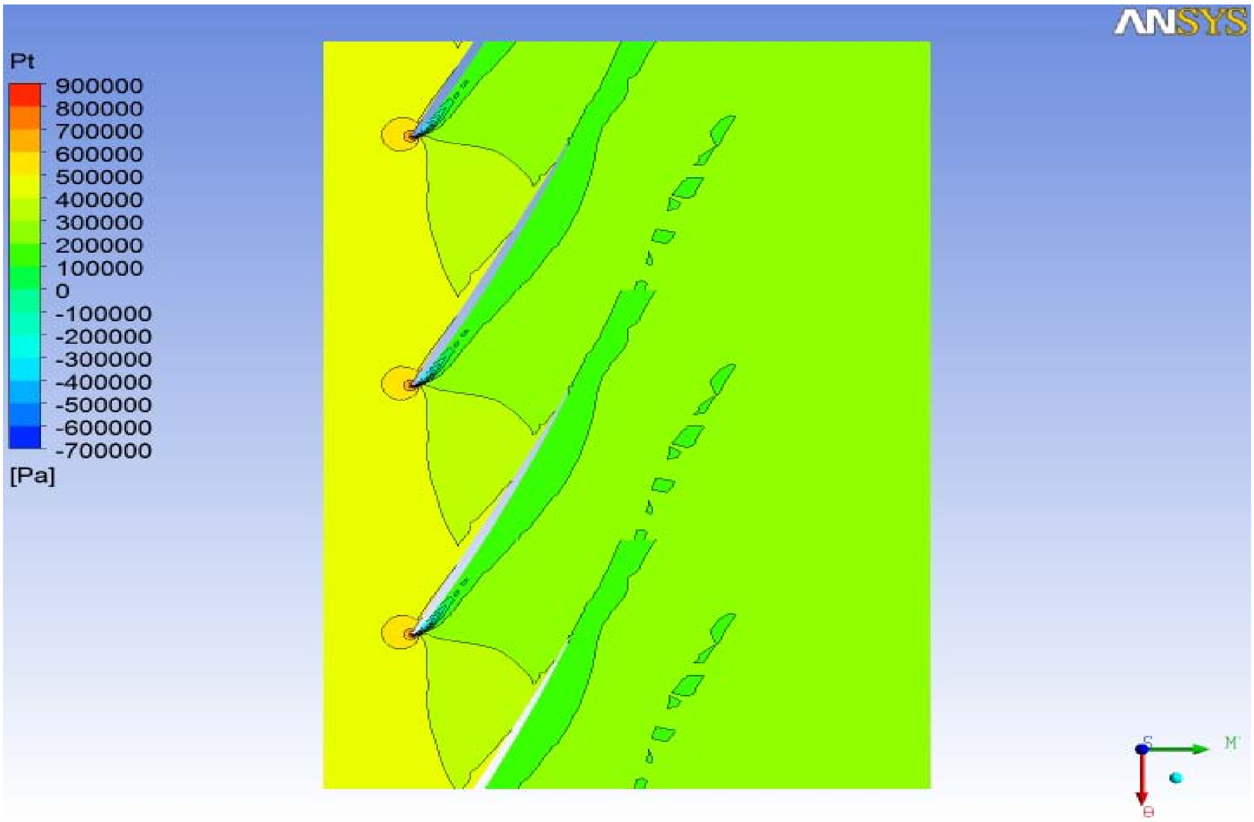

Figure 6.12 shows the pressure contours

The figure below shows presence of negative pressure developed at the suction side of the leading edge of the blade. This depicts cavitation formation during the operation of the turbine. Hence, to account for losses caused by cavitation.

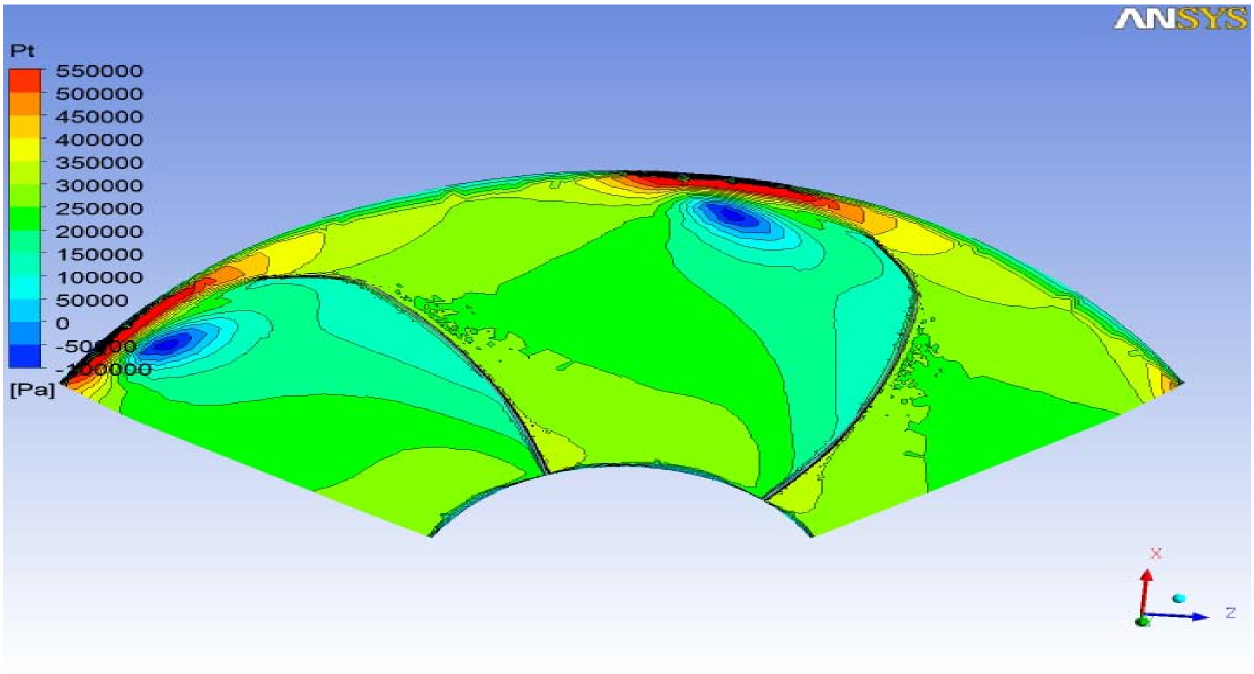

Figure 6.13a shows the negative pressure

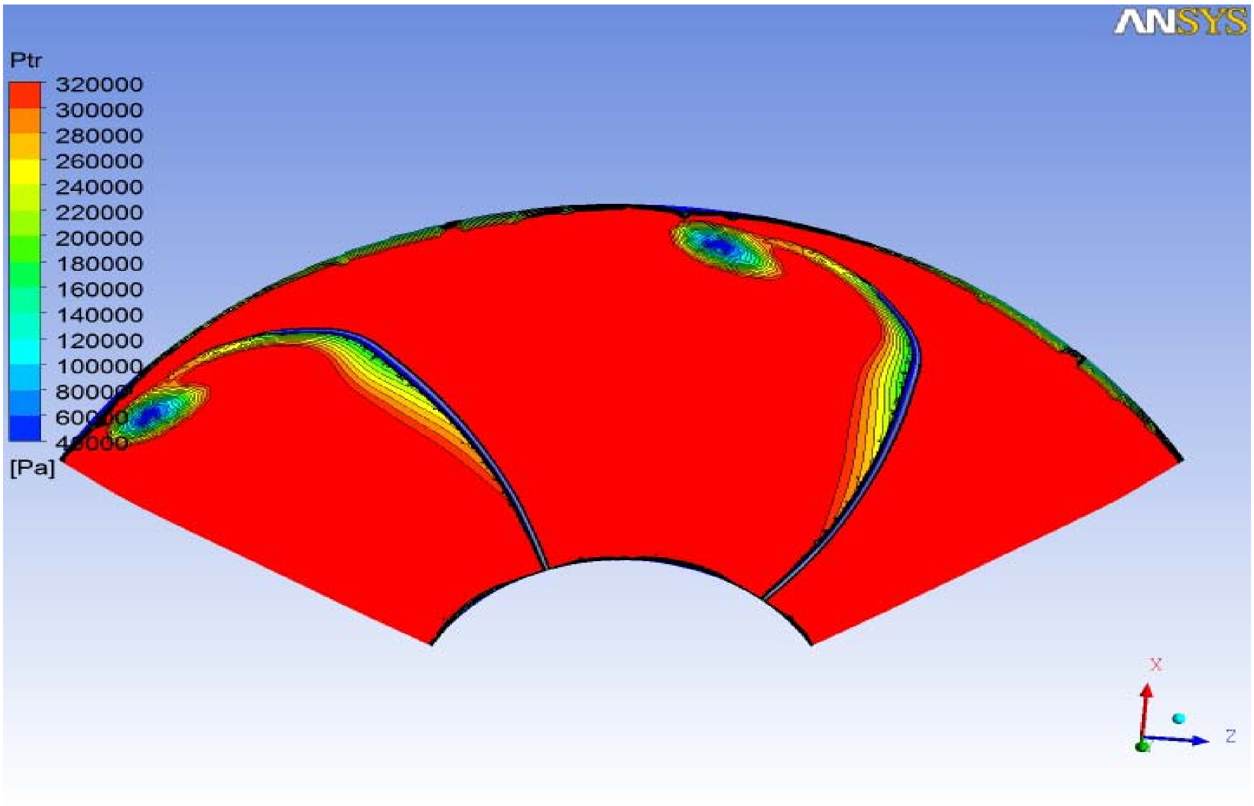

Figure 6.13b shows the pressure gradient

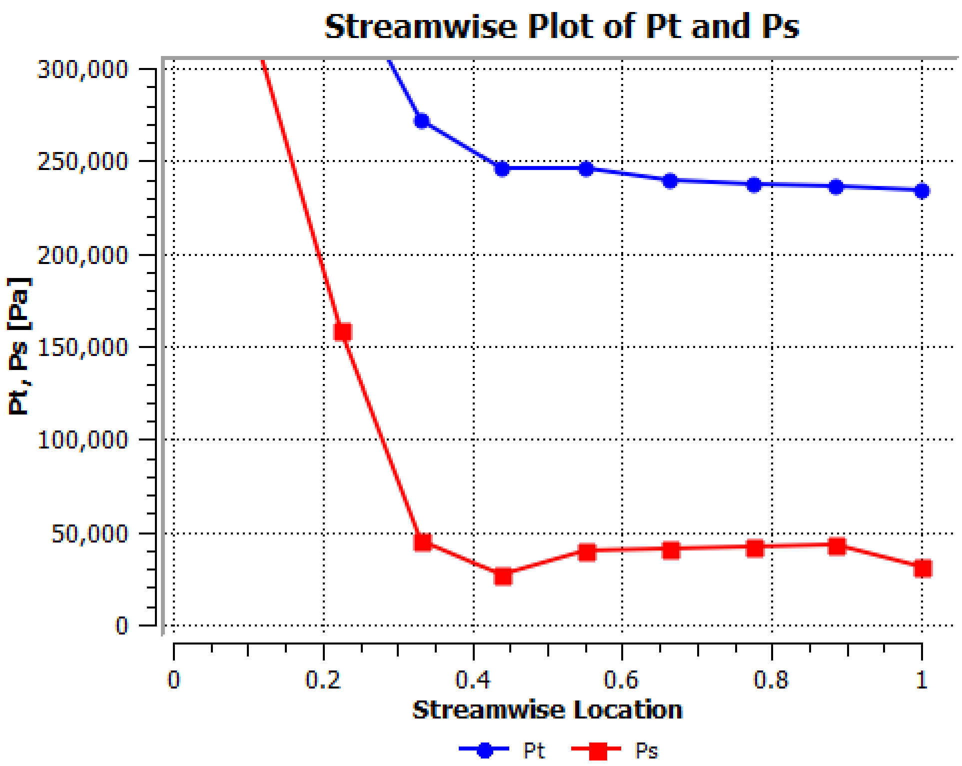

Figure 6.14 show the total pressure

Figure 6.15 shows the vapor in tip blade which matching with the actual eroded area in Roessiers turbine.

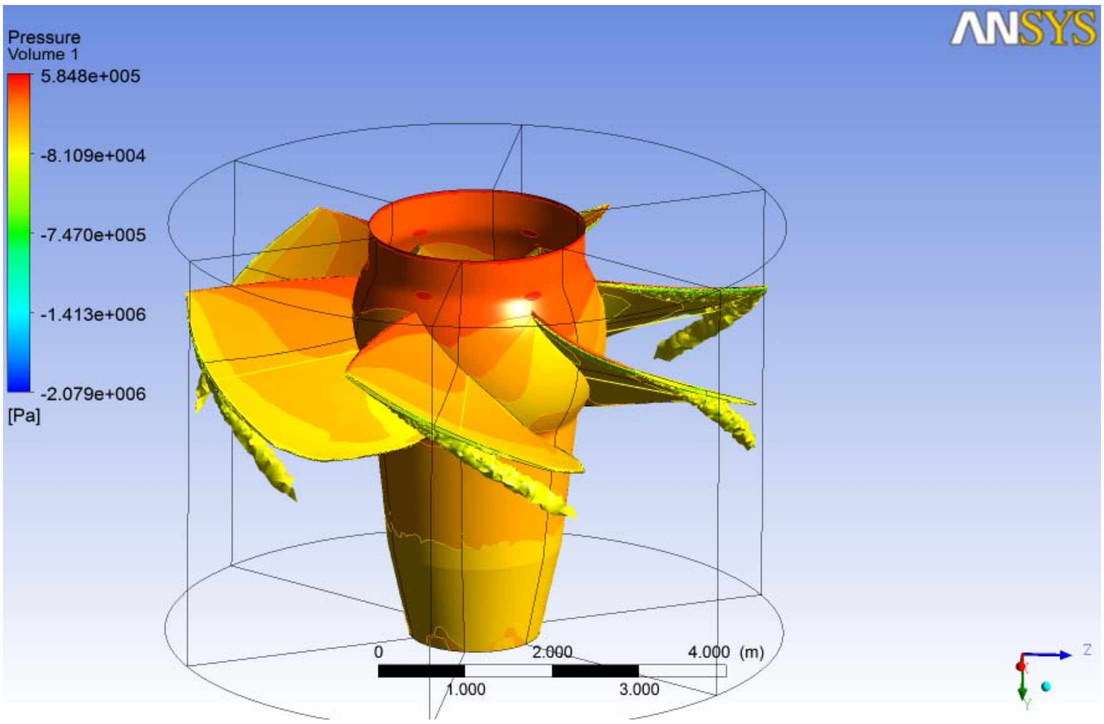

Figure 6.15 shows the vapor

6.3 Validation:

Validations of numerical results were measured and compared with Resoreis power plant turbines in Blue Nile state in Sudan. The turbine blades were shown in Figure 1 as a typical example of CFX model. Which indicate the erosion of the tip blades, and the clearance is increased due to tip clearance vortices.

Fig 6.16 illustrate the erosion in tip

7.0 Conclusion:

The CFD analysis success to prove the presence of the tip vortex with shape and intensity which could cause a pitting at the same location as on the actual case in Rosseries runner blade quite accurately. The danger of the pitting is mainly linked to the tip vortex intensity.

Generally, the flow within the gap (Tip clearance) has mainly negative effects. It decreases the efficiency of the runner because the water flowing through the gap does not generate any torque on the runner blades. Another negative effect is the cavitations' pitting which results from above mentioned phenomena associated with the tip clearance.

The mentioned tip clearance causes two types of the cavitation. The first type is called tip clearance cavitation and it takes place on the tip of the blade. The pressure difference between pressure and suction side of the blade causes flow through the tip clearance. The velocity of the water in the gap could be very high which is linked to the decrease of the pressure. In the case that the pressure drops below the saturated vapour pressure, the tip clearance cavitation occurs.

8.0 References:

[1]Abhijit Date, AliakbarAkbarzadeh. 2009. Design and Cost Analysis of Low Head Simple Reaction Hydro Turbine for Remote Area Power Supply. ELSEVIER.Renewable Energy. 34: 409–415.

[2]https://mechanicalengineeringstudymaterial.wordpress.com/

[3]http://www.turbinesinfo.com/history-of-hydroelectric-power/

[4]Ramon Pichs and etc.,2012, Renewable Energy Sources and Climate Change Mitigation: Special Report of the intergovernmental panel on climate change, Cambridge University, Chapter 5 page 437.

[5] Hunter, Louis C.: A History of Industrial Power in the United States, 1780 – 1930. The

University Press of Virginia, 1979.ISBN 0-8139-0782-9 (v.1).

[6] https://mechanicalengineeringstudymaterial.wordpress.com/

[7]Douglas, J. F., Gasiorek, J. M. and Swaffield, J. A. (1995).*Fluid Mechanics* (3rd edn), Longman.

[8]**Çengel, Yunus A and Cimbala, John M.** *Fluid Mechanics Fundamentals and*

*Applications.*Singapore : McGraw Hill, 2010. ISBN 978-007-128421-9.

[8]Tushar K. Ghosh, Mark A. Prelas. 2011. Energy Resources and Systems. *Renewable Resources*.Volume 2, Springer Netherlands, Chapter3, Online ISBN 978-94-007-1402-1.

[9]Performance analysis of Kaplan turbine with semi-spiral case at large flow conditions

Authors: Liao, Weili; Zhao, Yaping; Zhao, Qianyun; Ruan, Hui; Luo, Xingqi

Source: Transactions of the Chinese Society of Agricultural Engineering, Volume 30, Number 17, 2014, pp. 86-92(7)

Publisher: Editorial Office of Transactions of the Chinese Society of Agricultural Engineering

[10]S.L. Dixon, Fourth edition 1998, Fluid Mechanics, Thermodynamics of Turbomachinery,Butterworth-Heinemann , on line ISBN 0 7506 7059 2

[11]DragicaJošt* – Andrej Lipej, Numerical Prediction of Non-Cavitating and Cavitating

Vortex Rope in a Francis Turbine Draft Tube,Strojniškivestnik - Journal of Mechanical Engineering 57(2011)6, 445-456.

[12] Ting, L. (1991). Viscous Vortical Flows. Lecture notes in physics. Springer-Verlag.ISBN 3-540-53713-9.

[13] Kida, Shigeo (2001). Life, Structure, and Dynamical Role of Vortical Motion in Turbulence

(http://www.igf.fuw.edu.pl/IUTAM/ABSTRACTS/Kida.pdf) (PDF).IUTAM Symposium on Tubes, Sheets and Singularities in Fluid Dynamics.Zakopane, Poland.

[14] Vortex - Wikipedia, the free encyclopedia.

[15]*Clancy, L.J. (1975). Aerodynamics. London: Pitman Publishing Limited.* ISBN 0-273-01120-0.

[15*]Hart, D., D. Whale, and others. 2007. “A Review of Cavitation-erosion Resistant Weld

Surfacing Alloys for Hydroturbines.”Eutectic Castolin Web News Publications.
[16]**Ferziger, J H and Perić, M.** *Computational Methods for Fluid Dynamics.*

Heidelberg : Springer, 2002. 3-540-42074-6.

[17]https://en.wikipedia.org/wiki/Computational_fluid_dynamics

[18]BjörnJedvik,2012,Evaluation of CFD Model for Kaplan DraftTube,Luleå University of Technology,MASTER'S THESIS.

[19]Nils ReidarB.Olen,Dec2000,ISBN 827598-044-5, CFD Algorithms for Hydraulic Engineering.

[20] George W. Collins, 2003,FundamentalNumericalMethods and Data Analysis

[21]**L motycak, A Skotak and R Kupcik**,Kaplan turbine tip vortex cavitation -analysis and prevention.